

AXISVM

USER'S MANUAL

Copyright

Copyright © 1991-2027 Inter-CAD Kft. of Hungary. All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, for any purposes.

Trademarks

AXISVM is a registered trademark of Inter-CAD Kft.
All other trademarks are owned by their respective owners.
Inter-CAD Kft. is not affiliated with INTERCAD PTY. Ltd. of Australia.

Disclaimer

The material presented in this text is for illustrative and educational purposes only and is not intended to be exhaustive or to apply to any particular engineering problem for design. While reasonable efforts had been made in the preparation of this text to assure its accuracy, Inter-CAD Kft. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Changes

Inter-CAD Kft. reserves the right to revise and improve its product as it sees fit. This publication describes the state of this product at the time of its publication and may not reflect the product at all times in the future.

Version

This is an International Version of the product that may not conform to corresponding standards in a respective country and is available solely on an "as is" basis.

Limited warranty

Inter-CAD Kft. makes no warranty, either expressed or implied, including but not limited to any implied warranties of merchantability or fitness for a particular purpose, regarding these materials.
In no event shall Inter-CAD Kft. be liable to anyone for special, collateral, incidental, or consequential damages in connection with or arising out of purchase or use of these materials. The sole and exclusive liability to Inter-CAD KFT., regardless of the form of action, shall not exceed the purchase price of the material described herein.

Technical support and services

If you have questions about installing or using the AXISVM, check this User's Manual first - you will find answers to most of your questions here. If you need further assistance, please contact your software provider.

CONTENTS

1. New features of AXISVM Version 27 15
2. How to use AXISVM 17
2.1 Hardware requirements 17
2.2 Protection and installation 18
2.2.1 The AXISVM Launcher 18
2.2.1.1 Troubleshooting and error reference 20
2.2.1.1.1 MYAXISVM authentication errors 20
2.2.1.1.2 License errors 21
2.2.1.1.3 Launch and execution errors 21
2.2.1.1.4 Update errors 22
2.2.2 Cloud based license (Sentinel CL) 22
2.2.3 Hardware key 25
2.2.3.1 Sentinel SuperPro 25
2.2.3.2 Sentinel HL 26
2.2.4 Software key (Sentinel SL) 28
2.2.5 Installation 30
2.2.6 Running add-ons 36
2.3 Getting started 36
2.4 AXISVM user interface 37
2.5 Using the cursor, the keyboard, the mouse 38
2.6 Keyboard shortcuts 39
2.7 Quick Menu 43
2.8 Dialog boxes 44
2.9 Table Browser 44
2.9.1 Spring characteristics 50
2.10 Report Maker 52
2.10.1 Report toolbar 54
2.10.2 Report 55
2.10.3 Edit 57
2.10.3.1 Template-based reports 58
2.10.3.2 Editing a template 59
2.10.3.3 Filter-based report 62
2.10.4 Drawings 62
2.10.5 Gallery 63
2.10.6 Gallery and Drawings Library Toolbars 63
2.10.7 Text Editor 63
2.11 Stories 64
2.12 Layer Manager 64
2.13 Drawings Library 64
2.14 Save to Drawings Library 64
2.15 Export current view as 3D PDF 64
2.16 The Icon Bar 65
2.16.1 Selection 66
2.16.2 Zoom icon bar 68
2.16.3 Views 69
2.16.4 Display mode 70
2.16.5 Color coding 74
2.16.6 Geometric transformations on objects 77
2.16.6.1 Translate 77
2.16.6.2 Rotate 78
2.16.6.3 Mirror 79
2.16.6.4 Scale 79
2.16.7 Workplanes 80
2.16.8 Structural grids 80
2.16.9 Guidelines 83
2.16.10 Geometry tools 84
2.16.11 Dimension lines, symbols and labels 85
2.16.11.1 Orthogonal dimension lines 85
2.16.11.2 Aligned dimension lines 87
2.16.11.3 Angle dimension 88
2.16.11.4 Arc length 89
2.16.11.5 Arc radius 89

2.16.11.6 Level and elevation marks....89
2.16.11.7 Text box....90
2.16.11.8 Object info and result text boxes....91
2.16.11.9 Isoline labels....92
2.16.11.10 Dimension lines for footing....93
2.16.12 Editing background layers....93
2.16.13 Renaming/renumbering....94
2.16.14 Clipping regions....95
2.16.15 Turntable....96
2.16.16 Drone....97
2.16.17 Parts....101
2.16.18 Sections....103
2.16.19 Virtual beams....107
2.16.20 Find....110
2.16.21 Display options....110
2.16.22 Model options....116
2.16.22.1 Grid and cursor....117
2.16.22.2 Editing....117
2.16.22.3 Drawing....118
2.16.22.4 Load check....119
2.16.23 Model info....120
2.17 Smart command line....120
2.18 Speed buttons....123
2.18.1 Object snap....123
2.19 Information windows....124
2.19.1 Coordinate window....124
2.19.2 Info window....124
2.19.3 Color coding....125
2.19.4 Color legend window....125
2.19.5 Perspective window tool....128
3. The Main Menu....129
3.1 File 129
3.1.1 New model....129
3.1.2 Previous revisions....130
3.1.3 Open....131
3.1.4 Save....131
3.1.5 Save as....131
3.1.6 Export....131
3.1.6.1 Export to DXF....131
3.1.6.2 Export to IFC....132
3.1.6.3 Export to Revit....133
3.1.6.4 Export to Tekla Structures....136
3.1.6.5 Export to Nemetschek Allplan....136
3.1.6.6 Export to SAF file format....137
3.1.6.7 Export to other applications....138
3.1.6.8 Export to AXISVM....139
3.1.7 Import....139
3.1.7.1 Importing DXF files....139
3.1.7.2 Importing IFC files....140
3.1.7.3 Importing Revit RAE files....142
3.1.7.4 Import from Tekla Structures....147
3.1.7.5 Importing PDF files....147
3.1.7.6 Importing SAF files....148
3.1.7.7 Importing AXISVM AXS files....149
3.1.7.8 Importing files from other applications....149
3.1.8 Parametric design....150
3.1.8.1 Grasshopper/Rhinoceros add-on....150
3.1.8.2 Dynamo/Autodesk plugin....153
3.1.9 Tekla Structures – AXISVM connection – TI module....154
3.1.9.1 Two-way connection for Tekla Structures versions 2021 and later....154
3.1.9.1.1 Transferring model from AXISVM to Tekla Structures....154
3.1.9.1.2 Transferring analysis model from Tekla Structures to AXISVM....157
3.1.9.1.3 Workflow between AXISVM and Tekla Structures....161
3.1.9.2 One-way connection for Tekla Structures versions before 2021....163
3.1.10 Page header....167
3.1.11 Print setup....167

3.1.12 Print 167
3.1.13 Printing from file 169
3.1.14 Model Library 170
3.1.15 Material Library 171
3.1.16 Cross-Section Library 177
3.1.16.1 Cross-Section Editor 180
3.1.17 Spring characteristics library 193
3.1.17.1 Node-to-node spring 194
3.1.17.2 Seismic isolators 198
3.1.18 Exit 199
3.2 Edit... 200
3.2.1 Undo 200
3.2.2 Redo 200
3.2.3 Repeat last command 200
3.2.4 Select All 200
3.2.5 Restore previous selection 201
3.2.6 Copy 201
3.2.7 Paste 201
3.2.8 Copy / paste options 201
3.2.9 Delete 202
3.2.10 Table Browser 203
3.2.11 Report Maker 203
3.2.12 Saving drawings and design result tables 203
3.2.13 Weight Report 203
3.2.14 Assemble structural members 204
3.2.15 Break apart structural members 204
3.2.16 Convert beams to shell model 204
3.2.17 Create shell model for nodal connection 205
3.2.18 Convert surface loads distributed over beams 205
3.2.19 Convert loads of the selected load panels to individual loads 205
3.2.20 Convert automatic references 205
3.2.21 Make parametric ribs independent 205
3.2.22 Update support parameters based on the actual footings 205
3.3 Settings 206
3.3.1 Display options 206
3.3.2 Model options 206
3.3.3 Layer Manager 207
3.3.4 Storeys 208
3.3.4.1 Shear centers of storeys 209
3.3.5 Guidelines 210
3.3.6 Structural Grid 210
3.3.7 Design codes 210
3.3.7.1 Basis of structural design EN 1990 211
3.3.7.2 Loads and combinations EN 1991 212
3.3.7.3 Design of reinforced concrete structures EN 1992 215
3.3.7.4 Design of steel structures EN 1993 217
3.3.7.5 Design of composite structures EN 1994 220
3.3.7.6 Design of timber structures EN 1995 221
3.3.7.7 Design of masonry structures EN 1996 222
3.3.7.8 Geotechnical design EN 1997 222
3.3.7.9 Seismic design EN 1998 223
3.3.8 Units and Formats 225
3.3.9 Gravitation 225
3.3.10 Stiffness reduction 226
3.3.11 Preferences 227
3.3.11.1 Data integrity 228
3.3.11.2 Colors 229
3.3.11.3 Controls 230
3.3.11.4 Graphic symbols 231
3.3.11.5 Fonts 232
3.3.11.6 Dialog windows 233
3.3.11.7 Default names 233
3.3.11.8 Editing 234
3.3.11.9 Meshing 235
3.3.11.10 Toolbar 236
3.3.11.11 Display 237

3.3.11.12 Parts 238
3.3.11.13 Load group defaults 239
3.3.11.14 Analysis 239
3.3.11.15 Report 241
3.3.11.16 Update 243
3.3.12 Keyboard shortcuts 244
3.3.13 Language 245
3.3.14 Report language 245
3.3.15 Toolbars to default position 245
3.3.16 Dialog boxes to default position 245
3.3.17 Dialog windows to the primary monitor 246
3.3.18 Restore all warnings 246
3.4 View 246
3.5 Plugins 247
3.6 Window 247
3.6.1 Property Editor 248
3.6.2 Information Windows 248
3.6.3 Background picture 249
3.6.4 Split horizontally 249
3.6.5 Split vertically 250
3.6.6 Special window layouts 250
3.6.7 Close Window 251
3.6.8 Changing label font size 251
3.6.9 Preventing labels from overlapping 251
3.6.10 Drawings Library 251
3.6.10.1 Export drawings to a 3D PDF file - PDF module 254
3.6.11 Save to Drawings Library 254
3.7 AI Tools 255
3.7.1 AI Help 255
3.7.2 AI Selection 255
3.7.3 AI Model building 255
3.7.4 AI AddOn Development 256
3.7.5 Running Python scripts 256
3.8 License 258
3.9 Help 260
3.9.1 Contents 260
3.9.2 Guides 260
3.9.3 AXISVM Home Page 260
3.9.4 AXISVM Update 261
3.9.5 MYAXISVM Log in 261
3.9.6 COM server information 261
3.9.7 About 262
3.9.8 Error messages and warnings 262
3.9.9 Release information 262
3.10 Main toolbar 263
4. The Preprocessor 264
4.1 Geometry 264
4.2 The Model Editor 265
4.2.1 Multi-window mode 265
4.3 Coordinate systems 266
4.3.1 Cartesian coordinate system 266
4.3.2 Polar coordinates 266
4.4 Coordinate window 267
4.5 Grid 268
4.6 Cursor step 268
4.7 Editing tools 268
4.7.1 Cursor identification (object snap) 268
4.7.2 Entering coordinates numerically 269
4.7.3 Measuring distance 270
4.7.4 Constrained cursor movements 270
4.7.5 Locking coordinates 271
4.7.6 Auto intersect 271
4.8 Geometry Toolbar 271
4.8.1 Node (Point) 272
4.8.2 Line 272
4.8.3 Arc 273

4.8.4 Horizontal division 273
4.8.5 Vertical division 274
4.8.6 Replace selected arcs with straight lines 274
4.8.7 Quad/triangle division 274
4.8.8 Intersect 275
4.8.9 Remove node 275
4.8.10 Remove intermediate nodes 275
4.8.11 Extend lines to meet another line or plane 275
4.8.12 Trim lines to meet another line or plane 276
4.8.13 Extend/cut lines to their point of intersection 277
4.8.14 Line division 277
4.8.15 Normal transversal 278
4.8.16 Intersect plane with the model 278
4.8.17 Intersect plane with the model and remove half space 278
4.8.18 Domain intersection 278
4.8.19 Purge unnecessary lines and nodes 278
4.8.20 Geometry check 278
4.8.21 Surface 279
4.8.22 Modify, transform 279
4.8.23 Delete 281
4.9 Finite Elements 281
4.9.1 Materials 282
4.9.2 Cross-sections 283
4.9.2.1 Browse cross-section libraries 284
4.9.3 Add borehole samples – SOIL module 286
4.9.4 Direct drawing of objects 288
4.9.5 Direct drawing of supports 289
4.9.6 Domain 290
4.9.6.1 Defining a normal domain 291
4.9.6.2 Composite ribbed domain 293
4.9.6.3 Hollow core domain 294
4.9.6.4 Parametric ribbed plates 295
4.9.6.5 Trapezoidal steel deck 295
4.9.6.6 Domain with a custom stiffness matrix 297
4.9.6.7 CLT domain 298
4.9.7 Hole 300
4.9.8 Domain operations 300
4.9.9 Soil modeling domain – SOIL module 301
4.9.10 Line elements 302
4.9.10.1 Truss 303
4.9.10.2 Beam 305
4.9.10.2.1 Eccentricity of beams and ribs 309
4.9.10.3 7 DOF beam element – 7 DOF module 313
4.9.10.4 Rib 315
4.9.11 Modeling of cold-formed steel purlins 318
4.9.12 Purlin group definition 320
4.9.13 Surface elements 323
4.9.13.1 Membrane 323
4.9.13.2 Plate 324
4.9.13.3 Shell 325
4.9.14 Nodal support 326
4.9.14.1 Support stiffness calculation 328
4.9.14.1.1 Calculation from column parameters 329
4.9.14.1.2 Estimating support stiffness from soil profile and footing parameters 329
4.9.15 Line support 333
4.9.15.1 Support stiffness calculation 335
4.9.15.1.1 Calculation from wall parameters 335
4.9.15.1.2 Estimating support stiffness from soil profile and footing parameters 335
4.9.16 Surface support 336
4.9.16.1 Support stiffness calculation 337
4.9.16.1.1 Estimating support stiffness from soil profile and footing parameters 337
4.9.17 Edge hinge 338
4.9.17.1 Dilatation joints 338
4.9.18 Shear field 340
4.9.18.1 Trapezoidal sheet 340

4.9.18.1.1 Trapezoidal sheet library 341
4.9.18.2 Sandwich panel 342
4.9.18.3 Liner tray 344
4.9.18.4 Custom 344
4.9.19 Rigid elements 344
4.9.20 Diaphragm 345
4.9.21 Spring 346
4.9.22 Gap 350
4.9.23 Link 351
4.9.24 Nodal DOF (degrees of freedom) 354
4.9.25 References 356
4.9.26 Creating model framework from an architectural model 359
4.9.27 Modify 362
4.9.28 Delete 363
4.10 Loads 363
4.10.1 Load cases, load groups 363
4.10.2 Load combinations 370
4.10.3 Nodal loads 373
4.10.4 Concentrated load on beam 374
4.10.5 Point load on domain or load panel 374
4.10.6 Distributed line load on beam/rib 375
4.10.7 Edge load 377
4.10.8 Domain / Load panel line load 378
4.10.9 Surface load 380
4.10.10 Domain / Load panel area load 381
4.10.11 Surface load distributed over line elements 383
4.10.12 Load panels 384
4.10.13 Snow load – SWG module 387
4.10.14 Wind load – SWG module 395
4.10.15 Simulated wind used in wind tunnel analysis – WIND module 406
4.10.16 Generating wind load from a CFD (Computational Fluid Dynamics) file 411
4.10.17 Fluid load 412
4.10.18 Self-weight 412
4.10.19 Fault in length (fabrication error) 412
4.10.20 Tension/compression 413
4.10.21 Thermal load on line elements 413
4.10.22 Thermal load on surface elements 413
4.10.23 Forced support displacement 414
4.10.24 Influence line 415
4.10.25 Seismic loads – SE1 module 416
4.10.25.1 Seismic load calculation according to Eurocode 8 416
4.10.25.2 Seismic load calculation according to Swiss SIA 261 422
4.10.25.3 Seismic load calculation according to Italian NTC 2018 422
4.10.25.4 Seismic load calculation according to Romanian P100-1 423
4.10.25.5 Seismic load calculation according to Dutch NPR 9998:2018 423
4.10.26 Pushover loads – SE2 module 423
4.10.27 Global imperfection 425
4.10.28 Geometric imperfections based on buckling shapes 425
4.10.29 Tensioning - PS1 module 426
4.10.29.1 Post-tensioning of beam and rib elements 426
4.10.29.2 Post-tensioning of domains 434
4.10.30 Moving loads 437
4.10.30.1 Moving loads on line elements 437
4.10.30.2 Moving loads on domains or load panels 439
4.10.31 Dynamic loads (for time-history analysis) – DYN module 440
4.10.31.1 Dynamic nodal load 445
4.10.31.2 Dynamic concentrated load on domain or load panel 446
4.10.31.3 Mesh-independent dynamic area load on domains or load panels 446
4.10.31.4 Dynamic support acceleration 446
4.10.31.5 Dynamic nodal acceleration 447
4.10.32 Fire effect on steel line elements – SD8 module 447
4.10.33 Fire effect on timber line elements – TD8 module 454
4.10.34 Fire effect on concrete line elements – RC8-B module 457
4.10.35 Fire effect on concrete domains – RC8-S module 459
4.10.36 Split loads into separate load cases 460
4.10.37 Nodal mass 461

4.10.38 Modify 461
4.10.39 Delete 461
4.11 Wind tunnel – WIND module 462
4.11.1 Structure modeling options 463
4.11.1.1 Exported elements 463
4.11.1.2 Shrinkwrap 463
4.11.1.3 Surface roughness 464
4.11.2 Wind tunnel simulation 464
4.11.2.1 Setting analysis parameters (simple) 465
4.11.2.2 Setting analysis parameters (detailed) 465
4.11.2.3 Wind tunnel model 465
4.11.2.4 Mesh 467
4.11.2.5 Solution control 470
4.11.3 Wind tunnel analysis 470
4.11.4 Flow analysis results 472
4.11.5 Defining streamline seeds and streamline animation 473
4.12 Mesh 474
4.12.1 Mesh generation 474
4.12.1.1 Meshing of line elements 474
4.12.1.2 Meshing of domains 474
4.12.1.3 Meshing of soil modeling domains 476
4.12.2 Mesh refinement 476
4.12.3 Checking finite elements 477
4.12.4 Select free edges 477
4.12.5 Delete all meshes 477
4.13 Construction stages – STG module 478
4.13.1 General overview of the concept of construction stages 479
4.13.2 Defining stages 481
4.13.3 Editing stages 483
4.13.4 Display options for stages 485
4.13.5 Defining stage loads 485
5. Analysis 488
5.1 Static analysis 489
5.1.1 Construction stage analysis – STG module 495
5.2 Vibration 498
5.2.1 Vibration response factor 501
5.3 Dynamic analysis 503
5.4 Buckling 505
5.5 Finite elements 506
5.6 Main steps of an analysis 510
5.7 Error messages 511
5.8 Analysis parameters 512
6. The Postprocessor 513
6.1 Static 513
6.1.1 Minimum and maximum values 520
6.1.2 Animation 521
6.1.3 Soil section result animation 522
6.1.4 Diagram display 523
6.1.5 Pushover capacity curves 526
6.1.5.1 Capacity curves according to Eurocode 8 527
6.1.5.2 Acceleration-Displacement Response Spectrum (ADRS) 527
6.1.5.3 Drift 528
6.1.6 Result tables 528
6.1.6.1 Section segment result tables 529
6.1.7 Displacements 530
6.1.7.1 Nonlinear calculation of total deflection (wtot) for RC plates 532
6.1.7.2 Relative displacements 533
6.1.8 Nodal velocities 534
6.1.9 Nodal accelerations 534
6.1.10 Truss/beam internal forces 534
6.1.11 Rib internal forces 536
6.1.12 Virtual beam internal forces 538
6.1.13 Surface element internal forces 538
6.1.14 Support internal forces 541
6.1.15 Internal forces of line to line link elements and edge hinges 542

6.1.16 Spring element internal forces 542
6.1.17 Truss, beam and rib element strains 542
6.1.18 Truss, beam and rib element strains at stress points 544
6.1.19 Surface element strains 544
6.1.20 Surface element strains at stress points 546
6.1.21 Solid element strains 546
6.1.22 Spring element deformations 547
6.1.23 Truss/beam/rib stresses 547
6.1.24 Surface element stresses 552
6.1.25 Solid element stresses 556
6.1.26 Spring element nonlinear results 557
6.1.27 Spring element dynamic results 558
6.1.28 Nodal equilibrium 558
6.1.29 Influence lines 559
6.1.30 Unbalanced loads 560
6.1.31 Results of construction stages 560
6.2 Buckling 561
6.2.1 Creating imperfect geometry from buckling shapes – IMP module 563
6.3 Vibration 564
6.4 Dynamic 566
6.5 R.C. Design 567
6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module 567
6.5.1.1 Calculation of orthogonal x/y reinforcement according to Eurocode 2 571
6.5.1.2 Calculation of orthogonal x/y reinforcement according to DIN EN and SIA 262 572
6.5.1.3 Calculation of skew reinforcement according to Eurocode 2 and SIA 262 573
6.5.1.4 Calculation of reinforcement of composite slabs with trapezoidal sheeting according to EC and SIA standards 575
6.5.2 Actual reinforcement 578
6.5.2.1 Reinforcement for surface elements and domains 579
6.5.2.2 Mesh-independent reinforcement 580
6.5.3 Verification of actual reinforcement of RC surfaces 582
6.5.4 Actual reinforcement against biaxial bending (column) 583
6.5.5 Nonlinear analysis of reinforced concrete beam and column elements 587
6.5.6 Nonlinear analysis of RC surfaces 589
6.5.7 Cracking 590
6.5.7.1 Cracking calculation according to Eurocode 2 591
6.5.8 Shear resistance and reinforcement calculation for plates and shells – RC3 module 592
6.5.8.1 Calculation according to Eurocode 2 592
6.5.8.2 Calculation according to SIA 262 593
6.5.9 Punching reinforcement parameters for multiple elements 594
6.5.10 Column reinforcement – RC2 module 594
6.5.10.1 Eccentricity calculation 600
6.5.10.2 Check of reinforced columns according to Eurocode 2 (bending with axial force) 602
6.5.10.3 Check of reinforced columns according to SIA 262 (bending with axial force) 604
6.5.10.4 Shear and torsion check of reinforced concrete columns 605
6.5.10.5 Shear and torsion check according to Eurocode 2 607
6.5.10.6 Shear and torsion check according to SIA 262 608
6.5.10.7 Capacity design: calculation the design value of shear force according to Eurocode and SIA 609
6.5.11 Composite column design – RC2 module 610
6.5.11.1 Design check according to Eurocode and SIA standards 612
6.5.12 Beam reinforcement parameters (uniaxial bending) 613
6.5.13 Beam reinforcement design – RC2 module 618
6.5.13.1 Steps of beam reinforcement design 619
6.5.13.2 Checking calculated beam reinforcement 621
6.5.13.3 Checking actual beam reinforcement 624
6.5.13.4 Beam reinforcement according to Eurocode 626
6.5.13.5 Beam reinforcement according to SIA 262 628
6.5.13.6 Result components of beam reinforcement 629
6.5.14 Fire design of reinforced concrete elements – RC8 modules 629
6.5.14.1 Fire design of reinforced concrete columns and beams – RC8-B module 630
6.5.14.2 Fire design of reinforced concrete surfaces – RC8-S module 631
6.5.15 Punching analysis – RC3 module 632
6.5.15.1 Punching analysis according to Eurocode 634
6.5.15.1.1 Punching analysis of columns according to Eurocode 634
6.5.15.1.2 Punching analysis of wall ends and corners according to Eurocode 638
6.5.15.2 Punching analysis according to SIA 262 641
6.5.15.2.1 Punching analysis of columns according to SIA 262 641

6.5.15.2.2 Punching analysis of wall ends and wall corners according to SIA 262 644
6.5.15.3 Punching result components 645
6.5.16 Footing design – RC4 module 646
6.5.16.1 Pad footing design 646
6.5.16.2 Strip footing design 657
6.5.16.3 Soil profile editor 658
6.5.16.4 Soil profile library 659
6.5.17 Design of reinforced concrete cores and walls – RC5 module 660
6.5.17.1 Defining reinforcement 665
6.5.17.2 Check reinforcement 668
6.5.17.3 Calculation of eccentricity 669
6.5.17.3.1 Check reinforcement according to Eurocode 2 (bending with axial force) 670
6.5.17.3.2 Check reinforcement according to SIA 262 (bending with axial force) 672
6.5.17.4 Shear verification of reinforced concrete walls 673
6.5.17.4.1 Shear and sliding shear check according to Eurocode 2 674
6.5.17.4.2 Shear and sliding shear check according to SIA 262 675
6.5.17.5 Results 676
6.5.18 Stress-strain analysis of reinforced concrete structures – RC6 module 678
6.5.18.1 Stress-strain analysis of reinforced concrete sections 678
6.5.18.1.1 Stress-strain analysis parameters 680
6.5.18.1.2 Stress-strain analysis 682
6.5.18.1.3 Results 683
6.5.18.2 Stress-strain analysis of surface elements 684
6.6 Steel design 686
6.6.1 Steel beam design according to Eurocode 3 – SD1 module 686
6.6.1.1 ULS design parameters 693
6.6.1.1.1 Buckling (flexural) 694
6.6.1.1.2 Lateral-torsional buckling 698
6.6.1.1.3 Lateral supports 699
6.6.1.2 SLS design parameters 702
6.6.1.3 Diagrams and design calculations 704
6.6.1.4 Steel cross-section optimization - SD9 module 705
6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 -SD3 module 708
6.6.2.1 ULS design parameters 716
6.6.2.1.1 Connection verification 718
6.6.2.1.2 Buckling (flexural) 718
6.6.2.1.3 Lateral-torsional buckling 719
6.6.2.1.4 Lateral supports 719
6.6.2.2 SLS design parameters 719
6.6.2.3 Diagrams and design calculations 720
6.6.2.4 Steel cross-section optimization - SD9 module 721
6.6.3 Steel beam fire design according to Eurocode 3 – SD8 module 721
6.6.3.1 Fire design of hot-rolled steel design members 723
6.6.3.2 Fire design of cold-formed, thin-walled steel design members 730
6.6.4 Steel connection design - SC1 module 730
6.7 Timber design 732
6.7.1 Timber beam design – TD1 module 732
6.7.1.1 ULS design parameters 737
6.7.1.2 SLS design parameters 740
6.7.2 Timber beam fire design – TD8 module 743
6.7.3 Timber cross-section optimization – TD9 module 746
6.8 Design of CLT domains – CLT modul 747
6.9 Masonry wall design – MD1 module 750
6.9.1 Complex analysis of unreinforced masonry walls 750
6.9.1.1 Definition of masonry walls and their parameters 751
6.9.1.2 Parameters of a masonry wall strip 754
6.9.1.3 Check domain as a masonry wall 755
6.9.1.4 Design check of masonry walls, definition of design- and joint parameters 755
6.9.1.5 Display of masonry walls in the model 762
6.9.1.6 Results 763
6.9.1.7 Principles of checking 768
6.9.1.8 Design check according to Eurocode 6 768
6.9.1.9 Design check according to NTC 772
6.9.1.10 Design check according to SIA 775

7. Programming AXISVM – COM module 779
8. AXISVM Viewer and Viewer Expert 780
9. MYAXISVM – AXISVM Trial, Student, Viewer and Light version request 781
10. Examples 782
10.1 Linear static analysis of a steel plane frame 782
10.2 Geometric nonlinear static analysis of a steel plane frame 783
10.3 Buckling analysis of a steel plane frame 784
10.4 Vibration analysis (I-Order) of a steel plane frame 785
10.5 Vibration analysis (II-Order) of a steel plane frame 786
10.6 Linear static analysis of a reinforced concrete cantilever 787
10.7 Linear static analysis of a simply supported reinforced concrete plate 788
10.8 Linear static analysis of a clamped reinforced concrete plate 789
11. References 790

Available modules for AXISVM **27**

REINFORCED CONCRETE STRUCTURES	
RC1 – *Surface reinforcement parameters and reinforcement calculation*	6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module
RC2 – *Column reinforcement*	6.5.10 Column reinforcement – RC2 module 6.5.11 Composite column design – RC2 module
RC2 – *Beam reinforcement*	6.5.13 Beam reinforcement design – RC2 module
RC3 – *Punching analysis and shear capacity calculation*	6.5.15 Punching analysis – RC3 module
RC4 – *Pad & strip foundation design and geotechnical checks*	6.5.16 Footing design – RC4 module
RC5 – *Design of reinforced concrete cores and walls*	6.5.17 Design of reinforced concrete cores and walls – RC5 module
RC6 – *Stress-strain analysis of reinforced concrete structures*	6.5.18 Stress-strain analysis of reinforced concrete structures – RC6 module
RC8-B / RC8-S – *Fire design of reinforced concrete structures*	6.5.14 Fire design of reinforced concrete elements – RC8 modules
PS1 – *Analysis of post-tensioned beams and surfaces*	4.10.29 Tensioning - PS1 module
STEEL STRUCTURES	
SD1 – *Steel beam design*	6.6.1 Steel beam design according to Eurocode 3 – SD1 module
SD3 - *Design of cold-formed, thin-walled steel purlins and wall studs*	6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 -SD3 module
SD8 – *Steel beam fire design*	6.6.3 Steel beam fire design according to Eurocode 3 – SD8 module
SD9 – *Steel cross-section optimization*	6.6.1.4 Steel cross-section optimization - SD9 module
7DOF – *Beam elements with 7 degrees of freedom*	4.9.10.3 7 DOF beam element – 7 DOF module
SC1 – *Steel connections design*	6.6.4 Steel connection design - SC1 module
TIMBER STRUCTURES	
TD1 – *Timber beam design*	6.7.1 Timber beam design – TD1 module
TD8 – *Timber beam fire design*	6.7.2 Timber beam fire design – TD8 module
TD9 – *Timber cross-section optimization*	6.7.3 Timber cross-section optimization – TD9 module
CLT –*Cross laminated timber panel analysis and design*	6.8 Design of CLT domains – CLT modul
MASONRY STRUCTURES	
MD1 – *Masonry wall design*	6.9 Masonry wall design – MD1 module
MODULES FOR SEISMIC ANALYSIS	
SE1 – *Modal response spectrum analysis*	4.10.25 Seismic loads – SE1 module
SE2 – *Pushover analysis*	4.10.26 Pushover loads – SE2 module
DYN – *Time history analysis*	4.10.31 Dynamic loads (for time-history analysis) – DYN module
MODULES FOR DYNAMIC ANALYSIS	
FFA – *Calculation of vibration response factor for slabs*	5.2.1 Vibration response factor
DYN – *General dynamic analysis*	5.3 Dynamic analysis

GENERATION OF METEOROLOGICAL LOADS	
WIND –*Determination of wind loads through digital wind tunnel simulation*	4.11 Wind tunnel – WIND module
SWG – *Automatic generation of snow and wind loads*	4.10.13 Snow load – SWG module 4.10.14 Wind load – SWG module
BIM COLLABORATION	
SAF, IFC, TI, ALP, REV – *Connections with other CAD software*	3.1.6 Export, 3.1.7 Import
GENERAL PURPOSE MODULES	
STG – *Construction stages*	4.13 Construction stages – STG module 5.1.1 Construction stage analysis – STG module 6.1.31 Results of construction stages
SOIL – *Modelling the interaction between soil and structure*	4.9.3 Add borehole samples – SOIL module 4.9.9 Soil modeling domain – SOIL module
IMP – *Geometric imperfections based on buckling shapes*	4.10.28 Geometric imperfections based on buckling shapes

1. New features of AXISVM Version 27

GENERAL

AXISVM Launcher	2.2.1 The AXISVM Launcher
AXISVM themes (Classic / Dark / Light)	3.3.11.2 Colors
Mouse control profiles	3.3.11.3 Controls
Separate font for the Coordinate window	3.3.11.5 Fonts
Light direction, brightness, cast shadow settings in rendered view	2.16.4 Display mode
AI Tools	3.7 AI Tools

BIM AND LINKS TO OTHER PROGRAMS

New features in parametric design: GrasshopperToAXISVM V9.0	3.1.8.1 Grasshopper/Rhinoceros add-on
SAF 2.2.0 export / import	3.1.6.6 Export to SAF file format 3.1.7.6 Importing SAF files

ELEMENTS

New settings for material and cross-section tables *Display only the used materials / cross-sections* *Display only materials / cross-sections used in active parts or selection*	4.9.1 Materials 4.9.2 Cross-sections
Preset buttons to speed up the setting of stiffness components	4.9.13 Surface elements
Updating support parameters based on the actual footings imports the critical load used in settlement calculation if the stiffness was calculated from an estimated load	3.2.22 Update support parameters based on the actual footings
Modeling of cold-formed steel purlins	4.9.11 Modeling of cold-formed steel purlins
Purlin group panels to generate purlin layouts	4.9.12 Purlin group definition
Purlin and purlin fastener statistics	3.2.13 Weight Report
Shear field to include the stiffening effects of claddings	4.9.18 Shear field
Trapezoidal sheet library	4.9.18.1.1 Trapezoidal sheet library
Add / Overwrite option for custom CLT panel layer structures	4.9.6.7 CLT domain

LOADS

Optional abbreviation for load cases to display in critical combinations	4.10.1 Load cases, load groups
Critical load group combinations allow Excluded status for permanent groups as well	4.10.1 Load cases, load groups
*Shrinkwrap algorithm to speed up the convergence of wind tunnel simulations (**WIND** module)*	4.11.2 Wind tunnel simulation
*Surface roughness settings for wind tunnel simulation (**WIND** module)*	4.11.1.3 Surface roughness
*Wind tunnel bottom can be set higher than the terrain level (**WIND** module)*	4.11.2.3 Wind tunnel model
*Signed SRSS-like combination of modal responses in seismic analysis (**SE1** module)*	4.10.25 Seismic loads – SE1 module

CONSTRUCTION STAGES

*Stiffness components of elastic bedding, surface supports, edge hinges, line-to-line interface elements can be changed in construction stages (**STG** module)*	4.13.3 Editing stages
*Permanent load cases can also be deactivated (**STG** module)*	4.13.5 Defining stage loads
*Envelope for critical combination results in stages (**STG** module)*	

ANALYSIS

Highly optimized MKL / PARDISO solver	**3.3.11.14 Analysis**
ARPACK / PARDISO optimized solver for eigenvalues	**3.3.11.14 Analysis**
A constant load case can be specified for force- or displacement-controlled nonlinear analysis	**5.1 Static analysis**

RESULTS

Compound envelopes can be created from linear, nonlinear, dynamic and construction stage results	**6.1 Static**
Tables of regular and compound envelopes	**6.1 Static**
Result animations can be combined with turntable animation	**6.1.2 Animation**

DESIGN

*Design of cold-formed, thin-walled steel purlins and wall studs according to Eurocode 3 (**SD3** module)*	**6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 - SD3 module**
*Reinforced concrete beam design for multiple beams (**RC2** module)*	**6.5.13.6 Result components of beam reinforcement**
*Punching analysis of multiple columns, wall ends and corners (**RC3** module)*	**6.5.15.3 Punching result components**

REPORT

Drawing library items can be cropped	**3.6.10 Drawings Library**
Interactive report preview in the Report Maker	**2.10 Report Maker**
Exporting reports in DOCX format	**2.10 Report Maker**
Design calculation reports are exported line by line rather than page by page to allow deletion and rearranging parts	**2.10 Report Maker**
Optional design summary at the beginning of design calculation reports	**6.5.10 Column reinforcement – RC2 module** **6.5.13 Beam reinforcement design – RC2 module** **6.5.15 Punching analysis – RC3 module** **6.5.16 Footing design – RC4 module** **6.5.17 Design of reinforced concrete cores and walls – RC5 module** **6.6.1 Steel beam design according to Eurocode 3 – SD1 module** **6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 - SD3 module**

2. How to use AXISVM

Welcome to AXISVM!

AXISVM is a finite-element program AXISVM 27for the static, vibration, and buckling analysis of structures. It was developed by and especially for civil engineers. AXISVM combines powerful analysis capabilities with an easy to use graphical user interface.

Pre-processing

Modeling: geometry tools (point, lines, surfaces); automatic meshing; material and cross-section libraries; element and load tools, import./export CAD geometry (DXF); interface to architectural design software products like Graphisoft's ArchiCAD via IFC to create model framework directly.
At every step of the modeling process, you will receive graphical verification of your progress. Multi-level undo/redo command and on-line help is available.

Analysis

Static, vibration, and buckling

Postprocessing

Displaying the results: deformed/undeformed shape display; diagram, and iso-line/surface plots; animation; customizable tabular reports.
After your analysis, AXISVM provides powerful visualization tools that let you quickly interpret your results, and numerical tools to search, report, and perform further calculations using those results. The results can be used to display the deformed or animated shape of your geometry or the isoline/surface plots. AXISVM can liAXISVM 27arly combine or envelope the results.

Reporting

Reporting is always part of the analysis, and a graphical user interface enhances the process and simplifies the effort. AXISVM provides direct, high quality printing of both text and graphics data to document your model and results. In addition data and graphics can be easily exported (DXF, BMP, JPG, WMF, EMF, RTF, HTML, TXT, DBF).

2.1 Hardware requirements

The table below shows the minimum/recommended hardware and software requirements, so you can experience maximum productivity with AXISVM.

Recommended configuration

16 GB RAM
50 GB of free hard disk space
17" monitor (or larger), at least 1920x1080 pixels
A dual or multi-core processor over 2 GHz
Windows 11 64-bit operating system
Mouse or other pointing devices
Windows compatible laser or inkjet printer

Minimal configuration

4 GB RAM
10 GB of free hard disk space
15" monitor, at least 1024x768 resolution
Mouse
Windows 10 64-bit operating system

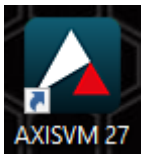
Supported operating systems

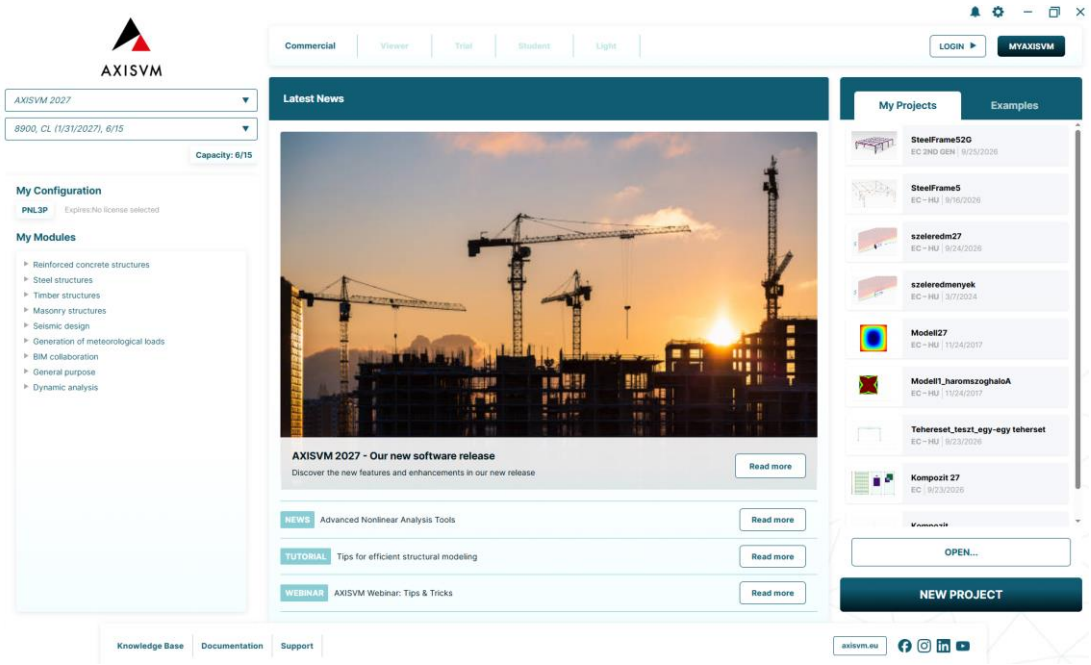
Windows11, Windows 10
AXISVM exclusively supports 64-bit operating systems.

2.2 Protection and installation

The software is protected by a hardware key, a software key or a cloud-based key.

2.2.1 The AXISVM Launcher





It is the central launcher of AXISVM integrating version selection, licence management, project handling and access to the latest technical information and updates.

Version selection

The Launcher detects and manages the various versions of AXISVM installed on the computer (starting with version 13). You can select the desired version and environment (commercial version / viewer / trial version / educational version / LT version) directly from the interface.

Licence management and authorisation checks

It supports cloud-based MYAXISVM authentication, as well as cloud-based, local hardware key and software key licences. The interface displays the number of licences currently in use and the total number available via a real-time capacity counter, expiry dates, the configuration and a list of the design modules associated with the licence. It manages both single-user and network licences.

My Projects

The panel on the right offers the option to open the most recently opened axs model files. The tooltip provides further information about the file.

Examples

The official AXISVM sample models are available here.

Integrated update management

The Launcher automatically notifies you of available program updates and new releases. When a new version or release becomes available, you can download and install it directly from the interface.

News feed and notification centre

The home page features the latest development articles, technical content and training information. The notification centre alerts you in good time to an impending licence expiry or the need for an internet connection.

☞ **To verify the licence, the programme must be launched at least once every 30 days whilst connected to the internet.**

Dragging the top bar, the Launcher window can be dragged to a different position, the usual icons allow maximized view or minimizing to the taskbar.

Top toolbar

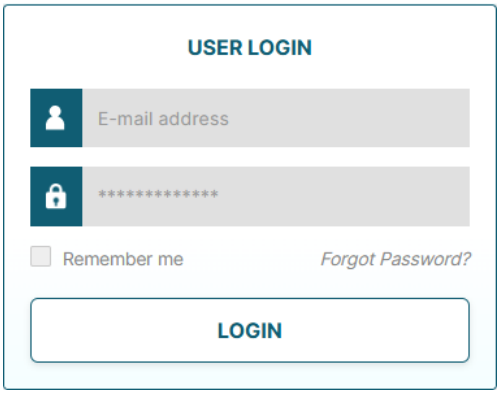
Displays the NOTIFICATIONS pop-up panel and indicates the number of unread messages.

Display the LAUNCHER SETTINGS pop-up panel.
In addition to the language, you can choose whether to display older versions of the programme, and whether the launcher should appear as a small icon in the Windows notification area (on the right-hand side of the taskbar, next to the clock) after launch.



Licence type selector, login

You can select the type of licence to launch.
On the right-hand side, click the LOGIN button to log in using the email address and password provided during MYAXISVM registration. If you tick the 'Remember me' option, the Launcher will automatically log you in each time it is launched.
If you have forgotten your password, clicking *'Forgot Password?'* on the login panel will open the MYAXISVM password recovery page. Once you have set a new password, you can log in via the launcher immediately.
The MYAXISVM button opens the user's personal web account in the browser.

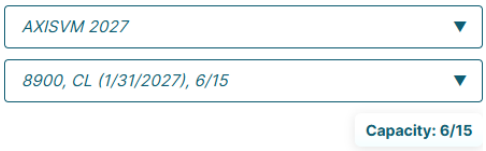


Upon successful login, the start panel loads the cloud licences associated with the account, updates the configuration and module information, the number of available licences and searches for downloadable sample models and updates.
The active profile appears in the header, along with a circular badge generated from the initial letters of the profile name.
If incorrect data is entered, the account is inactive, or in case of a network error, the system displays a pop-up message explaining the reason for the failure.

Log out

To log out, click on the circular badge and then on the LOG OUT button on the panel. This will cause the Launcher to release the reserved cloud licences, disable automatic login, and clear the version selector and module list on the control bar.

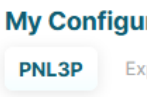
Control bar



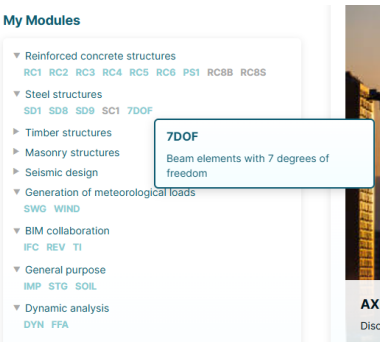
The first drop-down list shows the installed software versions, whilst the second lists the licences associated with the selected version. For cloud-based licences, it displays the number of seats currently in use real time and the total number of seats available to the company.

The Launcher compares the version installed on your computer with the latest available version. If a new service pack or update is available for the selected version, the live update panel will also appear; clicking the UPDATE button will start the download and installation.
If you have enabled the display of older versions in the LAUNCHER SETTINGS, you can also revert to a previous version (e.g. to accurately reproduce a previous calculation). In this case, the installer will restore the programme to the state it was in when that version was in use.

My Configuration



My Modules



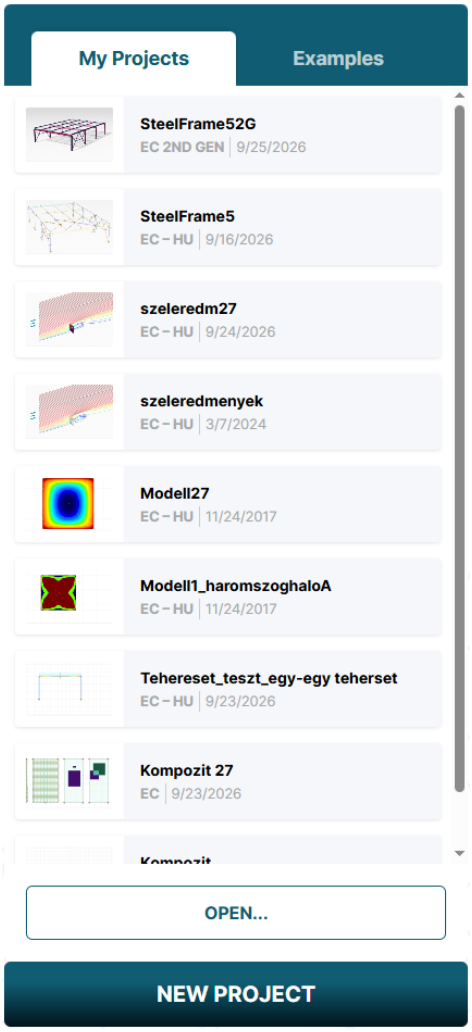
Besides the configuration corresponding to the selected version and licence, you can also check the validity period of time-limited licences or active plan packages. An approaching expiry date is indicated by an orange or red warning.
In the list of modules organised by category, the available modules are shown in turquoise, whilst the additional modules are shown in grey; their functions are explained in tooltips.

Latest news

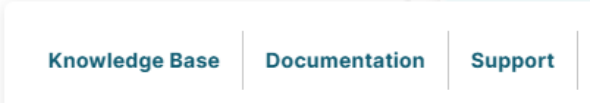
Provided you have an active internet connection, the central pane displays technical resources, lists of fixes and new features in the latest releases, development news and training events.

Model launcher

On the right-hand side, under the 'My Projects' tab, you can see a list of the most recently opened and edited models; the design code and the date of the last modification are also displayed. The tooltip provides further information.
The 'Examples' section contains AXISVM sample models; the available models can be saved directly to your computer and opened immediately using the 'Download' button.
At the bottom, the OPEN... button opens the standard Windows file open dialog. NEW PROJECT launches the selected version and licence type without loading a model. In both cases, the Launcher minimises to the taskbar.



At the bottom of the screen, you can access support channels. The *Knowledge Base* compiles frequently asked questions, modeling guides and articles; under 'Documentation', the official manual and various guides can be downloaded in PDF format; 'Support' provides direct access to the Inter-CAD customer service portal for reporting errors or asking for technical advice.



At the bottom right, you will find links to the official website and social media pages:



2.2.1.1 Troubleshooting and error reference

The AXISVM Launcher displays error dialogs or alerts when issues occur. This section details common error messages, their root causes, and recommended solutions.

2.2.1.1.1 MYAXISVM authentication errors

Errors encountered when signing in to MYAXISVM or refreshing an automated session token:

Invalid credentials

Message: *Incorrect email or password*
Root cause: A mistyped email address or incorrect password.
Solution: Verify your keyboard layout and Caps Lock status. If you cannot recall your password, click *Forgot Password?* in the login dialog to request a reset link via email.

Account unverified / suspended

Message: *Your account is not verified yet, please check your email.*
Account suspended, please contact the support team.
Root cause: The activation link in the confirmation email was not confirmed following registration, or the subscription is inactive for administrative reasons.
Solution: Check your email inbox (including Spam/Junk folders) for the automated confirmation message from InterCAD and click the activation link. If the account is suspended, contact InterCAD support to verify your account and contract status.

Session timeout / expired

Message: *Session expired, please log in again.*
Session ended, please log in again.
Root cause: The locally cached session token expired due to inactivity or a password change performed on another system.
Solution: Click LOGIN in the top right corner and re-enter your credentials. Check *Remember me* to enable automatic background login.

2.2.1.1.2 License errors

Issues encountered when validating operational privileges and design module access:

Seat limit reached

Message: *All seats in your license are currently in use, please try again later.*
Root cause: The maximum number of concurrent users allowed by the network or floating cloud license has been reached (e.g., capacity shows 5 / 5).
Megoldás: Check with colleagues to see if an idle AXISVM instance can be closed to free up a seat. Company administrators can review real-time seat assignments or purchase additional floating seats via the MYAXISVM web portal.

License expired

Message: *Your license has expired*
Root cause: A time-limited lease, evaluation trial, or active plan has passed its validity date.
Solution: Click on the MYAXISVM button in the header or footer to access subscription renewal options. If your organization holds multiple licenses, pick an active one using the license selector dropdown in the left sidebar.

License Server unreachable

Message: *License server is unreachable, please check your internet connection*
Root cause: Local network outage, restrictive corporate firewall rules, or an offline local network license service.
Solutions: Check your workstation's internet connection. For local network licenses, confirm that your workstation is connected to the office network (via LAN or VPN) and that the network license service is running on the host server.

2.2.1.1.3 Launch and execution errors

Errors encountered when launching AXISVM, opening models, or validating the runtime environment:

Already running

Message: *AXISVM is already running*
Maximum number of running AXISVM instances reached
Root cause: An instance of AXISVM is already active in the background, or an earlier process failed to shut down cleanly.
Solutions: Check the Windows taskbar and system tray, and click the tray icon to bring the existing window to the foreground. If the interface is not visible, open Windows Task Manager (Ctrl+Shift+Esc), locate the axisvm.exe process, terminate the task, and launch it again from the Launcher.

Administrator privileges required

Message: *Administrator privileges required, please run the launcher as administrator*
Access denied...
Root cause: The Launcher or the target AXISVM installation resides in a protected directory (e.g., C:\Program Files) where Windows requires elevated permissions for file access or configuration updates.
Solutions: Close the Launcher, right-click its desktop shortcut, and choose *Run as administrator*.
On corporate systems, contact your system administrator to ensure your user account has the necessary folder permissions.

Corrupted files

Message: *Corrupted files, please reinstall AXISVM.*
AXISVM not found, installed program is corrupted.
Root cause: An antivirus software quarantined the executable file due to a false positive, or an earlier update process was interrupted.
Solutions: Check your security software quarantine log and create an exception rule for the AXISVM installation directory. Run the installer for the affected AXISVM version and select the *Repair* option to restore missing or corrupted files.

Missing runtime dependencies

Message: *A required system component is missing, please reinstall AXISVM.*
Your graphics card or driver is unsupported...
Root cause: The system lacks the required Microsoft .NET Desktop Runtime or Microsoft Edge WebView2 runtime components, or the graphics driver does not support OpenGL 3.3.
Solutions: Install the latest 64-bit packages for Microsoft .NET Desktop Runtime and Microsoft Edge WebView2. Update your graphics adapter drivers to the latest version directly from the manufacturer (NVIDIA, AMD, or Intel).

2.2.1.1.4 Update errors

Issues encountered while using the integrated update manager:

Catalog unreachable

Message: *Update server unreachable, please check connection and try again.*
Root cause: Temporary internet disruption, an intervening corporate proxy, or scheduled maintenance on InterCAD update servers.
Solutions: Confirm your internet connection is working in a browser. Wait a few minutes, then reselect the target version in the version dropdown to query the update servers again.

Download interrupted

Message: *Update cancelled.*
An update download is already in progress, please wait for it to finish.
Root cause: The transfer was cancelled, network connectivity dropped during the download, or another update process is already running in the background.
Solutions: Verify you have a stable network connection, then click UPDATE to restart the download. If the download process remains locked, restart the AXISVM Launcher.

Installer launch failed

Message: *Installer downloaded, but failed to start, please run manually."*
Root cause: Windows User Account Control (UAC) blocked the automated installer start, or security software blocked executing the temporary .exe file.
Solutions: Navigate to the temporary downloads folder within your user profile directory and run the installer executable manually as an administrator. Relaunch the AXISVM Launcher with elevated administrator rights before initiating the update.

2.2.2 Cloud based license (Sentinel CL)

Sentinel CL license

Our software is delivered with a cloud-based license by default.
A cloud-based license is similar to an SL-type software license, but the license is not stored on the user's computer or network server, but on a logical server in the cloud.

The advantages of a cloud-based license are:

- The license can be accessed from anywhere over an internet connection, so it can be used from any number of computers (but not simultaneously, if it is not a multiple user network license).
- This key cannot be lost or damaged
- The license does not depend on computer hardware components, like the SL license

The disadvantage of a cloud-based license:

- Internet connection is required to access the license, so the program cannot be started offline. To avoid such situations, the license can be temporarily downloaded from the cloud to a specific computer. For details, see the description of the *License* menu item: **3.8 License.**

Cloud-based licences are managed and stored on a server rented by the software company. The server is not operated by the software company, so it cannot be held responsible for any loss of service due to downtime or server malfunctions, but it will do its best to restore service as soon as possible.
If you want to run the program on a location where web access is unavailable or unstable, or the program must be available at all times, you can download a license for a specific computer for temporary use. For more information, see **3.8 License.**
The program can be installed on any number of computers, but the number of copies that can be used simultaneously is limited by the number of licenses you have purchased. For information on the availability of licenses, see the *License* menu item in the *License Manager*.

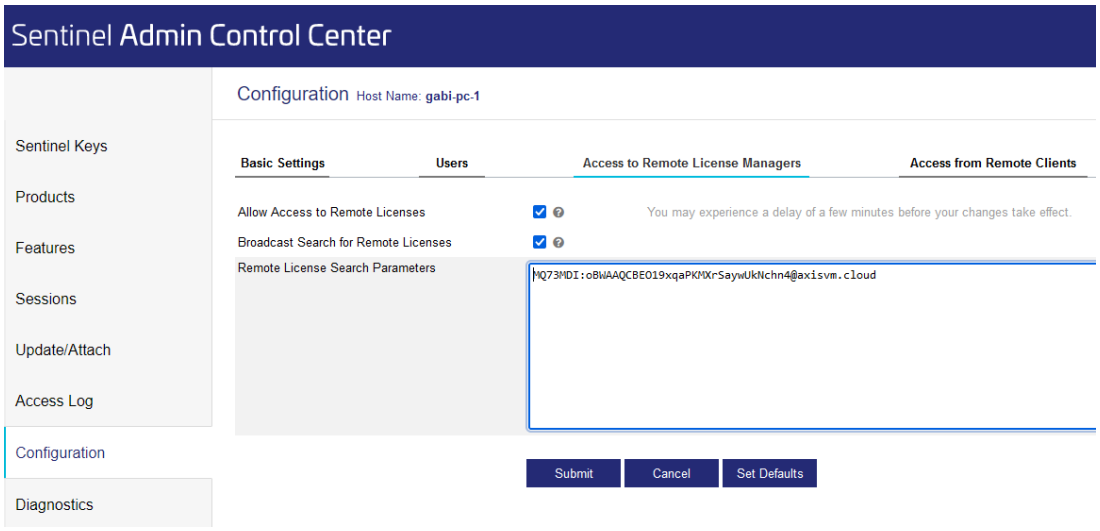
To access the license, you need the *Sentinel Admin Control Center* (ACC) service, which is installed automatically with AXISVM if the Sentinel driver is included in the installed components (it is always enabled by default).
To run the program, you also need the key file AXISVM_*[current_version]*_*[key number]*.key, which will be provided by the vendor. This file contains your registered username and the modules you have purchased.
After completing the installation and before starting AXISVM for the first time the cloud-based license must be registered. Open the confirmation email, click on the link, then click on the *Install* button on the webpage that is opened. If the registration was successful, and the license is available, the program can be started.
If you encounter an error accessing the license, check your firewall and ACC settings.

Checking the firewall

The ACC communicates with the cloud server via port 1947, which is automatically opened during installation. Check if this port is enabled for both TCP and UDP protocols on the firewall. After an operating system update the firewall settings may be changed, resetting the previous state. The solution is to reinstall the Sentinel Runtime Environment, as the installer run with administrator rights will reconfigure the firewall. The installer can be found in the Sentinel LDK folder under the AXISVM program folder as *InstallSentinelRuntime.exe*.

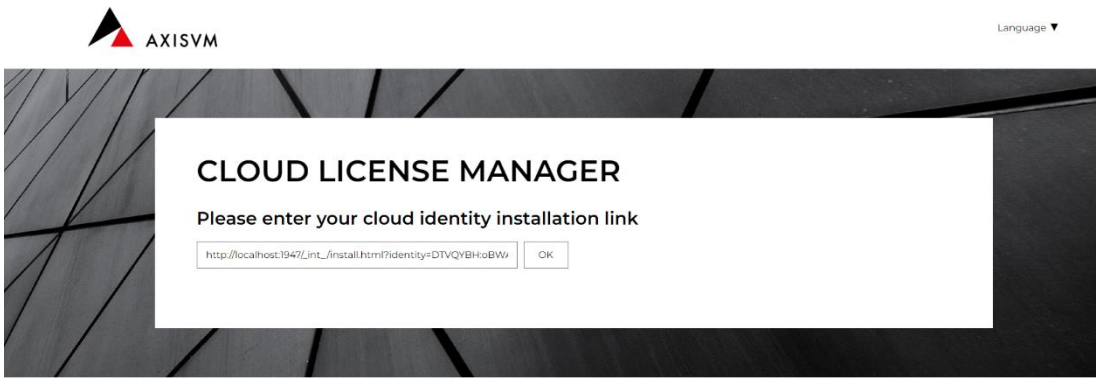
Checking the ACC

Type *localhost:1947* into your browser. This will bring up the ACC interface. Select *Configuration* from the menu on the left and *Access to Remote License Managers* at the top. The *Remote License Search Parameters* field should display your unique identifier.
If this field is blank, this identifier is missing. Ask your vendor for your unique identifier, then copy it here and press *Submit*. If the entry is successful, the ID will remain visible. If it disappears, the ID you entered is incorrect.
If the unique identifier appears in the field but the program does not start, either the cloud server is unavailable due to a service error or your license is disabled.
In this case, please contact your vendor.

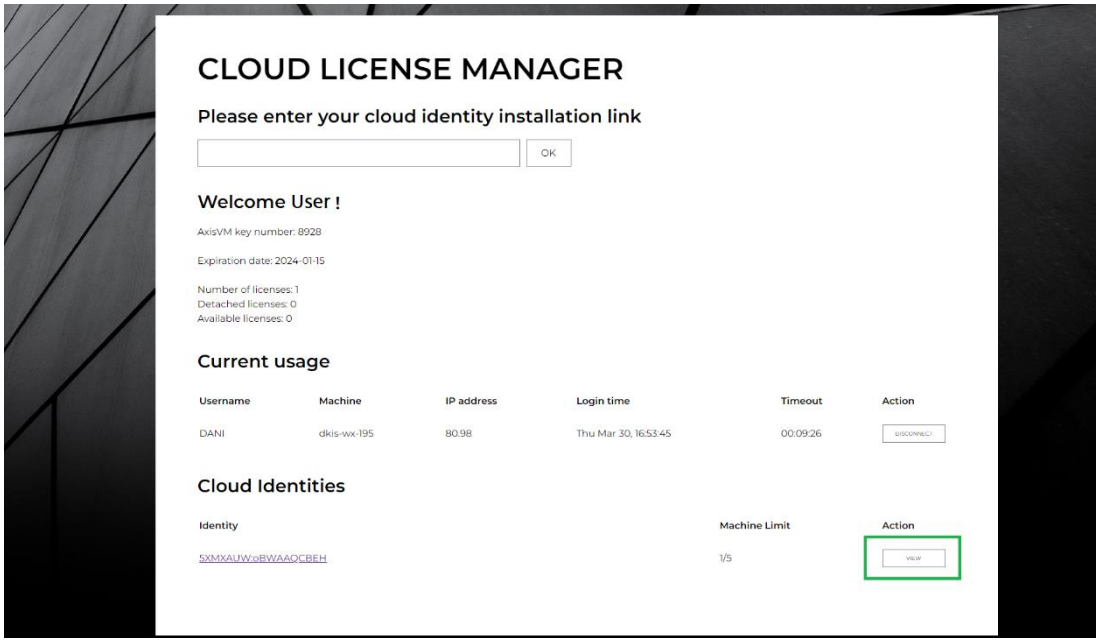


Cloud License Manager

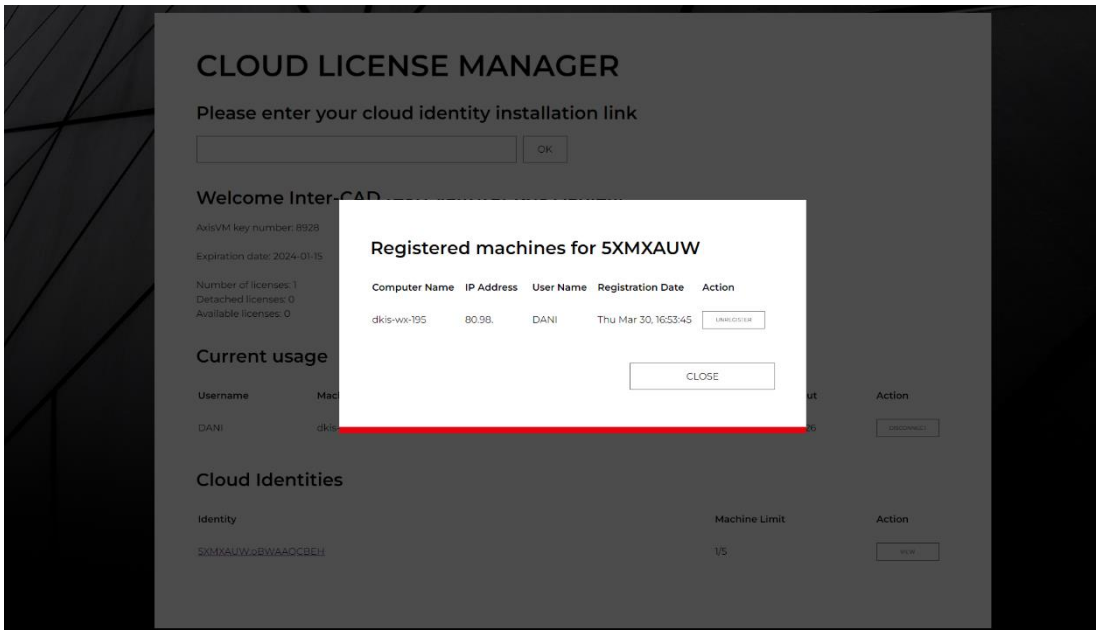
The Cloud License Manager is a dedicated management tool for AXISVM CL licenses.
It can be used to track and deactivate active sessions, list all identities associated with your license, and track and unregister the computers on which the identities have already been used.
It can be accessed at **cloud.axisvm.eu** where you will need to enter your Cloud Identity installation link that you received when you purchased your license.



After entering your correct installation link, you will be able to see basic information about your license: your key number, the time limit of your license (if there is one), the number of licenses that can be used at the same time, the number of licenses that have been detached, and the number of licenses that are available.
You can also see all the AXISVM sessions currently running on your Cloud license, which you can disconnect.



By clicking 'View' next to an Identity, you can see all the machines on which this Identity has been used on. You can deregister a computer if you don't want to use your license on it anymore.



2.2.3 Hardware key

Using a protection by hardware key the software can be installed on multiple computers but can be launched only if the hardware key is plugged in. Plug the key only after completing the installation as certain operating systems try to recognize the plugged device and this process may interfere with the driver installation.

2.2.3.1 Sentinel SuperPro

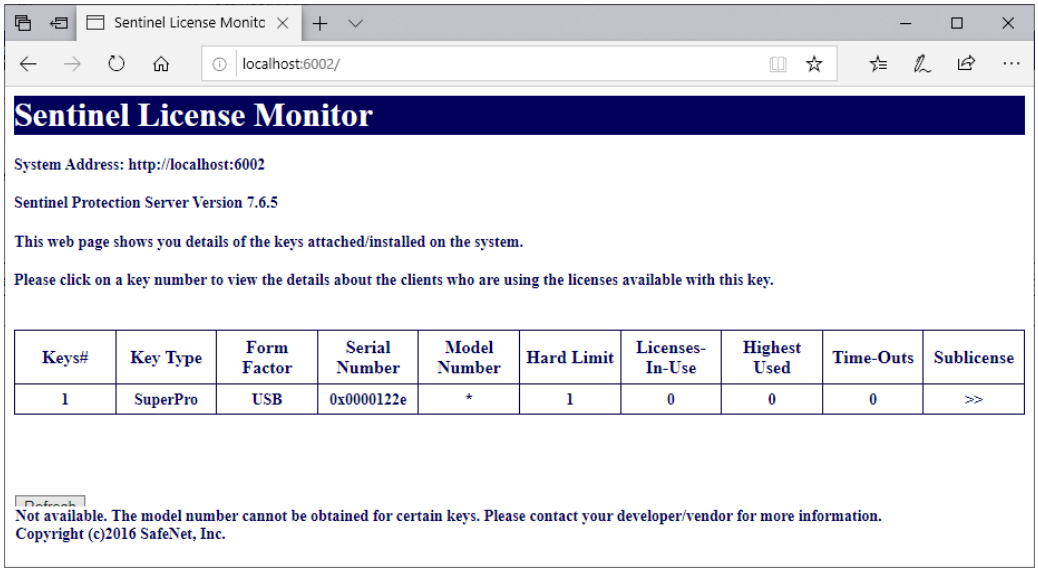
Super Pro hardware key

In earlier versions the most common protection device was the Sentinel SuperPro hardware key available as a single or a multi-user network license. A single license can be used on one computer only (on Windows Terminal Server networks on one terminal only).
The operation of the hardware key requires a driver. It is automatically installed by the AXISVM Installer but in case of a failure the installation can be done later.
After completing the AXISVM installation look for the *Sentinel* folder and run *Sentinel Protection Istaller* from there. If you have an installation USB flash drive run *Startup.exe* and choose ***Reinstall driver***.

Super Pro network hardware key

If you have a network license you must install the network key. In most cases AXISVM and the key are on different computers but to make the key available through the network the Sentinel driver must be installed on *both* computers using the *Sentinel Protection Installer*. Network keys can be used both on local networks (LAN) and remote networks (VPN).
Connect the key to the USB port of one of the computers. This way you select the AXISVM server.
After completing the installation a service called *Sentinel Protection Server* is launched to establish a connection between the hardware key plugged into the server machine and the client machines. Network licenses allow running as many AXISVM copies on the network as the purchased number of licenses. Launched copies are registered into the hardware key. Closing the copy will remove the registration making the license available on other machines. If the application becomes unresponsive for some reason the registration is not removed immediately. The license will be available only after 3 minutes when the lack of response is detected by the server.

The hardware key manufacturer provides a tool called *Sentinel License Monitor* to check the currently registered network licenses, the number of free licenses and remove the unresponsive licenses if necessary. To run this tool start you internet browser and type in the local network IP address of the server (i.e., the machine where the hardware key is plugged in) and append a reference to the port 6002, e.g.: **http://192.168.18.167:6002.** If you run the internet browser on the machine where the hardware key is plugged in you can type simply **http://localhost:6002.**



☞ **FIREWALL SETUP: Enable TCP / IP and UDP protocols on ports 6001 and 6002 on the firewall to ensure network operation.**

Access of Sentinel SuperPro network keys

This setting is required for a VPN connection.
If AXISVM is launched on a client machine it begins to search the network for available network keys checking each computer running Sentinel Pro Server regardless if the key is plugged or not. It may slow down the search process. To improve the connection speed, it is recommended to create an NSP_HOST environment variable on the client machine, specifying the local network IP address of the computer with the key, e.g.: NSP_HOST = 192.168.0.23.

In case of more than one network key it is possible to set the NSP_HOST1, ..., NSP_HOST5 environment variables identifying computers with keys. The maximum number of keys that can be handled this way is five.

☞ **To run AXISVM on any computer on the network SuperPro Server must be running on the server. If it stops all running AXISVM programs stop.**

2.2.3.2 Sentinel HL

Sentinel HL hardware key

The **Sentinel HL** key is a new kind of protection device that does not require driver installation. Plugging it into the USB port the Windows operating system identifies it as a HID (Human Interface Device).
Sentinel HL keys also have single user and network models. A single license can be used on one computer only (on Windows Terminal Server networks on one terminal only).

Sentinel HL network hardware key

Network keys can be used on local (LAN) or remote networks (VPN) as well. Connect the key to a machine to use it as a key license server and install AXISVM copies on client machines. Sentinel Runtime Environment must be installed both on the server and on all clients by launching *InstallSentinelRuntime* located in the *Sentinel LDK* folder under the AXISVM folder created during program installation.
The Sentinel Runtime Environment provides a connection between the hardware key plugged into the server machine and the client machines. If the environment is not running AXISVM cannot be launched on client machines.
To purchase further licenses for a network key run the *Admin Control Center* (see below), choose *Sentinel Keys* from the menu on the left and choose the appropriate network key from the list. Click on the *C2V* icon on the right to save a *c2v* file. Send this file to your distributor with your license order. Your distributor will send a *v2c* file with the additional licenses. Save it to your machine. To update the number of licenses choose *Update/Attach* from the menu on the left then browse for the *v2c* file you saved. Click the *Open* button to transfer license information to the network key. The new licenses will become available.

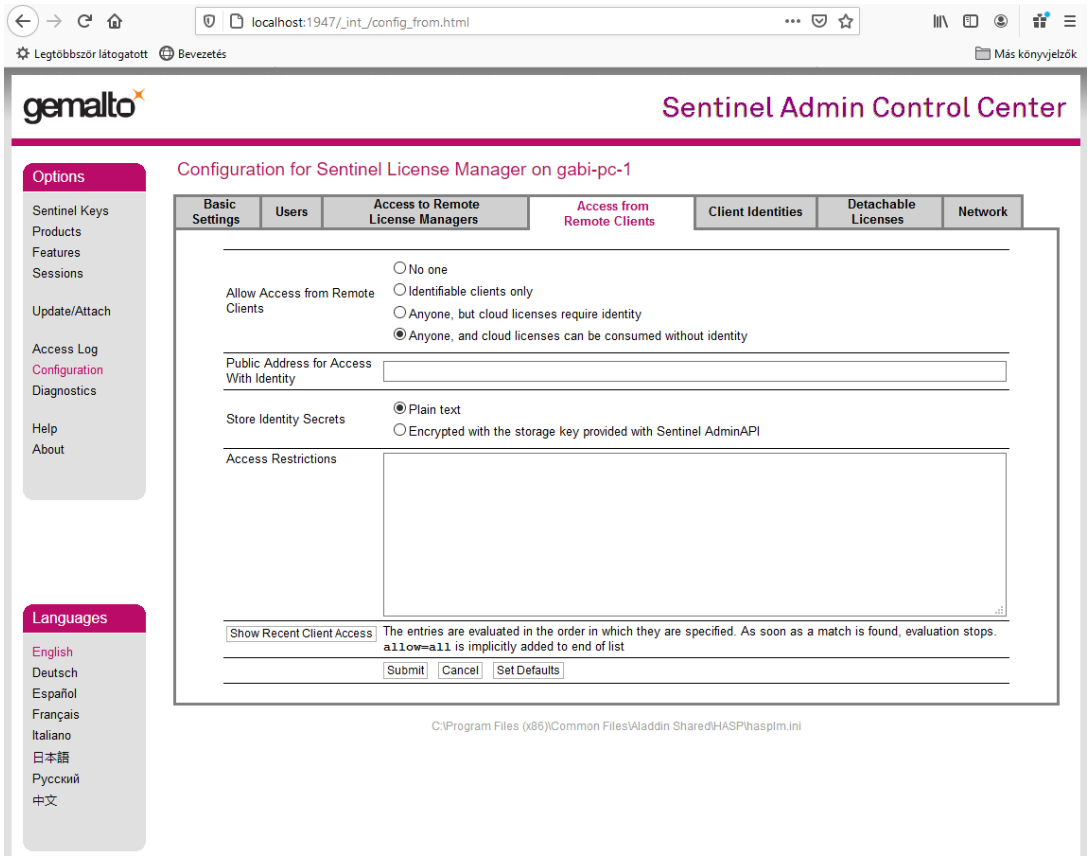
☞ **FIREWALL SETUP: Enable TCP / IP and UDP protocols on port 1947 on the firewall to ensure network operation.**

Configuration of server and client computers

To ensure the proper functioning of a Sentinel HL key, the Sentinel Admin Control Center must be configured on server and client machines.

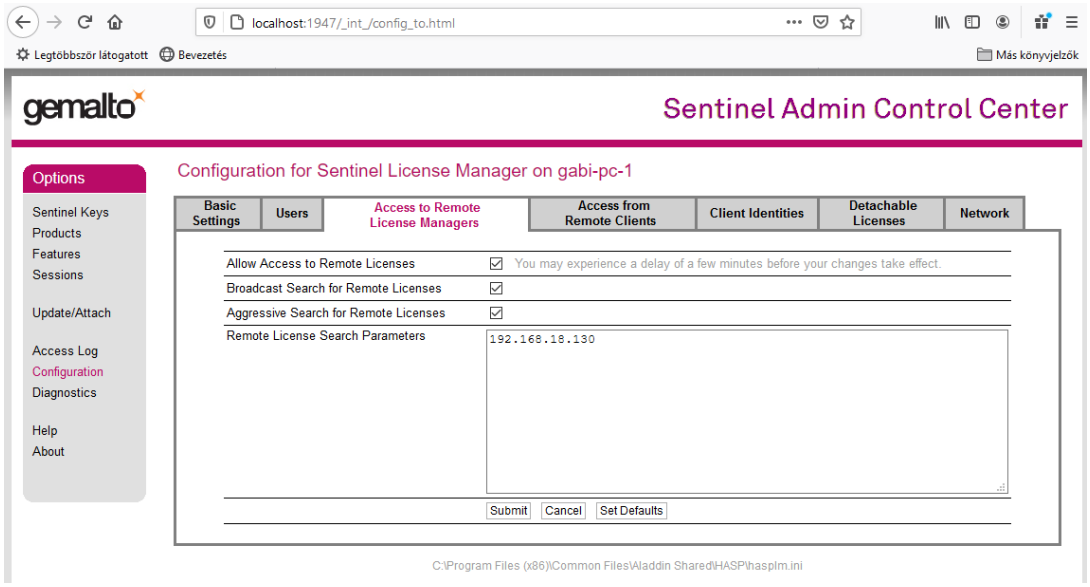
Server

Open the web browser and type localhost:1947 to get to the Sentinel Admin Control Center. Go to the Access from Remote Clients tab and set Anyone under Allow Access from Remote Clients.



Client

Open the web browser and type localhost:1947 to get to the Sentinel Admin Control Center. Go to the Access to Remote License Manager tab and enable Allow Access to Remote Licenses, Broadcast Search for Remote Licenses and Aggressive Search for Remote Licenses. Enter the IP address of the server to the Remote License Search Parameters field and click on the Submit button.
If both the server and the clients are properly configured, choosing Sentinel Keys from the Options list on the left displays a list of all Sentinel HL and SL keys installed.



2.2.4 Software key (Sentinel SL)

Software key protection by an installed license replaces the hardware key plugged into a USB port. Software key licenses are also available in a local or a network version. A local license allows running AXISVM on one computer, where the software key is installed. A network license allows a limited number of concurrent users on the local network.

Software keys can be transferred to another machine using a tool (see later). After transferring a software license, the program will run only on the other machine and fails to run on the first one.

A software license is linked to the hardware elements of the computer. Therefore, any change in the hardware (replacement of the motherboard, CPU, Ethernet card, hard drive, will invalidate the license. To prevent such loss, the license must be transferred temporarily to another machine **before** changing the hardware, replace the hardware element then transfer the license from the other machine

Sentinel SL software key

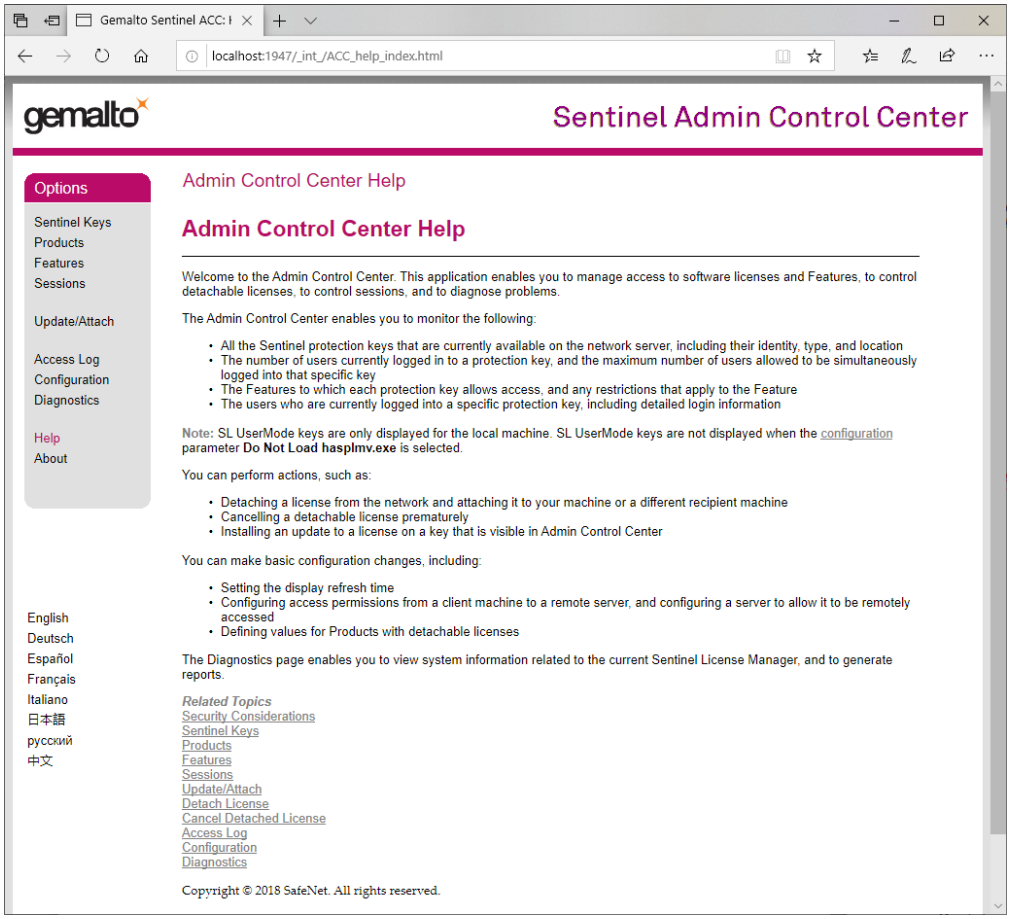
Sentinel SL is not a physical device attached to a port but a software key specific to the given hardware. It cannot be copied directly but can be transferred to another machine using the RUS tool of the protection package (see details below).
Software key may be a reasonable solution if storage, connection or protection of the physical key raises problems.
As the software key is stored on the hard drive / SSD of the machine, viruses, malware or flaws in the update of the operating system may corrupt the key. In this case another software key must be purchased.

Installing Sentinel HASP Run-time Environment

A software key requires the installation of the Sentinel Run-time Environment.
Download it from **www.axisvm.eu** (find it under Downloads / Software Updates / Sentinel driver) or look for *InstallSentinelRuntime.exe* in the Sentinel LDK folder of your AXISVM installation.

Sentinel Admin Control Center

To check the status of the enviroment run your internet browser on the server then type **http://localhost:1947.** The Sentinel Admin Control Center will be displayed.



The *Sentinel Admin Control Center* provides overview of the Sentinel HL and SL keys, their usage information and helps disconnecting clients from licenses. Choose *Sentinel Keys* form the menu on the left to display the list of available keys on the network.
If more than one hardware key is plugged and you are not sure which one belongs to AXISVM select *Products* from the menu on the left. The list of keys will be reorganized by protected products. Look for the one protecting AXISVM.

Requesting a Sentinel SL software license

A software key requires the installation of the Sentinel Run-time Environment. See *Installing Sentinel HASP Run-time Environment* above for the details. After completing the installation run the *RUS_AxisVM_En.exe* application from the *SentinelRuntime* subfolder in the AXISVM installation folder.
To request a software license, select the option *Installation of new protection key* at the bottom and click on the *Collect information* button.
Save the file with *c2v* extension generated by this application and send it attached to a license request email to your AXISVM distributor.

Applying a Sentinel SL software license

Software license is a *v2c* file sent by your distributor. Save it to your computer.
Run the *RUS_AxisVM_En.exe* application from the *SentinelRuntime* subfolder in the AXISVM installation folder. Choose the *Apply License File* tab and enter the file name with full path to the *Update File* field or click on the [...] button to browse.
Clicking on the *Apply Update* button activates the software license.
Check your license using the *Admin Control Center* tool (see above). Choose *Sentinel Keys* from the menu on the left, and check if your Sentinel SL license appears in the list.

Sentinel SL License transfer

A software license can be transferred to another machine. After completing the transfer AXISVM will run only on the other machine (works just like a hardware protection key removed and plugged into the other machine). To transfer the license run the *RUS_AxisVM_En.exe* application on the other machine too, select the *Transfer License* tab and generate a recipient information file *(*.id)* and copy that file to the first machine. Go back to the first machine, select the *Transfer License* tab, go to the *Read the recipient information file from* field, click on the [...] button and load the *.*id* file you have copied from the other machine, fill out the *Generate the license transfer file to* field then click on the *Generate License Transfer File* button. Copy the generated file with *h2h* extension to the other machine, go there, select the *Apply License File* tab, go to the *Update File* field, browse for the *h2h* file you have copied and click on *Apply Update*.

Configuration of the server and client machines

To ensure the proper functioning of a software key, the Sentinel Admin Control Center must be configured on server and client machines the same way described at the end of the previous chapter.

2.2.5 Installation

Preparing AXISVM installation on a Mac

Most professional structural analysis software programs support only Windows. The following methods will enable AXISVM use on a Mac:

1. **Boot Camp**

Apple's **Boot Camp** allows for **installing Windows on a Mac.** This will provide the best performance for running AXISVM on a Mac.

2. **Virtual Machines**

Software such as *Parallels* or **VMWare Fusion** use virtual machines to install Windows on a Mac, allowing AXISVM to run.

3. **Remote Desktop**

The **Remote Desktop software** on a Mac computer allows for remote access of a Windows computer running AXISVM.

Installation under Vista/Windows

If you have downloaded the installer application from our website, run it. Otherwise, connect the AXISVM installation USB flash drive. The Startup program starts automatically if the autoplay option is enabled. If autoplay is not enabled, click the Start button, and select *Run...*. Open the Startup.exe program from the USB flash drive. Select AXISVM *X8 Setup* and follow the instructions.

Installation under Windows Vista/Windows 7/Windows 8/Windows 10 operating systems:

- You need the latest Sentinel driver. You can download it from *www.axisvm.eu / Downloads – Software updates / Sentinel Driver update on the right tab)*
- Right click on the program icon after the installation of AXISVM
- Choose the *Properties* menu item from the Quick Menu.
- Select the *Compatibility* tab on the appearing dialog and turn on the *Run as administrator* checkbox.

By default the application and the example models will be installed on drive C: in

`C:\AxisVM_27` and `C:\AxisVM_27\Examples`

folders. You can specify the drive and the folders during the installation process. The setup program creates the AXISVM program group that includes the AXISVM application icon.

The application can be installed to the usual C: \ Program Files \ AxisVM_27 folder. However, in this case the *Run as administrator* property must be set for AxisVM_x64.exe \ IDTFConverter \ DTFConverter.exe. Find these files through *Start Menu / Computer,* right click on the files and choose *Properties* from the popup menu, go to the *Compatibility* tab, find *Privilege level* and turn on the above option. Users without administrative rights has to ask the administrator to set write access to the C: \ Program Files \ AxisVM_27 folder (see *Permissions* under the *Security* tab).

☞ **It is not recommended to install AXISVM under the c:\Program Files folder as the program placed there can be started only with administrative rights and there may be failures in running libraries like the 3D PDF generator.**

Silent installation

The software can be installed in silent mode using command line. The Setup program accepts optional command line parameters. These can be useful to system administrators, and to other programs calling the Setup program.
setup.exe /VERYSILENT /SUPPRESSMSGBOXES /NORESTART /Dir="C:\AxisVM_27" lang=nl NoIcons=0 types=64bits Tasks=extensions1,extensions2

AXISVM setup is made by Inno Setup installer. The most important optional command line parameters are listed below. For more information please see the Inno Setup help.

/SP-

Disables the This will install... Do you wish to continue? prompt at the beginning of Setup. Of course, this will have no effect if the DisableStartupPrompt [Setup] section directive was set to yes.

/SILENT, /VERYSILENT

Instructs Setup to be silent or very silent. When Setup is silent the wizard and the background window are not displayed but the installation progress window is. When a setup is very silent this installation progress window is not displayed. Everything else is normal so for example error messages during installation are displayed and the start-up prompt is (if you haven't disabled it with DisableStartupPrompt or the '/SP-' command line option explained above).

If a restart is necessary and the '/NORESTART' command isn't used (see below) and Setup is silent, it will display a Reboot now? message box. If it's very silent it will reboot without asking.

/SUPPRESSMSGBOXES

Instructs Setup to suppress message boxes. Only has an effect when combined with '/SILENT' and '/VERYSILENT'.

The default response in situations where there's a choice is:

- Yes in a 'Keep newer file?' situation.
- No in a 'File exists, confirm overwrite.' situation.
- Abort in Abort/Retry situations.
- Cancel in Retry/Cancel situations.
- Yes (=continue) in a DiskSpaceWarning/DirExists/DirDoesntExist/NoUninstallWarning/ExitSetupMessage/ConfirmUninstall situation.
- Yes (=restart) in a FinishedRestartMessage/UninstalledAndNeedsRestart situation.

5 message boxes are not suppressible:

- The About Setup message box.
- The Exit Setup? message box.
- The FileNotInDir2 message box displayed when Setup requires a new disk to be inserted and the disk was not found.
- Any (error) message box displayed before Setup (or Uninstall) could read the command line parameters.
- Any message box displayed by [Code] support function MsgBox.

/LOG

Causes Setup to create a log file in the user's TEMP directory detailing file installation and [Run] actions taken during the installation process. This can be a helpful debugging aid. For example, if you suspect a file isn't being replaced when you believe it should be (or vice versa), the log file will tell you if the file was really skipped, and why.

The log file is created with a unique name based on the current date. (It will not overwrite or append to existing files.)

The information contained in the log file is technical in nature and therefore not intended to be understandable by end users. Nor is it designed to be machine-parsable; the format of the file is subject to change without notice.

/LOG="filename"

Same as /LOG, except it allows you to specify a fixed path/filename to use for the log file. If a file with the specified name already exists it will be overwritten. If the file cannot be created, Setup will abort with an error message.

/NOCANCEL

Prevents the user from cancelling during the installation process, by disabling the Cancel button and ignoring clicks on the close button. Useful along with '/SILENT' or '/VERYSILENT'.

/NORESTART

Instructs Setup not to reboot even if it's necessary.

/LANG=language

Specifies the language to use. language specifies the internal name of the language as specified in a [Languages] section entry.

When a valid /LANG parameter is used, the Select Language dialog will be suppressed.

/DIR="x:\dirname"
Overrides the default directory name displayed on the Select Destination Location wizard page. A fully qualified pathname must be specified.

/GROUP="folder name"
Overrides the default folder name displayed on the Select Start Menu Folder wizard page. If the [Setup] section directive DisableProgramGroupPage was set to yes, this command line parameter is ignored.

/NOICONS
Instructs Setup to initially check the Don't create a Start Menu folder check box on the Select Start Menu Folder wizard page.

/TYPE=type name
Overrides the default setup type.

If the specified type exists and isn't a custom type, then any /COMPONENTS parameter will be ignored.

/COMPONENTS="comma separated list of component names"
Overrides the default component settings. Using this command line parameter causes Setup to automatically select a custom type. If no custom type is defined, this parameter is ignored.

Only the specified components will be selected; the rest will be deselected.

If a component name is prefixed with a "*" character, any child components will be selected as well (except for those that include the dontinheritcheck flag). If a component name is prefixed with a "!" character, the component will be deselected.
This parameter does not change the state of components that include the fixed flag.
Example:
Deselect all components, then select the "help" and "plugins" components:
/COMPONENTS="help,plugins"
Example:
Deselect all components, then select a parent component and all of its children with the exception of one:
/COMPONENTS="*parent,!parent\child"

/TASKS="comma separated list of task names"
Specifies a list of tasks that should be initially selected.

Only the specified tasks will be selected; the rest will be deselected. Use the /MERGETASKS parameter instead if you want to keep the default set of tasks and only select/deselect some of them.

If a task name is prefixed with a "*" character, any child tasks will be selected as well (except for those that include the dontinheritcheck flag). If a task name is prefixed with a "!" character, the task will be deselected.
Example:
Deselect all tasks, then select the "desktopicon" and "fileassoc" tasks:
/TASKS="desktopicon,fileassoc"
Example:
Deselect all tasks, then select a parent task and all of its children with the exception of one:
/TASKS="*parent,!parent\child"

Silent uninstall

On installation, a file **unins000.exe** is created in the program directory, which can be used to uninstall the program.
The command line parameters are as follows:

/SILENT, /VERYSILENT
When specified, the uninstaller will not ask the user for startup confirmation or display a message stating that uninstall is complete. Shared files that are no longer in use are deleted automatically without prompting. Any critical error messages will still be shown on the screen. When '/VERYSILENT' is specified, the uninstallation progress window is not displayed.

If a restart is necessary and the '/NORESTART' command isn't used (see below) and '/VERYSILENT' is specified, the uninstaller will reboot without asking.

/SUPPRESSMSGBOXES
Instructs the uninstaller to suppress message boxes. Only has an effect when combined with '/SILENT' and '/VERYSILENT'. See '/SUPPRESSMSGBOXES' under Setup Command Line Parameters for more details.

/LOG

Causes Uninstall to create a log file in the user's TEMP directory detailing file uninstallation and [UninstallRun] actions taken during the uninstallation process. This can be a helpful debugging aid.

The log file is created with a unique name based on the current date. (It will not overwrite or append to existing files.)

The information contained in the log file is technical in nature and therefore not intended to be understandable by end users. Nor is it designed to be machine-parsable; the format of the file is subject to change without notice.

/LOG="filename"

Same as /LOG, except it allows you to specify a fixed path/filename to use for the log file. If a file with the specified name already exists it will be overwritten. If the file cannot be created, the Uninstall will abort with an error message.

/NORESTART

Instructs the uninstaller not to reboot even if it's necessary.

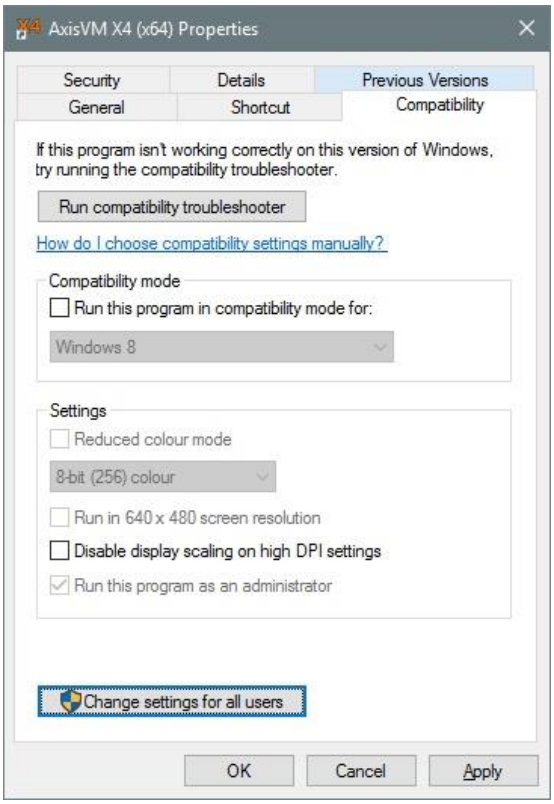
Troubleshooting

The setup program creates desktop shortcut for AXISVM 27 automatically.
Click on the icon to launch AXISVM.
Depending on the security settings of the operating system a security warning may appear. Click on Yes and continue.
If AXISVM does not start set user rights as follows.

- Click the right mouse button on the AXISVM icon.
- Select *Properties* from the popup menu
- Go to the *Compatibility* tab, and check *Run this program as an administrator*

If you have administrator rights but the program does not start, check the following issues:

1. ***If a message is displayed with an error code***, please check if the hardware key is properly connected and the green LED is lit. If no light can be seen the USB port may be out of operation.
2. ***If no message appears***, but the program does not start check your antivirus program (see below).



False virus alarms

Certain antivirus products running on the PC can send a false alarm during installation. This is caused by heuristic algorithms searching for virus-like activities. These algorithms may detect the operation of the special protection system of AXISVM and send a false alarm. If this happens you can do the following

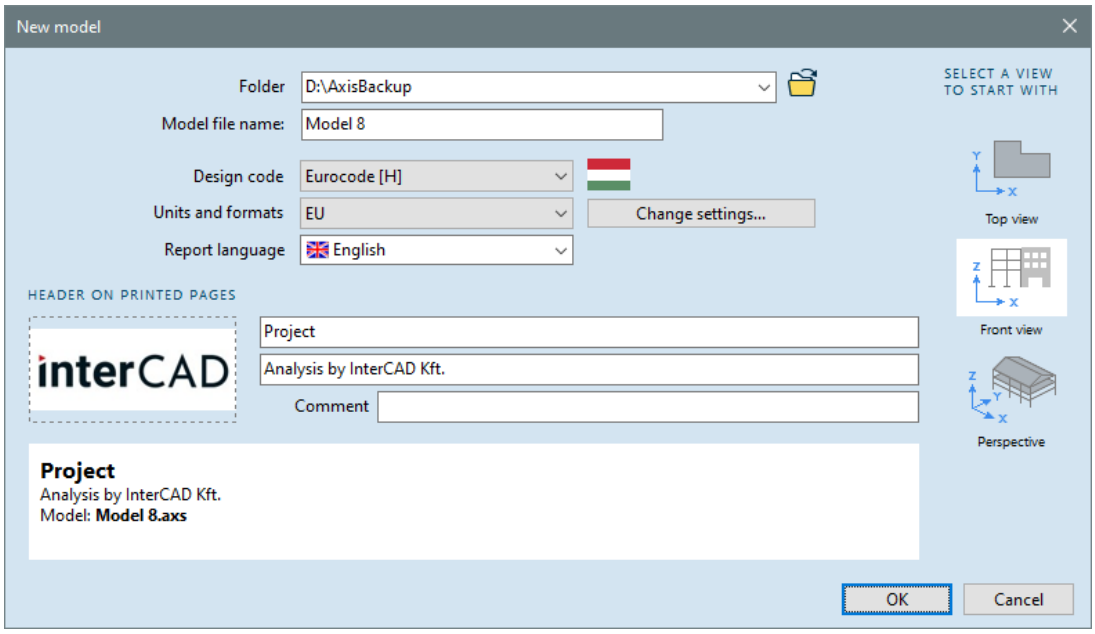
- *If the antivirus product put AxisVM.exe into quarantine restore it*
- *Add AxisVM.exe to the exceptions (files not checked by the antivirus software)*
- *Reduce the sensitivity of the heuristic check on the control panel of the antivirus product*

The VirusTotal website offers antivirus check of 47 different products.

Starting AXISVM



Click the Start button, select Programs, AXISVM folder, and click the AXISVM icon.
At startup a splash screen is displayed then a welcome screen is shown where you can select a previous model or start a new one.
Clearing the checkbox at the bottom turns the welcome screen off for the future. To turn it on choose the *Settings / Preferences / Data Integrity* dialog and check the *Show welcome screen on startup* checkbox.
When setting up a new model the following dialog is displayed.



Running AXISVM in safe mode

Both AxisVM.exe can be started in safe mode entering `axisvm.exe /SAFE` in the command line.
It is recommended to start AXISVM in safe mode in the following cases: (1) graphic card or driver problems, (2) if problems are detected in multithreaded mode (3) if AXISVM hangs when trying to recover the latest file damaged in a crash (4) if a plugin or addon module causes errors.

Launching AXISVM in debug mode

If a problem is encountered during startup or AXISVM cannot be launched, run AxisVM with the /USERDEBUG and /NETDONGLELOG switches. In the case of a network license, first, launch AXISVM with /USERDEBUG then with /NETDONGLELOG. In case of a local single license, it is enough to use the /USERDEBUG switch.
These switches activate logging of the startup process, creating a log file. Send the log file to your distributor to find how to fix the problem.

Program and file types

	File type	*File extension*	*Can be opened by* *Standard*	*Viewer*	*Academic*	*Trial*	*Light*
created by	Standard	.axs/.axe	yes	yes	yes	yes	no
	Viewer	.axv/.axw	no	yes	no	no	no
	Academic	.axs/.axe	WM	yes	yes	WM	DEN
	Trial	.axs/.axe	yes	yes	yes	yes	DEN
	Light	.axd/.axr	yes	yes	yes	yes	yes

WM	watermark appears on the printed output
DEN	depends on the number of elements in the model (Light version limit)

Upgrading

It is recommended to install the new version to a new folder. This way the previous version will remain available.

Converting earlier models

Models created in a previous versions are recognized and converted automatically. Saving files will use the latest format by default. Saving files in the file format of one of the previous is possible but this way the information specific to the newer versions will be lost.

Steps of an analysis

The main steps of an analysis using AXISVM are:

Creating the Model (Preprocessing)
↓

Analysis			
Static (linear/nonlinear)	Vibration (first/second-order)	Dynamic (linear/nonlinear)	Buckling

↓

Evaluating the Results (Postprocessing)

Capacity

Practically, the model size is limited by the amount of free space on your hard disk.

The restrictions on the model size and on the parameters of an analysis are as follows:

Professional

Entity		Maximum
Nodes		Unlimited
Materials		Unlimited
Elements	Truss	Unlimited
	Beam	Unlimited
	Rib	Unlimited
	Membrane	Unlimited
	Plate	Unlimited
	Shell	Unlimited
	Support	Unlimited
	Gap	Unlimited
	Diaphragm	Unlimited
	Spring	Unlimited
	Rigid	Unlimited
	Link	Unlimited
Load cases		Unlimited
Load combinations		Unlimited
Frequencies		Unlimited

Small Business

Entity		Maximum
Nodes		Unlimited
Materials		Unlimited
Elements	Only trusses	500
	Truss+Beam+Rib *	250
	Rib on the edge of a surface	1500
	Any combination of membrane, plate or shell	2000
	solid element (SOIL module)	20000
	Support	Unlimited
	Gap	Unlimited
	Diaphragm	Unlimited
	Spring	Unlimited
	Rigid	Unlimited
	Link	Unlimited
Load cases		Unlimited
Load combinations		Unlimited

***** If there are beams or/and ribs in the structure

2.2.6 Running add-ons

You can use add-ons created for AXISVM by external developers – or yourself. To run these programs the AXISVM COM server must be registered in the Windows Registry. If you installed AXISVM with administrative rights this registration is already completed. If the registration failed you can run! Register_AxisVM.bat (on 32-bit operating systems) or !Register_AxisVM_x64.bat (on 64 bit operating systems) with administrative rights.

☞ **32-bit add-ons launch the 32-bit version and are compatible only with that. 64-bit add-ons can be used only on 64-bit versions.**

By default, the 64-bit version of the program is installed, registering only the 64-bit COM server. If the 32-bit version is required, the installation will request the 32-bit version to be installed, which will also install the 32-bit COM server.

If add-ons do not run because the registration failed during the installation, the COM server registration can be checked and modified in Help / COM Server information: 3.9.6 COM server information

2.3 Getting started

The best way to discover AXISVM is to build a simple model, analyse it and evaluate the results.
The steps of model building are described in the AXISVM Step by step tutorial.

There are three major steps in a modeling process:

Geometry

The first step is to create the geometry model of the structure (in 2D or 3D).
Geometry can be drawn by hand or can be imported from other CAD programs. It is also possible to draw elements (columns, beams, walls, slabs) directly.

Elements

If you chose to draw the geometry first you must specify material and element properties, mesh the geometry into elements (assigning the properties and a mesh, to the wire-frame model), and define the support conditions.

Loads

In the third step you must apply different loads on the model.

The end result will be a finite element model of the structure.
Once the model is created it is ready for analysis.

You can find the steps of model building in detail in the AXISVM *Step by step* tutorial made available from the *Help* menu. This tutorial helps to build and analyse simple structures.

- Beam model
- Frame model
- Plate model
- Membrane model (with parametric mesh)
- Membrane model (with domains)
- Shell model

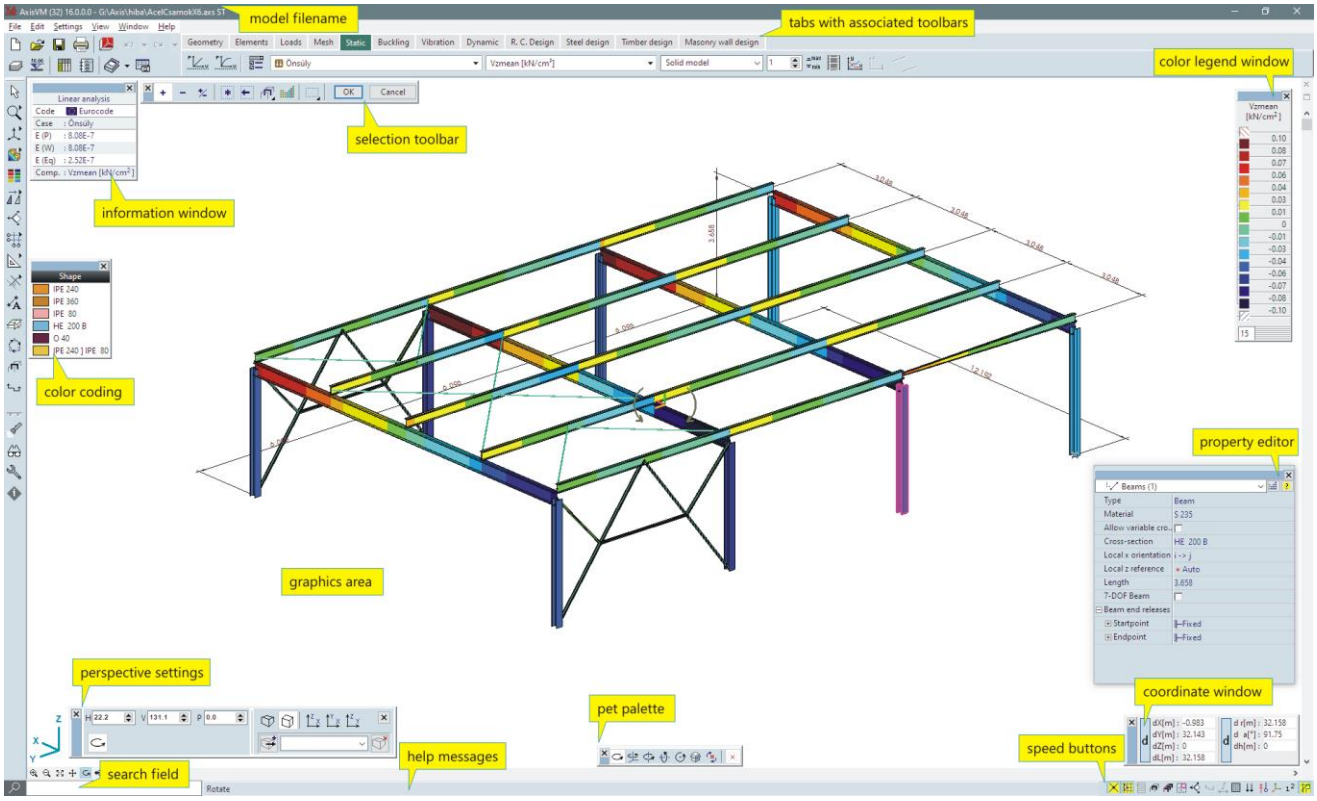
AXISVM Step by step (for advanced users) guides the user through dynamic and seismic analysis of two advanced models.
English language video tutorials for the example models can be found on the **AXISVM YouTube channel**

It is recommended that you read the entire User's Manual at least once while exploring AXISVM.
In **Chapter** 1 you can find the timely, new features of the version.
Chapter 2 contains general information about using AXISVM.In other chapters the explanation follows the pre- and postprocessor menu structures. Please, consult this *User's Manual* every time you are using AXISVM.

2.4 AXISVM user interface

This section describes the working environment of the full AXISVM graphical user interface. Please read these instructions carefully. Your knowledge of the program increases the modeling speed and productivity.

AXISVM screen Elements of the user interface



The parts of the AXISVM screen are briefly described below.

Graphics area The area on the screen where you create your model.

Graphics cursor The screen cursor is used to draw, select entities, and pick from menus and dialog boxes. Depending on the current state of AXISVM, its shape may change.

Top menu bar Each item of the top menu bar has its own dropdown menu list. To use the top menu bar, move the cursor up to the menu bar. The cursor will change to a pointer. To select a menu bar item, move the pointer over it, and press the pick button to select the item. Its associated dropdown menu will appear.

Tabs with associated toolbars Tools of model building, analysis and evaluation of results are grouped on tabs following the logical order of these operations.

Active icon The active icon on these toolbars represents the command that is currently selected.

Icon bar The icons on the left represent working tools in a pictorial form. These tools are accessible during any stage of work. The icon bar and flyout toolbars are draggable and dockable.

Coordinate window The window on the graphics area displaying the graphics cursor coordinates.

Color legend window The window shows the color legend used in the display of the results. Appears only in the post-processing session.

Info window The window shows the status of the model and results display.

Context sensitive help Provides a help message that depends on the topic under process.

Property Editor The Property Editor offers a simple way to change certain properties of the selected elements or loads.

Pet palette Pet palettes appear when modifying geometry according to the type of the dragged entity (node, straight line, arc). See **4.8.22 Modify, transform**

Search field In the bottom left-hand corner, a smart command line helps you find the function you're looking for or select elements. See **2.17 Smart command line**

Speed buttons Speed buttons in the bottom right provide the fastest access to certain switches (parts, sections, symbols, numbering, workplanes, etc.)

The model With AXISVM you can create and analyze finite element models of civil engineering structures. Thus the program operates on a model that is an approximate of the actual structure.
To each model you must assign a name. That name will be used as a file name when it is saved. You may assign only names that are valid Windows file names. The model consists of all data that you specify using AXISVM.

The model's data The model's data are stored in two files: the input data in the **filename.axs** and the results in the **filename.axe** file.
If construction stages are defined (see **4.13 Construction stages – STG module**), filename.axe contains results of the final state. Results of intermediate stages are stored as **filename.stg1, filename.stg2, ... filename.stgN,** where N is the total number of stages.
If the analysis fails at a stage, results of the previous (completed) stages remain available. If stage *k* is modified after an analysis, the results of stages 1, 2, ... k-1 will remain available.
AXISVM checks if AXS and AXE files belong to the same version of the model.

2.5 Using the cursor, the keyboard, the mouse

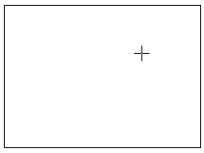
Unicode is a computing industry standard for the consistent encoding, representation and handling of text expressed in most of the world's writing systems. AXISVM provides full Unicode support.
All windows appear according to the current Windows theme.

Graphics cursor

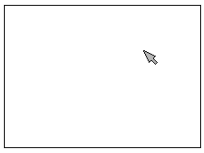


As you move your mouse, the graphics cursor symbol tracks the movement on the screen. To select an entity, an icon or menu item, move the cursor over it and click the left mouse button. The shape of the cursor will change accordingly (see **4.7.1 Cursor identification**), and will appear on the screen in one of the following forms:

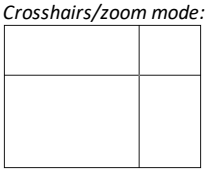
Crosshairs:



Pointer:



Crosshairs/zoom mode:



If you pick an entity when the cursor is in its default mode (info mode), the properties of that entity will be displayed as a tooltip.

Tooltip Depending on the menu your cursor is on, you may get the properties of the following entities:

Geometry	node (point) coordinates, line length
Elements	finite element, reference, degree-of-freedom, support, length, mass, material, cross-section
Loads	element load, nodal mass
Mesh	meshing parameters
Static	displacement, internal force, stress, reinforcement, influence line ordinate
Vibration	mode shape ordinate
Dynamic	displacement, velocity, acceleration, internal force, stress
R.C. Design	specific reinforcement values
Steel Design	displacement, internal force, utilization results and resistances
Timber Design	displacement, internal force, utilization factor results and resistances

The keyboard You can also use the keyboard to move the cursor:

Arrow keys, Moves the graphics cursor in the current plane.

[Ctrl] + Arrow keys, Moves the graphics cursor in the current plane with a step size enlarged/reduced by a factor set in the Settings dialog box.

[Shift]+ [↑][↓][←][→],	Moves the graphics cursor in the current plane on a line of angle n·Δα , custom α or α +n·90°.
[Home] [End]	Moves the graphics cursor perpendicular to the current plane.
[Ctrl]+ [Home], [End]	Moves the graphics cursor perpendicular to the current plane with a step size enlarged/reduced by a factor set in the Settings dialog box.
[Esc] or right button	Interrupts the command and/or returns to an upper menu level.
[Enter]+[Space] left button	Selects an item from a menu, executes a command, and selects entities. These are termed command buttons.
[Alt]	Activates the main menu
[Tab]	Moves the focus from control to control in a dialog.
[+] [-]	Performs fast zoom in/out and pan. The zoom and pan parameters are defined by the current position of the graphics cursor in the graphics area, and by the magnification factor set in *Settings / Options / Zoom Factor*. Center of the fast zoom in/out is always the current graphics cursor position.
[Insert] *or* **[Alt]+[Shift]**	Moves the relative origin (i.e., the reference point of the relative coordinates) to the current graphics cursor position.
wheel	Roll forward to zoom in Roll backwards to zoom out Press the wheel and drag to drag the drawing area Centre of zoom in and zoom out is the current position of the cursor.
Hot Keys	Keyboard combinations to access frequently used functions faster. See **2.6 Keyboard shortcuts**
right button	Displays the Quick Menu. See **2.7 Quick Menu**

2.6 Keyboard shortcuts

Keyboard shortcuts can be assigned to common operations, toolbar buttons, menu items. The default settings of AXISVM are the following. The main toolbar button shortcuts are context-sensitive, i.e., the same shortcut can perform different tasks on different tabs (*Geometry, Elements, Loads,* etc.).

General commands

Open	Ctrl+O
Save	Ctrl+S
Print	Ctrl+P
Undo	Alt+BkSp
Redo	Shift+Alt+BkSp
Layer Manager	F11
storeys	F7
Table Browser	F12
Report Maker	F10
Drawings Library	F6
Save to Drawings Library	F9
Delete	Del
Set relative origin	Ins
Previous load case	Ctrl+PgUp
Next load case	Ctrl+PgDn

Icon bar

Selection	S
Color coding	Shift+C
Translate / Copy	Shift+T
Rotate	Shift+R
Mirror	Shift+M
Scale	Shift+S
Dimension lines	Ctrl+Alt+D
Edit background layers	Ctrl+Alt+E
Part	Shift+P
Section lines	Shift+Ctrl+X
Find	F3
Display options	Ctrl+Y

Model information ..... Shift+I
View
Zoom in ..... Ctrl+ü
Zoom out ..... Shift+Ctrl+ü
Fit in window ..... Ctrl+W
Pan ..... Ctrl+M
Rotate ..... Ctrl+R
Front view ..... Ctrl+1
Side view ..... Ctrl+3
Top view ..... Ctrl+2
Perspective ..... Ctrl+4
Wireframe ..... Alt+F5
Hidden line removal ..... Alt+F6
Rendered ..... Alt+F7
Texture ..... Alt+F8
Guidelines
Structural grid ..... Shift+G
Perpendicular ..... Alt+V
Parallel ..... Alt+P
Bisector ..... Alt+B
Dividing point ..... Alt+M
Intersection point of two lines ..... Alt+I

Toolbar
[Geometry] ..... Shift+F1
[Elements] ..... Shift+F2
[Loads] ..... Shift+F3
[Mesh] ..... Shift+F4
[Static] ..... Shift+F5
[Buckling] ..... Shift+F6
[Vibration] ..... Shift+F7
[Dynamic] ..... Shift+F8
[R. C. Design] ..... Shift+F9
[Steel design] ..... Shift+F10
[Timber design] ..... Shift+F11
Geometry
Node ..... N
Line L
Polygon ..... P
Rectangle ..... R
Arc ..... A
Arc based on three points ..... B
Horizontal division ..... H
Divides lines ..... D
Intersect I
Elements
Material ..... Shift+Ctrl+M
Cross-section ..... Shift+Ctrl+C
Draw objects directly F4
Draw supports directly F5
Domain ..... D
Hole..... H
Line elements L
Nodal support ..... T
Line support ..... U
Node to node interface element I
Line to line interface element ..... J
Loads
Load cases and load groups L
Load combinations..... C
Nodal loads ..... N
Concentrated loads on beams ..... B
Domain point load ..... A
Load along line elements ..... J
Surface edge loads ..... E
Domain line load I
Distributed surface load ..... H
Distributed load on domain ..... D
Derived surface load over trusses/beams/ribs ... K
Load panel ..... P
Snow load ..... O
Wind load ..... W
Fluid loads ..... F
Self Weight ..... G
Moving line load definition ..... T
Mesh
Domain meshing ..... G
Static
Linear static analysis L

- Nonlinear static analysis ..... N
- Result display parameters ..... D
- Min, Max values ..... Ctrl+X
- Animation ..... A

Buckling

- Buckling analysis L
- Result display parameters ..... D
- Min, Max values ..... Ctrl+X
- Animation ..... A

Vibration

- Vibration analysis L
- Result display parameters ..... D
- Min, Max values ..... Ctrl+X
- Animation ..... A

Dynamic

- Dynamic analysis L
- Result display parameters ..... D
- Min, Max values ..... Ctrl+X
- Animation ..... A

R. C. Design

- Reinforcement parameters ..... P
- Actual reinforcement ..... T
- Result display parameters ..... D
- Min, Max values ..... Ctrl+X
- Animation ..... A
- Column reinforcement C
- Beam reinforcement design ..... B
- Plate punching analysis ..... U
- Pad footing design ..... F
- Strip footing design I

Steel design

- Design parameters ..... P
- Result display parameters ..... D
- Min, Max values ..... Ctrl+X
- Joint design ..... J
- Joint design ..... K
- Steel cross-section optimization ..... O

Timber design

- Design parameters ..... P
- Result display parameters ..... D
- Min, Max values ..... Ctrl+X
- Animation ..... A

Menu

File

- Open ..... Ctrl+O
- Save ..... Ctrl+S
- Print ..... Ctrl+P
- Exit ..... Ctrl+Q

Edit

- Undo ..... Alt+BkSp
- Redo ..... Shift+Alt+BkSp
- Select all ..... Num *
- Copy ..... Ctrl+C
- Paste ..... Ctrl+V
- Delete ..... Del
- Table Browser F12
- Report Maker F10
- Drawings Library F6
- Save to Drawings Library F9
- Weight report F8
- Piano Sisma ..... Ctrl+Alt+P
- Assemble structural members ..... Shift+A
- Break apart structural members ..... Shift+B
- Reverse local coordinate system ..... Ctrl+E

Settings

- Symbols ..... Ctrl+Y
- Switches ..... Ctrl+D
- Layer Manager F11
- storeys F7
- Guidelines setup ..... Ctrl+G
- Structural grid ..... Shift+G
- Keyboard shortcuts ..... Ctrl+K
- **Preferences**
  - Fonts ..... Shift+Alt+F
  - Analysis ..... Shift+Alt+B

View

- Front view ..... Ctrl+1
- Top view ..... Ctrl+2

Side view ..... Ctrl+3
Perspective ..... Ctrl+4
One storey up ..... PgUp
One storey down ..... PgDn
Zoom in ..... Ctrl+ü
Zoom out ..... Shift+Ctrl+ü
Fit in window ..... Ctrl+W
Pan ..... Ctrl+M
Rotate ..... Ctrl+R
Wireframe ..... Alt+F5
Hidden line removal ..... Alt+F6
Rendered ..... Alt+F7
Texture ..... Alt+F8

Window

Property Editor ..... Shift+Alt+P
Status ..... Shift+Alt+I
Color coding ..... Shift+Alt+C
Coordinates ..... Shift+Alt+K
Color legend ..... Shift+Alt+L
Load background picture ..... Ctrl+B
Split horizontally ..... Shift+H
Split vertically ..... Shift+V
Close window ..... Ctrl+F4
Decrease label font size ..... Ctrl+Alt+I
Increase label font size ..... Ctrl+Alt+O
Drawings Library F6
Save to Drawings Library F9

Help

Contents F1

Speed buttons

Auto intersection ..... Ctrl+I
Mesh display on/off ..... M
Show only selected elements ..... Ctrl+F
Load display on/off ..... Ctrl+L

Result display options

None ..... Ctrl+F5
Diagram ..... Ctrl+F6
Isolines ..... Ctrl+F7
Isosurfaces 2D ..... Ctrl+F8
Section line ..... Ctrl+F9
Isosurfaces 3D ..... Ctrl+F10
Diagram + average values ..... Ctrl+F11
List of result components ..... Q

Coordinates

X X
Y ..... Y
Z ..... Z
L ..... Shift+Ctrl+L
R ..... Shift+Ctrl+R
A ..... Shift+Ctrl+A
H ..... Shift+Ctrl+H
B ..... Shift+Ctrl+B
Temporary workplane ..... Shift+Ctrl+W
Lock X[m] : ..... Ctrl+Alt+X
Lock Y[m] : ..... Ctrl+Alt+Y
Lock Z[m] : ..... Ctrl+Alt+Z
Lock L[m] : ..... Ctrl+Alt+L
Lock r[m] : ..... Ctrl+Alt+R
Lock a[°] : ..... Ctrl+Alt+A
Lock h[m] : ..... Ctrl+Alt+H
Lock b[°] : ..... Ctrl+Alt+B
Relative / global coordinates ..... Shift+D
Relative / global polar coordinates ..... Shift+E

Hot Keys in Tables

Ctrl+L	Browse Libraries
Alt+F4	Exit
Ctrl+Insert	New line
Ctrl+Del	Delete line
Ctrl+A	Select all
F5	Jump to line
Ctrl+D	Default format
Ctrl+Alt+F	Set column format

Ctrl+R	Set result display mode (for result tables)
Ctrl+G	Edit new cross-section (for cross-section tables)
Ctrl+M	Modify cross-section (for cross-section tables)
F1	Context sensitive help
F9	Add table to the report
F10	Report Maker

Hot keys in the Report Maker

Ctrl+T	Insert text
Ctrl+Alt+B	Insert Page Break
Ctrl+W	Export to RTF file
F3	Report Preview
Ctrl+P	Print
Ctrl+Del	Delete

Mouse wheel commands

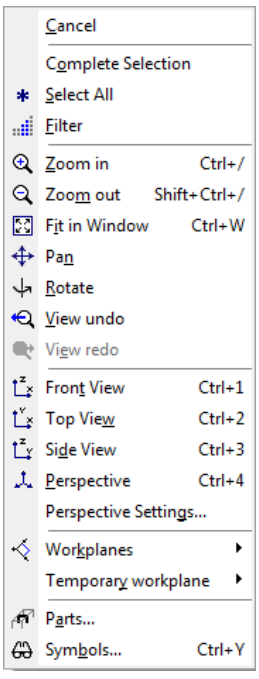
Scroll up	Zoom in
Scroll down	Zoom out
Wheel down + move	Pan (slow)
Wheel down + ALT + move	Rotate
Wheel down + CTRL + move	Pan (fast)

2.7 Quick Menu

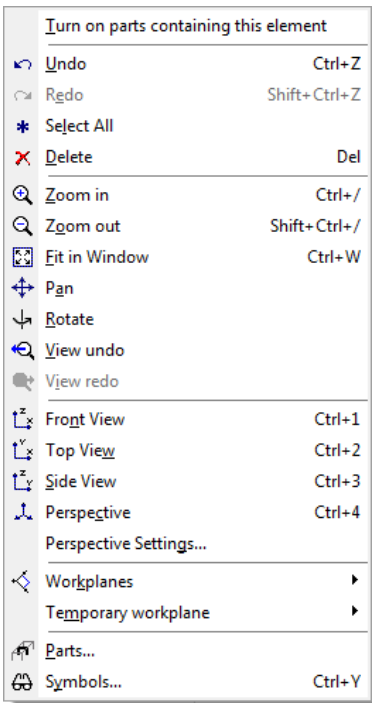
right button

When the cursor is over the graphics area, by pressing the right mouse button a **quick menu** appears in accord with the current command in use.

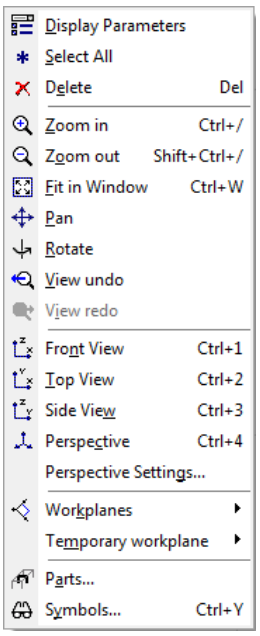
Selection



Geometry / Elements / Loads



Results



2.8 Dialog boxes

After selecting a function usually a dialog box appears on the screen. These dialog boxes can be used the same way as any other Windows dialog.
The dialog font can be changed by selecting the *Settings / Preferences / Fonts* dialog and clicking the font sample label *Dialog boxes*.
You can change the position of all dialog windows. The program saves the latest position and displays the dialog on the same position next time.

2.9 Table Browser



[F12]

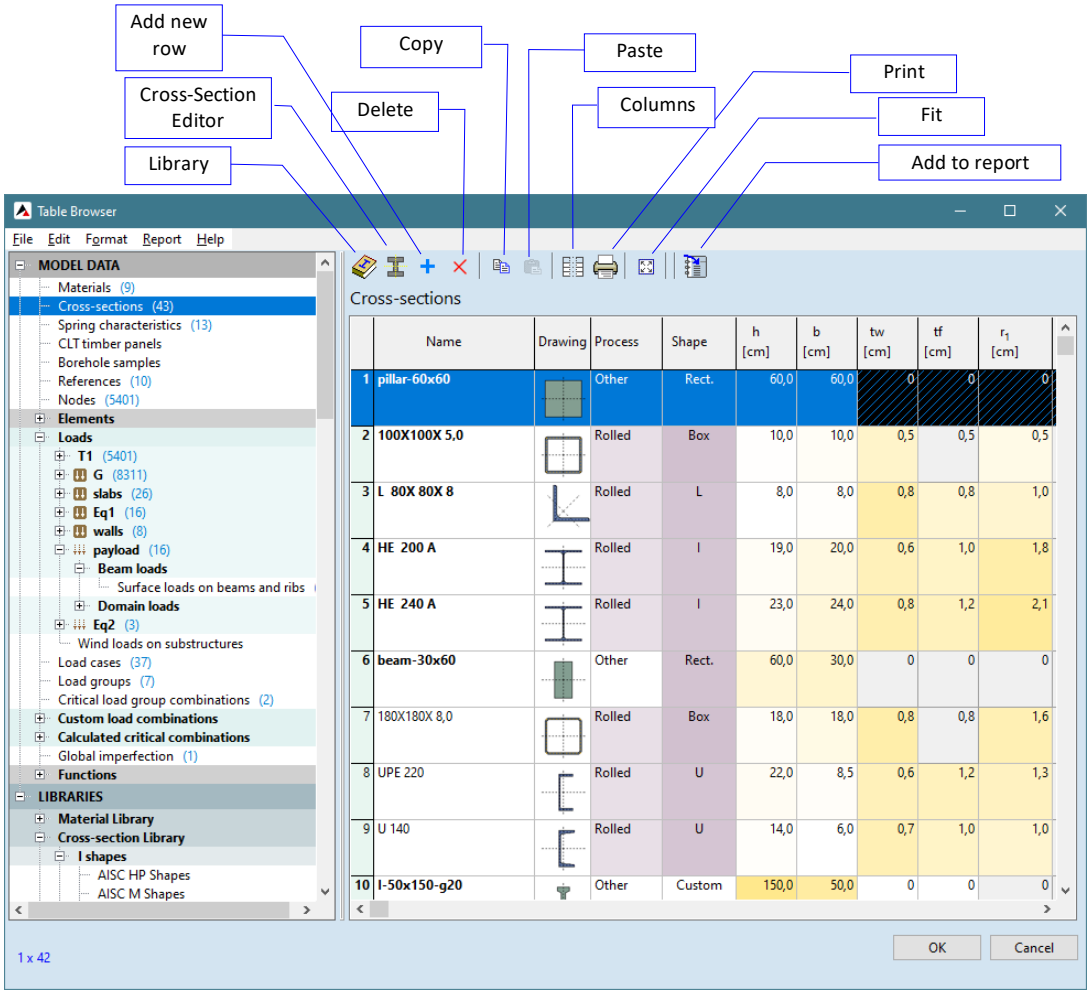
AXISVM uses tables to display numerical information on the screen allowing changes in formatting. The tables operate the same way independent of the content displayed. All the tables AXISVM creates are available through the Table Browser dialog box by clicking its button or pressing **[F12].**
The model data to be displayed in the Table Browser can be selected from the tree structure in the left side of the browser. If you use Table Browser while working in the pre-processor (before the *Static* tab), only input model data is displayed. While working in the post-processor (on the *Static* tab or to the right), the model results are also displayed.

☞ **Only the data of the current selection (if any) or of the active (i.e., displayed) part is listed by default.**

The tree view on the left lists element / load data, result tables and libraries in a hierarchy and can also be used as a model overview. If the model includes construction stages, a *Construction stages* subfolder is displayed under the *Elements* folder. You can find further subfolders for stages containing tables of elements present in the respective stage.

Color coding of table cells

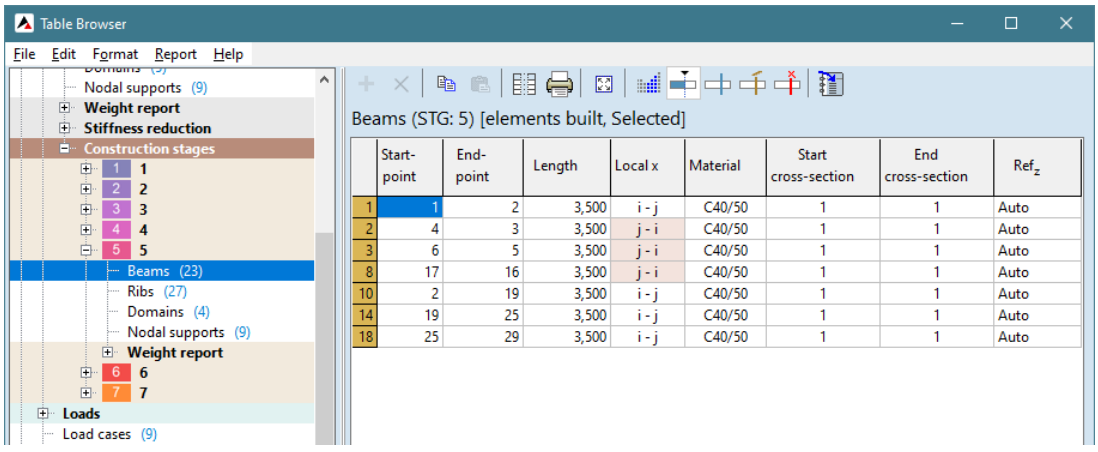
Tables of the *Table Browser* apply color coding to the displayed values (e.g., domain thickness, beam cross-section, beam length, result components, etc.). Cell background colors appear only on the screen. It helps to find common, smaller of bigger values.
Result tables show negative values in shades of blue, positive values in shades of red. Efficiencies are displayed in shades of green under the limit and in shades of red above it. Color coding can be disabled, see *Format / Color coding of table cells*.



Display options for construction stages



If the model includes construction stages (see **4.13Construction stages – STG module**), and a table under *Elements / Constructions stages / <Stage name>* is selected, additional buttons appear on the toolbar. These are display options for the construction stage (elements built, added, modified or removed from the stage). For more information see **4.13.4 Display options for stages.**



Using the table

A table can contain more rows and/or columns than can be displayed at the same time. It can be viewed in its entirety using the scroll bars and/or using the keyboard as follows:

Arrow keys
left button — Moves the edit focus up and down, to the left and to the right, and scrolls the table along the rows or columns. Clicking an editable cell moves the edit focus to that cell.

[Home] — Moves the focus to the first cell of the row.

[End] — Moves the focus to the last cell of the row.

[Ctrl]+[Home] Moves the focus to the first cell of the first row

[Ctrl]+[End] Moves the focus to the last cell of the last row.

[Page Up] Displays the previous page of rows.

[Page Down] Displays the next page of rows.

[Ctrl]+ [→] Moves the focus to the next (to the right) page of columns (only in tables where more columns can be displayed at the same time).

[Ctrl]+ [←] Moves the focus to the previous (to the left) page of columns (only in tables where more columns can be displayed at the same time).

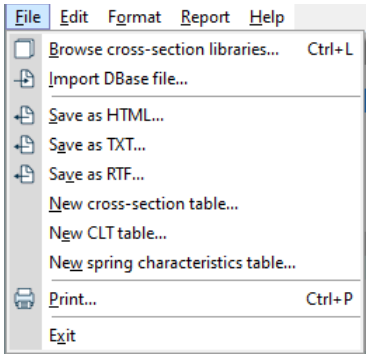
[Enter] Ends the current editing in the edit box storing the data entered and moves the edit box a column to the right or to the first column of the next row.

[Esc] Aborts the current editing in the edit box.

right button

[Shift] While the **[Shift]** key is down all direction keys will select cells instead of moving the edit focus. You can also select cells by dragging the mouse. Clicking a fixed (topmost) cell of a column selects the column. Clicking a fixed (leftmost) cell of a row selects the row. Clicking the top left cell selects the entire table. Selected cells can be copied to clipboard as a table. If selection is within an editable column you can set a common value for the selected cells.
See **Set Common Value** below.

File



Browse from libraries
[Ctrl]+ [L] If the table of *Materials, Cross-sections, Spring characteristics* or *CLT timber panels* is selected the first menu item allows browsing of the built-in material library, cross-section library, spring characteristics library or CLT timber panels library.

Import DBase File Imports a DBase file *name.dbf* into the current table. The program checks the values of the fields and sends an error message if an incompatible value is found.

Save As DBase File Exports the current table into a Dbase file *name.dbf*. The field names are generated based on the names of the columns. The fields will be of text type.

Save As HTML Exports the current table into an HTML file *name.htm*. This file can be imported as a table into Word or can be opened in web browser applications. Some formatting information of the columns will be lost.

Save As TXT Exports the current table into a TXT (ASCII) file *name.txt*.

Save As RTF Exports the current table into an RTF file *name.rtf* using the current template file. You can import this file into Microsoft Word or any other word processor which can import RTF files.
See **2.10.2 Report**

New cross-section table Creates a new cross-section data file *name.sec*. The table created will be placed together with the cross-sections of the same type.
You can store cross sections of any type in these tables. Type of the table determines only the position of the table in the Cross-section Library.

Cross-section table properties — You can modify properties (table's name, cross.section type) of a user defined table.

Delete cross- section table — You can delete a user defined table.

New CLT table — Creates a new *CLT* timber panel layer table under *LIBRARIES / CLT timber panels.*

Rename CLT table — You can change the name of a user defined table within *CLT timber panels*.

Delete CLT table — You can delete a user defined table within *CLT timber panels.*

New spring characteristics table — Creates a new spring characteristics table under LIBRARIES / *Spring characteristics library.*

Rename spring characteristics table — If a custom spring characteristics table is selected within the *Spring characteristics library* this menu item appears and the name of the table can be changed.

Delete spring characteristics table — If a custom spring characteristics table is selected within the *Spring characteristics library* this menu item appears and the table can be deleted.

Print **[Ctrl + [P]**

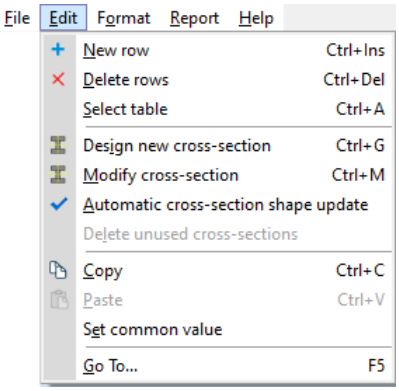


Prints all the information displayed in the table to the selected printer or to a file, with the page header and comment row previously set with the *File/Header* menu command.
Description of table columns adds an explanation of columns at the bottom of the table.

Name: Cross-section name; **Process:** Manufacturing process; **h:** Cross-section height; **b:** Cross-section width; **tw:** Web thickness; **tf:** Flange thickness; r_1, r_2, r_3: Rounding radius; **Ax:** Cross-section area; **Ay, Az:** Shear area; **Ix:** Torsional inertia; **Iy, Iz:** Flexural inertia; **Iyz:** Centrifugal inertia; I_1, I_2: Principal flexural inertia; **α:** Principal directions; **Iω:** Warping constant; $W_{1,el,t}$, $W_{1,el,b}$, $W_{2,el,t}$, $W_{2,el,b}$: Elastic modulus; $W_{1,pl}$, $W_{2,pl}$: Plastic modulus; i_y, i_z: Radius of inertia; **Hy:** Dimension in local y direction; **Hz:** Dimension in local z direction; y_G: y coordinate of the center of gravity; z_G: z coordinate of the center of gravity; y_s: y coordinate of the shear (torsion) center relative to the center of gravity; z_s: z coordinate of the shear (torsion) center relative to the center of gravity; **S.p.:** Stress calculation points;

Exit **[Alt]+ [F4]** — Exits the table in the same way as the *Cancel* button (the changes are not saved).

Edit



New row **[Ctrl]+ [Insert]**



Adds a new row to the list, and allows you to fill all the editable cells with data in a fixed order from left to right.

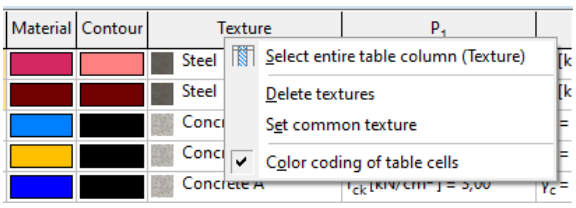
Delete rows **[Ctrl]+ [Del]**



Deletes the selected rows. Also available in the popup menu.

Delete textures — Available only in the table of materials. Removes texture from the selected materials. Available in the popup menu.

Set common textures — Sets the same texture for the selected rows. Available only in the table of materials through the popup menu.



Select table **[Ctrl]+ [A]** — Selects the entire table. Clicking the top left cell does the same.

Design new custom cross-section



[Ctrl]+[G]

Starts the graphics Cross-Section Editor, allowing the input of a new custom cross-section.

Modify custom cross-section



[Ctrl]+[M]

Starts the graphics Cross-Section Editor, allowing the modification of a custom cross-section previously created with the graphics Cross-Section Editor.

Automatic cross-section shape up-date

If this function is on changing section parameters in the table leads to the recalculation of geometry and cross-section parameters.

Delete unused cross-sections

Unused cross-sections will be deleted from the table.

Copy



[Ctrl]+ [C]

Copies selected cells to the Clipboard as a table. Also available in the popup menu.

Paste



[Ctrl]+ [V]

Pastes table cells from the Clipboard overwriting cell values.
If any of the values is unacceptable Paste aborts.
If entire rows were cut or copied and the table allows inserting new rows you can also add clipboard data to the end of the table instead of overwriting the existing rows.

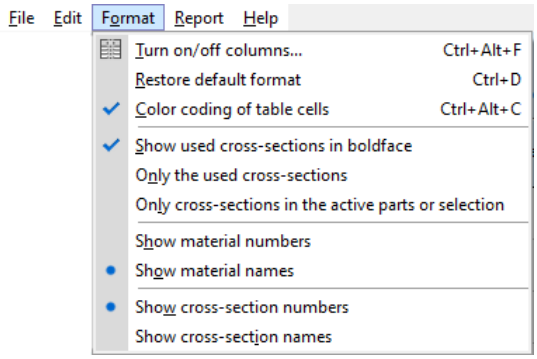
Set common value

Sets a common value for the selected cells within a column.
Example: you can set the Z coordinate of all nodes to the same value making the model absolutely flat.
Available from the *Table Browser Menu / Edit / Set common value*.
Also available in the popup menu.

Go to
[F5]

Jumps to a specified row in the table.

Format
Data entry tables



Turn on/off columns



[Ctrl]+ [Alt]+ [F]

You can specify whether a column is visible or not, by setting the check boxes of the corresponding columns. If some columns are turned off, information on hidden columns appear below the table. Checking the *Save as default* option makes the column status the default for that type of table.

The display format is set according to the settings in the Units/Settings dialogue window (See **3.3.8 Units and Formats**).

Many cells require the entry of a numeric value. When entering real numbers you can use the following characters:

+ - 0 1 2 3 4 5 6 7 8 9 0 E

and the standard Windows decimal separator specified in *Start / Settings / Control Panel / Regional Settings / Number / Decimal symbol* field.
In some cases you cannot enter a negative number so the - key is deactivated while entering these kind of values. If an integer value is required you cannot use the decimal separator and E.

Restore default format
Ctrl+D

Restores the default format of the entire table (column visibility and decimals).

Color coding of table cells
Ctrl+Alt+C

Color coding of cell backgrounds can be enabled or disabled here.

Order of load cases...

The display order of load cases can be customized.
See **4.10.1 Load cases, load groups**

Show material names / Show material numbers

If a column refers to a material, you can set here whether the number or the name should be displayed.

Show cross-section names / Show cross-section numbers

If a column refers to a cross-section, you can set here whether the number or the name should be displayed.

During result query

In case of result query new items appear on the *Format* menu and the Toolbar.

Result Display Options
[Ctrl]+[R]

You can control finding the extremes for result components and set to show results (Result) and/or just the extremes (Extremes).
See in detail... **6.1.6 Result tables**

Results On/Off
[Ctrl]+[T]

Display of results can be turned on / off.

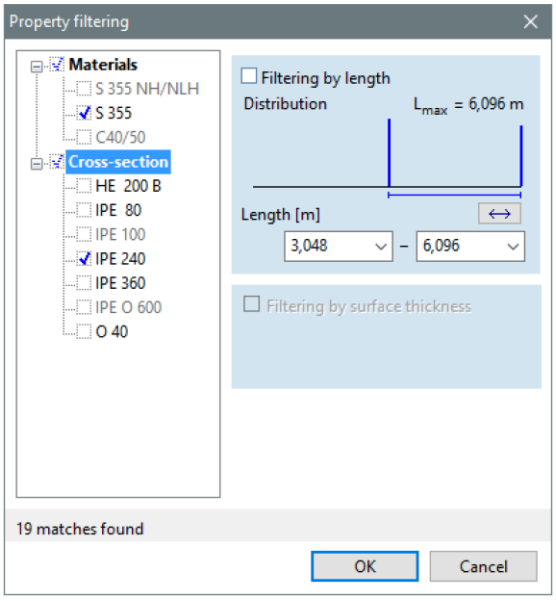
Extremes On/Off
[Ctrl]+[E]

Display of extremes can be turned on / off.

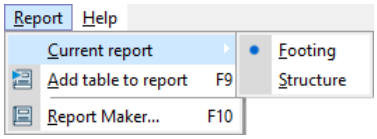
Property Filtering

[Ctrl]+[Q]

Property filtering helps you to select elements to include in the table.



Report



Current report

You can set the current report. Tables will be added to this report. See **2.10 Report Maker**

Add table to report

[F9]

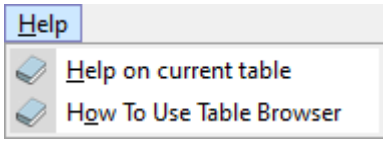
Adds the current table to the current report. If the selected node in the treeview has sub-nodes (e.g., MODEL or Loads) all tables under that node will be added. If the current table is a result table and is set to display extremes only all sub-tables will display extremes only. See **2.10 Report Maker**

Report Maker

[F10]

Opens Report Maker.

Help



Help on Current Table Displays info about the table.

Help to Use Table Browser Displays info about the table browser operation.

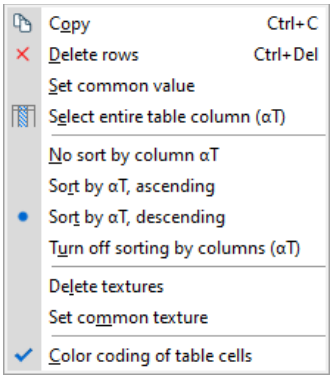
OK Saves the data and closes the table.

Cancel Closes the table without saving the data.

☞ **Result tables also display the extremes (minimum and maximum values) of the data if you select this option in the Display options dialog when you enter Table Browser. Displaying both the individual values and the extremes is the default setting.**

Popup menu

Right-clicking on the table displays the popup menu. Its content depends on the table and whether there are cells selected or not. Some of the functions also appear in the main menu, others available only here:



Select the entire table column: Select cells of the column that was right-clicked.

Delete textures: Available only in the table of materials. Removes texture from the selected rows.

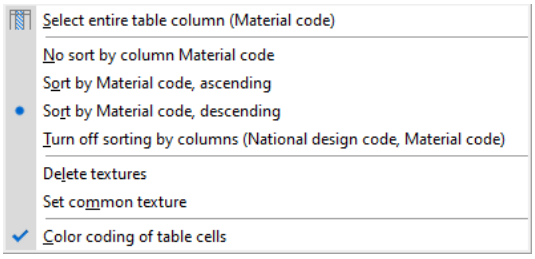
Set common texture: Available only in the table of materials. Sets the same texture for the selected rows.

Sorting by columns

Data entry and result tables can be sorted by most of the columns. Hovering over the header of a sortable column makes the header cell color change. Clicking on the header turns the cell to light blue and the sorting order changes (ascending, descending, no sort). Table rows will be sorted accordingly.

Most data entry tables can be sorted by multiple criteria. Sorting the table of nodes by Z then by X means that nodes with an equal Z coordinate will be sorted by their X coordinate.
The order of criteria depends on the order of clicking. Result tables allow sorting by one result component column only. But there may be certain columns that can be used as the first criterion (like *Cross-section* in the *Beam force* tables, i.e., you can sort the results first by cross-section then by a force component).

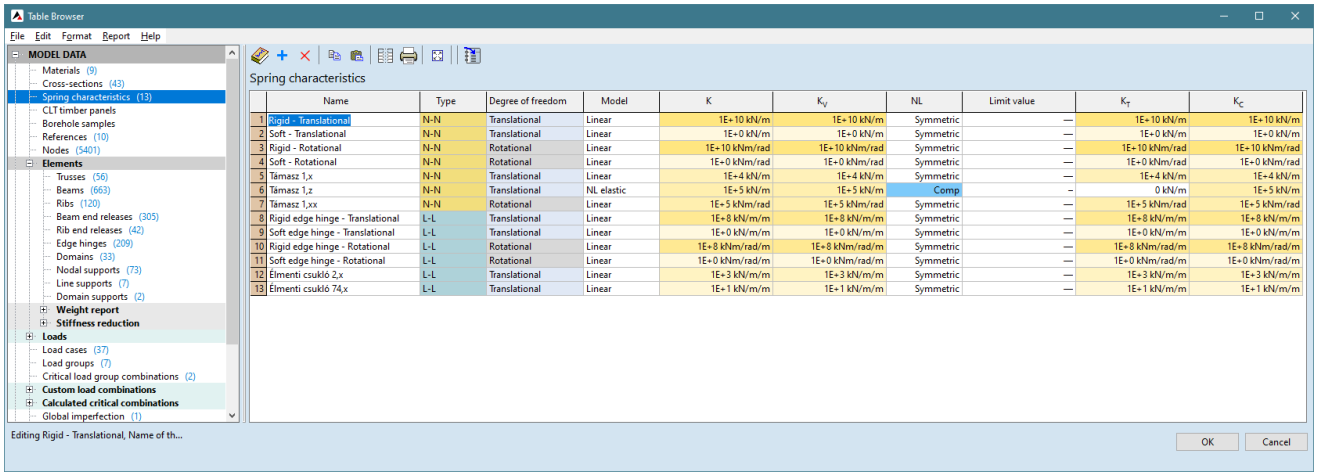
In multiple criteria are applied the order of criteria can be checked by right clicking the header cell and checking this item of the popup menu: *Turn off sorting by columns*



2.9.1 Spring characteristics

Define spring characteristics

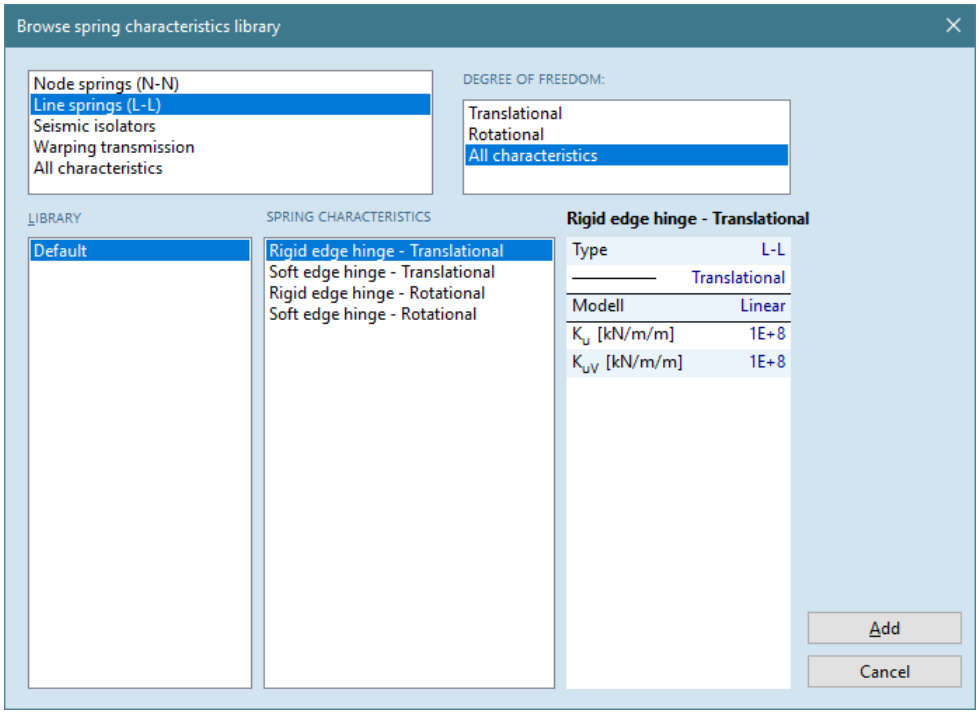
New spring characteristic can be entered in *Table Browser* under *Spring characteristics*. This table shows the spring characteristics specified in the current model. The unused characteristics can be deleted with the *Edit/Delete unused springs* function.



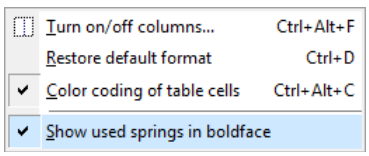


Browse Spring characteristic library

Spring characteristic can be added from the *Spring characteristic library*. A window with libraries and the characteristics is displayed. The spring characteristics are listed depending on the selected degree of freedom (translational, rotational, all). The properties of selected characteristic are shown in the third column.



Activating *Format / Show used springs in boldface* helps to avoid deleting a spring characteristic when it is in use.
Listed but unused springs can be easily removed by the command *Edit / Delete unused springs*.





New spring characteristic

Specify a new spring characteristic is described in **3.1.16 Cross-Section Library.**

Properties of spring characteristics

Existing spring characteristics can be modifeid by clicking to a non-editable column. In the appearing dialog all properties of the spring carateristic can be defined or changed.
The properties of spring characteristics are described in **3.1.16 Cross-Section Library**

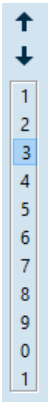
2.10 Report Maker

[F10]

Report Maker is a tool to compile a full report of a project using report items (tables / drawings / pictures created by AXISVM and user-defined text blocks). Reports and their components are displayed in the tree on the left-hand side of the dialog. Reports are stored in the model file *(*.axs)* and can be printed or exported as a DOCX or RTF file. Both files can be processed by other programs (e.g., Microsoft Word or LibreOffice).

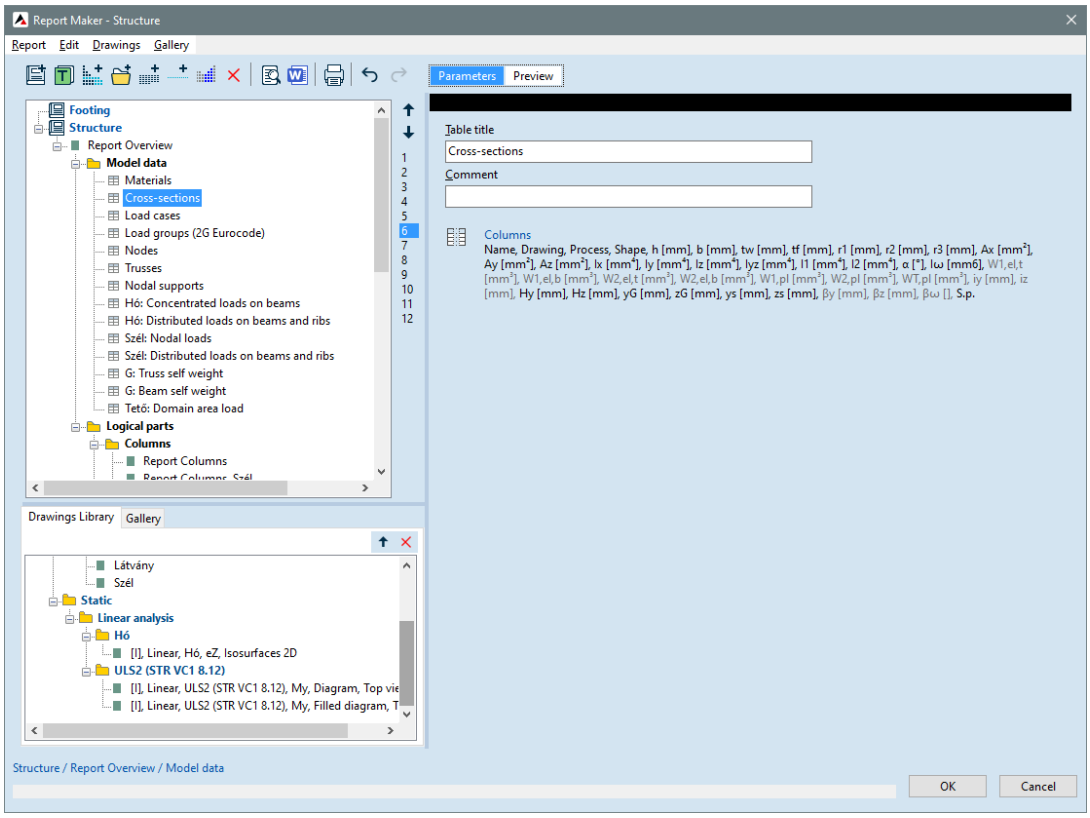
You can configure and check the layout before printing by clicking the *Print* icon.
To set the page size, orientation, margins, etc in the *Report printer* dialog box, set the desired values, then tick the Preview option. Do not print at this stage; click the Cancel button to exit. The Report Maker will operate according to these settings.

Tables exported from Table Browser (see **2.9 Table Browser**) are automatically updated if the model has been changed or some of its parts were deleted.
Report Maker can handle several different reports for the same project. The structure of reports is displayed in a tree view on the left. The properties of the selected report item are shown on the right side of the window.
Use the arrows to move the selected item(s) up or down in the tree.
You can drag the selected items to a new location with the mouse; the exact insertion point and its level depth are indicated by a black horizontal line at the target position.
Clicking on any number button opens up the entire tree, but only to that depth.



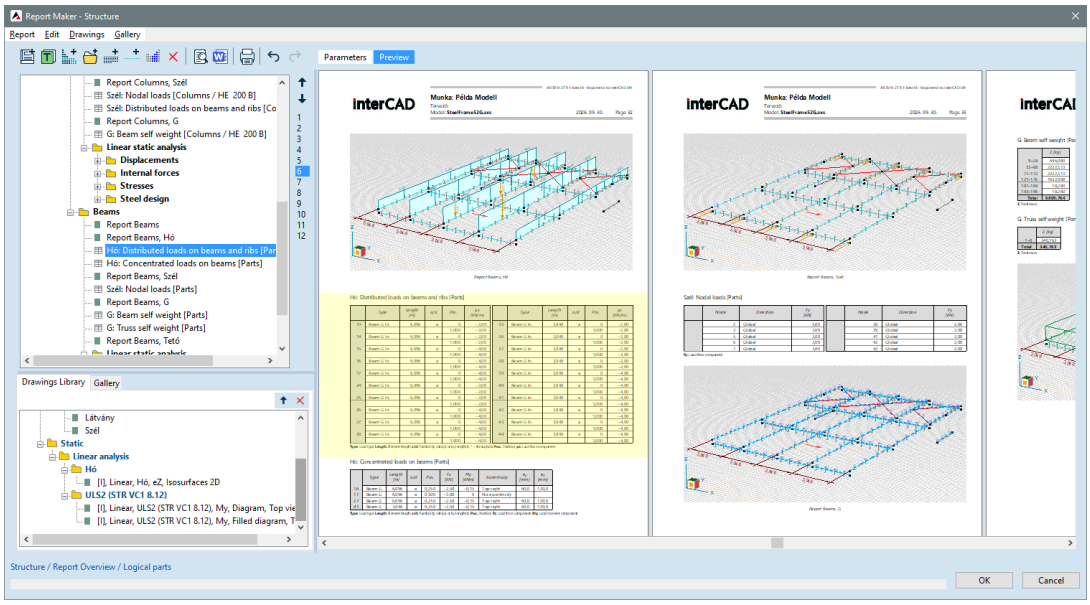
Parameters

When *Parameters* is selected, the parameters of the currently selected report item are displayed on the right-hand side (see below for details).



Preview

Selecting *Preview*, a report preview is displayed at the position of the item selected.
The window can be resized, allowing you to adjust the size of the preview. Just as in the *Report preview*, you can drag the preview image with the mouse, or by flicking it, make it scroll smoothly depending on the speed of the flick. The initial rendering of drawings takes some time, so at first scrolling may show some lagging.

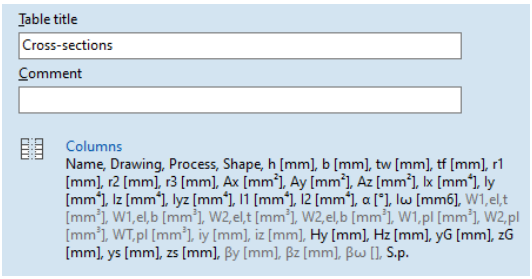


The selected item is displayed with a yellow background in the preview. Clicking on an image, table or folder name in the preview selects it both in the preview and the tree. Ctrl+click selects multiple items scattered throughout the preview and the tree; while Shift+click select elements in a continuous range starting from the previously selected element.
If you delete or add items, drag items to a new location, or modify text, titles or properties, the entire documentation is automatically recomposed.

Parameters

Folder If a folder is selected its name can be edited on the right.

Table If a table is selected, its comment text, column titles and other properties are shown. Click on the icon left to *Columns* to turn columns on and off. Title and comment can be edited but custom titles are not translated when changing report language.



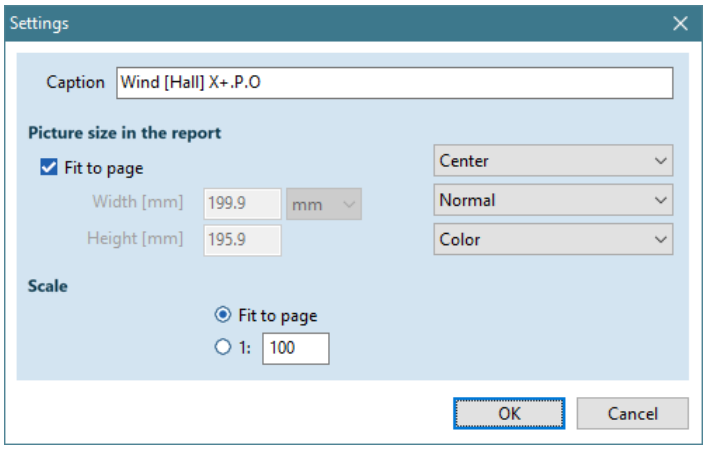
Text If a text block is selected the text is shown on the right. Click the button *Edit text...* to make changes.

Picture or Drawing If a picture or drawing is selected it is shown on the right. Its size, alignment and caption can be set by clicking the *Settings* button (see below).

Drawings Library By clicking the *Drawings Library* tab you can browse the saved drawings and add the selected ones to the report. Unlike the pictures in the Gallery these drawings are not graphics files, but view settings stored to recreate the drawing at any time. This way drawings will be automatically updated if we change and recalculate the model.
See in detail **3.6.10 Drawings Library** and **3.6.11 Save to Drawings Library.**

Gallery By clicking the *Gallery* tab you can browse the saved pictures (BMP, JPG, WMF, EMF) located in a folder named *Images_modelname* and add the selected ones to the report. This folder is automatically created as a subfolder of the model folder.
See in detail **2.10.5 Gallery**

Settings



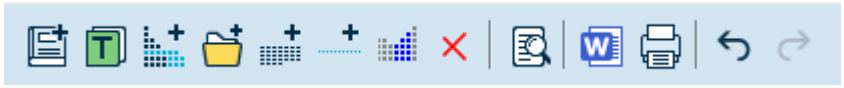
Click the *Settings...* button to change the caption, size, justification, rotation color mode or scaling of drawings.

You can save the current drawing on screen or the result tables in design modules with the function of *Edit\ Saving drawings and design result tables* in main menu.
See **3.2.12 Saving drawings and design result tables**

One or more selected pictures in the Gallery can be inserted into a report by selecting menu item *Gallery/Add pictures to the report* or clicking the arrow button above the Gallery or by drag and drop.

In printed reports Report Maker automatically builds a table of contents and inserts it to the beginning of the report. Tables are listed according to their titles. Text blocks are listed only if they were formatted using one of the Heading styles in the Text Editor. Pictures are listed only if they have a caption.

2.10.1 Report toolbar




Creates a new report
See **2.10.2 Report**


Creates a new report based on a template
See **2.10.3.1 Template-based reports**


Creates a new report based on filters
See **2.10.3.3 Filter-based report**


Inserts a folder
See **2.10.3 Edit**


[Ctrl]+[T]
Inserts formatted text
See **2.10.3 Edit**


[Ctrl]+[Alt]+[B]
Inserts a page break
See **2.10.3 Edit**


Selection filter
See **2.10.3 Edit**


[Del], [Ctrl]+[Del]
Deletes selected reports or report items
See **2.10.3 Edit**


[Ctrl]+[R]
Preview of the entire report
See **2.10.2 Report**


[Ctrl]+[W]
Exports the report as an RTF file
See **2.10.2 Report**



[Ctrl]+[P]

Print
See 2.10.2 Report



[Ctrl]+[Z]

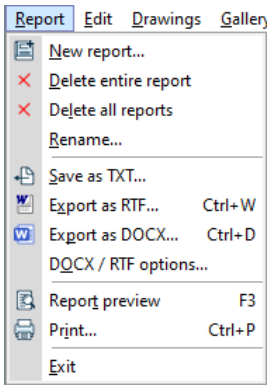
Undo
See 2.10.3 Edit



[Shift]+[Ctrl]+[Z]

Redo
See 2.10.3 Edit

2.10.2 Report



New report



Creates a new report. Report names can be 32 characters long.

Delete entire report / all reports



[Del], [Ctrl]+[Del]

Delete entire report deletes the current report (i.e., the report which contains the selected item).
Delete all reports deletes all reports.
Pictures used in the report are not deleted from Gallery.

Rename

Gives a new name to an existing report.

Save As TXT

Exports the report into a ASCII text file. Drawings or pictures are not included.

Export as RTF



Saves the report as *name.rtf* using the current template. If you save the file to a folder different from the model folder all picture files used in the report are copied to an automatically created subfolder ***Images_modelname***. It is necessary because pictures are only linked and not saved into the RTF document. To print the RTF report on a different machine make sure that picture files are also copied to a subfolder ***Images_modelname.***
Character and paragraph formatting of text blocks will be exported. The only exception is the character color. Tables will be exported as RTF tables. Table titles are formatted with Heading 3 style so it is easy to build a table of contents automatically using Microsoft Word.

Export as DOCX



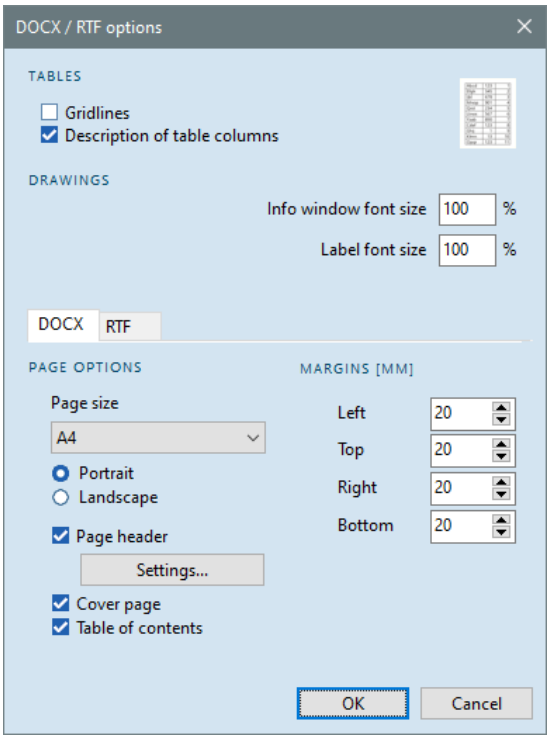
Export the documentation to a DOCX file. Unlike the RTF export, this export does not use a template file, but instead uses the same header and company logo settings as printing. All images are embedded. Table of contents can also be included.

DOCX / RTF settings
DOCX options

The dialog is to control the output page size, page orientation and margins, the display of the header and title page, and the generation of the table of contents.
The header content and the company logo can be customized with the *Settings...* button See **3.1.10 Page header**, **3.3.11.15 Report**

Gridlines of exported tables can be turned on/off. *Description of table columns* can be added to the end of tables (see below).

As the design calculations also contain formulae, these are inserted into the document line by line as graphical content, which allows sections to be moved or deleted, but not modified.



DOCX/RTF settings
RTF options

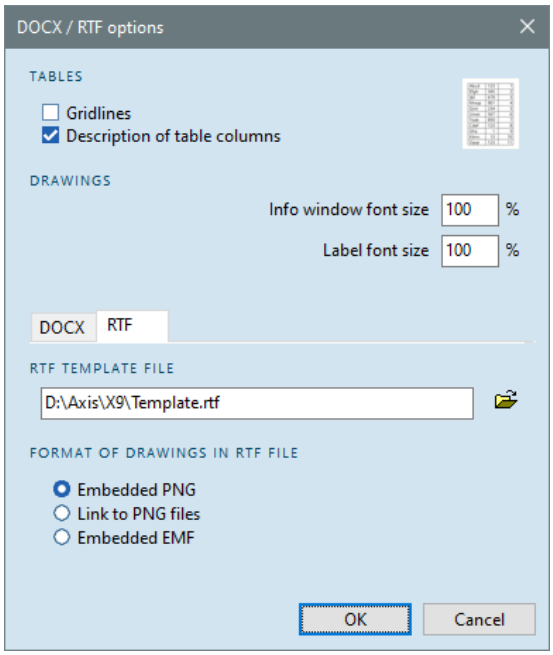
AXISVM saves reports to RTF files using a template (the default one is *Template.rtf* in the program folder). You can use other templates as well. When changing a template you can create your own cover sheet and header / footer for the report. Read the text of the template file carefully before changing it.

Format of drawings in RTF file can also be set:

Embedded PNG: Drawings are embedded into the file. This method provides portability with a smaller size due to lossless compression.

Link to PNG files: This option keeps the RTF file even smaller as drawings are stored in external files. Drawings appear only if pictures are present in an *Images_modelname* subfolder relative to the folder of the RTF file.

Embedded EMF: Drawings are embedded into the file as EMF (Enhanced Windows Metafile). It improves portability but can result in huge file size.



Label font size: Here the label font size used on exported drawings can be fine-tuned.

Gridlines: Gridlines of exported tables can also be turned on/off.

Description of table columns: Adds a description of columns at the end of each table. See *Print* below.

Report preview
[F3]



Displays a print preview dialog. You can set the zoom factor between *10%* and *500%* (*Page Width* and *Full Page* is also an option). Click the buttons or use the keyboard to move backward and forward between pages (**[Home]** = first page **[←]** = previous page, **[→]** = next page, **[End]** = last page.
Report preview can display multiple pages. **[PgUp]** **[PgDn]** steps back and forward according to the number of pages displayed.
The image can be dragged to left or right and continuous (inertial) scrolling is also implemented (like on smartphones). Generating drawings the first time when they are needed can make the scrolling lag.

Print
[Ctrl]+[P]

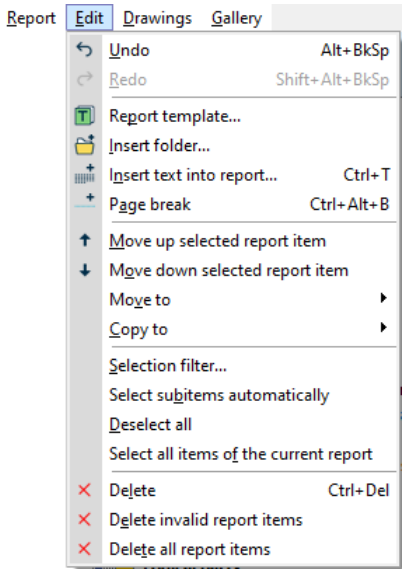
A dialog to set printing parameters and print a report. The options are the same as the table printing options. Turning on ***Description of table columns*** an explanation of columns appears at the bottom of each table.

Name: Cross-section name; **Process:** Manufacturing process; **h:** Cross-section height; **b:** Cross-section width; **tw:** Web thickness; **tf:** Flange thickness; r_1, r_2, r_3: Rounding radius; **Ax:** Cross-section area; **Ay, Az:** Shear area; **Ix:** Torsional inertia; **Iy, Iz:** Flexural inertia; **Iyz:** Centrifugal inertia; I_1, I_2: Principal flexural inertia; **α:** Principal directions; **Iω:** Warping constant; $W_{1,el,t}$, $W_{1,el,b}$, $W_{2,el,t}$, $W_{2,el,b}$: Elastic modulus; $W_{1,pl}$, $W_{2,pl}$: Plastic modulus; i_y, i_z: Radius of inertia; **Hy:** Dimension in local y direction; **Hz:** Dimension in local z direction; y_G: y coordinate of the center of gravity; z_G: z coordinate of the center of gravity; y_s: y coordinate of the shear (torsion) center relative to the center of gravity; z_s: z coordinate of the shear (torsion) center relative to the center of gravity; **S.p.:** Stress calculation points;

Exit

Quits the Report Maker.

2.10.3 Edit



Some of the functions in the *Edit* menu are also available in the popup menu after clicking right mouse button on a report item.

Undo

Undoes the effect of the previous command.

Redo

Executes the command which was undone.

Report template

See 2.10.3.1 Template-based reports

Insert folder

Inserts a new folder into the tree, below the current item. The current folder name appears on the right side under the folder icon.
The number of expanded levels (1-7) of the report tree can be set with the level-adjustment bar.

Insert text into report
[Ctrl]+[T]

Starts a built-in Text Editor to create a new text block. The formatted text will be inserted after the selected report item.

Page break
[Ctrl]+[Alt]+[B]

Inserts a page break after the selected report item.

Move up/down selected report item

Moves up/down the selected report item by one.

Move to / Copy to

Moves / copies the selected report item to the end of another report.

Selection filter

Determines which types of report items can be selected (report, table, drawing, picture, text, page break, folder).

Select subitems automatically If you turn this checkbox on and select a folder all subitems will be selected automatically.

Deselect all Deselects all selected items in the documentation.

Select all items of the current report Every report item of the current report will be selected.

Delete

[Del], [Ctrl]+[Del] Deletes the selected report item (text block, picture, table, page break). If the current selection in the tree is a report it deletes the entire report.

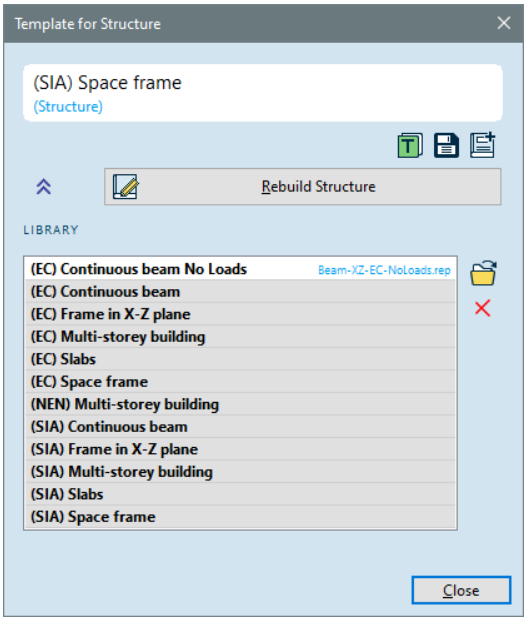
Delete all report items Deletes all items from the current report but does not delete the report itself.

2.10.3.1 Template-based reports

Report templates can be used to generate reports based on certain pre-sets, filters and preferences. Generated reports consist of drawings and tables. Templates can be saved as files and reused to generate a report for other models.
The range of included elements, model data and result components can be set by filters.

Clicking on the icon opens the template administrator dialog. If the current report was generated from a template, the template is loaded. If the current report was not based on a template a new default template appears.
If the dialog is opened up (see below) a list of predefined (grey background) and user-defined (on white background) templates appears in the lower part. Clicking on a list item loads the template.
Templates are listed with their names specified in the template editor (See **2.10.3.2 Editing a template**).

Templates are saved to and loaded from the following folder:
c:\ Users \ [user name] \ AppData \ Roaming \ AXISVM \ [version number] \ Templates.



Rebuild report If the model has been extended and the report should be updated (for example the steel member design has been completed) click on the *Rebuild* button.
Any report item inserted by the user will be removed.

This button is to open or close the bottom part of dialog with the list of templates.

Toolbar icons have the following functions.

Edit template
Content filters and views used on drawings can be edited. See **2.10.3.2 Editing a template**

Save this template to a file
The current template can be saved to a file to reuse it in another model. Report templates have a **.rep* extension and are saved to the templates folder described above.

Create a new report
Builds a new report based on the current template.

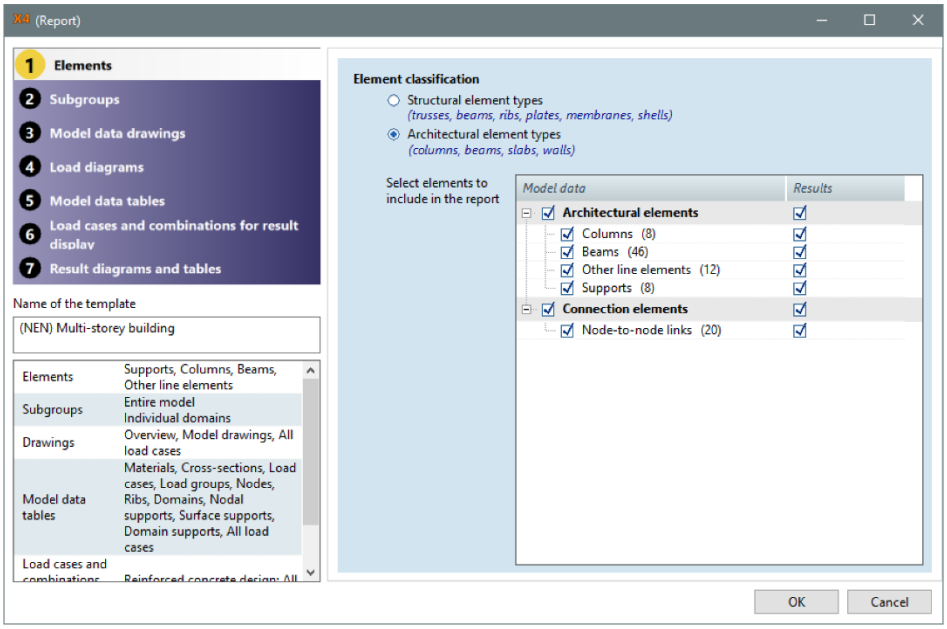
Import...
Template files *(*.rep)* can be imported from other locations to the folder of the template library. Imported templates can be used the same way as the predefined ones.

Delete
Removes the selected template from the template library.

2.10.3.2 Editing a template

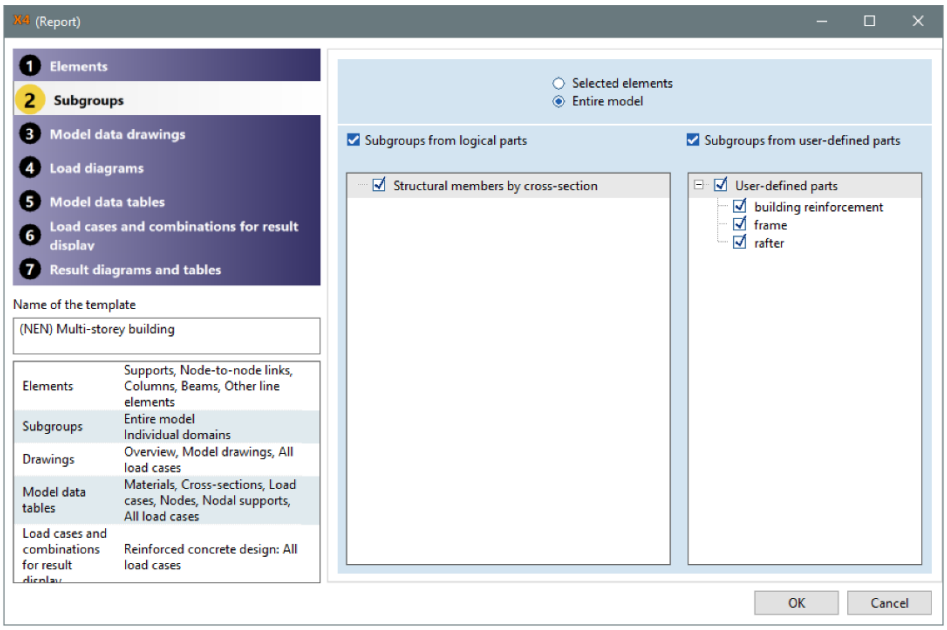
It is a complex task to create a report template. The steps of this process are listed on the left. Clicking on these items we can edit filter options and other settings. An edit box under the list allows entering a name for the template. The template administrator dialog lists templates by their names.

Elements



The first step is selecting element types to include in the report and choosing the element classification. If *Structural element types* is selected, elements will be classified by their finite element type. If *Architectural element types* is selected, elements will be classified by their architectural type (determined from the element geometry).
Element data and results can be selected separately for reporting.
Next steps will display tables and drawings based on this selection.

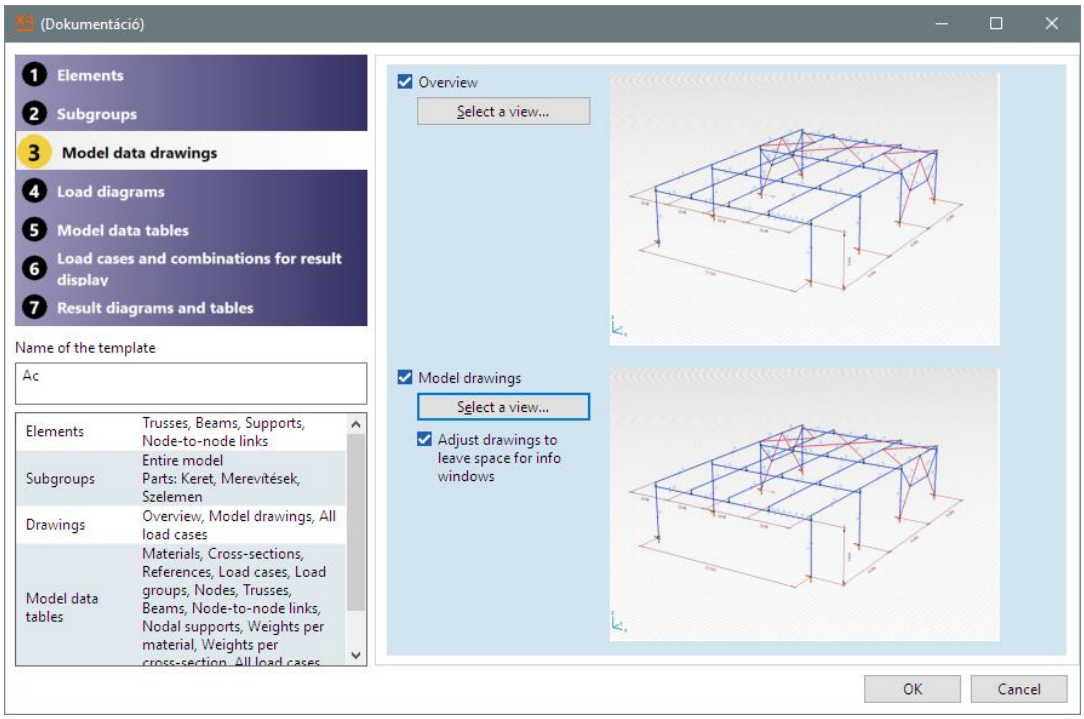
Subgroups



The second step is to set the subgroups for the reported elements. A complete sub-report will be built for each subgroup.
If the option *Selected elements* is activated only elements selected before opening Report Maker will be reported.
If the option *Subgroups from logical parts* is activated subgroups can be created from domains with the same thickness, structural members with the same cross-section or storeys (only the selected storeys will be reported).

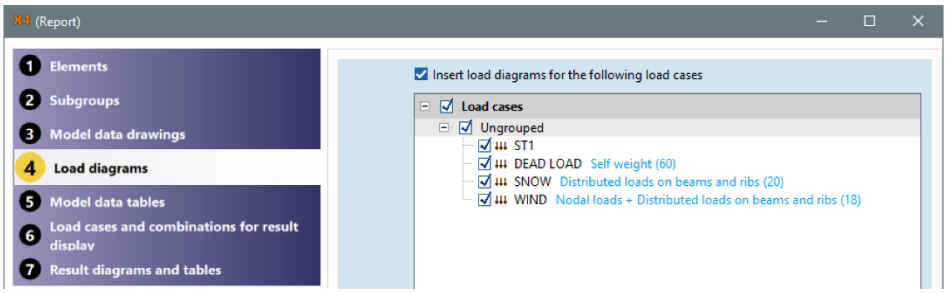
If *Report domains one by one* is checked each domain will be reported separately. Internal domains (being entirely within another domain) are reported with the outer domain even in this case.
If the *Entire model* is selected subgroups can be created also from user defined parts.

Model data drawings



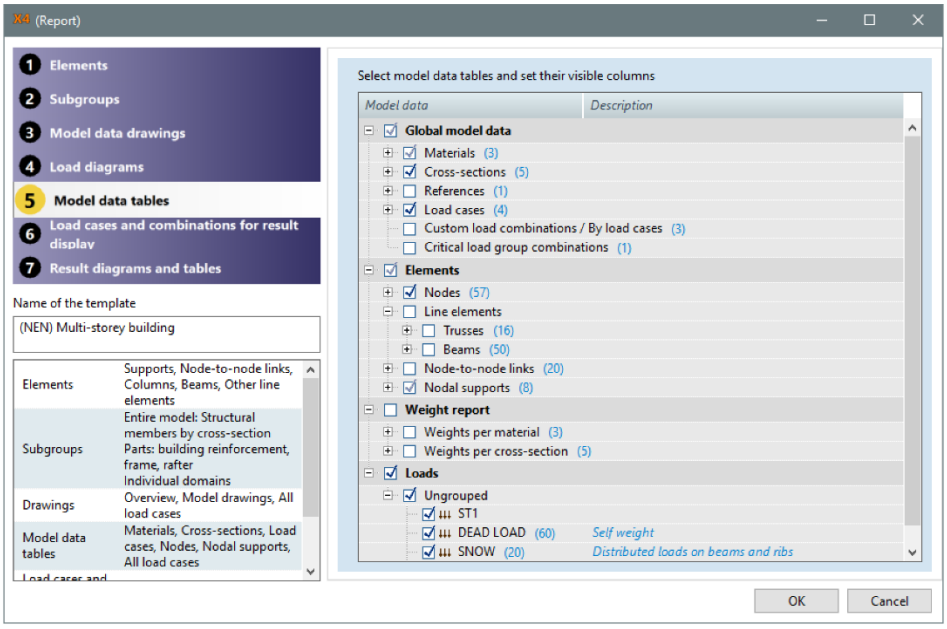
If *Overview* is checked a view of the model will be inserted at the beginning of the report. Click *Select a view...* to choose a view from the Drawings Library.
Check *Model drawings* if you want to include automatically created load and result diagrams. Click *Select a view...* to choose a view from the Drawings Library.
Generated diagrams will inherit all settings of the selected drawing (point of view, status of graphics symbols, numbering, labelling etc.) with minor adjustments. If no drawing is selected (e.g., the Drawings Library is empty) drawings will follow the current view in the active window.
If a diagram is displayed only on a certain part of the structure the view is zoomed to fit drawing to the window. The point of view and the status of graphics symbols will remain unchanged.
Checking *Adjust drawings to leave space for info windows* sets the zooming to fit the drawing within the maximum rectangular space available among the info windows.

Load diagrams



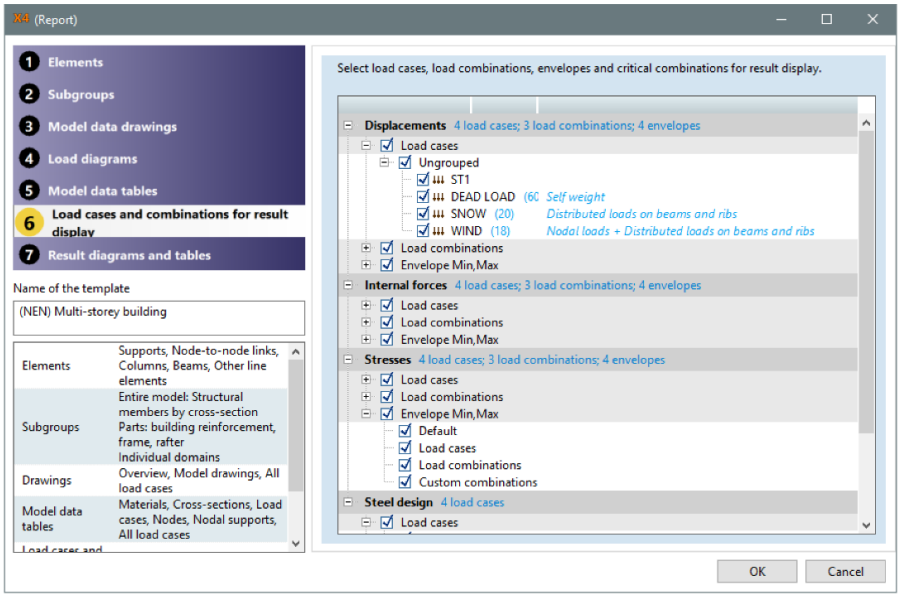
If *Insert load diagrams for the following load cases* is checked, select load cases to add their load diagrams to the report. Load diagrams will be generated from the view set for *Model drawings* in the previous step.

Model data tables



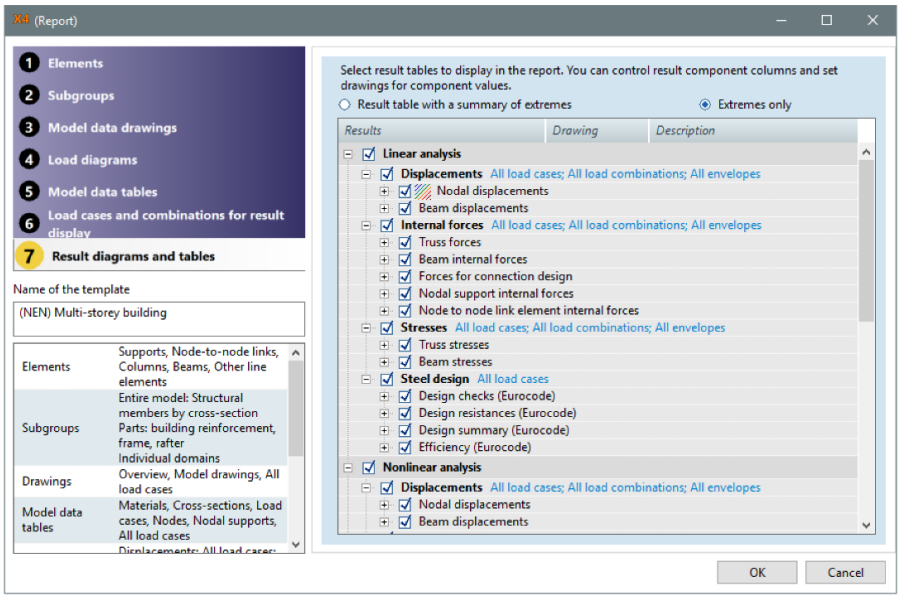
Select model data tables to add to the report. You can set the visible columns for tables to fine tune the report content.
Under *Elements* you will find only those elements you selected in the first step.
Under *Loads* you can select load cases to add their load data tables to the report.

Load cases and combinations for result display



AXISVM provides a huge number of results. It is important to decide which load cases, combinations, envelopes or critical combinations should contribute to the report for displacements, internal forces, stresses, reinforcement values, steel or timber design checks.

Result diagrams and tables



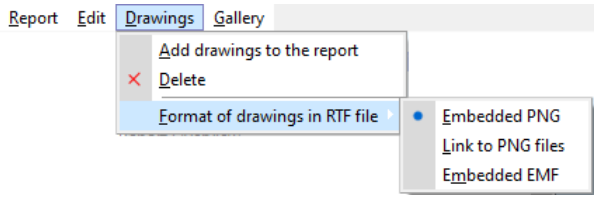
The last step is to select from the possible result tables and control the visibility of their columns. It is also possible to generate result diagrams for result components. Click in the *Drawing* column in a row of a result component. You can choose a drawing mode for that component from the dropdown list even if you leave the checkbox unchecked (hiding the respective column). Result diagrams will be generated from the view set for *Model drawings* and will be inserted before the table.

2.10.3.3 Filter-based report



Creating reports from filter options is an earlier method which is less configurable and its content is not updated but can be used in simple cases. Select element and load types, load cases and result components from the filter tree on the left to control report composition. The resulting report appears in the tree on the right. Its individual items can be checked or unchecked. Only checked items will be included in the generated report.

2.10.4 Drawings



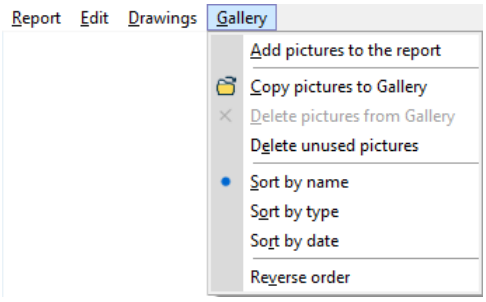
Add drawings to the report

Inserts the selected drawing(s) from the Drawings Library into the selected report. Place of insertion is determined by the selected item of the report tree. Effect of this function is the same as that of the  button on the Drawings Library tab.

Format of drawings in RTF file...

See 2.10.2 Report

2.10.5 Gallery



Add pictures to the report — Inserts selected pictures into the current report.

Copy pictures to Gallery — You can copy bitmaps *(.BMP, .JPG, .PNG)* and Windows Metafiles *(.WMF)* to the folder *Images_modelname*.

Delete pictures from Gallery — Deletes selected pictures from the Gallery. Files are permanently deleted.

Delete unused pictures — Deletes pictures which are not used in the reports.

Sort by name / type / date — Gallery sorts pictures by filename / by type *(.BMP, .EMF, .JPG, .WMF)* or by date.

Reverse order — If checked pictures are sorted in descending order. Otherwise pictures are sorted in ascending order.

2.10.6 Gallery and Drawings Library Toolbars



You can perform certain tasks faster using these small toolbars.



Deletes selected pictures or drawings from the *Gallery/Drawings Library*.



Inserts selected pictures or drawings into the current report.
Place of the insertation is determined by the selected item in the report tree.



Copies pictures from other locations to the Gallery. This function is not available on the Drawings Library tab.

2.10.7 Text Editor

After selecting *Insert text to report* a formatted text can be created in a simple WordPad-like text processor.

File

Open **[Ctrl]+[O]** — The main purpose of this function is to load a Rich Text file written in Text Editor. If you open an RTF file created in another word processor it may contain special commands (e.g., tables, paragraph borders, Unicode characters) which are not supported this simple editor. As a result you may get a series of rtf control commands instead of formatted text.

Save **[Ctrl]+[S]** — Saves the text into an RTF file.

Exit — Quits Text Editor.

Edit

Undo / Redo **[Alt]+[BkSp] / [Shift]+[Alt]+[BkSp]** — Undoes / redoes the last editing action.

Cut **[Ctrl]+[X]** — Cuts the selected text and places it to the Clipboard.

Copy **[Ctrl]+[C]** — Copies the selected text to the Clipboard.

Paste
[Ctrl]+[V]
Pastes the content of the Clipboard at the current position.

Find
[Ctrl]+[F]
You can search for any text in the document. You can search from the beginning or from the current position. You can search whole words only and turn on and off case sensitivity.

Find next
[F3]
If a match was found you can get the next match with this function.

Select all
[Ctrl]+[A]
Selects the entire text.

Character

Bold
[Ctrl]+[B]
Applies bold formatting to the selected text.

Italic
[Ctrl]+[I]
Applies italic formatting to the selected text.

Underline
[Ctrl]+[U]
Applies underline formatting to the selected text.

Color
[Ctrl]+[Alt]+[C]
Sets the character color of the selection.

Paragraph

Left justify
[Ctrl]+[L]
Justifies the selected paragraphs to the left.

Centered
[Ctrl]+[E]
Justifies the selected paragraphs to the centerline.

Right justify
[Ctrl]+[R]
Justifies the selected paragraphs to the right.

Bullet
[Ctrl]+[Alt]+[U]
Places bullets before the selected paragraphs.

2.11 Stories



See in detail **3.3.4 Storeys**

2.12 Layer Manager



See in detail **3.3.3 Layer Manager**

2.13 Drawings Library



See in detail **3.6.10 Drawings Library**

2.14 Save to Drawings Library



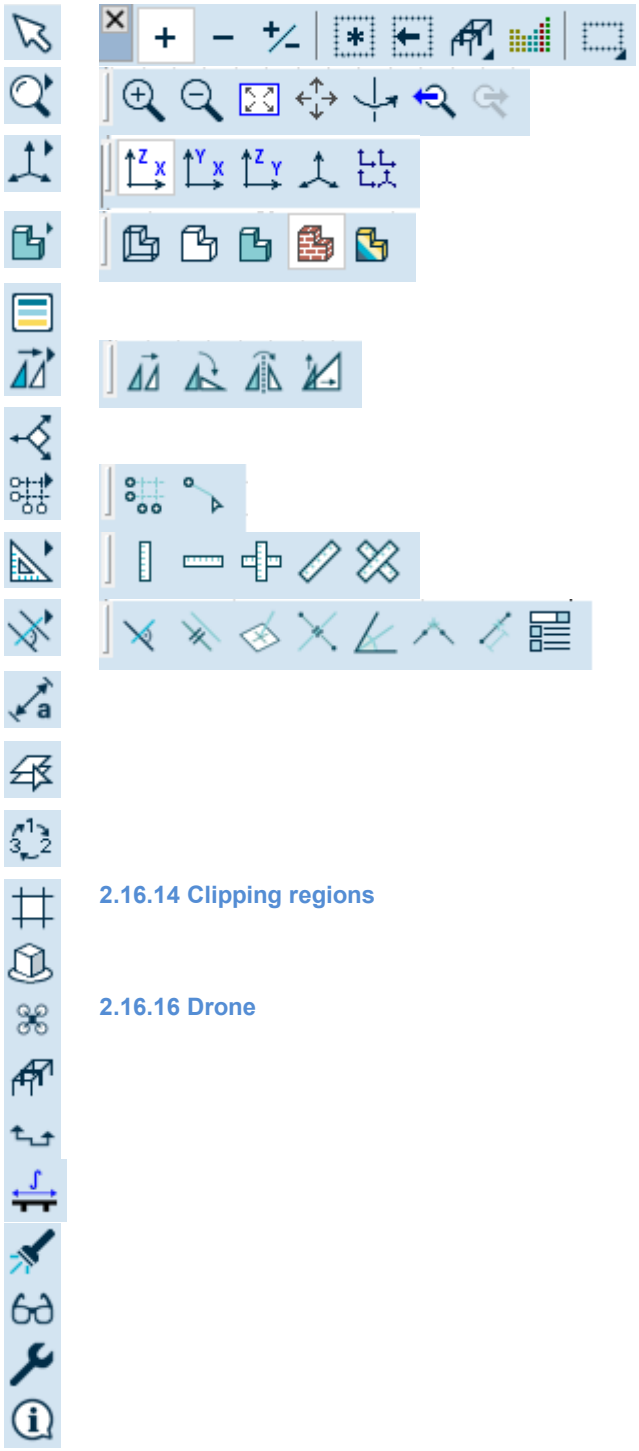
See in detail **3.6.11 Save to Drawings Library**

2.15 Export current view as 3D PDF



Saves the current view as a 3D PDF file.
The result is a PDF file containing a 3D view. Adobe Acrobat Reader supports zooming and rotating the model since the updated 8.1 version.
The file contains the actual reinforcement as well.

2.16 The Icon Bar



2.16.14 Clipping regions

2.16.16 Drone

Dragging and docking the Icon bar and the flyout toolbars

The left-side icon bar and any flyout toolbar can be dragged and docked.

Dragging and docking of the Icon bar

If you move the mouse over the handle of the Icon bar (on its top edge), the cursor will change its shape (moving). You can drag the Icon bar to any position on the screen. If you drag the Icon bar out of the working area through its top or bottom edge the Icon bar becomes horiozontal. If you drag it to the left or right edge it becomes vertical.

If the Icon bar is horizontal you can dock it at the top or at the bottom. You can change the position and the order of docked toolbars by dragging. In the Cross-Section Editor and in Beam and Column Reinforcement dialogs the Icon bar cannot be docked. Closing a floating Icon bar restores its original position docked on the left.

Dragging and docking of flyout toolbars

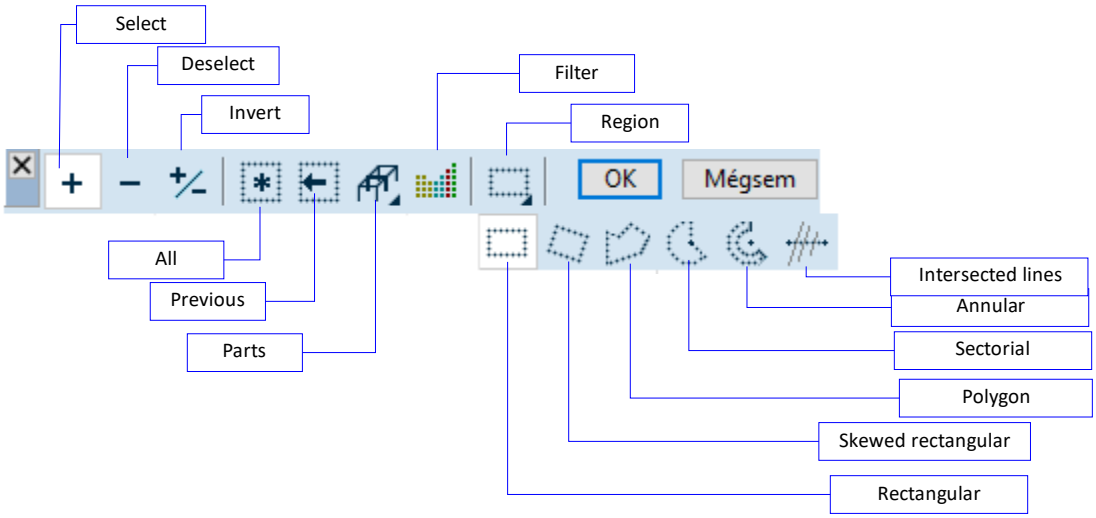
You can also separate flyout toolbars from the Icon bar by dragging their handle. Closing or dragging them back to the Icon bar restores their original position. Floating flyout toolbars can be docked at the top or at the bottom.

☞ **The Icon bar and the flyout toolbars can be restored to their original position by selecting Settings / Toolbars to default position from the menu.**

2.16.1 Selection



Activates the selection mode and displays the selection icon's bar.



Lets you select a set of entities (nodes (points), lines, finite elements and loads) for processing. When you execute commands you can use the Selection icon to specify the entity set to which to apply the command to. If the *Parts* check box (See section **2.16.17 Parts**) is enabled the selection will refer only to the active (visible) parts.
You can change the view settings or continue selection in another window pane during the selection process. These allow you to select elements in the most convenient view. The selected entities are displayed in magenta in the graphics area.
The selection process is considered finished when the **OK** button is pressed.

Select



Adds the currently selected entities to the set of selected entities.

Deselect

Removes the currently selected entities from the set of selected entities.

Invert



Inverts the selection status of the currently selected entities.

All



Applies the current selection mode (add, remove, or invert) to all filtered entities.

☞ **All selection commands change the selection status of elements chosen in the selection filter (see below).**

Previous



Restores the previous selection set.

Selection of parts



Clicking the button and a part from the list will select elements of the chosen part.

Filter



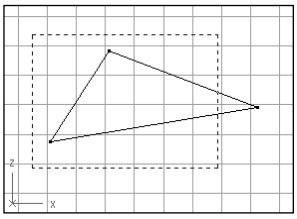
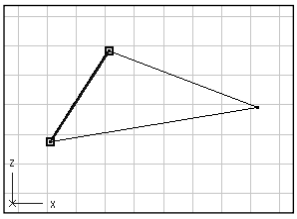
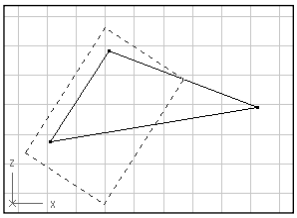
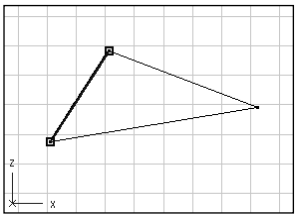
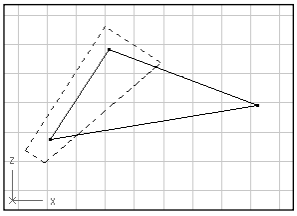
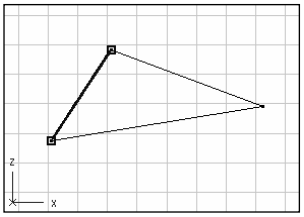
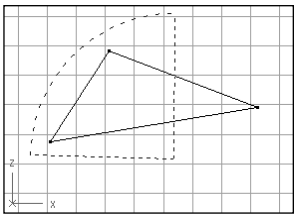
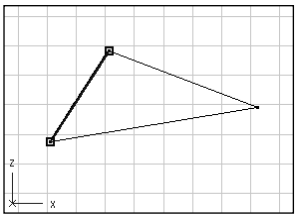
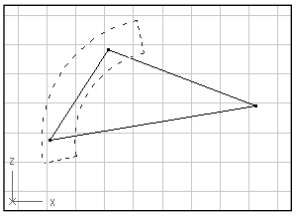
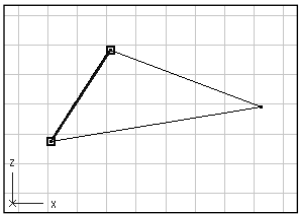
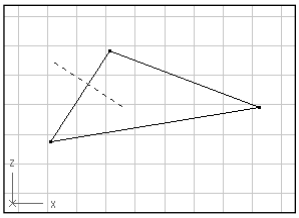
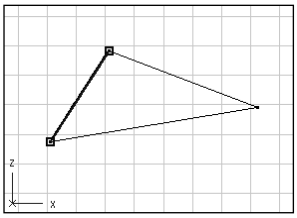
Lets you specify filtering criteria to be used during selection. Check element types to select.
Property filtering lets you apply further criteria (beam length, cross-section, material, surface thickness, reference). All elements matching certain criteria can also be selected using the smart command line (see **2.17 Smart command line**).

Method



Selects entities using different methods (selection shapes). Rectangular, skewed rectangular, sectorial or ring selection shapes are available. In the followings examples of the application of various selection shapes are provided:

When drawing a rectangular frame, the following rules apply:
Dragging the selection frame from left to right selects elements falling entirely within the frame.
Dragging the selection frame from right to left selects elements that are not entirely outside the frame (i.e., they are inside the selection rectangle or intersect it).

	Selection:	*Result:*
Rectangular 		
Skewed rectangular 		
Polyline 		
Sectorial 		
Annular 		
Intersected lines 		

OK Ends the selection, retaining the selected set for use.

Cancel Ends the selection, discarding the selected set.

☞ **If an entity is hidden by another entity you cannot select it by simply clicking on it. In such a case, you have to change view to select it.**

The selected nodes are marked with a surrounding magenta rectangle. Sometimes it is necessary to double-select nodes. In this case these nodes are marked with an additional blue rectangle surrounding them.

Selections can also be made, without using the Selection Icon Bar. Pressing and holding the **[Shift]** button while selecting with the will add entities to the selection and pressing and holding the **[Ctrl]** button while selecting with the will remove entities from the selection.

Double selections can be made by pressing and holding the **[Alt]** button while double clicking on the entities with the .

During selection you can modify the view or switch to another view or perspective.

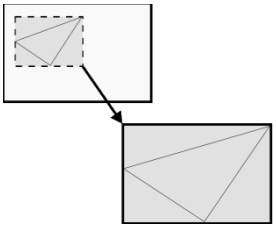
2.16.2 Zoom icon bar



Zoom in



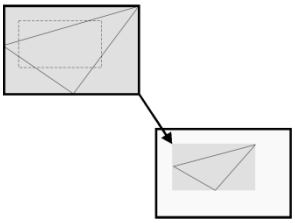
Displays an area of the model drawing specified by two points (two opposite corners) on the graphics area defining a rectangular zoom region. As a result, the apparent size of the model displayed in the graphics area increases.



Zoom out



Displays the model drawing from the graphics area on the area specified by two points (two opposite corners) defining a rectangular zoom region. As a result, the apparent size of the model displayed in the graphics area decreases.



Zoom to fit



Scales the drawing of the model to fit the graphics area, so you can view the entire model.

Pan



Moves the drawing. Press and hold the left button of the while moving the mouse, until the desired position of the drawing is obtained on the screen.

Quick Drag:

You can use the mid mouse button to drag the model drawing at any time (without the the *Pan* icon).

1. Click the *Pan* icon.
2. Drag the model to its new position.

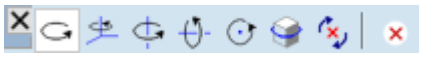


This cursor shape indicates that you can pan the model.

Rotate



After clicking this icon you can rotate the model around the centre of the encapsulating block of the model by dragging. During the rotation the following pet palette appears at the lower part of the screen:



Rotation methods:



Free rotation around the horizontal axis of the screen and the global Z axis.



Rotation around the global Z axis.



Rotation around the vertical axis of the screen.



Rotation around the horizontal axis of the screen.

Rotation around an axis perpendicular to the screen.

Rotation around the bounding box of on-screen elements

Rotation around a selected point. Click on a point to use it as the center of rotation.

Controls the display of the rotation center symbol.



This cursor shape indicates that you can rotate the model.

Undo view / Redo view

Undoes / redoes the action of up to 50 view commands.

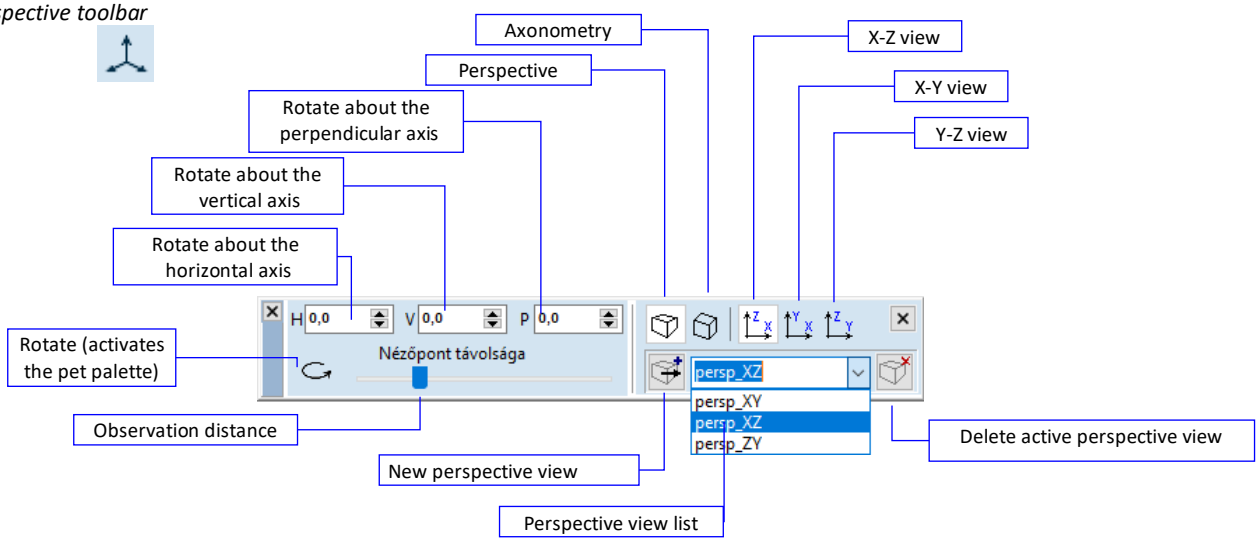
2.16.3 Views

Displays the projection of the model on the X-Z plane (front view).

Displays the projection of the model on the X-Y plane (top view).

Displays the projection of the model on the Y-Z plane (side view).

Perspective toolbar



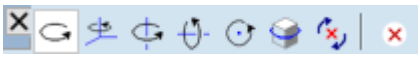
Sets the parameters of the perspective display. The proper view can be set by rotating the model drawing around the three axes, and by setting the observation distance. Rotation angles can be set with a precision of 0.1 degrees. You can assign a name to each setting that you want to save for later use. Type a name into the combo and click on the icon on the left of the combo to save the settings. To delete a perspective setting choose it from the dropdown list and click on the Delete icon on the right side of the combo. Palette settings are stored.

Observation distance

Observation distance is the distance between the viewpoint and the centre of the encapsulating block of the model.

Rotation

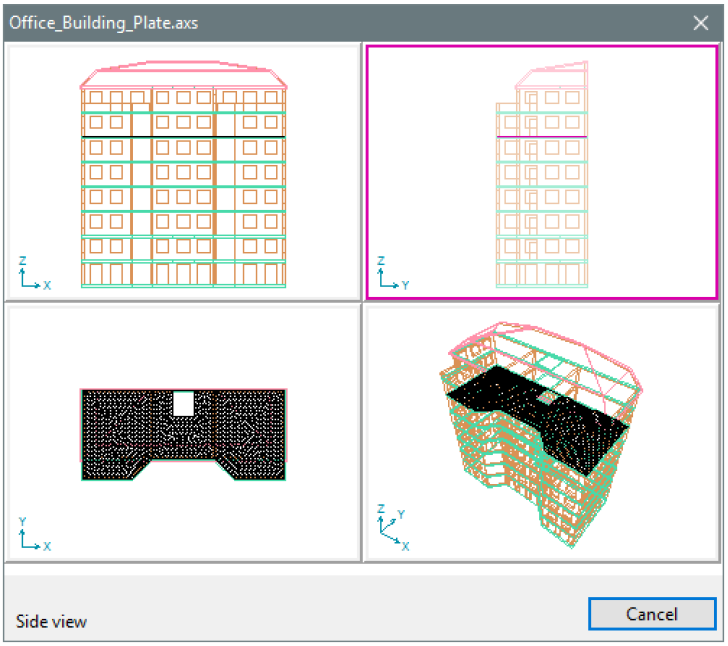
After clicking on the rotate icon a pet palette appears as described earlier (**Zoom***Rotate*).



Views, perspective

Displays three projection views and the perspective view of the model, and allows you select the view that you want to display. Click the view you want to select.

X-Z: Front view, Y-Z: Side view, X-Y: Top view

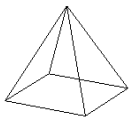


2.16.4 Display mode



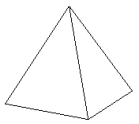


Wireframe:
Displays a wireframe model drawing. In this mode the axis of the line elements and the mid-plane of the surface elements are displayed.



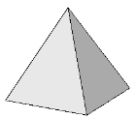


Hidden:
Displays a wireframe model drawing with the hidden lines removed.



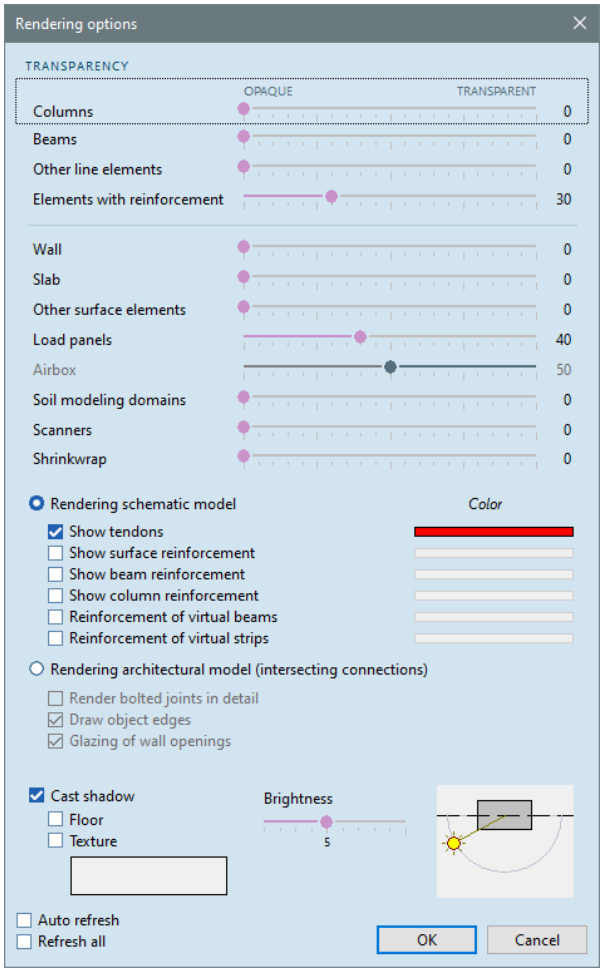


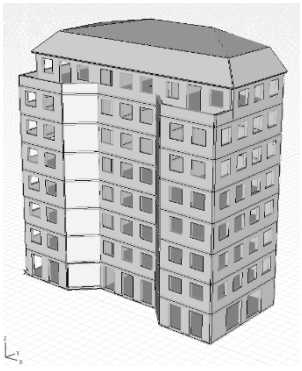
Rendered:
Displays a rendered model drawing. The line elements are displayed with their actual cross-section and the surface elements with their actual thickness. The elements colors are displayed corresponding to colors assigned to their materials. Rendered view is smoother and shows the details of thin-walled cross-sections.



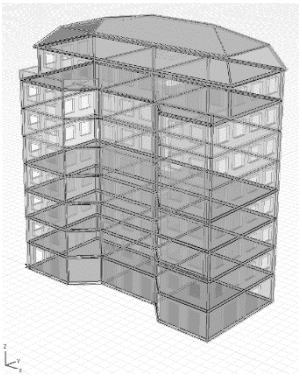
Rendering options

In *View / Rendering options...* transparency of element types can be set. Element types are determined by geometry.
Vertical line elements are considered to be columns, horizontal ones are handled as beams, horizontal domains as floors, vertical domains as walls.





Opaque



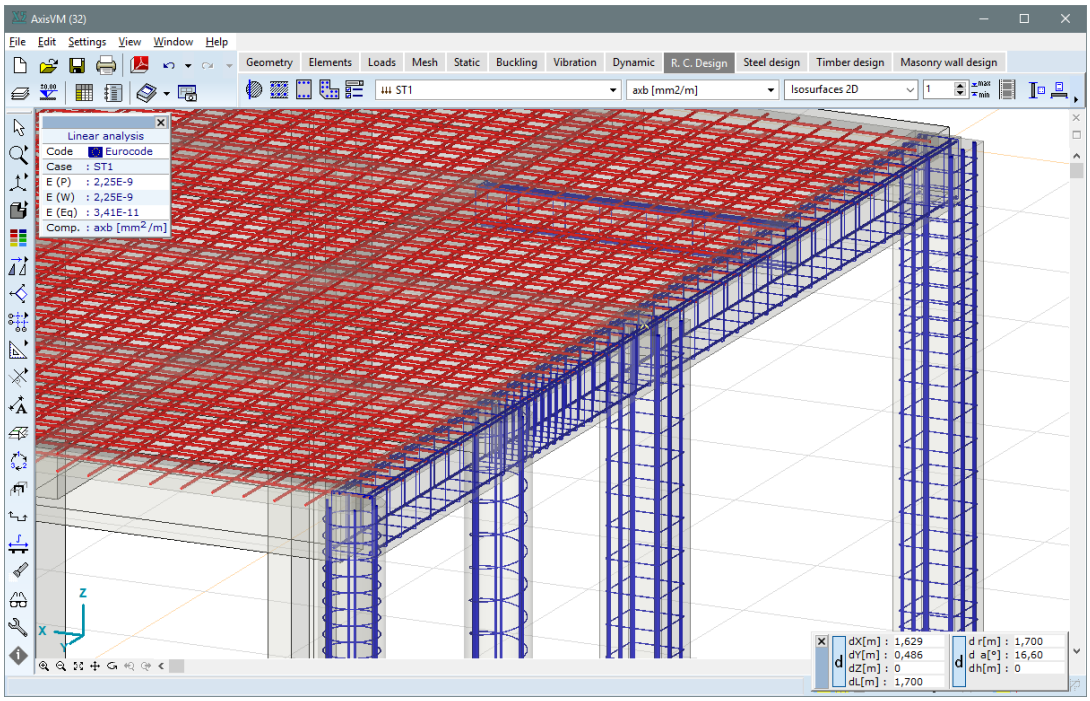
Transparent

Rendering type

Two rendering types are available:

Rendering schematic model

Realistic display of structures. Display of tendons and surface/column/beam reinforcement bars is optional. Click on the color rectangles to change colour.

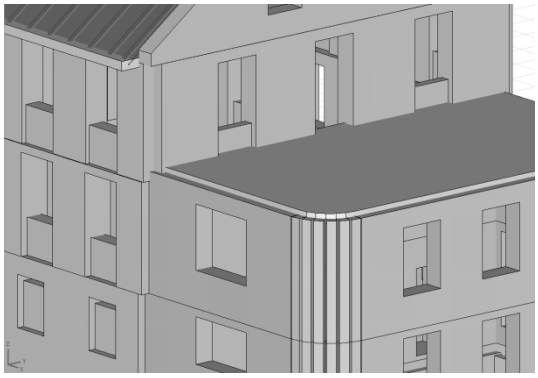


Rendering architectural model

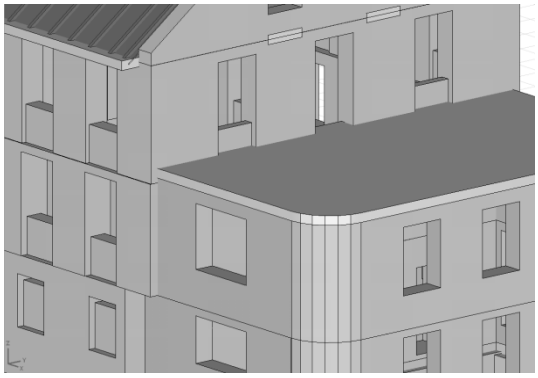
Instead of drawing the structural framework this rendering mode intersects connections getting closer to the final look of the model.

Render bolted joints in detail turns on detailed rendering of designed bolted joints.

Draw object edges turns on/off object edges.



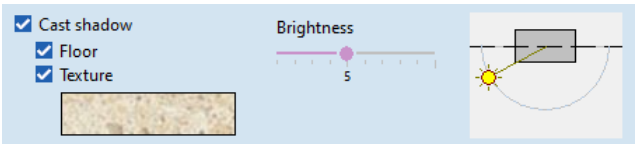
Schematic model

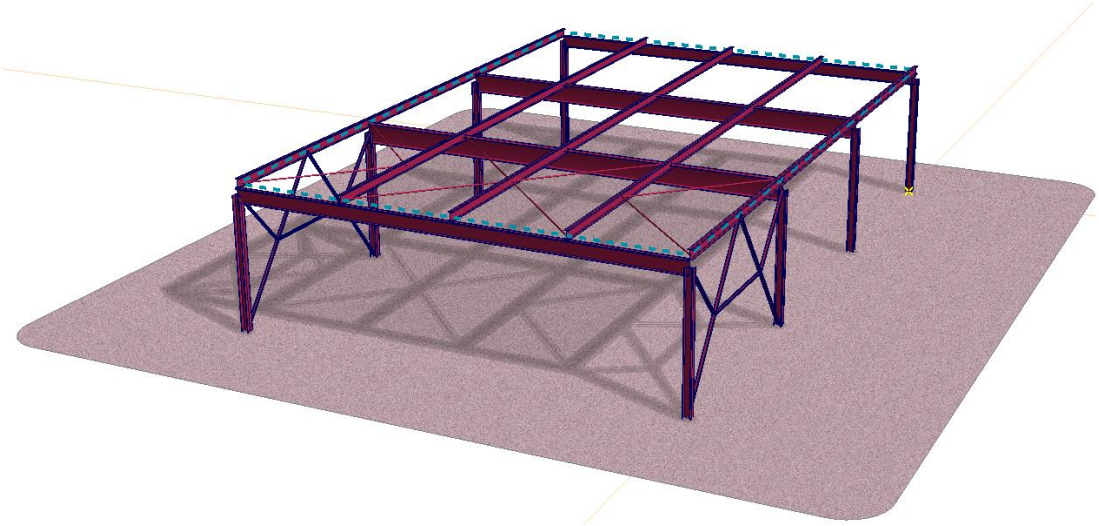


Architectural model

When *Cast shadow* is enabled, a floor plane appears in the viewport, showing the model's cast shadow. The floor plane is merely a visual element; it is not part of the model. A texture can also be displayed on the floor plane (see below), even if the model's elements are not otherwise textured. The higher the brightness, the greater the contrast of the shadow.

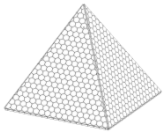
Regardless of the *Cast shadow* settings, both the brightness and the direction of the light (and thus the shadow) can be adjusted. Drag the yellow light source symbol in the schematic diagram to a new position to change the direction of the light.

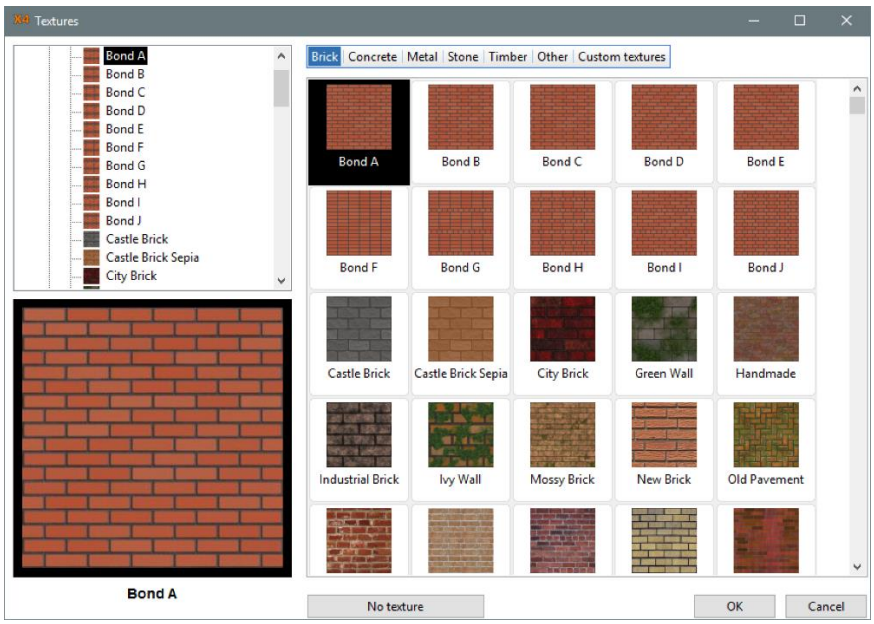






Texture: A rendered view using textures assigned to individual materials. Textures can be assigned to materials by clicking the *Texture* field in the table of materials or in the material database and choosing a texture from the library of textures. It contains predefined textures and let the user define custom textures as well. If more than one row is selected in the table texture will be applied to all selected materials.

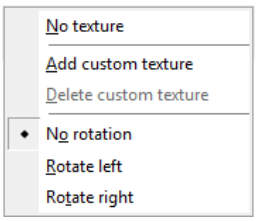




Branches of the tree view on the left and the horizontal list above the texture thumbnails show the material types (brick, concrete, metal, stone, timber, other). The last type (custom) is for the user-defined textures. Textures of the current type are displayed as thumbnails. The selected texture appears in the preview window with a thick black frame.

Popup menu

After clicking the texture with the right mouse button, a popup menu appears with the following functions:



Removing the texture from the material
Defining or deleting a custom texture
Rotation settings

No texture Removes the texture from the current material

Add custom texture 24-bit True Color bitmaps (JPG or BMP) can be converted to textures of 64 x 64, 128 x 128 or 256 x 256 pixels. If the bitmap was not rectangular it will be cropped into a rectangle.

Delete custom texture

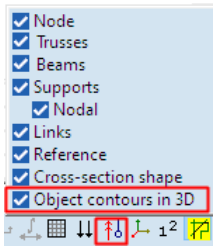
Predefined textures cannot be deleted from the library, only the assignment can be removed. User-defined textures in the *Custom* category can be deleted.

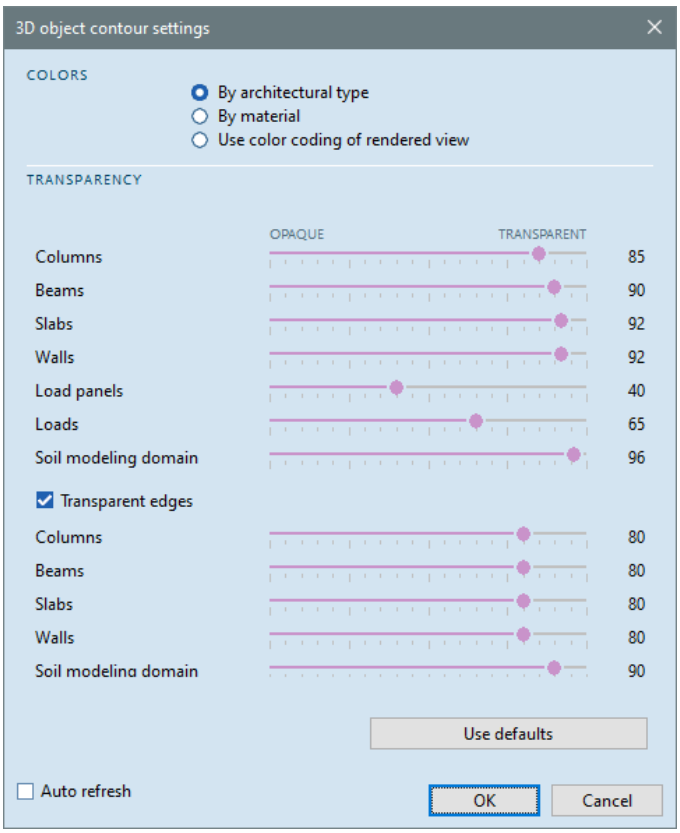
Rotation settings

Textures are mapped to the elements according to their local coordinate system. Sometimes it can lead to undesirable results (e.g., in case of brick walls). Texture rotation can solve these problems without changing the local system of elements. By default, the textures are not rotated. The other two options are *Rotate left* and *Rotate right* rotating the bitmap by 90°. Rotation is indicated in the table by a < or > character appearing at the end of the texture name.

3D object contour settings

Turning on display of 3D object contours (see the speed button below) a semi-transparent solid model will also be displayed. Transparency of elements can be configured in *View / 3D object contour settings*.





Colors

By architectural type — The solid model will be displayed using the colors assigned to walls, slabs, beams, etc.
See **3.3.11 Preferences / Graphical symbols**

By material — The colors assigned to the material will be used.

Use color coding of rendered view — Colors will be determined by the current color coding set for rendered view.

Transparency

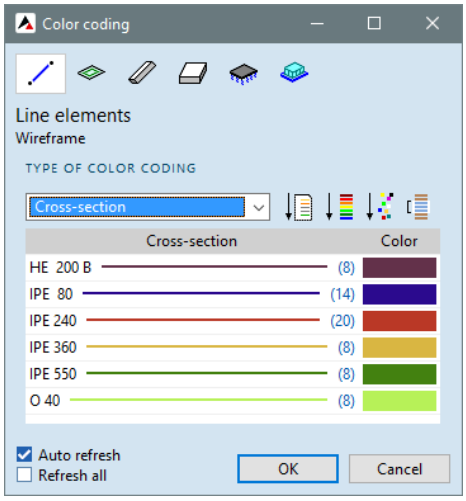
Move trackbars to set the desired transparency. Click *Use defaults* to return to the default values. Check *Auto refresh* to see the changes immeadiately on the model view.

Results displayed on the rendered model. If a result component is displayed with the Solid model display mode (see **6.1 Static**), results will color the rendered model where elements are displayed with their actual physical dimensions. For settings, see **3.3.11.11 Display**.

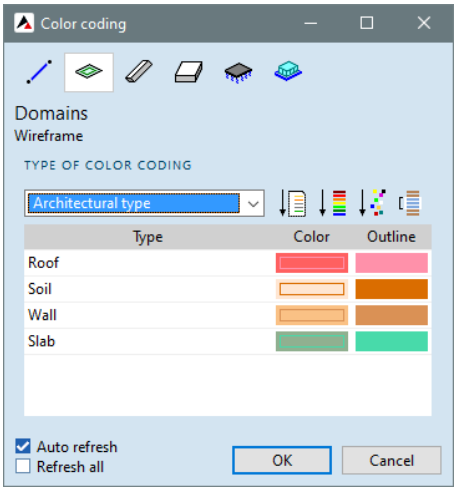
2.16.5 Color coding

Color coding helps to get an overview of element properties. Different color coding can be set for the rendered and wireframe display modes. Type of color coding can be chosen from a dropdown list.
The program automatically associates different colors to different properties, but colors can be changed.

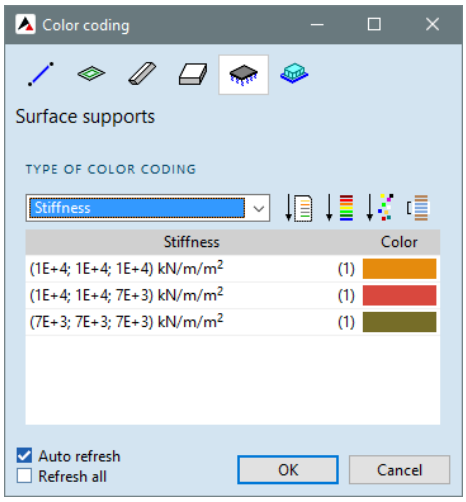
Line elements



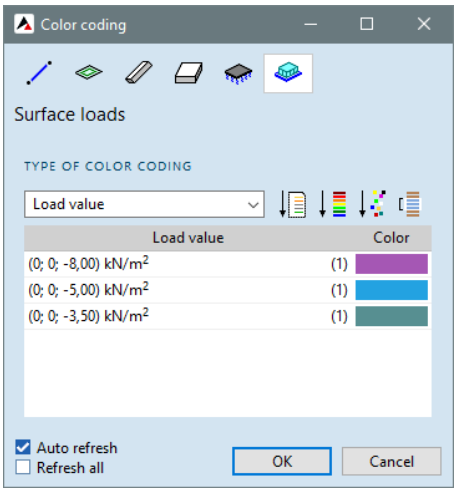
Domains



Surface supports



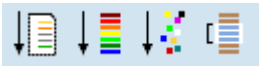
Surface loads



Type of color coding

Default	Uses default colors.
Type	Finite element type (truss, beam, rib for line elements, shell, plate, membrane for domains and surface elements) determines the element color.
Architectural type	Architectural type (column, beam and miscellaneous for line elements, slab, wall, roof for domains) determines the element color.
Material	Element colors are assigned by material
Thickness	Element colors are assigned by domain thickness
Cross-section	Element colors are assigned by line element cross-section
Stiffness	Element colors are assigned by surface support stiffness
Eccentricity	Element colors are assigned by rib or domain eccentricity
Eccentricity groups	Element colors are assigned by eccentricity groups
End releases	Element colors are assigned by end releases on beam ends
Load value	Surface load colors are assigned by load intensity. Areas where load polygons overlap are colored according to the sum of the individual loads.
Load group values	Surface load colors are assigned by load case and load intensity. This is a special color coding as it makes all loads of load cases in the current load group visible. This option can be used to check loads split into separate load cases (see **4.10.36 Split loads into separate load cases**). Though loads from multiple load cases are displayed, only loads from the current load case can be selected, modified or removed.
Uniform	Uniform color for all elements

Setting colors



Click any color cell to change the color.
Toolbar buttons change more than one cell.



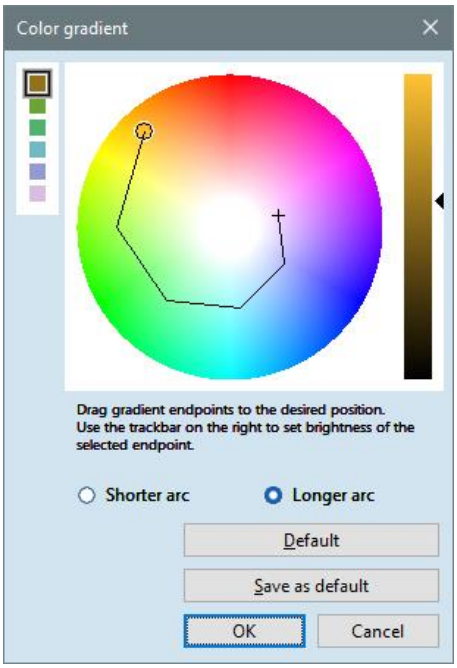
Default
Restores the default values (default element type color, default material color).



Color gradient

Enter the start and end color of the gradient by dragging arc end points on the hue / saturation circle to the desired position. The program picks up the necessary number of colors between the end points.
Use the trackbar on the right to set brightness for the selected end point.
Shorter arc connects colors with the smallest possible hue changes. *Longer arc* goes around the hue circle in the other (longer) way.

The *Save as default* button saves the current gradient endpoints and arc length option as a default setting.
Clicking on the *Default* button loads these values.





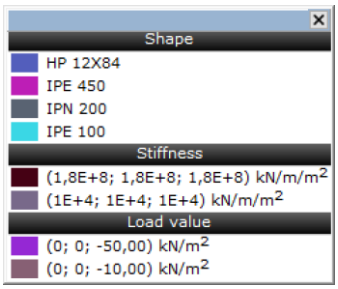
Random colors
Program selects random colors but ensures that colors are not too close.



Set a common color
Pressing the Shift key before clicking you can select a range of color cells. Selected color cells appear with a thick black outline. This tool allows assigning the same color for the selected cells.

☞ **Rendered and wireframe colors are handled separately but can be synchronized. Clicking with the right mouse button on the color list a popup menu appears. Select** *Apply colors assigned to rendered view (wireframe view)* **to import the color set from the other display mode.**

The current color coding is displayed as a separate info window.
You can turn on and off this window from the main menu *(Window / Color coding)*



Auto Refresh Views are automatically updated after changes.

Refresh all Applies changes to all views. If unchecked only the active view is changed.

2.16.6 Geometric transformations on objects

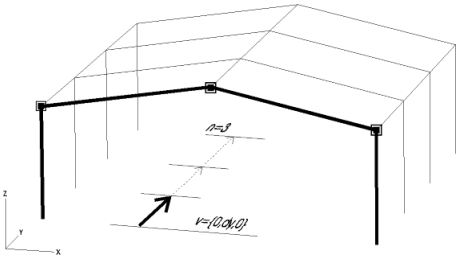



2.16.6.1 Translate

Translate



Makes multiple copies of, or moves the selected geometric entities or loads, by translation along a vector. You must specify the translation vector (dX, dY, dZ), and the number of copies (*N*).



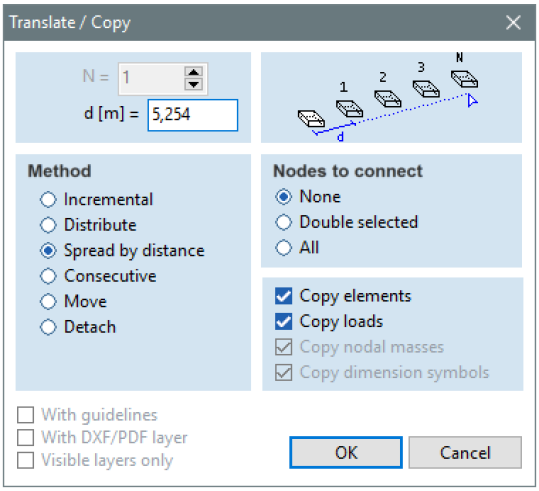
Translation options

Incremental: makes *N* copies of the selected entities by the distance *dX, dY, dZ.*

Distribute: makes *N* copies of the selected entities along the distance *dX, dY, dZ* (by *dX/N, dY/N, dZ/N* increments).

Spread by distance: makes copies of the selected entities spread by distance *d* in the direction of the translation vector. The number of copies depends on how many copies will fit into the length defined by the translation vector *dX, dY, dZ.*

Consecutive: makes *N* consecutive copies of the selected entities by different distances *dX, dY, dZ.*



Move: moves the selected entities by the distance *dX, dY, dZ.* Lines running into the moved nodes remain connected.

Detach: moves the selected entities by the distance *dX, dY, dZ.* Lines running into the moved nodes are detached.

None: No nodes will be connected.

Double selected: Holding the **[Alt]** key pressed you can double select nodes. These nodes will be connected.

All: All nodes to be copied will be connected.

Switches

Copy options

Copy elements: You can specify the finite elements assigned to the geometric entities to be copied as well.

Copy loads: You can specify the loads assigned to the geometric entities to be copied as well.

☞ **Loads can be copied separately (without the elements).**

Copy nodal masses: You can specify the nodal masses to the geometric entities to be copied as well.

Copy dimension lines: The dimension lines will be copied only if the nodes to which they are assigned are selected.

With guidelines

All rulers will also be moved (useful when moving the entire model).

With DXF layer

With this option checked the transformations will be performed on the objects of the DXF layer as well. If individual layer elements are selected the transformation will be applied only to the selected elements. If nothing is selected the entire layer is transformed.

Visible layers only

With this option checked only the visible layers will be transformed.

Steps of translating

The translation consists of the following steps:

1. Click on the Translate icon
2. Select the entities or loads to be copied
3. Click **OK** on the Selection Window (or Cancel to interrupt the selection and translation commands)
4. Select your options from within the Translate Window.
5. Click **OK**
6. Specify the translation vector by its start and end point

The command can be applied in the 2-3-1-4-5-6 sequence as well.

☞ **If you have repetitive parts in your model, you should first create these (including the definition of finite elements, support conditions, loads, and dimension lines), and then make copies of them.**
You can use any existing point when you have to specify the translation vector.

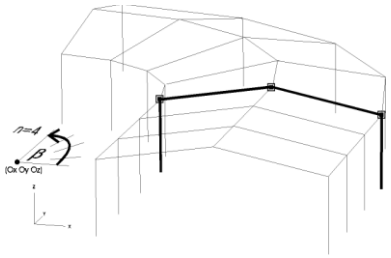
Selected loads can be copied or moved to another load case if load case is changed to the target load case during the operation.

2.16.6.2 Rotate

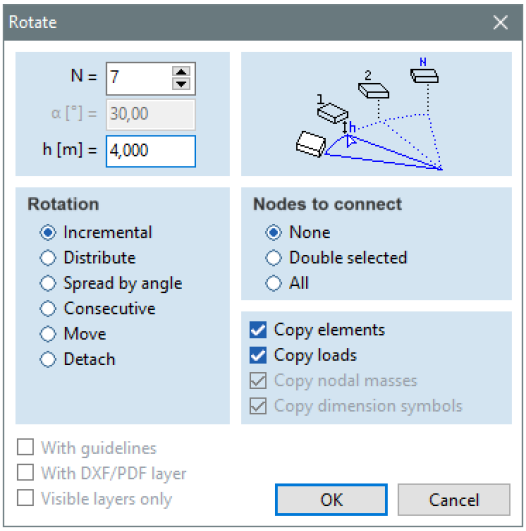
Rotation



Makes multiple copies of, or moves the selected geometric entities or loads, by rotation around a center. In X-Y, X-Z or Y-Z views the rotation axis is normal to the current view plane. In perspective view rotation axis is always the Z axis.



You can specify the method of rotation. Parameters depend on the method: rotation angle *α* the number of copies *(N)* and an additional translation *h* along the rotation axis (each copy will be shifted by this distance).
Click the rotation center (OX, OY, OZ), the rotation arc start point and draw the cursor angle.



Rotation options

Incremental: makes *N* copies of the selected entities by the cursor angle.

Distribute: makes *N* copies of the selected entities by cursor angle/*N* increments.

Spread by angle: makes copies of the selected entities spread by a given angle α specified in the dialog. The number of copies depends on how many copies will fit into the cursor angle.

Consecutive: makes *N* consecutive copies of the selected entities at different cursor angles.

Move: moves the selected entities by the cursor angle. Lines running into the moved nodes remain connected.

Detach: moves the selected entities by the cursor angle. Lines running into the moved nodes are detached.

Nodes to connect

See 2.16.6.1 Translate

Switches

See 2.16.6.1 Translate

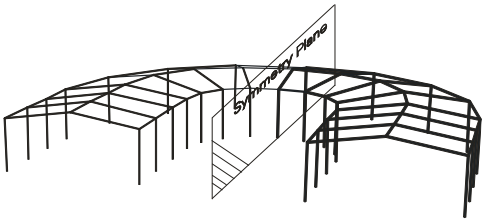
In perspective view, the centerpoint, start point and end point can be specified only using existing points or other identified 3D locations (i.e., a point on a line). In perspective view, cursor angle is determined by the global X and Y coordinates only.

2.16.6.3 Mirror

Mirror



Makes a copy of, or moves the selected geometric entities or loads, by mirroring. Specify two points of the symmetry plane. The symmetry plane is always parallel to a global axis depending on what view you are in.



Mirror options

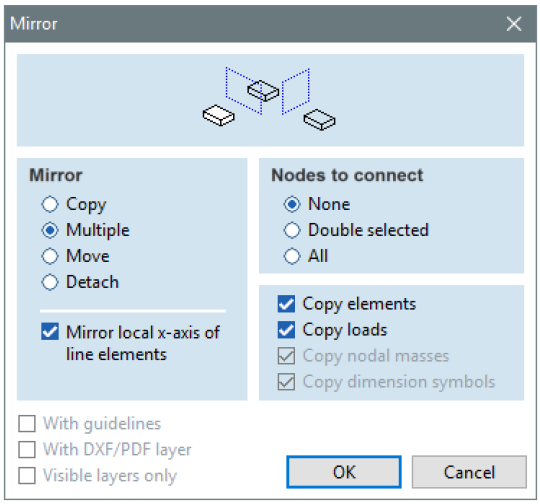
Copy: reflects a copy of the selected entities over the mirror plane.

Multiple: makes consecutive copies of the selected entities over different mirror planes.

Move: moves the selected entities across the mirror plane. Lines running into the moved nodes remain connected.

Detach: moves the selected entities across the mirror plane. Lines running into the moved nodes are deatched.

Mirror local x axis of line elements: This option controls if local x orientation of the mirrored line elements will be inverted or not.



Nodes to connect

See 2.16.6.1 Translate

Switches

See 2.16.6.1 Translate

In perspective view, the mirroring is possible only across a plane parallel to the global Z axis.

2.16.6.4 Scale

Scale



Makes multiple copies of, or moves the selected geometric entities, by scaling from a center. You must specify the scaling center, a point of reference and its new position after scaling (coordinate ratios will determine the scaling factors).

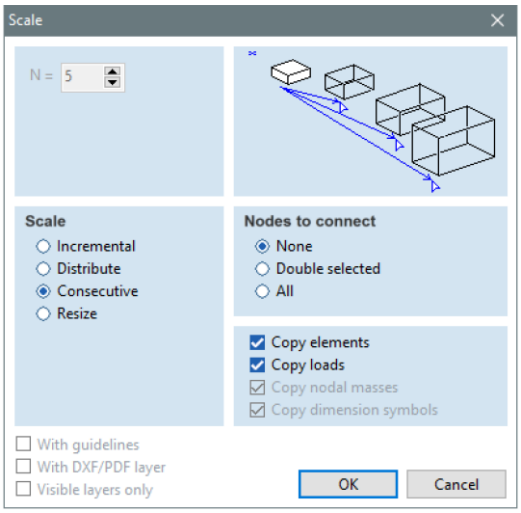
Scale options

Incremental: makes N scaled copies of the selected entities by repeating the scaling N times.

Distribute: distributes N scaled copies of the selected entities between the original and the scaled image.

Consecutive: makes differently scaled copies of the selected entities in consecutive steps.

Resize: redefines the selected entities by scaling**.**



Nodes to connect

See 2.16.6.1 Translate

Switches

See 2.16.6.1 Translate

2.16.7 Workplanes



Workplanes (user coordinate systems) makes it easier to draw on oblique planes. Consider a hole for a skylight on an oblique plane of a roof. The plane of the roof can act as a workplane so drawing can be performed in two dimensions. In case of workplanes altitudinal coordinate means the distance along the axis normal to the workplane.

☞ **All drawing/editing functions are available in workplane mode.**
Using multi-window mode a different workplane can be set for each window.

Global X-Y, Global X-Z, Global Y-Z workplanes
General workplanes

These workplanes are parallel with a global coordinate plane so their position is defined by a single coordinate. Useful when drawing storeys of a building.

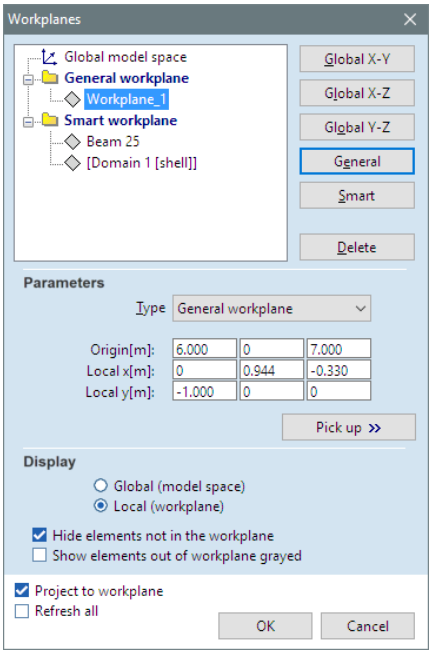
These workplanes are defined by an origin and two vectors for the local *x* and *y* axes.

Smart workplanes

These workplanes follow the local system of a truss, beam, rib or domain. The origin is the first point of the element, local *x* and *y* axes are parallel to the local *x* and *y* axes of the local system of the element.

☞ **Changing the local system of the finite element the workplane is also changing. Deleting the finite element you delete the workplane as well.**

Clicking the workplane speed button the workplane can be selected from a list. Workplanes are also available from the main menu by selecting *View* \ *Workplanes* or from the popup menu by selecting *Workplanes*.



Clicking the workplane speed button the workplane can be selected from a list. Workplanes are also available from the main menu by selecting *View* \ *Workplanes* or from the popup menu by selecting *Workplanes*.

Display options

A workplane can be displayed in the global coordinate system or in its local system. After checking *Hide elements not in the workplane* only those elements are displayed that are in the workplane. After checking *Show elements out of workplane grayed* elements out of the workplane appears grayed.

Changing workplane parameters

If you select a workplane from the tree, its parameters are displayed. Editing them and clicking the **OK** button or selecting another workplane will change the parameters of the selected workplane.

Delete

Deletes user defined workplanes.

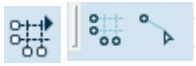
Pick Up >>

Lets you define workplane parameters (origin or axes) graphically.

Project to workplane

If this option is activated all editing will be projected to the workplane. To work in the coordinate system of the workplane but out of the workplane (e.g., setting z to 3 meters) turn off this option.

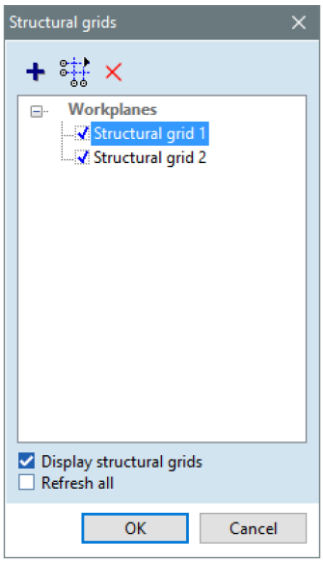
2.16.8 Structural grids



Two buttons fly out from the Icon bar button. The first one is to define a structural grid, and the second one is to define individual gridlines.

To define a structural grid set an X_0, Y_0, or Z_0 origin then enter the ΔX, ΔY or ΔZ relative spacing values. For example, with X_0 = 0 entering 4*3.5; 2*5; 7.5 into the ΔX field gridlines will appear at the following X positions: 3.50; 7.00; 10.50; 14.00; 19.00; 24.00; 31.50.
The structural grid can be rotated by a custom α angle.
The length of lines is determined by the minimum and maximum coordinate positions of gridlines in the other direction, so the shape of the grid is always a rectangle.

Structural grids

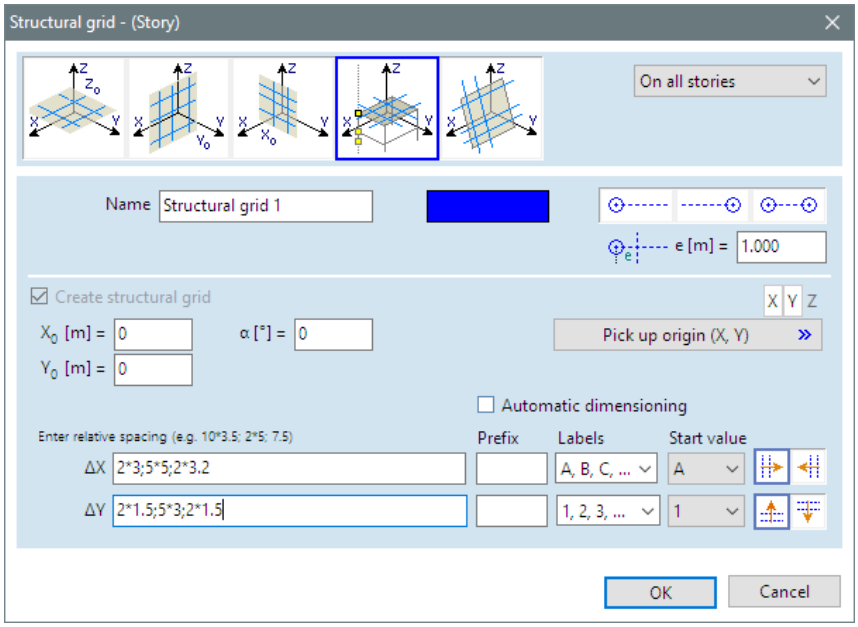


Structural grids are sets of coloured lines in a common plane, with a given length and a label helping the model building process. These sets of gridlines can be parallel to the global X-Y, X-Z or Y-Z planes, workplanes or storeys.
Structural grids are displayed in a tree, organized by the grid plane.

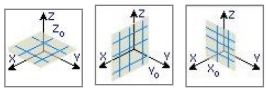
Display structural grids
Turns on/off the display of structural grids in the model.
If it is turned off all grids disappear.
If it is turned on all grids matching the following two criteria will be displayed: 1) it is checked 2) the rule associated to the grid allows the display of the grid. Grids assigned to workplanes and storeys can hide themselves if their workplane or storey is not active.

Refresh all
Update structural grids for all views.

New structural grid



Grid plane



For grids parallel to a global plane the X_0, Y_0, or Z_0 distance between the grid plane and the global plane can be set



Story grid

If the model has storeys different structural grids can be assigned to each storey. The grid can be associated to all storeys by selecting *On all storeys* from the dropdown list. If the grid is associated to a specific storey (e.g., storey 1) and *Display only if the storey is active* is checked the grid remains hidden until storey 1 is activated.



Workplane grid

Structural grids can be assigned to workplanes (if there are workplanes in the model). If *Display only if the workplane is active* is checked the grid remains hidden until the workplane is activated.

Name Name of the structural grid

Color Click the button to change the color of the gridlines.

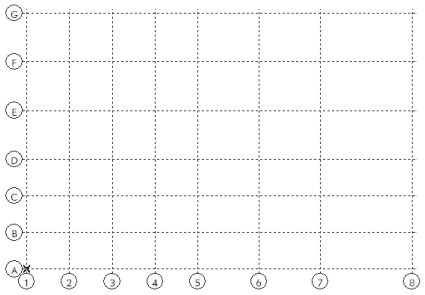
X_0 / Y_0 / Z_0 [m] Origin of the structural grid relative to the global origin.
The origin can be picked up by clicking on the *Pick up the origin* button and clicking on anywhere on the model. By activating or deactivating the *X, Y, Z* fields above the button the user can control which coordinates to pick up.

α[°] Angle of rotation around the origin of the structural grid

Create structural grid

Grid spacings, prefixes, labels, directions can be defined.

Gridline labels can be consecutive numbers (1, 2, 3, ...) or letters (A, B, C, ...) according to the *Labels* dropdown lists. *Start value* defines the first label. A common prefix can be set to create labels like 1A, 1B, 1C, ... or F1, F2, F3. The order of grid lines can be set by selecting one of the icons (left to right, right to left in X direction and bottom to top, top to bottom in Y direction).





Order of creating vertical grid lines

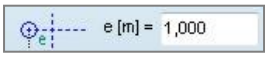


Order of creating horizontal grid lines



Labels can be positioned to the start point, end point or both.

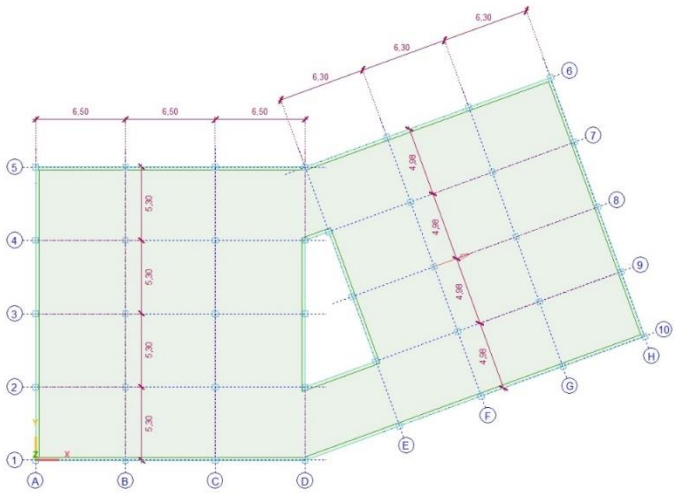
e[m]



It is recommended to set a nonzero Grid line extension so that the labels fall outside the rectangle of gridlines

Automatic dimensioning

If the option *Automatic dimensioning* is checked dimension lines are also created with the structural gridlines. These dimension lines are associative, e.g., follow changes of the structural grid.
It is also possible to place smart dimension lines manually between intersection points of structural gridlines (see **2.16.11.1 Orthogonal dimension lines**). It means that activating smart mode and placing a dimension line between A1 and G1 will generate a group of dimension lines: A1-B1, B1-C1, C1-D1, D1-E1, E1-F1, F1-G1.



Modify structural grid



Name, labels, positions, color of a grid can be modified. If the *Create structural grid* is checked the entire grid will be recreated with the new parameters. In this case all custom gridlines associated to the grid (see below) will be removed.

Copy structural grid



Copies will be made from the selected grid.
Enter a root to generate names for the copies. Copies will be named as *root_1, root_2, root_3...*
Enter the required number of copies, the start and endpoint of the translation vector.

Delete structural grid

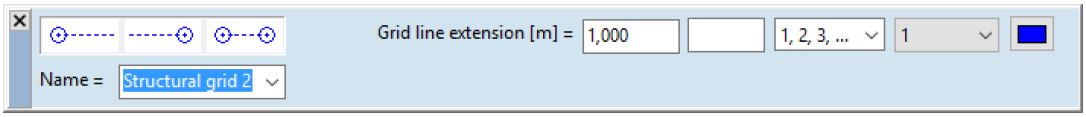


Selected structural grids will be deleted.
Pressing **Ctrl** or **Shift** during mouse clicks more than one grid can be selected.



Custom gridlines

Custom gridlines can also be defined by clicking on the start point and the end point. Properties of the gridline (label position, extension, prefix, labels, colour) can be set on a pet palette.
Custom grid lines must be associated to a structural grid and can be turned on / off with that grid. Recreating the grid with new parameters deletes all associated custom gridlines.

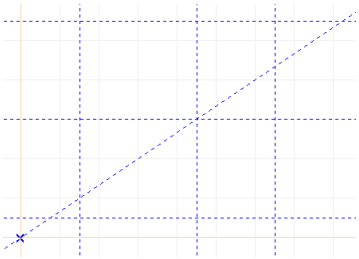


2.16.9 Guidelines



Helps in editing the geometry of the model. Guidelines can be defined in the global coordinate system. This way an arbitrary grid can be created, intersections can be determined and distances can be set. The cursor identifies the guidelines.
See 4.7 Editing tools

The guidelines are displayed as blue dashed lines. The display of the guidelines can be enabled or disabled in the Display Options menu (or icon) in the Switches section.





Places a vertical guideline at the current position of the cursor.

Places a horizontal guideline at the current position of the cursor.



Places a vertical and a horizontal guideline at the current position of the cursor.



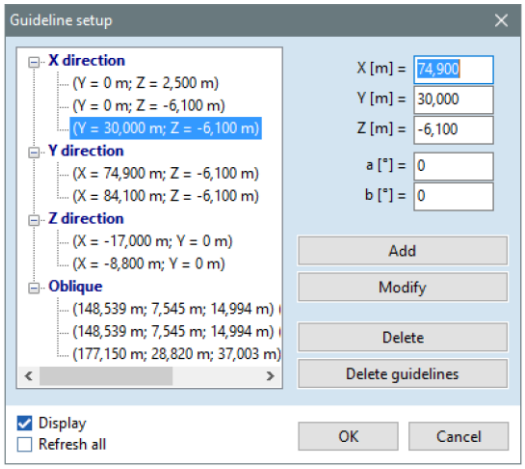
Places an oblique guideline at the current position of the cursor.

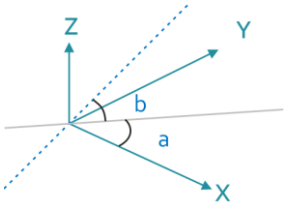


Places a pair of orthogonal oblique guidelines at the current position of the cursor.

In perspective view all the guidelines are displayed but only oblique guidelines can be placed. You can change the position of a guideline with the mouse by dragging it to a new position. You can remove (delete) a guideline by dragging it off the graphics area.

Guidelines can be entered numerically by coordinates. Clicking with the mouse on a guideline or selecting *Settings / Guidelines Setup* command from the main menu, the following dialog is displayed:





a: is the angle of the guideline's projection on the X-Y plane and the X axis.

b: is the angle of the guideline and its projection on the X-Y plane.

Display Turns on / off the display of guidelines.

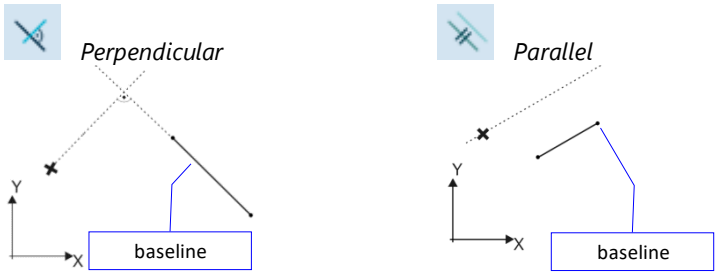
Refresh all If checked changes will be applied to all views otherwise only in the active view.

2.16.10 Geometry tools





The icons of **Geometry tools** allow you to lock the direction of drawing a line.



Begin to draw a line. Click the *Perpendicular* or *Parallel* icon then click an existing line or click two points to define the direction. The cursor will move perpendicular or parallel to this baseline.



Perpendicular to a plane

Begin to draw a line. Click the *Perpendicular to a plane* icon then click the domain defining the plane. The cursor will move perpendicular to the plane. The plane can also be defined by clicking three points.

These icons can be conveniently used while editing the geometry of the model or defining section planes.



Line towards a midpoint

Begin to draw a line then click start point and end point of another line. Midpoint will determine the direction.



Bisector

Begin to draw a line then click the two legs of an angle. Bisector will determine the direction of the line.



Point of intersection

Begin to draw a node or a line then click the icon, click the two lines or their start and end point. A node or line point is created at the point of intersection. Any of the lines (or both) can be an arc. In this case there may be more than one point of intersection. If so, calculated points are marked with small circles. The required point has to be selected by clicking.



Dividing point

Begin to draw a node or a line then click the icon and click the two nodes. Specify the division by ratio or by distance in the popup dialog. A node or line point is created.



Point constraint operation

The action for *Point of intersection* and *Dividing point* can be set here. Two options are available: creating a node or moving the relative origin to the position calculated.

2.16.11 Dimension lines, symbols and labels



This group of functions lets you assign associative orthogonal and aligned dimension lines or strings of dimension lines to the three dimensional model, as well as angle, arc length, arc radius, level and elevation marks, labels for result values. Click on the Dimensions icon to display the Dimension Toolbar. That will allow you to select the proper dimension tool. Click on the left-bottom icon of the Dimension Toolbar to set the parameters of the selected tool.

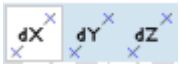


You can change the position of dimension lines or labels at any time by dragging them to their new position. If the dimension lines were associated with the model their position and dimension will be continuously updated as you modify the geometry of the model.

2.16.11.1 Orthogonal dimension lines



Associative orthogonal dimension lines or strings of dimension lines, parallel to the global X, Y, or Z axes can be assigned to the model by following the next steps:



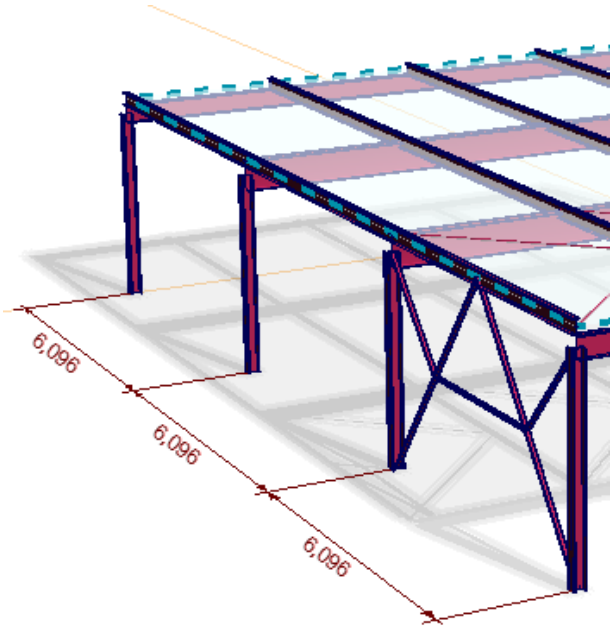
1. Click on dimension line start point and on the end point. If these points are connected by a line you can just click on the line.
2. Move the mouse. The position of the dimension line depends on the direction in which you moved the mouse. There is one exception: when the segment is not parallel to any global plane and the editing is in the perspective view. In this case you have to select the direction dX, dY, or dZ from the toolbar.
3. Click the left mouse button to set the final position of the dimension line.

To insert a string of dimension lines, click on the points in the corresponding order or on the lines if any. Steps 2 and 3 are the same as for the individual dimension lines. A string of dimension lines can be selected at once if you click on one of them while depressing the Shift key. It allows you to move it as a group. To change the position of a group segment individually select it using the selection rectangle and drag it to its new position. As a result this dimension line will be removed from the group (it can be moved individually).



Smart dimension lines

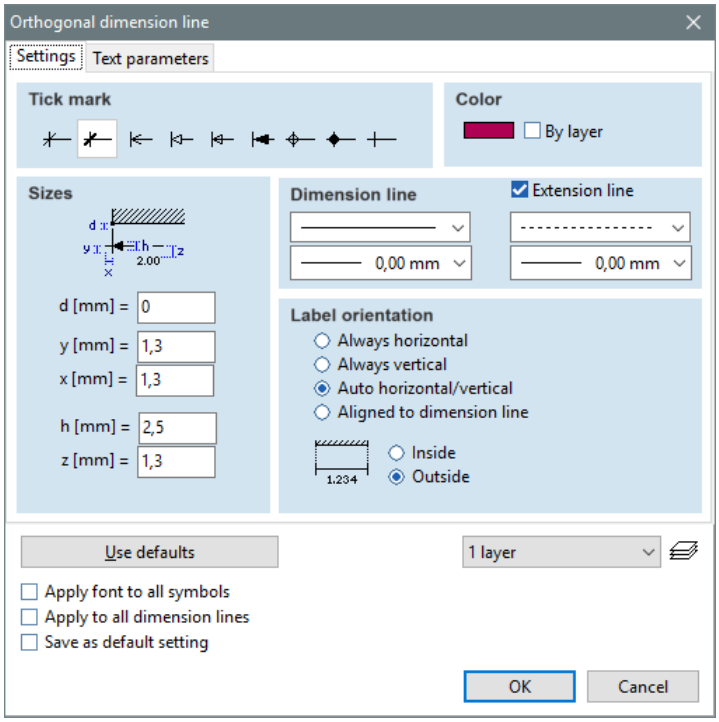
A string of dimension lines can also be created by turning on the smart dimension lines. If you enable this function by pressing the button, you have to select only the end points of the string, assuming that the intermediate points were not generated by a domain mesh command. All intermediate dimension lines will be created automatically.



An example of smart dimension lines

If the dimension line is assigned to the points of a model, it will always behave in an associative way (e.g., will move with the model when the model is changed or resized or moved).

Orthogonal and Aligned Dimension Line Settings



Tick mark Lets you set the tick marks of the dimension lines. You can select from nine predefined symbols.

Color Lets you set the color of dimension lines individually. You can get the color from the active layer. The dimension lines, marks, and texts are placed on the Dimensions layer by default but you can change it any time.

Sizes Lets you set the drawing parameters of the dimension line.

Dimension style/ Extension style Lets you to set the type and thickness of a dimension or extension line. You can choose a predefined value or get it from the active layer. You can turn on/off the display of extension lines.

Label orientation Lets you set the orientation of the text labels of the dimension lines *(Always horizontal, Always vertical, Auto horizontal/vertical*, or *Aligned to dimension line)* inside or outside the dimension line.

Use defaults Lets you restore the default setting.

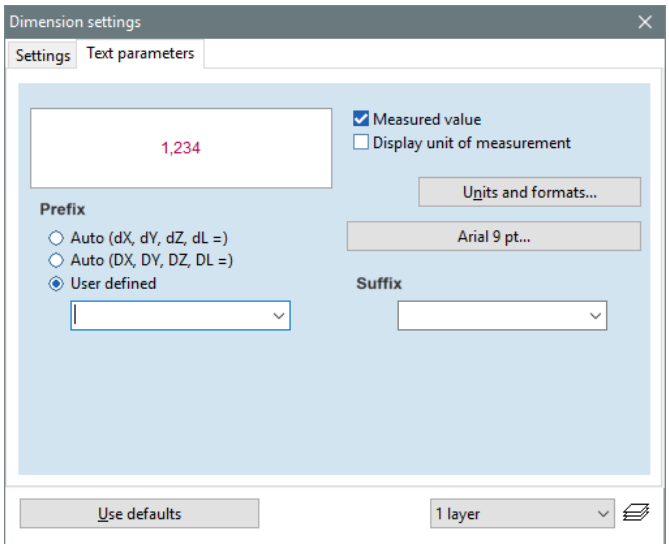
Apply font to all symbols Apply the same font to every dimension line.

Save as default setting Lets you save the current setting as default setting.

Apply to all dimension lines Applies the current setting to all existing orthogonal or aligned dimension lines to ensure a uniform look.

Layers Lets you select/define/set layers where the dimension lines will be placed. If there are no layers defined when you start defining dimension lines, a Dimension layer will be automatically created.
See **3.3.3 Layer Manager**

Text Parameters



Allows to you to define the settings of the text on the dimension lines.

Measured value Allows you to place the measured value on the dimension line, using the current prefix and suffix settings. By clicking the Units and formats button the number format can be set in the Dimensions section of the *Settings / Units and formats* dialog box.

Display unit of measurement Display of the unit of measured value.

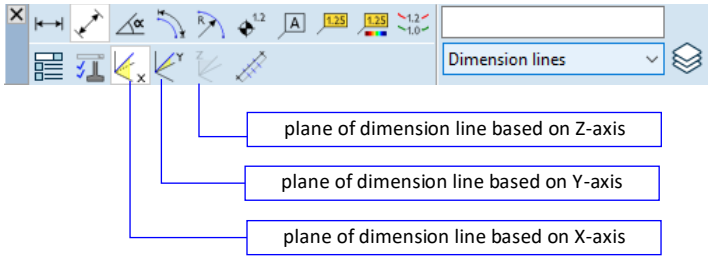
Units and Formats... To change the current font parameters, click the button below the *Units and formats...* button.

Prefix Sets the prefix used with the text on the dimension lines. You can choose from the following options:
Auto (dX, dY, dZ, dL = [depending on the direction])
Auto (DX, DY, DZ, DL = [depending on the direction])
User defined (this option will require you to enter the prefix).

Suffix Sets the suffix used with the text on the dimension lines.

2.16.11.2 Aligned dimension lines

Assigns aligned dimension lines or a string of dimension lines to the model.



The steps are the same as the steps of creating an orthogonal dimension line.
See **2.16.11.2 Aligned dimension lines**
The plane of the parallel dimension line is determined automatically. There is one exception: when the segment is not parallel to any global plane and the editing is in the perspective view. In this case you have to select the direction X, Y, or Z from the toolbar. The plane of the section line will be defined by the segment and the selected global axis.

Sets the dimension line settings. See **2.16.11.1 Orthogonal dimension** .
For aligned dimension lines the automatic prefix is always dL= or DL=.

An example of associative dimension lines (orthogonal and aligned):



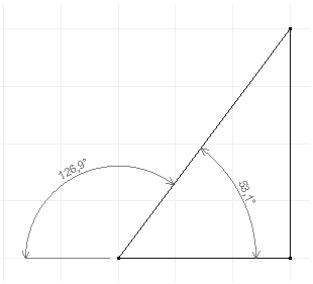
Before Scale command *After Scale command*

2.16.11.3 Angle dimension

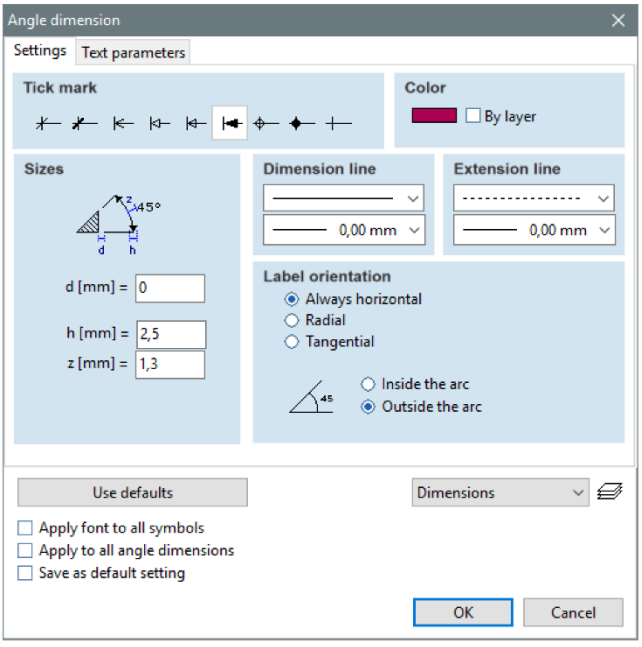


Associative angle dimensions, as the symbol of the angle between two segments, can be assigned to the model in the following steps:

1. Click on start point and on the end point of the first segment. If the points are connected by a line you can just click on the line.
2. Click on start point and on the end point of the second segment. If the points are connected by a line you can just click on the line.
3. Move the mouse. The position and radius of the angle dimension will be determined by the mouse movement. Based on the position of the mouse, the angle, supplementary angle or complementary angle dimension can be entered.
4. Click the left mouse button to set the angle dimension in its final position.







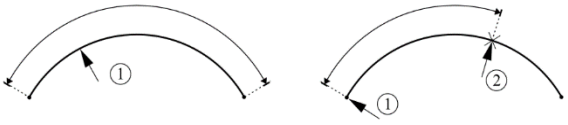
By clicking the *Text parameters / Units and formats...* button the angle number format can be set in the Dimensions section of the *Settings / Units and Formats* dialog box.

2.16.11.4 Arc length



Creates arc length dimension symbols in your model.

To assign this symbol to a full circle click any point of the circle and drag the symbol.
To assign this symbol to an arc click any point of the arc and drag the symbol.
To assign this symbol to a part of an arc click any end point of the arc, click the middle point of the arc and drag the symbol.



2.16.11.5 Arc radius



Creates arc radius dimension symbols in your model.
To assign this symbol to an arc click any point of the arc drag the symbol.

2.16.11.6 Level and elevation marks



Creates associative level and elevation marks in your model.
By clicking the Units and formats button the number format can be set as the unit of Distance in the Geometry section of the *Settings / Units and Formats* dialog box. This is the unit and format used in the Coordinate Window. See **3.3.8 Units and Formats**

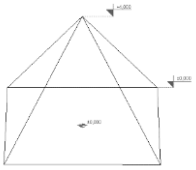


Level marks can be placed in top view, by clicking on the desired point. The top view is defined as the view in the direction of gravity (You can change it in the *Settings / Gravitation* dialog).
See **3.3.9 Gravitation**



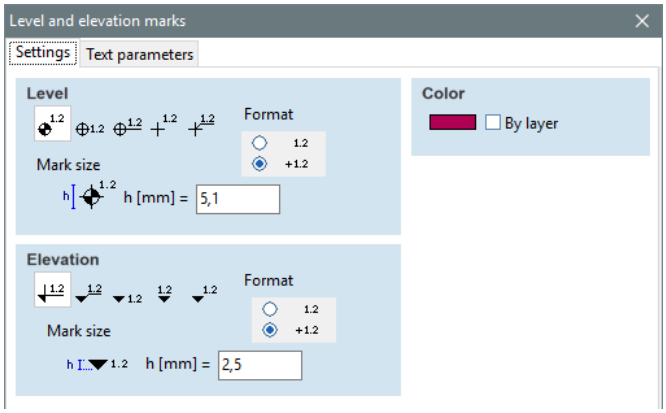
Elevation marks can be placed in front view, side view, or in perspective, by following the next steps:

1. Click on the point you want to mark.
2. Move the mouse in the direction you want to place the elevation mark, and click to set the symbol in its final position.





Level and elevation mark parameters

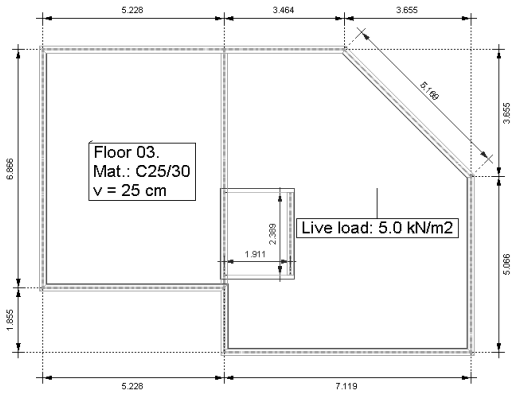


Level Selects the level mark symbol, and sets its size and format.

Elevation Selects the elevation mark symbol, and sets its size and format.

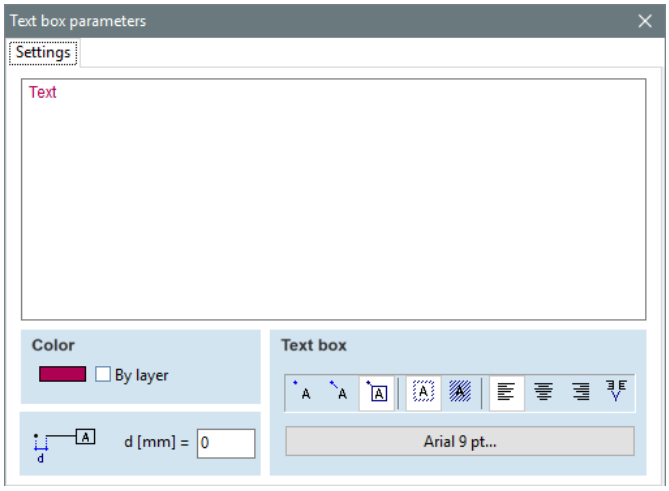
2.16.11.7 Text box

Creates an associative text box in your model. You can enter multiline text in a text box. The text will use the same text formatting within a text box.



You can create a text box in the following steps:

1. Enter the text in the Text box parameters window, or in case of a single line text enter it directly into the edit field of the Toolbar.
2. Click on the point to which you want to assign the text box.
3. Move the mouse to the desired position and click to set the text box in its final position.



Color Sets the color of the text, frame, and extension line. You can get the color from the layer.

Text box These switches set the drawing parameters of the text box, frame, and extension line, the transparency and alignment of the text, and the *d* distance of the extension line from the reference point (to which the text box is assigned to).

Font Sets the text font, style and size.

You can reload and change default settings, apply text box or font parameters to all existing text boxes

Active Links Active links can be placed in text boxes to attach any external information to the model. If the text contains a file reference or a link to a web page clicking the text box launches the application associated to the file or URL instead of opening the above dialog. To change the text, select text box first (e.g., *Shift+click*) then click into the box.

File reference A file reference is made of the -> characters and a file name. E. g.:
`->C:\MyModel\Reports\Details.doc`
If no full path is specified AXISVM starts from the folder of the model, so if our model is in *C:\MyModel* we can enter: **-> \Reports\Details.doc**
Clicking the text box starts the application associated to the file type. This way we can attach pictures, movies, sounds, Excel tables or other documents to any part of the model.

URL Supported protocols and link formats are:

`http://..., ftp://..., https://..., file://..., www. ...`
After clicking the textbox, the default web browser launches and opens the web site or file.
If the text contains more than one URL, the first one is used.

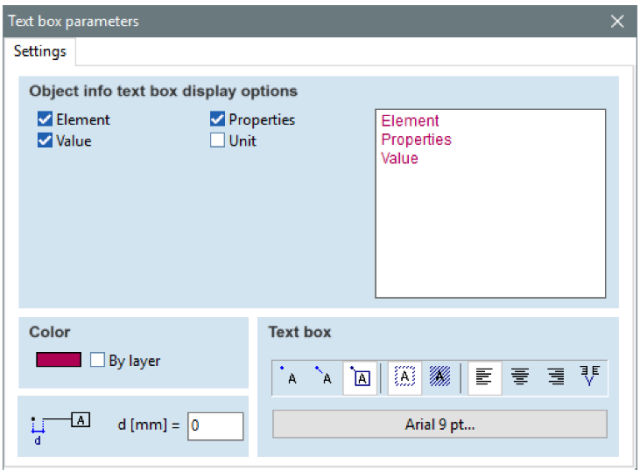
2.16.11.8 Object info and result text boxes

Object info text box



Element or load properties appear in the text box depending on the current tab *(Geometry, Element or Loads)*. Information text box parameters can be set in a dialog:





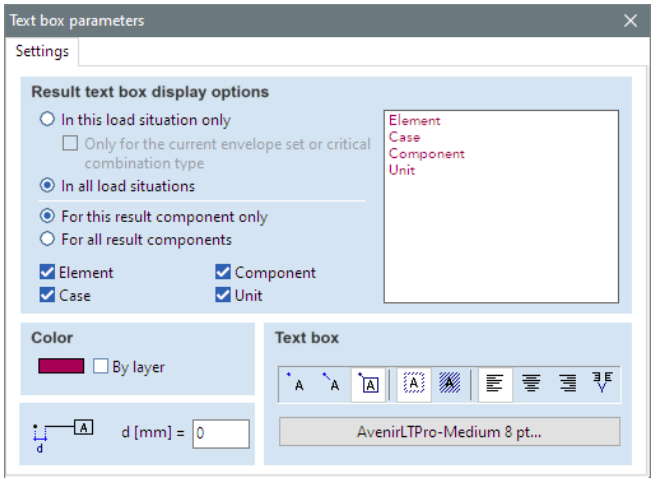
Result text box



When displaying results the cursor determines the value of the current result component on nodes, mid-side nodes, surface centers, or intermediate points of beams or ribs and shows it as a tooltip. The text of the tooltip is automatically entered in a text box.
The steps of result labeling are similar to creating a text box.
The result text box is visible only when the selected result component is the same as the one that was selected when the result text box was created. For example an *My* result text box is displayed only when the *My* component is selected as the current result component.

Result text box options can be set in a dialog box:





In this load situation only

The result label is visible only in the load situation (load case or combination, envelope or critical combination) in which it was created.

Only for the current envelope set or critical combination type

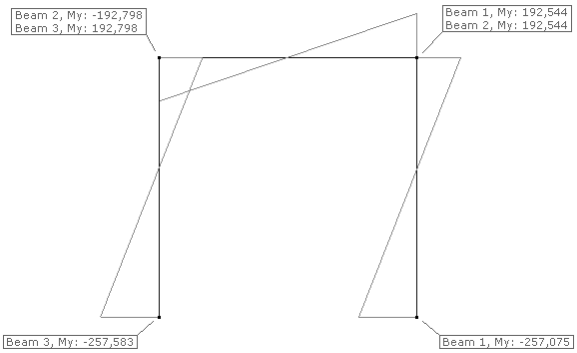
If an envelope or critical combination is selected this additional constraint allows the display of the result label only if the same envelope set (e.g., All load combinations) or critical combination type (e.g., SLS Quasi permanent) is selected. If unchecked, the result label becomes visible when any envelope is displayed (if it was defined with an envelope selected), or when any critical combination is displayed (if it was defined with a critical combination selected).

In all load situations

The result label remains visible regardless the load case, combination, envelope or critical combination displayed. The actual values will be updated on changing the case.

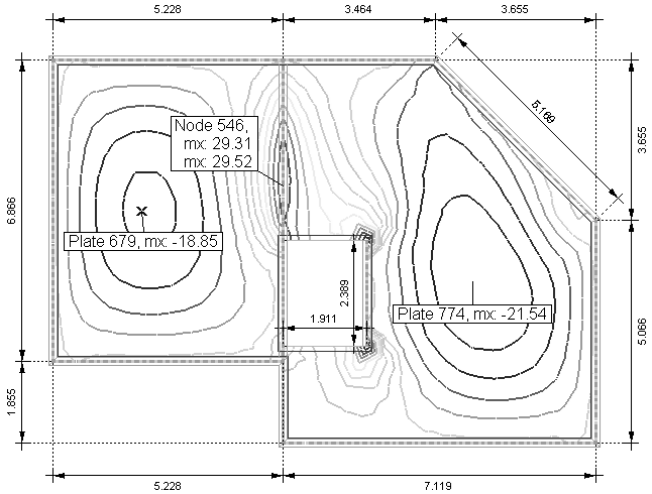
For this result component only
Result label is visible only if its result component is displayed.

For all result components
Result label remains visible regardless the displayed result component.



Result label text options:

Element: Include element type and number.
Component: Include result component name.
Case: Include name of the load case, combination or description of the critical combination.
Unit: Include unit name.



Below the button of *Use defaults* three checkboxes helps to customize the text box:

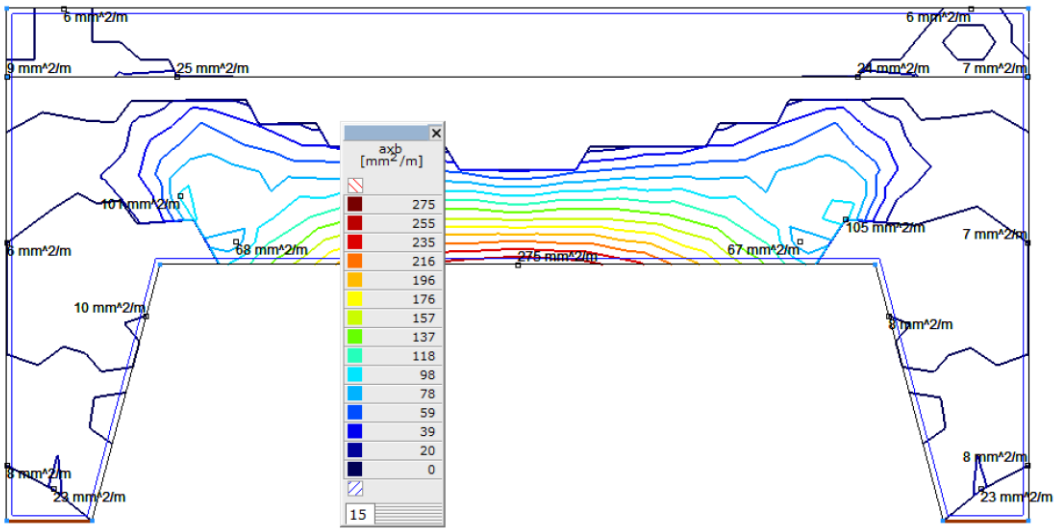
Apply font to all text box
After clicking the **OK** button only the font of all text boxes will change.
Save as default setting
New text boxes will appear using the current settings as default.
Apply parameters to all text box
After clicking the **OK** button parameters of all text boxes will be set to these values.

Layer Manager
[F11]
Lets you create new layers or modify existing ones.
This function is also available from the menu as *Settings / Layer Manager*.
See **3.3.3 Layer Manager**

2.16.11.9 Isoline labels

Lets you place a series of labels to isolines.

1. Click to the *Isoline labels* icon
2. Enter two points defining a line segment
3. The labels are placed at the intersections of the segment and the isolines



Automatic labeling of isolines can be activated in the color legend window. See **2.19.4 Color legend window**

2.16.11.10 Dimension lines for footing



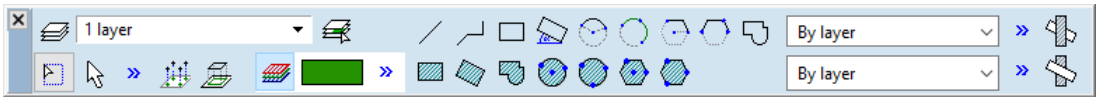
Sets the properties of dimension lines for designed footings.
Settings are the same as for normal dimension lines.

2.16.12 Editing background layers



Background layer editor

This editor allows making changes in the imported DXF and PDF layers and adding new shapes. Background layers contain only geometry information and play no role in the structure.





Layer Manager
Opens the Layer Manager. See **3.3.3 Layer Manager**

Selecting a layer

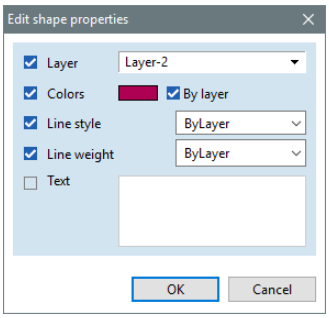
Select a layer for editing from the dropdown tree.
To create a new layer, open the Layer Manager and create a new layer and click OK. Then you can select the new layer.



Another way to select a layer is to click on this button beside the dropdown tree then click on a shape. The layer associated to that shape will be selected.



Selection
Click on this button to activate shape selection.
Click the outline of the shape or drag a frame around shapes and click on the outline of a selected shape then set properties in the *Edit shape properties* dialog.
Using special selection functions choose the next button of the toolbar *(Special selection modes).*
Right clicking on a layer object displays a popup menu. *Isolate layer for editing* locks all the other layers and allows editing of the containing layer. *Delete layer* allows deleting of the containing layer.





Special selection modes
Clicking on this button displays a palette for selection options. See **2.16.1 Selection**
Click OK if the selection is finished and click on any selected shape to set shape properties.



Pick up properties
Click on this button and to activate pick up. Clicking on a shape picks up all properties of that shape (i. e. all subsequent drawing functions will use these properties).



Convert points of the selected shapes to AXISVM nodes



Convert selected shapes to AXISVM lines
After clicking this button, a selection toolbar appears. Click OK if the selection is finished. All selected shapes will be copied as regular AXISVM lines.



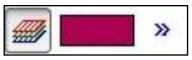
Send selected objects to back



Bring selected objects to front

Delete shapes To delete shapes first select them then press the *Delete* key of the keyboard.

Pen color Pen color is used to draw the outline of shapes and also to fill the interior of filled shapes.
There are three ways to set the current pen color.





Set the layer color as pen color.



Choose a color from a dialog.



Pick up the pen color of an existing shape by clicking on it.

Line style
Line weight Two dropdown lists on the right show the available line styles (top) and line weweights (bottom). Select the desired values.
These settings don't have any effect on filled shapes as they don't have an outline.



Pick up line style or line weight of an existing shape



Toolbar for drawing lines and outlined shapes.

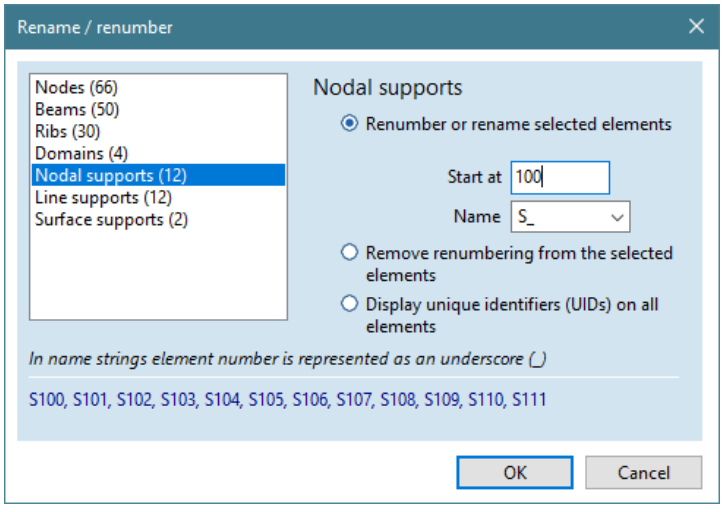


Toolbar for drawing filled shapes.

2.16.13 Renaming/renumbering



Nodes, trusses, beams, ribs, domains, nodal, line and surface supports of the model, steel and timber design members can be renumbered and renamed (their numbering follows creation order by default).
To rename and renumber nodes or elements select them first then click on the icon on the Icon bar.



The list on the left shows the number of selected nodes and elements. Choose what you want to rename / renumber.
You can use one of the three renumbering functions:

Renumber or rename

The first option is to rename or renumber the selected elements by setting the fields below.

Start at

Enter the starting number. Selected elements will be renumbered in an order determined from their position. Renumbering can affect elements not selected as two nodes or elements cannot have the same number.

Name

In this field element number is represented as an underscore (_). For example: if starting number is 1, and the *Name* field contains *T_*, the names of the selected elements will be *T1, T2, T3* ... If only one element is selected it is not necessary to include _ in the *Name*. Otherwise, it must be included as elements must have different names.
If the *Name* field is empty, the name will be the number itself.

Remove renumbering from the selected elements

If this option is checked clicking the OK button restores the original numbers of the selected elements and clears their names. Element type must be selected from the list on the left.
Renumbering can affect elements not selected as two nodes or elements cannot have the same number.
To turn on/off the display of numbers / names of elements open the *Display options* dialog (see **2.16.21 Display options**) or use the speed button for numbering (see **2.18 Speed buttons**).

Display UIDs on all elements

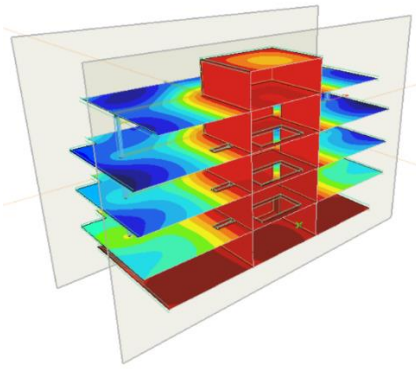
The last option is to display unique identifiers (UIDs) on **all** elements of this type. This feature ensures that the number assigned to an element remains unchanged during its lifetime. It can be useful when you export data from AXISVM to external applications for further processing.

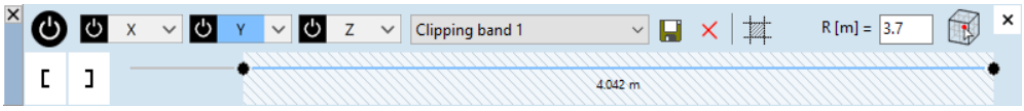
2.16.14 Clipping regions



Clipping regions are to display slices of the model interactively between pairs of clipping planes parallel to the global planes or custom directions, allowing visualization of internal parts of the structure and displaying results on them.

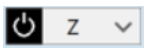
Clipping band configurations can include one, two or three pairs of clipping planes. These configurations can be saved under a name.



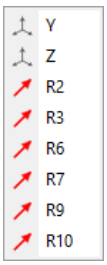




Switch clipping bands on/off
Use the switch to turn the entire clipping band configuration on and off.



Normal vector of the clipping band
The user can set up to three different clipping bands by choosing a normal vector for each cutting plane from the dropdown list. The normal vector of the cutting planes can be one of the axes of the global coordinate system or a user-defined reference vector. Each band has its own activating switch.
Clicking on the normal vector name field sets it to blue and selects the clipping band for interactive editing. Drag the sliders to set the position of clipping planes or drag the hatched part to move the slice along the normal vector without changing its width.





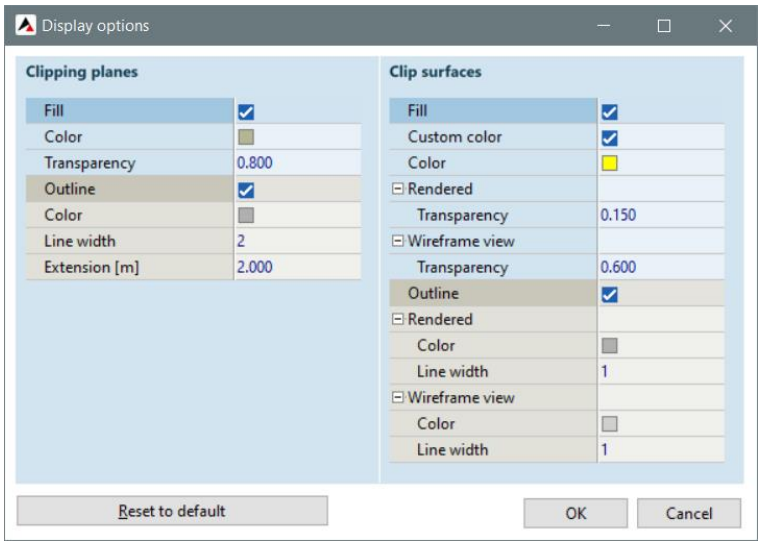
Save current clipping bands
Saves the current clipping band configuration under a name. It is stored in the model.



Delete the saved clipping band configuration
Deletes the clipping band configuration chosen from the dropdown list.



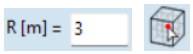
Display options for clipping bands



The *Display options* dialog is to adjust the display properties of clipping planes and clip surfaces. Changes are immediately applied in the main window.
Click on *Reset to default* to set the default properties.

Clipping panes The fill and outline of clipping planes can be turned on/off separately.
Extension controls how far the drawing of the clipping plane extends beyond the bounding rectangle of the structure projected onto the clipping plane.

Clip surfaces Clip surfaces are the cut surfaces of elements intersected by the clipping plane. Their fill and outline can be turned on/off separately. Some properties can have different values for rendered or wireframe view.



Clipping cube around a selected point
Set the *R* distance of cutting planes from the given point. Click on the icon to pick up a point from the model. Cutting planes will be parallel to the global planes.

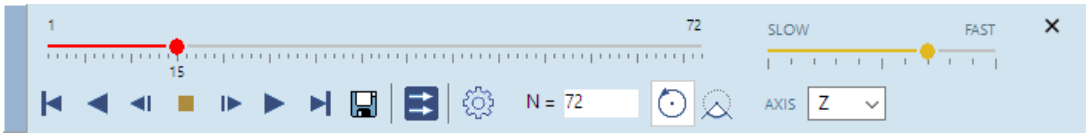


Apply the first/second clipping plane
The first or second cutting plane can be switched on or off. Turning off one of the clipping plane makes the clipping band unbounded on one side.

2.16.15 Turntable



Displays the Turntable toolbar to rotate the model around different axes in the active window. It can be used in data input or results display mode.
It is similar to the toolbar described in section **6.1.2 Animation**: two sliders at the top, the red one shows the rotation phase. Dragging it rotates the structure around the chosen axis. When playing an animation, you can control the animation speed with the yellow slider. The Esc key stops the animation.



Animation controls on the toolbar are explained in **6.1.2 Animation**.

Number of frames (rotated positions) generated for the animation.

Two types of turntable animation are available.



Rotating 360°
The animation rotates the structure by 360°



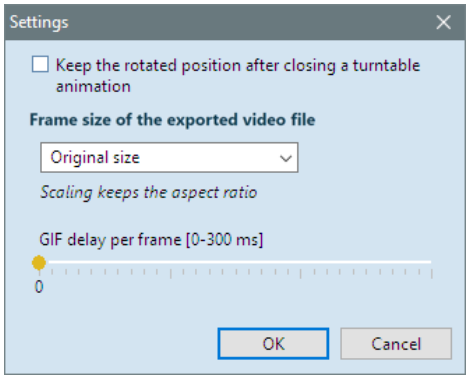
Rotating back and forth within a given range
The angle range can be specified in the δ field.

Axis The axis of rotation. It can be one of the global X, Y, Z axes, or one of the reference vectors and reference axes defined in the model (see **4.9.25 References**). If a workplane is activated, global axes are replaced in the list with the x, y, z axes of current workplane.



Settings Settings are similar to those available in *Animation* (see **6.1.2 Animation**). You can specify the frame size used when saving the animation to a video file, and an additional frame delay for GIF exports.

Turntable animation offers a specific option: *Keep rotated position after closing a turntable animation*. When turned off, the model returns to the original position after closing the animation. When turned on, the model retains the rotated position when the animation is closed.

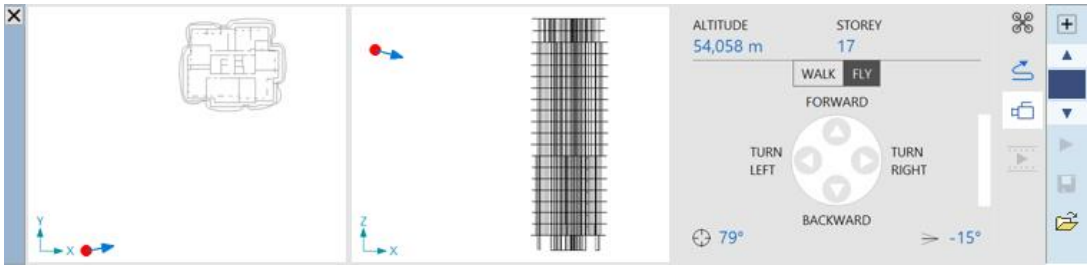


2.16.16 Drone



The drone helps to check the structural model, and in particular the inside of the structure, and is also a good tool the demonstrate the structure. Its operation is similar to that of a real drone: it can not only fly around the model, but, unlike other modes of visualisation, it can also penetrate the structure and visualise it from a point inside.

When switched on, the drone displays a dashboard showing its position and status. The first two panels show a top view and a front view of the structure, the drone position is marked with a red dot.



The third panel shows the current altitude (Z coordinate of the drone) in the upper bar, with the corresponding storey if storeys have been defined in the model. Below that is the drone mode (see later), the status of the main controls, and below that the track angle (orientation of the drone's line of sight projected onto the X-Y plane (the Y direction is 0°) and the pitch angle (angle of the line of sight relative to the X-Y plane, 0° is in a horizontal plane).
The drone's line of sight and the direction of back-and-forth motion are always the same.

Toolbar There are two vertical toolbars on the right, the first one has a grey background, the second one has a blue background. Functions of the grey toolbar are

 *Views and settings*

 *Recording drone flight*

 *Recording camera path as a sequence of camera positions*

 *Save flight animation to a file*

The functions of the blue toolbar depend on whether you choose to record the flight or set up a camera path animation, which will be explained below.

Drone modes In *Walk* mode the vector of the drone's line of sight and movements is restricted into a horizontal plane. The drone can still be moved up and down and sideways. In *Flight* mode, this vector may be out of the horizontal plane, but the angle relative to the vertical Z direction must not be less than 30°.

Control The drone can be controlled in two ways: with the keyboard or an XBOX compatible controller.
When the controller is connected to the computer, control from the keyboard is automatically disabled. When the controller is removed, keyboard control is automatically switched back on.

Keyboard control

↑ or *W*	Move forward
↓ or *S*	Move backwards
← or *A*	Turn left
→ or *D*	Turn right
Ctrl + ↑ or *T*	Pitch up
Ctrl + ↓ or *G*	Pitch down
Shift + ↑ or *I*	Rise in Z direction
Shift + ↓ or *K*	Sink in Z direction
Shift + ← or *N*	Move sideways to the left
Shift + → or *M*	Move sideways to the right
PgUp	Turn left by 15°
PgDn	Turn right by 15°
1	Front view (X-Z)
2	Top view (X-Y)
3	Side view (Y-Z)
F	Switch between *Walk / Fly* modes
+ / - or *Q / E*	Speed up / keep the speed of the drone movement
Shift + + / - or *Shift* + *Q/E*	Speed up / keep the speed of moving sideways
Space	Stop the drone
Home	Move the drone outside of the structure
Esc	Interrupt playback of a flight or a camera path animation

When keys of movement or rotation are pressed, the drone will accelerate the movement or rotation up to an upper speed limit. When the button is released and pressed again, the process restarts.
To move the drone at a slower constant speed, press and hold the - key on the numeric keypad or the E key when the drone reaches the desired speed.
Movements can be combined, e.g. ↑ + ← = turning in flight, W + I = moving forward with climb.
If you want to adjust the pitch angle to horizontal, but want to keep the *Fly* mode, use the F key to switch between the *Walk* and *Fly* modes, keeping *Fly* mode active.

If the drone doesn't respond to keystrokes, click into the main window to return control.

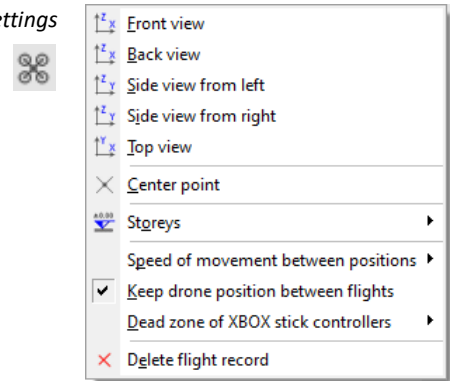
XBOX compatible controllers



1	Right trigger	Throttle lever, moves the drone forward
2	Left trigger	Throttle lever, moves the drone backwards
3	Left stick	Change the direction of the drone's view and movement (turn, pitch)
4	Right stick	Make the drone rise, sink or move sideways
5	X or Y button	*Walk* mode
6	A or B button	*Flight* mode
7	Left bumper	See *Home* key
8	Right bumper	See *Home* key

If you want to set the pitch angle to horizontal while in flight, but want to continue in *Fly* mode, press the X or Y button (5) to change the flight to horizontal, then press the A or B button (6) to return to *Fly* mode.

Views and settings



Click on the icon to move the drone to one of the predefined global views or to the center point of the entire model. If storeys have been defined, selecting a storey from the submenu will move the drone to the center plane of the selected storey.
Speed of movement between positions can be set from 1 to 10 in the submenu. This affects the speed of movement between the predefined views or camera positions recorded on the camera path.
The *Dead zone of the XBOX stick controllers* can similarly be adjusted from 1 to 10. This can be used to control the minimum amount of stick movement the drone will respond to. The lower the value, the finer the launch. But due to the characteristics of the controller if the dead zone is set too low, the controller may give a signal even if the stick is in its centre position. This may trigger a drifting motion. The proper size of the dead zone is highly device dependent.
If *Keep drone position between flights* is checked the drone will start from the position it was left at the end of the previous flight. The value is stored with the model.
Delete flight record or *Delete camera path* deletes the current flight record or camera path and makes playback inactive. If the recording or camera path has been saved under a name, the saved recording will not be deleted.

Recording drone flight

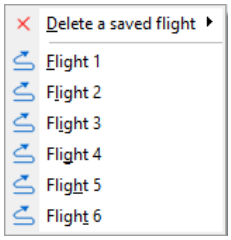
Use the buttons of the blue toolbar to record the trajectory of your drone flight and replay it in real time. Recorded flights can be saved to the model under a name and recordings can be reloaded later.

Recording the flight
Press the button to turn the icon fully red. From then on, the program records drone movements. Pressing the button again will end the recording. The recording can be played back by pressing the next button. The recording can be deleted in the *Views and settings* menu.

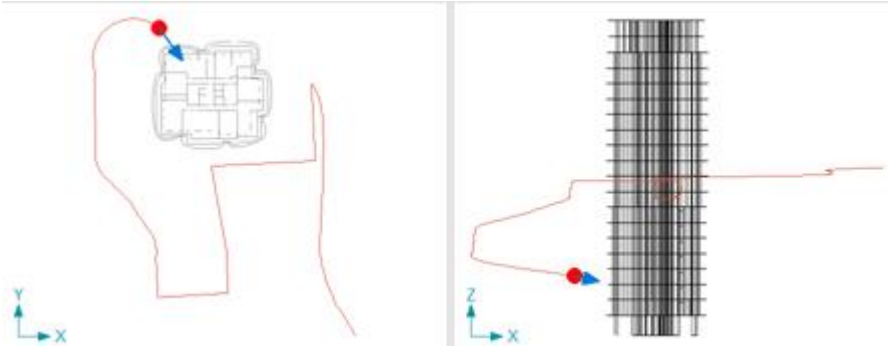
Replay recorded flight

Save recorded flight under a name

Load or delete a recorded flight
You can select a saved flight from the menu to load it, or *Delete a saved flight* from the submenu.
A flight that has been recorded but not saved can be deleted from the *Views and settings* menu.



The projections of the drone trajectory are also displayed on the view panels.



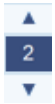
Recording camera path as a sequence of camera positions

Use the blue toolbar to edit the camera path. The camera path consists of successive camera positions. The drone moves along this path at the speed selected in the *Views and settings* menu (*Speed of movement between positions)*.

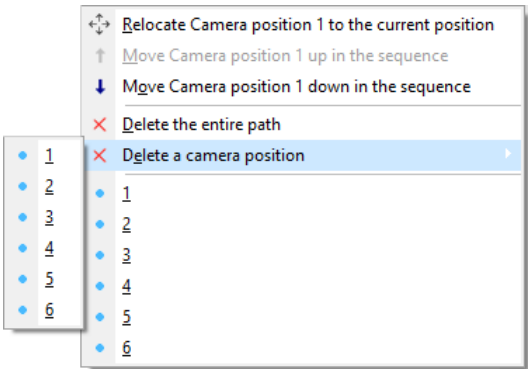
If camera positions has been added to the path and one of them is selected, the toolbar will show the current camera position number. You can move the drone from that camera position, the value will not change.



Insert current camera position into the sequence after the current position
After moving a drone to a new position it can be saved into the sequence. It will be placed after the position displayed on the number button.



Camera position on the path
Arrows move the drone to the previous or next camera position.
Click on the number button to access simple editing options.





Fly along the camera path

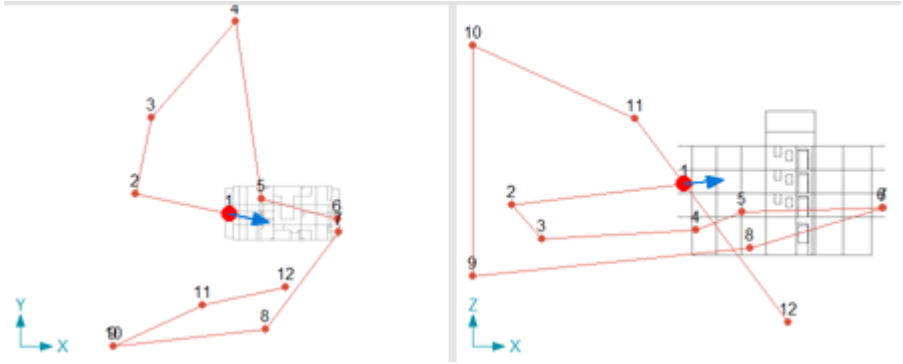


Save camera path under a name



Load or delete a camera path
Load a saved camera path or select a camera path to delete from the *Delete a saved camera path* submenu. A camera path that has been edited but not saved can be deleted in the *Views and settings* menu.

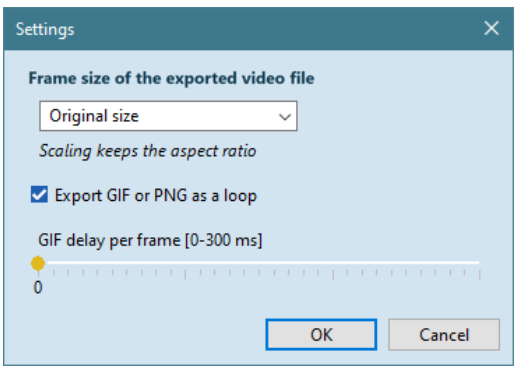
The projections of the camera path are also displayed on the view panels.



Save animation to a file



The currently recorded or loaded flight or camera path animation can be played into an animated GIF or animated PNG file. These formats are supported by most browsers (or Skype, for example): dropping the file on the browser window will start playback.
The settings are similar to animation export settings (see **6.1.2 Animation**).
Reducing the image size specified in the *Settings* window does not necessarily reduce the file size, because images captured in their original resolution are compressed more efficiently.



Resizing the AXISVM window changes the frame size and also affects the size of the generated file.

If *Export GIF or PNG as a loop* is checked, the playback of the file will be repeated automatically.

Finding elements

If you use the *Find* function from the Icon Bar **(2.16.20 Find)** in drone mode, the drone will fly to the item. If the *Select element* option is checked, the item will be selected.
The drone will also fly to the result of the search from the smart command line **(2.17 Smart command line)**. If you have entered several numbers separated by commas, i.e. you searched for several items at once, the drone will visit all items found.

☞ **Since the drone stops and turns towards the element at a distance depending on the element size, it is possible that another element blocks the view. Moving forward the drone can help to clarify the situation.**

☞ **The drone does not stop at elements, it flies through all objects. As it can get inside e.g. a reinforced concrete column, the rear faces of the column behind the rebars may not be displayed if the option *Settings / Preferences / Colors / Simplified OpenGL drawing also in drone mode* is checked. Uncheck this option to remove this speed optimization and enforce drawing of rear faces of objects in drone mode.**

Minimum and maximum values

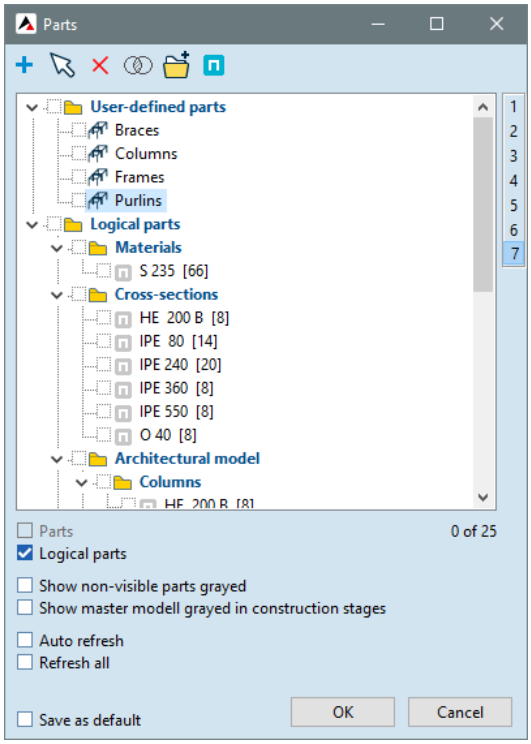
If Drone mode is activated when displaying results, finding minimum and maximum values (**6.1.1 Minimum and maximum values**) sends the drone to the first position where the extreme value is found. Positioning is performed the same way as for *Finding elements*.

Selection and object snap

Items are identified with object snap in drone mode. Their properties are displayed in a tooltip bubble and they can also be selected by Shift+click. The current load or result component set for the view can also be modified.

2.16.17 Parts

Lets you create sets of structural elements called parts. Working with parts makes the pre- and post-processing easier.
AXISVM allows you to display one or more parts, called active parts, at the same time. In addition, if the Parts check box is enabled the commands will only affect or refer to the entities of the active parts. The name of the current part is displayed in the Info window. If more than one part is turned on *n parts* is displayed, where *n* is the number of active parts.
There are two types of parts: user-defined parts and logical parts. *User defined parts* are created by the user selecting elements belonging to the part. *Logical parts* are created automatically by the program sorting the elements into categories by different criteria (material, cross-section, thickness, element type, storey or optimization groups, eccentricity groups).
You can activate an existing part by clicking its name in the list box.



Parts can also be activated without opening this dialog box by simply clicking the *Parts* speed button (at the bottom of the screen).

Depth of the tree expansion can be set by clicking on the numbers on the right-hand side of the window.

New

Creates a new user-defined part (a set of model entities).
You must assign a name to each new part. You must then define the new part by selecting entities (using the Selection Icon Bar if necessary) in the active display window.

Modify

Lets you modify the selected user-defined part. When the selection menu appears, the entities of the model that are in the part are displayed as selected.

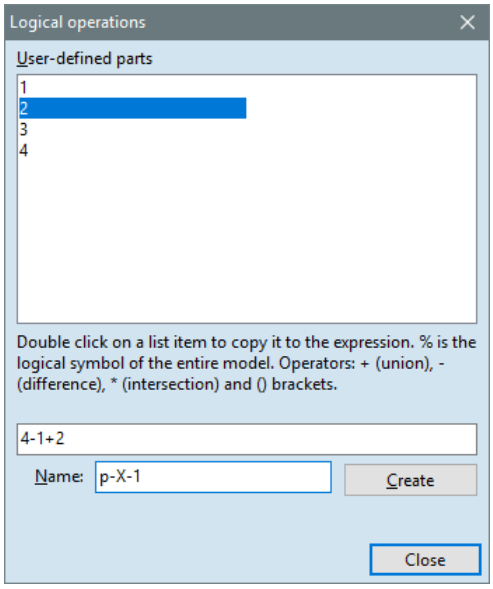
Delete

Lets you delete the selected user-defined part from the list. This command will not affect the model.

☞ **If section segment result tables are selected only section segments within the active parts are listed.**

Logical set operations

Creates a new part by performing logical set operations on the user-defined parts of a model. You have to specify the set operations. To enter the name of a part, double click on the respective name in the list. Use the % symbol to include the entire model.
For example: %-Columns will create the part that will include the entire model less the part named Column.
Clicking on the Create button, you can enter in the Name field the name of the newly created part. If you want to use the +, -, , (,) characters in the name of a new part, you need to put the name between "" marks (example: "floor +12.00").

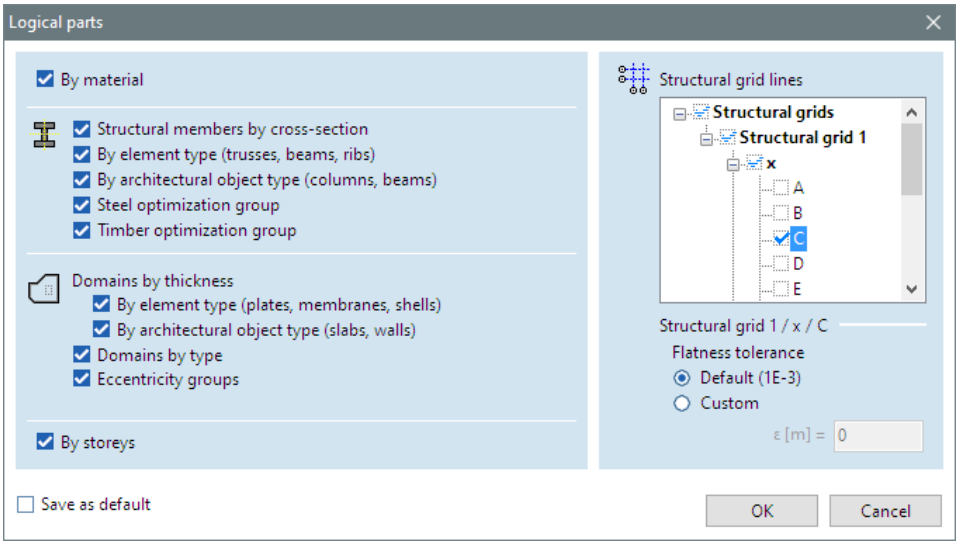


Creating new folders

Creating folders offer a way of sorting user-defined model parts. Parts can be moved and rearranged by dragging them to a new position. **[Ctrl]** and **[Shift]** allow multiple selection in the usual way.
Turning folders on/off turns on/off the parts within the folder.

Logical parts

This dialog is to set criteria for creating logical parts.
Architectural objects are defined by their geometry. Vertical beams, ribs and trusses are considered to be columns, horizontal ones are considered to be beams. Domains in horizontal plane are slabs, domains in planes perpendicular to horizontal planes are considered to be walls.
If there are storeys defined logical parts can be created by storeys.
Parts created from timber and steel optimization groups make it easy to change cross-section of an optimization group. Parts can be created from *Domains by type* (parametric ribbed domains, trapezoidal steel decks, CLT domains, etc.) or by eccentricity groups see **4.9.6 Domain**.
If the model contains structural grids, it is also possible to define logical parts based on structural gridlines. These parts contain elements within the plane obtained by sweeping the gridline perpendicular to the plane of the grid. Line elements and domains are included only if all their nodes fall within the plane. Plane tolerance can be customized for each gridline. Multiple gridlines can be selected (*Ctrl+click* or *Shift+click* on the gridline name) and a common value can be set.



Display switches

Display switches work in the following way:

Parts If it is on only the parts checked in the list are displayed.
If it is off the entire model is displayed.

Logical parts Turns on/off display of logical parts.

☞ **When working on parts, only the data of the active parts will appear in the tables by default.**

Auto Refresh
If it is on turning on or off parts will immediately cause a redraw. If it is off the screen is updated only after clicking the **OK** button.

Refresh all
If it is on parts will be turned or on off in all window panes in multi-window mode. If it is off part settings will be updated only in the active panel.

Show non-visible parts greyed
If it is on the entire modell wireframe is also displayed in gray to help identification of model parts.

Save as default If this option is checked the current state of the *Logical parts, Refresh all* and the *Show non-visible parts greyed* will be saved as the default setting when closing the dialog.

2.16.18 Sections



Lets you create section lines, planes and segments through any surface model, that can be used to process the results (displacements, internal forces, etc).

Sections are defined by selecting existing line elements and/or surface finite element edges. The current result component is plotted along these lines. Their use is not recommended for domains, as creating a new mesh will remove the previous finite elements.

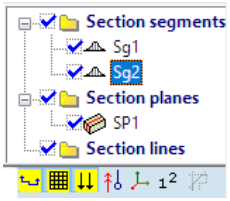
Section planes and section segments are generated by plotting the values of the result components along the intersection lines of the given plane with the model domains.
The section plane cuts through the entire model and plots the result on every intersection line.

Section segments have a length where the result is to be plotted, thus enabling the calculation of various average values and resultant values; by assigning a bandwidth the result diagram is constructed from values integrated over sections taken perpendicular to the section segment.

Sections can be organised into groups, allowing us to show or hide them all at once.

If a truss, rib or beam is within an active section plane and the result component has values on these elements a diagram is displayed on these line elements too.

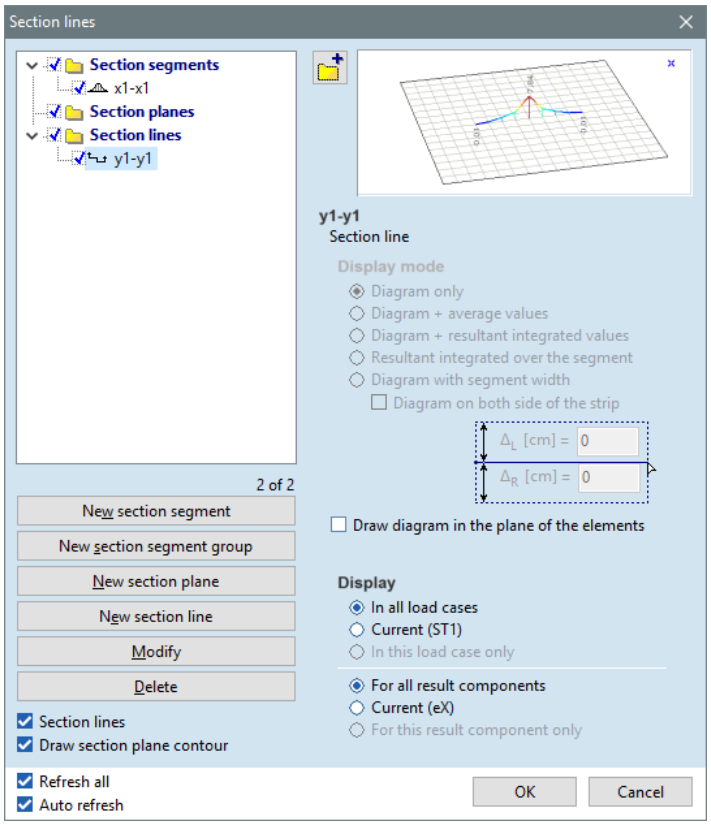
Section segment results can be listed in the Table Browser.
See **6.1.6.1 Section segment result tables**



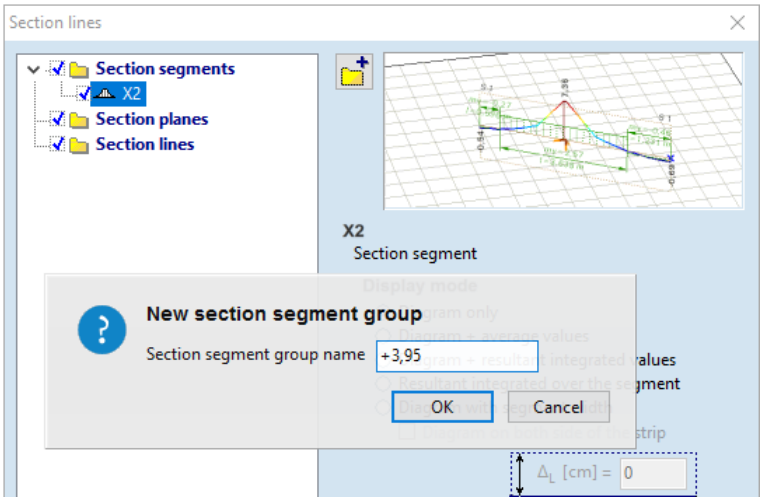
The dialog works similar to the *Parts* dialog.
Section lines, planes and segments can also be turned on and off using a speed button at the bottom toolbar.
If the result display mode is *Section* result diagrams are displayed only on section lines, planes and segments.
To reduce the complexity of drawings display of individual sections lines, planes or segments can be controlled to appear only in a certain load case and/or for a certain result component. Section segments, planes and lines are automatically sorted into three different folders (type groups).

☞ **Items cannot be dragged into a group with different type.**

☞ **If section segment result tables are selected only section segments within the active parts are listed.**



Creating a section segment group



Section segment groups can be created to make it easier to turn on/off several section segments together. Click *New section segment group*, enter a name for the group *(name)* and define any number of section segments. End definition by pressing **[Esc]**. Section segments will be numbered *(xx)* and get into the *name* folder as *name_xx*. Extension of an existing section segment group is also possible selecting the folder or a member of that before starting this function.

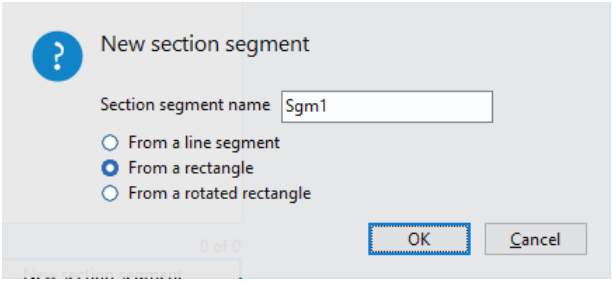
Creating new folders



Creating folders offer a way of sorting sections. Segments can be moved and rearranged by dragging them to a new position within its own type group. **[Ctrl]** and **[Shift]** allow multiple selection in the usual way. Turning folders on/off turns on/off the segments within the folder.

New section segment

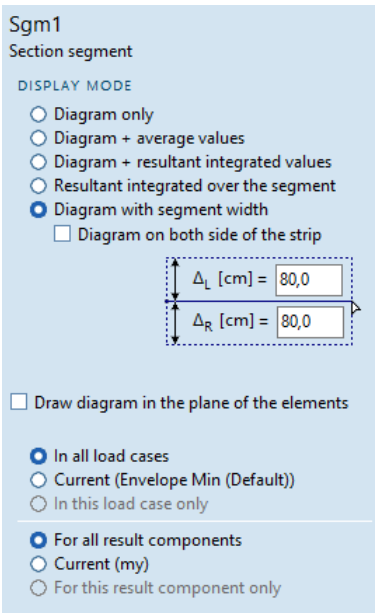
There are several ways to define a section segment.



From a line segment:
Enter two points of a domain or on domains in the same plane. Setting the radio buttons you can control how the internal forces diagram will be displayed. The average and resultant integrated values are calculated for subsegments defined between adjacent changings of the sign of the diagram. Left or right segment width can also be specified. In this case the diagram values are computed by averaging along the segment width. For the more expressive visualization of the segment width the diagram can also be displayed on both side of the strip.

From a rectangle:
In regions parallel to one of the global planes, we select the cross-section band by drawing a rectangle by clicking on the opposite corner points. The cross-section will be formed by joining the midpoints of the shorter sides, with the same width on both sides.

From a rotated rectangle:
A variant of the previous method that can be used in any plane.

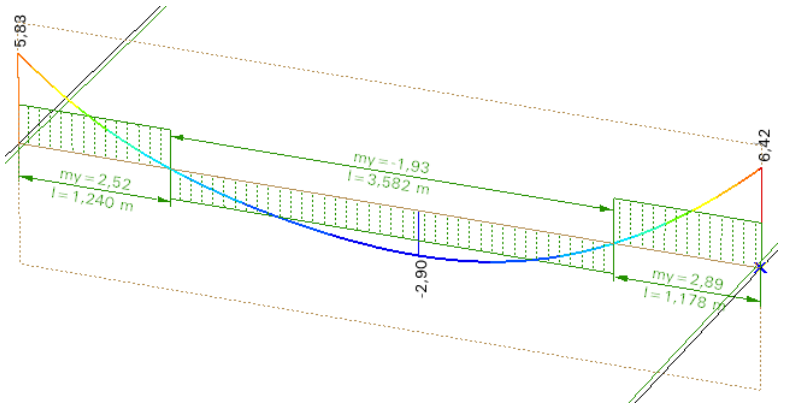


Section segments defined by rectangles automatically set the *Diagram with segment width* display mode.
In the case of envelope or critical results, the programme first calculates the extreme value at each point and then integrates them. In the case of aggregated earthquake results (X, Y, Z, +, -), the programme first calculates the value aggregated using the CQC or SRSS method at each point and then integrates them.

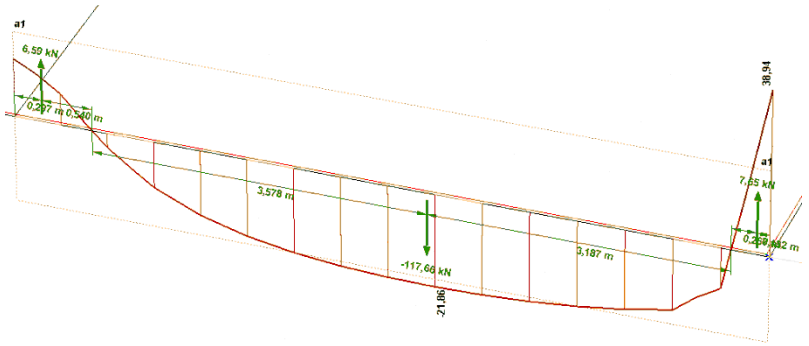
☞ **In these cases the integrated value is not an actual one in the sense that it is computed from not simultaneous values.**

Diagrams are usually displayed perpendicular to the element plane but checking the option *Draw diagram in the plane of the elements* rotates the diagram into the plane. In the *Display parameters* dialog this parameter can be turned on/off for all section segments.

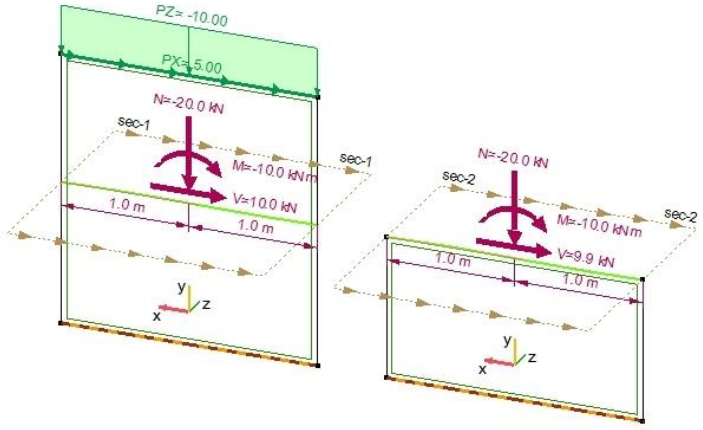
☞ **If the option *Draw diagram in the plane of the elements* is selected, the option *Diagram with segment width* cannot be used.**



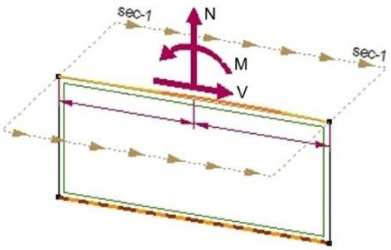
Display of the average values



Display of the resultant integrated values



Resultant integrated over the segment



The section segment divides the domain into two parts. For the calculation one of them is removed. The removed part is selected such a way that the local axis perpendicular to the section segment (axis y denoted by yellow in the picture) points outward from the remaining part. The arrows are displayed on the side of the removed part. They show the forces and moment acting on the remaining part by the removed one. The direction of the section segment is shown by the arrows along the section plane contour. The section segment is directed from the first to the second point picked at its definition. The picture on the right shows the positive direction of resultant components.

☞ **The resultant integrated over the segment is displayed only for section segments parallel with the local x or y axis.**

☞ **If *Envelope* or *Critical* is selected the resultant components are displayed in *Min* or *Max* mode only. The values of the three resultant components are not simultaneous.**

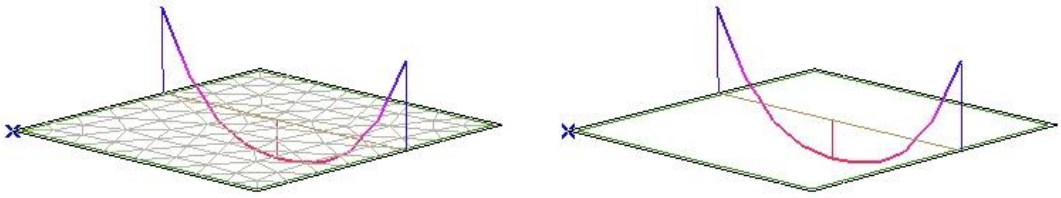
☞ **If *Envelope* or *Critical* is selected the resultant value is integrated for each envelope or critical load combination over the segment and then the minimum or maximum is shown. Therefore the displayed result belongs to only one load combination and it is independent from the displayed diagram that contains non-simultaneous values.**

☞ **If combined seismic result (X, Y, Z, +, -) is selected the resultant value is integrated for each modal load case over the segment and then the combined result according to the CQC or SRSS method is shown. Therefore, the result is independent from the displayed diagram that contains non-simultaneous values.**

New section plane

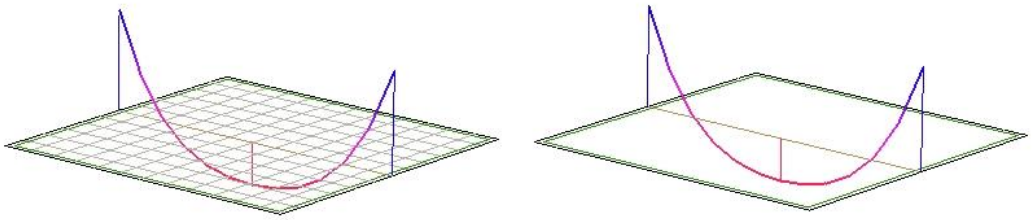
Click *New section plane* and assign a name to the section. This type of section is based on a plane. Click or enter two points to set the section plane. Then click OK in the Selection Icon Bar to save. In perspective view you have to click or enter three points to set the section plane. Section planes are displayed as rectangles of dotted lines. You can enable/disable the display of section plane rectangles.

Section planes are useful when you want to display results only along a certain line through the entire structure.



New section line

Click *New section line* and assign a name to the section. You then have to select surface edges or beam elements that define the section line. Then click OK in the Selection Icon Bar to save. Section lines can be discontinuous.

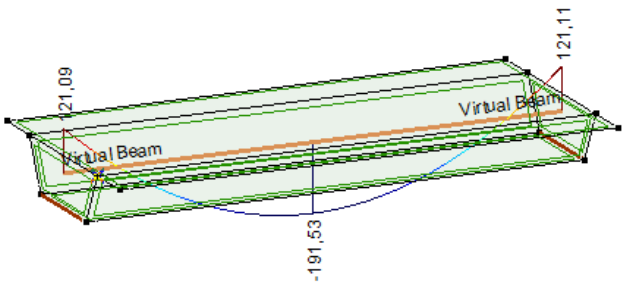


The checked section lines, planes and segments are active.
You can use Auto Refresh and Refresh All checkboxes, New, Modify and Delete buttons the same way as in the *Parts* dialog.

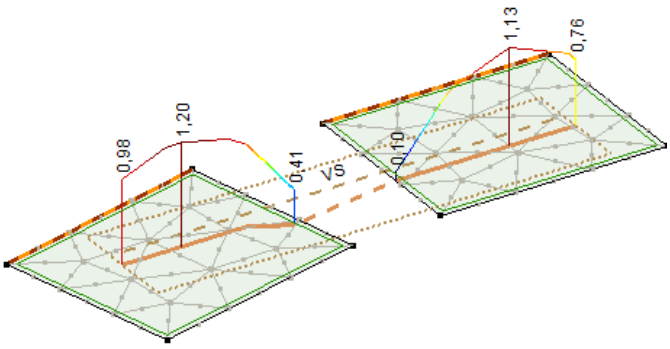
☞ **The tracelines of the section lines are not correlated with the directions of the result components displayed.**

2.16.19 Virtual beams

Virtual beams are used for beam-like representation of results of domains and attached ribs. After calculating the surface forces the program creates a finite number of sections perpendicular to the direction of the virtual beam and reduces the section forces to the center of gravity of the section.



If active parts contain domains with virtual beams and a result component is selected from Virtual beam internal forces (**6.1.12 Virtual beam internal forces**) the diagram will be displayed along the centerline of the virtual beam.
If a virtual beam and the corresponding domains are turned on, and compatible force-like results are selected, then the reduced forces will be plotted on the axis of the virtual beam.



Virtual strips are like virtual beams but sections do not run until the domain edge (they are limited by a fixed strip width on the left and on the right side, Δ_L and Δ_R).

Table of virtual beam results can be displayed in the **Table Browser**.
See **6.1.6.1 Section segment result tables**

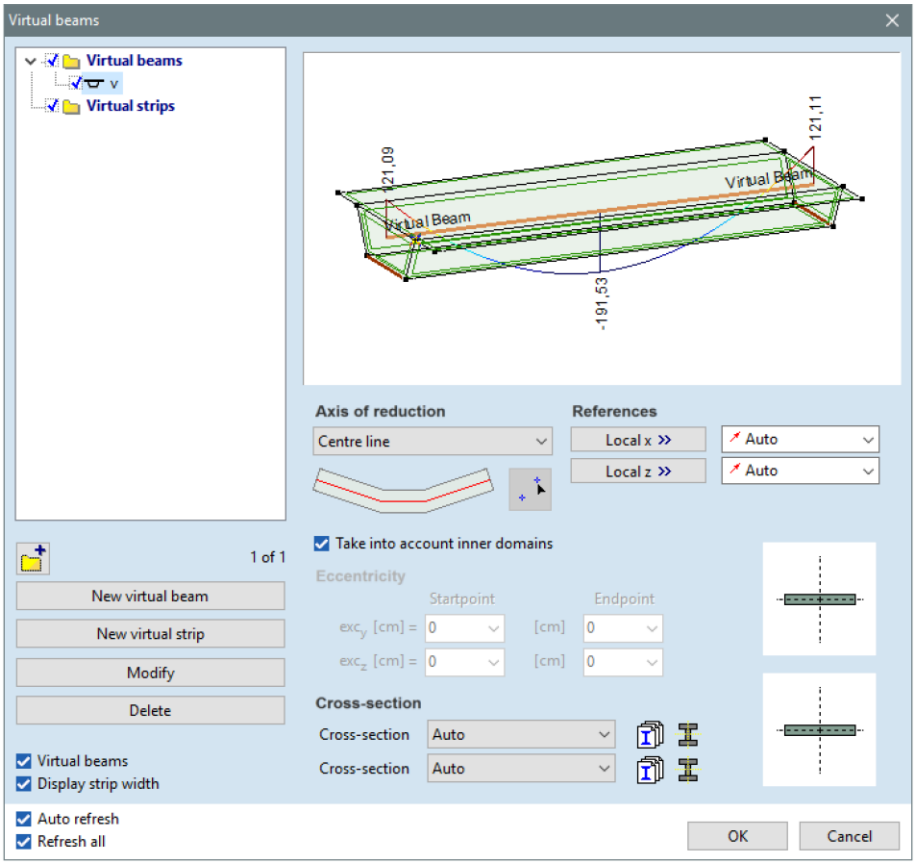
New virtual beam Enter a name and select domains where the virtual beam should run. The axis of the virtual beam is not necessarily a continuous line and its type can be set (see below). See **4.9.25 References.**

New virtual strip Enter a name then the start-point and end-point of the strip. These points define the axis of the virtual beam. If the two points are on different domains, domains must be in the same plane. The Δ_L and Δ_R strip width values of a new strip are set automatically to 0.5 m, but it can be changed.

☞ **Virtual beam sections are created from both the meshed and unmeshed domains. Virtual strips require a mesh.**

☞ **The definition of virtual beams and strips is much quicker on unmeshed domains, since then the program calculates the intersections with all the finite elements only after mesh generation, not at the moment of definition.**

☞ **Items cannot be dragged into a group of different type.**



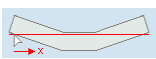
Axis of reduction of virtual beams



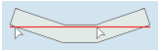
Centre line: The axis of reduction is the centre line of the virtual beam. The local system of the virtual beam can be set independently from the axis of reduction by setting the local *x* and *z* direction under *References*.



Straight line: The program slices the domains along the local x direction, perpendicular to the *x* direction. The cross-section at a certain point consists of the intersection of the domains and the section plane. The first and last non-empty cross-section defines the start point and end point of the virtual beam. The axis of reduction is a straight line between the center of gravity of the first and last section. The axis of reduction does not necessarily run parallel with the local x direction but depends on it.

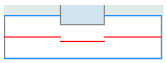


Point and reference vector: The axis of reduction will be a line going through a given point and parallel with a selected reference vector.

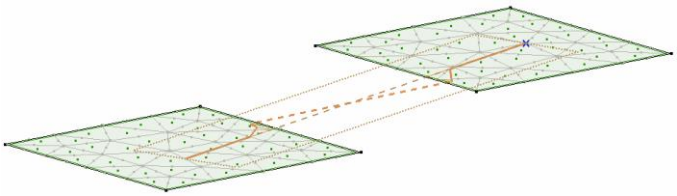


Two points: The axis of reduction goes through two selected points.

Axis of reduction of virtual strips

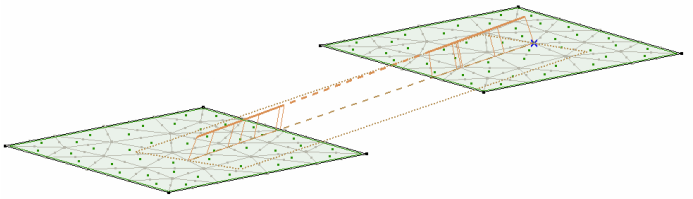


Centre line: The axis of reduction coincides with the centre line of the virtual strip, which is the collection of centres of gravity of the sections perpendicular to the line defined by the two end points of the strip.



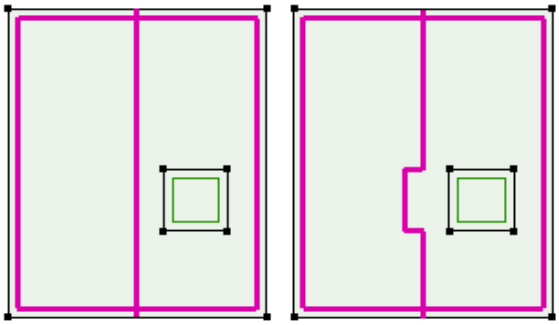


Eccentric: The creation of segments is identical, but the internal forces are reduced to a line segment, eccentric to the one connecting the end points of the strip. The eccentricity is governed by the transition of the end points of the strip.



Take into account inner domains

If checked, the program also takes into account the unselected inner domains when finding the centre line and integrating the results. If unchecked these domains do not contribute to the centreline and they are ignored during the integration as well.



The same virtual beams taking into account the inner domains (left) and ignoring them (right).

References

References are set automatically. By default, the local x direction of the virtual beam is parallel with the longest side of the bounding box of the selected domains in the global space. The local y and z directions are set the same way as the automatic references for beam elements. The references can be customized. (See **4.9.25 References**).

Cross-section

A custom cross-section can be defined for virtual beams and strips. This cross-section can be read through the COM interface and can be used in external design programs.

Creating new folders



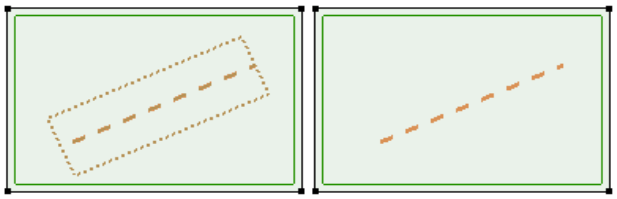
Creating folders offer a way of sorting virtual beams. Virtual beams can be rearranged by dragging them to a new position within its own type group. Turning folders on/off turns on/off all virtual beams within the folder. **[Ctrl]** and **[Shift]** allows multiple selection in the usual way.

Virtual beams

This checkbox controls the display of virtual beams in the model.

Display strip width

This checkbox controls the display of virtual strip width.

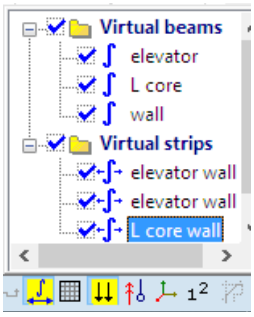


The same virtual strip with (left) or without (right) displaying strip width.

Speed button

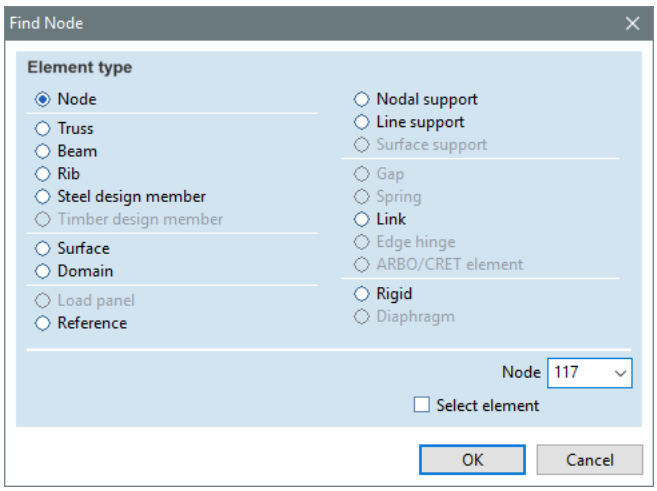


A dedicated speed button at the bottom right allows turning on/off virtual beams and strips.



2.16.20 Find





Finds the entity having a specified index, and moves the cursor over it. If *Select element* is turned on the element found will also be selected (displayed in purple).

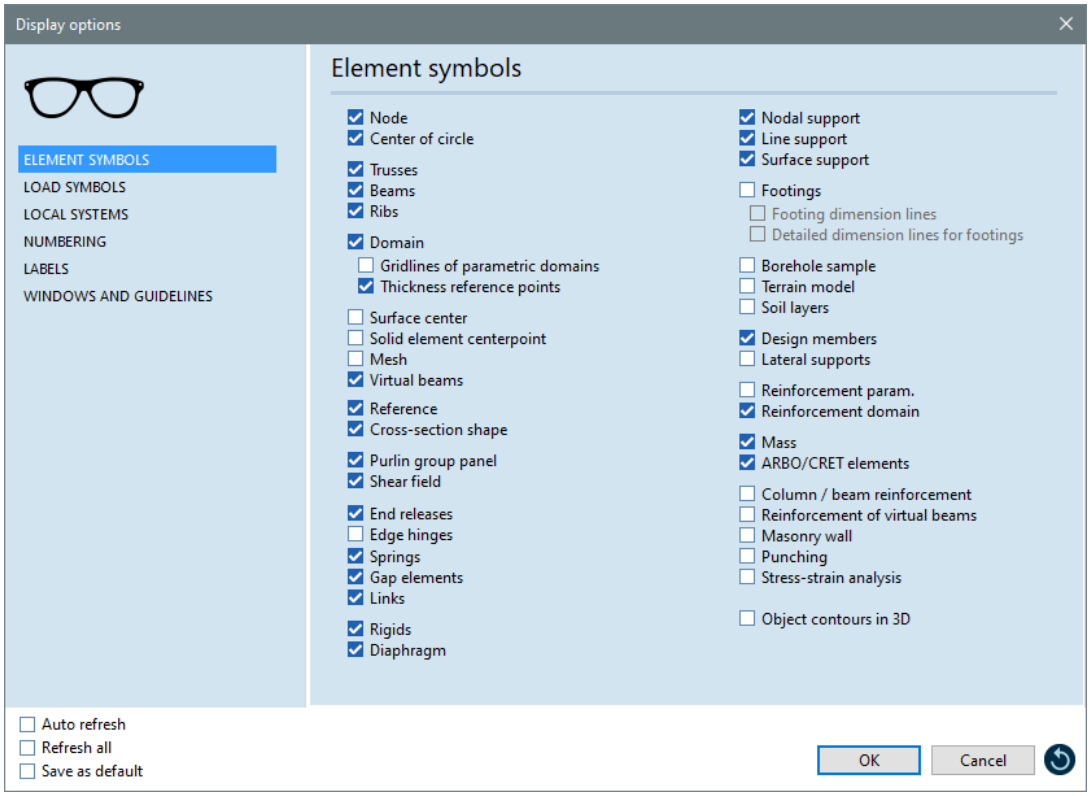
You can also use the smart command line to search for items, where you can search for (select) multiple items at once. See **2.17 Smart command line.**

Using the function in *Drone* mode (see **2.16.16 Drone**) sends the drone to the element found.

2.16.21 Display options



Element symbols



Enables/disables display of element symbols.

Common symbols can be turned on/off using a speed button. 
Symbols can be customized. See *Settings / Preferences / Graphic symbols*.

Node
Enables display of the nodes (small black rectangles).

Trusses, Beams, Ribs
Enables display of line elements

Center of circle
- Enables display of centers of circles as a small cross.

Domain
Enables display of the domain's contour.
- The color of the domain is the same as of the surface type.
  Color codes: plate = red, membrane = blue, shell = green.

Gridlines of parametric domains
- Grid lines are displayed in the colour of the surface type. Gridlines of parametric ribbed domains or composite ribbed domains are thick lines, grid lines of hollow core slabs are thin dashed lines.

Thickness reference points
- Thickness reference points specified when defining variable thickness domains are shown as thin x crosses drawn with a brown dashed line and connected by thin dashed brown lines. The value of thickness is displayed only if *Properties / Thickness* on the *Labels* tab is enabled.

Virtual beams
- Axis of the virtual beam is drawn as a thick light brown line.

Surface center
Enables display of the center point (selection point) of the surface elements.
- Color codes: plate = red, membrane = blue, shell = green.

Mesh
Enables display of the inner mesh lines.
- When disabled the generated mesh lines are not displayed.

Nodal support
Enables display of the nodal supports.
- Nodal supports appear as thick axes.
  Color codes: axial displacement=yellow, axial rotation= orange.

Line support
Enables display of line supports.
- The line supports appear as a thick edge.
  Color codes: axial displacement=yellow, axial rotation= orange.

Surface support
Enables display of the surface supports.
- Surface supports appear as a light brown hatch .

Footings
- Footings designed on the R. C. Design tab appear with their calculated or specified shape and size.
  Dimension lines
  - Enables display of footing dimension lines.

Links
Enables display of link elements.
- *Node-to-node link elements* are displayed as solid green lines with an arrowhead showing the location of the link.
  Line-to-line link elements are displayed as solid green lines with an arrowhead showing the location of the link and dashed green lines at the line end points.

Rigids
- Enables display of rigid bodies. They appear as thick black lines.

Diaphragm
- Enables display of diaphragms as dashed thick purple lines.

Reference

Enables display of the references.

- Red vector, crosshairs or triangle.

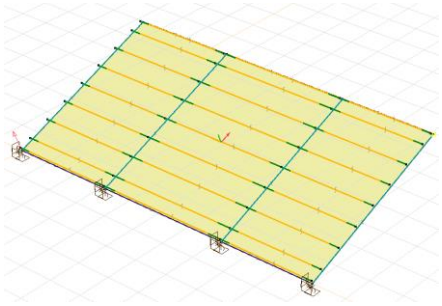
Cross-section shape

Enables display of the shape of the cross-section of the truss/beam/rib elements.

- The user-defined cross-sections will be displayed as rectangles that circumscribe the shape of the cross-sections.

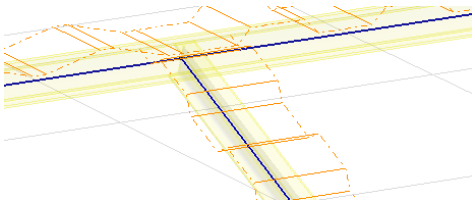
Purlin group panel

- The beam group panel is indicated by a pale-yellow transparent fill; the beams are orange, and their overlapping sections are green.
  As the panel is just a logical unit, it does not appear on a rendered view.



Shear field

- Elements with a shear field attached are displayed with a trapezoidal sheet pattern with a dashed outline. The cladding itself is not displayed.



End releases

Enables display of the end release and edge hinges.

End release:

- Blue circle: hinge / roller
  Blue circle + cross semi-rigid hinge
  Red circle: spherical hinge
  Solid blue circle: plastic hinge

Edge hinges

- Circles on the edges.

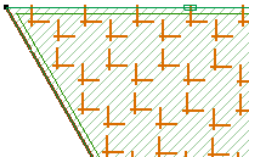
Design members

Enables display of steel or timber design members.

- An orange line along the member and the number of the member.

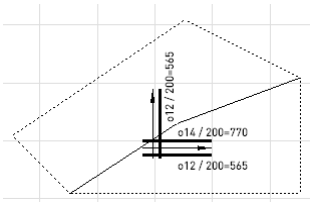
Reinforcement param.

- Enables display of two intersected straight lines at surface centers where reinforcement parameters are assigned. The direction of these lines indicates the defined reinforcement directions. (See **6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module**).



Reinforcement domain

- Enables display of mesh independent reinforcement domains as dashed brown outlines.
  Top and bottom x and y reinforcements are also displayed.
  Two vertices of the polygon are connected to the center by brown lines.



Mass

Enables display of the symbol of the concentrated masses.

- Double red circle.

ARBO-CRET elements

Aschwanden ARBO-CRET elements placed into the model.

- A schematic drawing of the element is displayed.

AIRDECK elements

AIRDECK elements placed into the model.

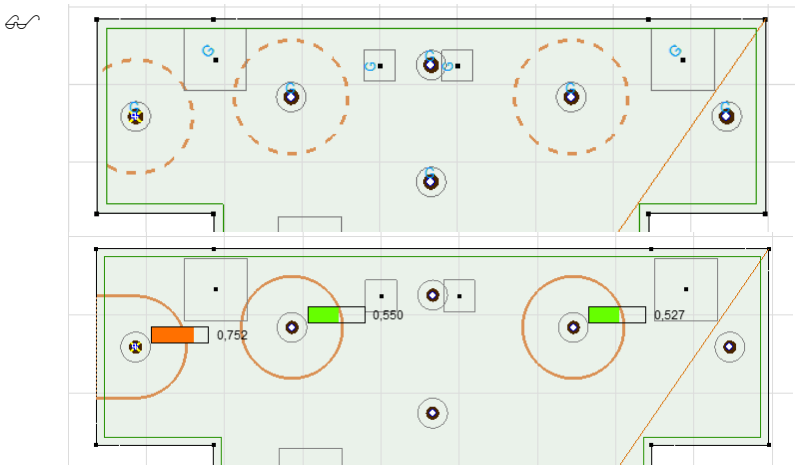
- Void formers are displayed as circles in wireframe mode and balls in rendered view.

Reinforcement of cores and walls

For detailed description, please see **6.5.17 Design of reinforced concrete cores and walls – RC5 module**

Punching

For columns, wall corners and wall ends where punching analysis parameters are defined a dashed brown outline is displayed around the element. If the actual punching verification has been carried out, its utilization and the basic control perimeter are displayed.



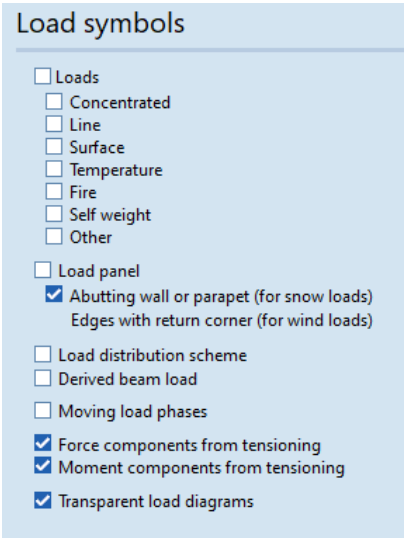
Stress-strain analysis parameters

Structural elements with stress-strain parameters are shown in the main window with zigzag lines.

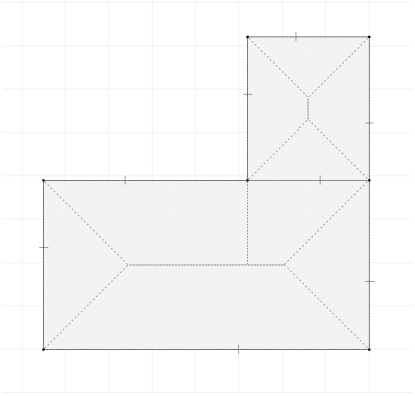
Object contours in 3D

Displays static model with a 3D wireframe look (cross-section, thickness, eccentricity can be checked). Colors depend on architectural type (column, beam, wall, slab).

Load symbols



Display of load symbols can be set separately for each load type (concentrated, distributed along a line, distributed on surface, temperature, fire, self weight, moving load, miscellaneous (length changing, tension / compression).
To display of surface loads distribution to beams (see the diagram on the right) check *Load distribution scheme*. To display the derived beam loads, check *Derived beam load*.



Load panel Load panel outlines are thick dashed turquoise lines. If load panels are attached to domains small rectangles appear along the outline showing the linkage.

Abutting wall or parapet / Edges with return corner Load panel edges with abutting wall or parapet can be selected when defining snow loads. These edges are displayed with a light brown rectangle along the edge. Edges with return corner can be selected when defining wind loads.

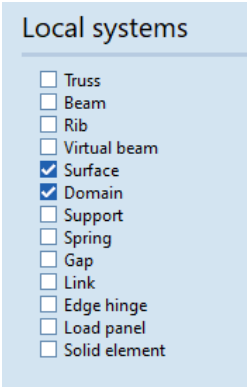
Load distrubution scheme / Derived beam load Display of the load distribution scheme and the derived beam loads (see above).

Moving load phases If this option is turned on all phases of moving loads are displayed in grey. If this option is turned off the moving load is displayed only in the position determined by the current load case.

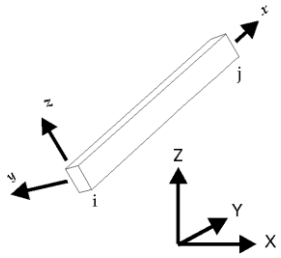
Force / Moment components from tensioning It displays the force and moment components exerted by the tendons of post-tensioned structures on the structure, based on the results obtained from the PS1 module.

Transparent load diagrams It controls the transparency of the fill of load diagrams.

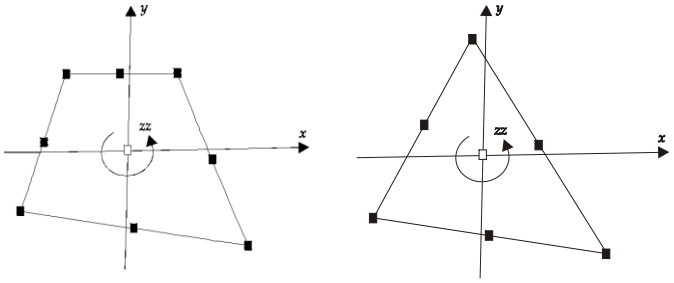
Local systems



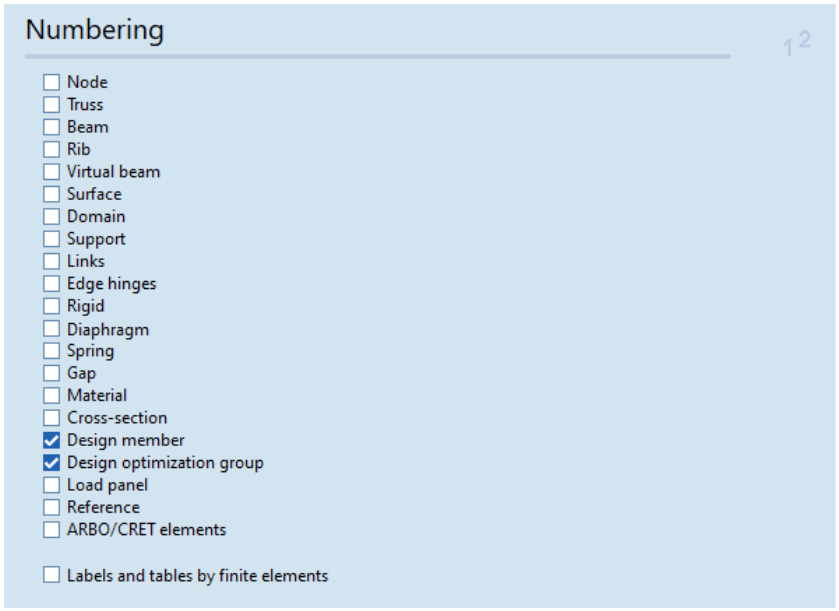
Enables display of axes of the elements in the local coordinate system.

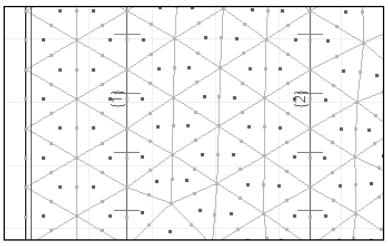


Beam element local coordinate system

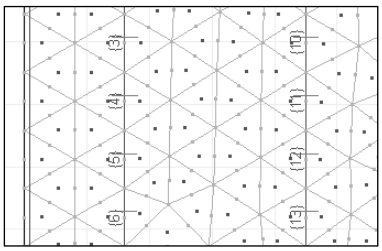


Surface element local coordinate systems





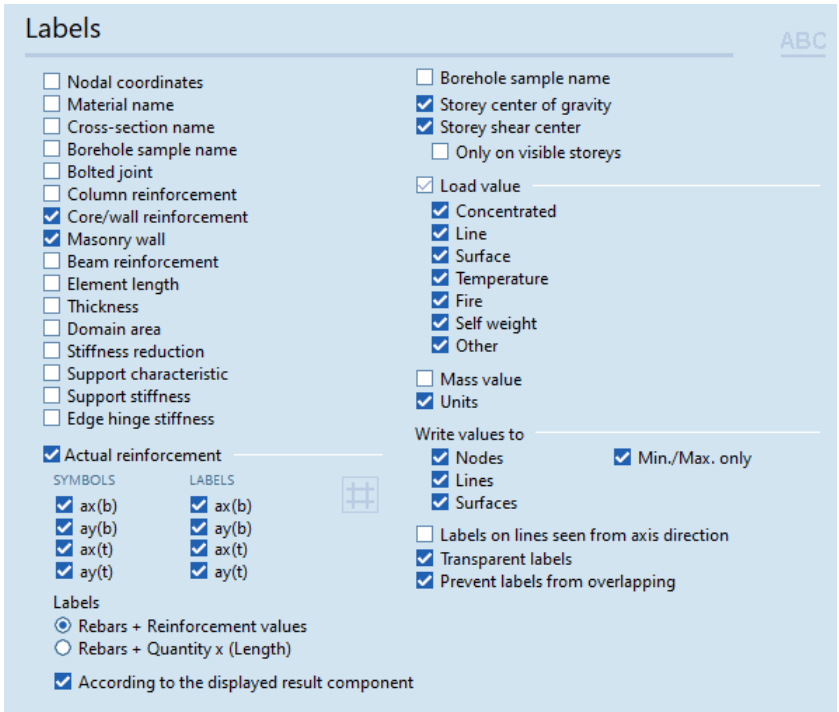
Displaying the number of nodes, elements, materials, cross-sections, references.



For meshed line elements checking *Labels and tables by finite elements* displays the number of finite elements instead.

Labels and tables by finite elements

Turning on this switch replaces structural member numbers on diagrams with finite element numbers. Tables also display results on a finite element basis and not for structural elements.
See **3.2.14 Assemble structural members, 3.2.15 Break apart structural members**



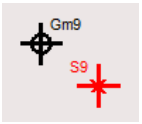
Labels on lines seen from axis direction

Checking/unchecking *Labels on lines seen from axis direction* turns on/off labels on lines seen from the direction of their axis (seen as points).

Transparent labels If this option is checked all labels are transparent. Otherwise, labels are drawn on a rectangle filled with the background color.

Prevent labels from overlapping If this option is checked the program prevents labels from overlapping by moving and hiding less important labels. In the latter case zooming in may increase the number of visible labels. If large number of labels are to be displayed this optimization may be a time-consuming process.

Story center of gravity Enables display of center of gravity of each storey. The center of gravity is only displayed if there are seismic loads (valid seismic spectra and storeys defined), and static results. AXISVM converts loads of load cases used to calculate the vibration shapes for seismic analysis into masses then calculates the center of gravity for each storey. The centers are displayed as black +s in black circles with a label Gm*i* where *i* is the level number.



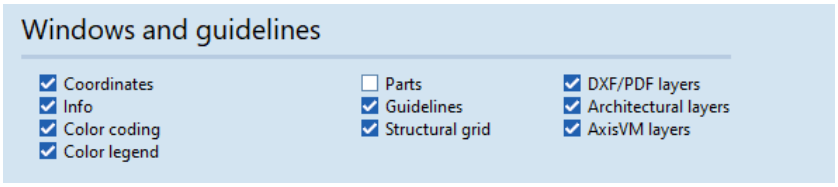
Story shear center Enables display of shear center of each storey. AXISVM calculates storey shear centers by finding wall sections and using the same method as for thin-walled cross-sections. See **3.3.4.1 Shear centers of storeys.** The centers are displayed as red +s, with a label S*i*, where *i* is the level number.

If the *Only on visible storeys* is checked centerpoints of hidden storeys are not displayed.

Properties Enables display of the name and values of materials properties, cross-sections, element lengths or thicknesses, load values, masses, etc.
If the Units option check-box is enabled, the labels will include the units as well.

Write values to Controls how the result labels are placed over the drawing.

Actual reinforcement Enables labeling for top and bottom reinforcement in *x* and *y* directions independently and sets the labeling mode.
Turning on *According to the displayed result component* makes the current reinforcement component the only displayed component.



Information Windows

Coordinates	See **2.19.1 Coordinate window**
Info	See **2.19.2 Info window**
Color coding	See **2.19.3 Color coding**
Color legend	See **2.19.4 Color legend window**

Display The display of the actual parts and guidelines can be turned on and off.

Parts	Enables/disables the display of user-defined and logical parts.
Guidelines	Enables/disables the display of guidelines.
Structural grid	Enables/disables the display of the structural grid.
DXF/PDF layers	Enables/disables the display of background layers.
Architectural layers	Enables/disables the display of imported architectural objects.
AXISVM layers	Enables/disables the display of layers defined within AXISVM.

Auto Refresh If it is turned on any change in settings will make the active panel redrawn immediately.

Refresh All Changes will affect all panels in multi-window mode.

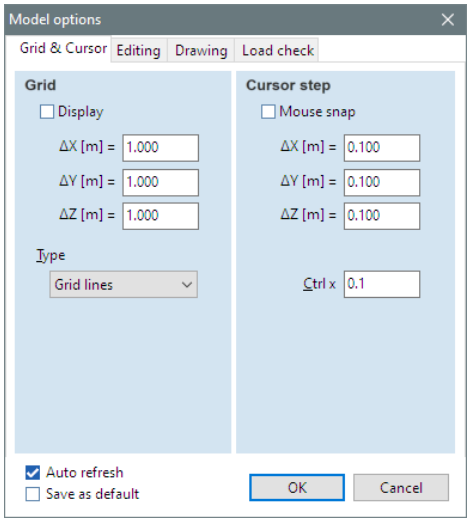
Save as default Saves the current symbol display option settings as default for new models.

2.16.22 Model options

Allows the selection of the options for the settings of the grid, cursor, editing, drawing parameters and design code.

2.16.22.1 Grid and cursor

Grid The grid consists of a regular mesh of points or lines and helps you position the cursor to provide a visual reference. Depending on its *Type* the grid is displayed as:



Dot grid Axes are displayed with yellow crosses, points in gray.

Grid lines Axes are displayed in yellow, lines in gray.

Display Displays or hides the grid.

ΔX, ΔY, ΔZ Sets the spacing of the dots/lines of the grid in the direction X, Y or Z.

Cursor step Allows snapping the mouse to an invisible dot mesh (not the grid).
You can set the cursor step parameters as follows:

Mouse snap Restricts the movement of the mouse cursor to an invisible grid specified by the cursor step values below.

ΔX, ΔY, ΔZ Restricts the cursor movement to regular intervals. Each time you press a cursor movement key the cursor moves in the corresponding direction (X, Y or Z) one step (ΔX, ΔY or ΔZ respectively).

Ctrl x Sets the value of a factor that increases or decreases the cursor step size if you press the **[Ctrl]** key when you move the cursor. This allows you to achieve adequate positioning accuracy.

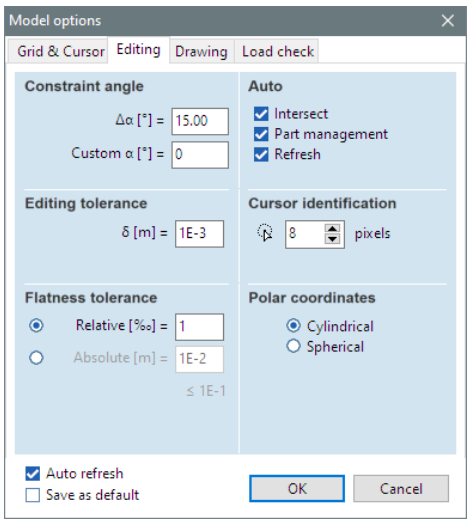
☞ **The cursor step is ignored if you position the cursor on a line not parallel to global coordinate axes. In such a case, the cursor will move along the line.**
If the editing tolerance is greater than the cursor step, the mouse will follow an invisible grid specified by the editing tolerance.

When using with constraints, the cursor step is applied in the constrained direction with the DX value.
See 4.7.4 Constrained cursor movements

☞ **If the grid step and the cursor step is set to the same value, nodes will be placed snapped to the grid.**

2.16.22.2 Editing

Constraint angle During the model editing the movement of the cursor can be constrained.
Using the **[Shift]** key while moving the cursor, the movement direction can be set. In this case the constrained movement of the cursor will be based on two types of angles (for other type of constrained movements see 4.7.4 Constrained cursor movements).



Auto Sets commands that are applied automatically if the corresponding check-box is enabled.

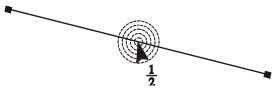
Intersect
Sets the line intersection handling. At intersection points of lines a node will be generated and lines will be bisected. If surfaces are intersected by lines, they will be split, and the resulting elements will have the same material and cross-sectional properties as the original.

Part management:
Any entity drawn or modified after the check-box is enabled will be associated with all of the active parts.

Refresh
Sets the display refresh mode to automatic.

Editing tolerance If two nodes are closer than the value set as the editing tolerance, they will be merged in the case of a mesh check. This value is also used when comparing surface thickness or beam length.

Cursor identification



The element under the cursor is identified if it is within an adjustable cursor identification distance. The unit for cursor identification distance is pixels.

If more than one element is within this range the closest one will be identified.
See 4.7.1 Cursor identification

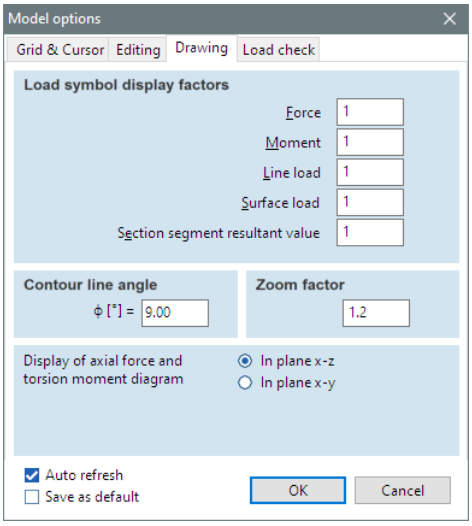
Plane tolerance Nodes of domains and surfaces must be in plane. If a node of a domain or surface deviates from this plane more than the given value the element will be deleted. Plane tolerance can be specified in two ways:

Relative [‰]	per thousand of the biggest extension of the element polygon
Absolute [m]	a given value

Auxiliary coordinates Cylindrical or spherical.
See 4.3.2 Polar coordinates

2.16.22.3 Drawing

Load symbol display factors Sets the display size of the load symbols. This factor is applied when the checkbox in *Symbols / Graphics Symbols / Load* is enabled. These values do not affect load values.



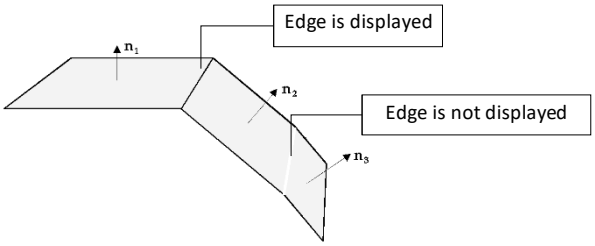
Force
Sets the display size of the symbol of concentrated force loads.

Moment
Sets the display size of the symbol of concentrated moment loads.

Line / surface load
Sets the display size of the symbol of line / surface loads.

Section segment resultant value
This factor determines the size of the arrow representing the resultant value over a section segment.

Contour line angle Sets the display of the inner mesh lines (between adjacent surface elements). The common edge of two or more surface elements is displayed if the angle enclosed by the normal to the planes of the elements is larger than the value set here.

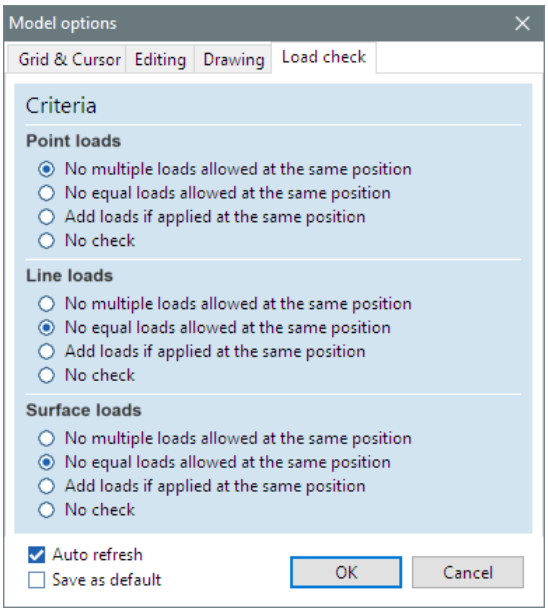


Zoom factor Sets the scale of magnification/reduction of the zoom commands associated to the **[+]** and **[-]** keys.

Display of axial force and torsion moment diagram Until Version 12 normal force and torsion moment diagrams were always drawn in the local x-z plane. From Version 13 it can be set to the local x-y plane as well.

2.16.22.4 Load check

Criteria Criteria for checking loads can be set here. These checks run automatically after defining loads. The category *Point loads* refer to nodal loads and concentrated loads on line elements or domains, the category *Line loads* refer to line loads applied to line elements or domains, while *Surface loads* include area loads placed on domains or load panels.



Options:

No multiple loads allowed at the same position
If a load is placed on the position of an existing load, it is deleted.

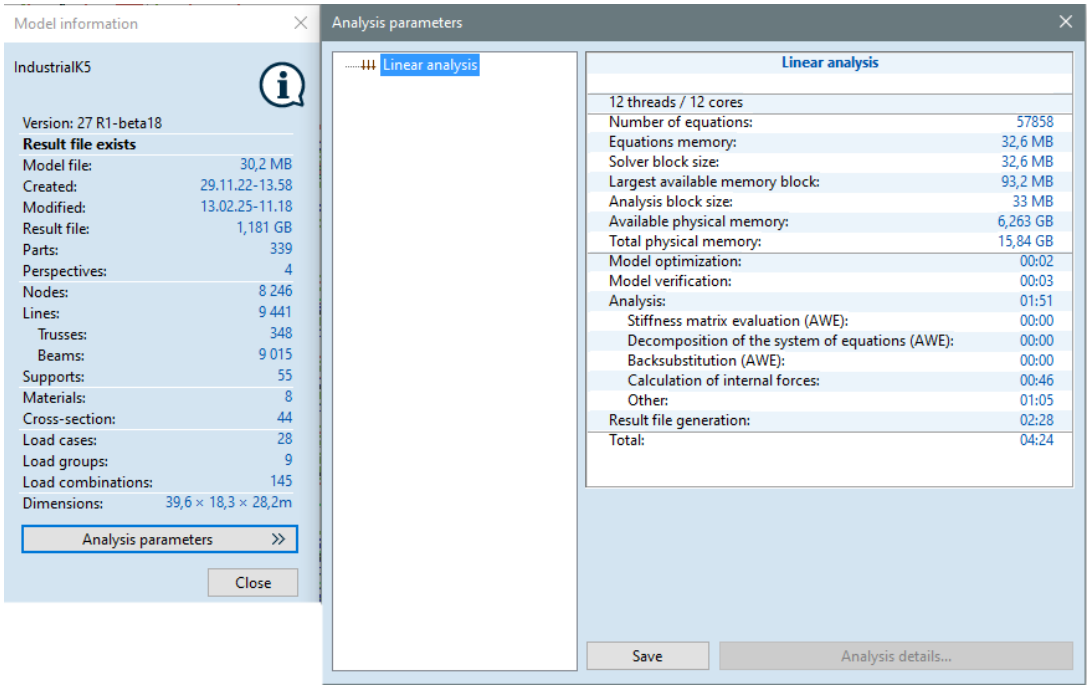
No equal loads allowed at the same position
If a load is placed on the position of an existing load having the same intensity, it is deleted.

Add loads if applied at the same position
If a load is placed on the position of an existing load, it is not created but its intensity will be added to the intensity of the existing load instead.

No check:
Any number of loads can be defined at the same position.

2.16.23 Model info

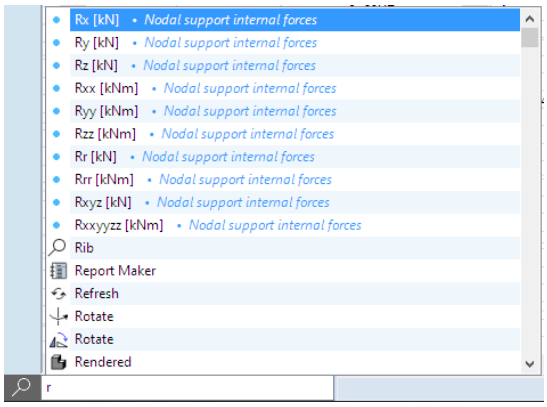
Shows the main parameters of the model.
Clicking the *Analysis parameters* button certain parameters of the latest analysis (memory usage, running time) can be studied.

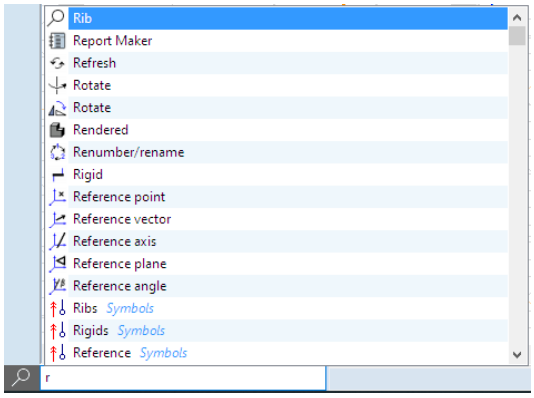


2.17 Smart command line



Most of the commands and settings of AXISVM is made available directly through a simple search field called smart command line. Click into the edit field or press *Ctrl+F* to see commands and settings matching the entered characters. This function finds the typed letters also inside expressions but items beginning with them appear at the top of the list.
The range of available commands depends on the active tab (e.g., result components are available on the *Static* tab but not on the *Loads* tab while a load case can be selected on both).





Certain keys have a special function

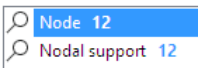
↑ ↓	Move up or down within the list
Enter	Activate the command for the selected item
Esc	Cancel the search

The list includes the following commands and settings:

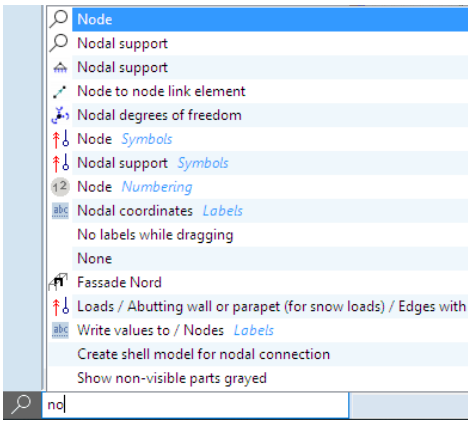
- All commands in the main menu, on the icon bar, coordinate window, etc. that can have a keyboard shortcut assigned
- All the symbols, labels and switches configurable on the *Display options* dialog (**2.16.21 Display options**)
- Load cases (on the *Loads* tab or on result or design tabs)
- Result components load situations (on tabs displaying results)
- Settings available at *Settings / Preferences* (**3.3.11 Preferences**)
- Model options available at *Settings / Model options* (**2.16.22 Model options**)
- Elements can be found by their number or name
- Elements can be selected with or without filter criteria

Find elements

Example: *Searching for Node 12.*
Upon entering **no** a list of matches appears. Find commands are listed at the top. Typing a space and **12** will narrow down matches to the following



Pressing Enter will select Node 12 and move the mouse pointer there. In short, you can find Node 12 by typing **no 12**[Enter].



Finding multiple elements

More than one numbers can be entered, separated by commas.
E. g. to select Node 105, 107 and 111 type **no 105,107,110**[Enter]

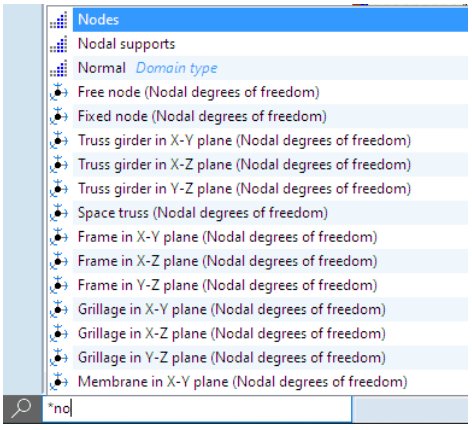
As the smart command line finds the term inside expressions, nodal support 5 can be found also by typing **sup 5**[Enter], taking advantage of the fact, that *Nodal support* appears at the top of the list after narrowing down the matches.

Selection

It is also possible to select or deselect certain elements from the smart command line. If parts are activated, these commands operate on the visible parts only.

Simple selection

If the first character is ***, the list displays the selectable elements only and the command will select the chosen item.
Typing ***-** at the beginning will deselect the chosen item.
To select everything type ***[Enter], to deselect everything type ***-**[Enter]
To select all nodes type ***no**[Enter]
Nodes with a given degree of freedom can also be selected. Selection of other elements like beams, trusses, ribs, domains work the same way.



Selection by a property

Type * and start typing the name of a material or cross-section (or type **mat** or **cr**) to select elements made of a certain material or having a certain cross-section. Typing ***Bottom rib, *Rib in midplane, *Top rib** the appropriate ribs can be selected. It is enough to type the first characters of these expressions and choose from the list. Similarly, typing ***Variable cross-section** selects these line elements, while typing ***Variable thickness** selects these domains. Domains of a special type can also be selected by typing ***Ribbed, *Hollow core, *Composite ribbed slab, *Trapezoidal steel deck, *CLT, * Custom stiffness matrix.** Architectural type can also be used (***Columns, *Beams, *Walls, *Slabs**).

Selection by a parameter

Elements can also be selected by a parameter value. An example: Select line elements longer than 5 m.

After typing ***le***[Enter] the command line changes to *Length (m) and the list is replaced by a help message

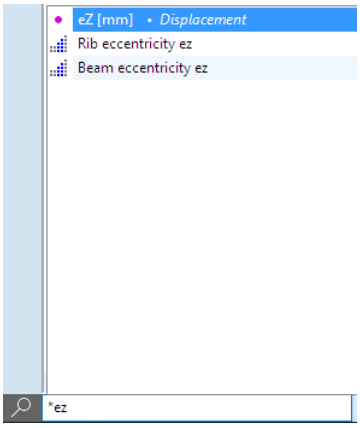
Type the filter condition then press Enter.
Examples: >0, >=10, <>20, =30, <=40, <50

Continue typing to get ***Length (m) > 5***[Enter]. The appropriate line elements will be selected.
The condition can include one of the >, >=, <>, =, <= and < symbols and a numerical value.
Supported parameters are: ***Length, *Beam eccentricity ey, *Beam eccentricity ez, *Rib eccentricity ey, *Rib eccentricity ez,*** (line elements), ***Area**, ***Thickness**, ***Eccentricity*** (domains)**.**

Selection by result value

If the active tab is *Static* or any design tab to the right, elements can be selected by a result component value. The appropriate elements matching the condition in every cross-section or finite element point will be selected.

An example: *Select elements where eZ < 0.*
Type ***ez**[Enter]. The command line changes to ***eZ [mm]**. Continue typing to get ***eZ [mm] < 0**[Enter]. All nodes, midside nodes, surface centers, beams and ribs where the global *Z* component of the displacement is negative will be selected.
If the active view already displays the *eZ* component the command is even shorter. Simply type ***<0**[Enter]. If no result component is entered, the current result component is used to check the values.
Another example: *Select failed design members on the Steel design tab.*
Type ***u**[Enter]. The command line changes to ***Utilization []**. Continue typing to get ***Utilization[] >1**[Enter]. All steel design members where utilization exceeds 1 will be selected.



Selection by multiple criteria

To select elements based on multiple criteria click the right mouse button over the magnifier icon and choose *Selection based on multiple criteria* from the popup menu. Entering multiple criteria is more complex so in simple cases *Selection based on one criterion* is recommended.
An example: *Select all columns where normal force is below –5 kN.*
Choose *Selection based on multiple criteria* from the popup menu. Type ***co**[Enter]. In the other mode this command would select all columns. But now the command line changes to ***Columns |** waiting for further conditions. Pressing another [Enter] would select all columns.
Continue typing ***Columns | nx**< to get ***Columns | Nx[kN]** <,
then continue ***Columns | Nx[kN] < -5**[Enter].
Now a second criterion is added: ***Columns | Nx < -5.00 kN |***
This is the last one. Pressing [Enter] selects all columns where the normal force is below –5 kN.

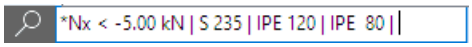
Interpretation of multiple criteria



Criteria relating to different properties (like those of normal force, material and cross-section) are always connected with an implied AND operator, i.e., all criteria must be met. Criteria for result component values are also connected with an implied AND operator. Selecting elements with two result criteria in an OR relation can be completed only in two consecutive selections:



Multiple criteria for materials or cross-sections are connected with an implied OR operator *within their category* (i.e., elements with any of the entered cross-sections will be selected) but is in AND relation with other types of conditions. E. g.

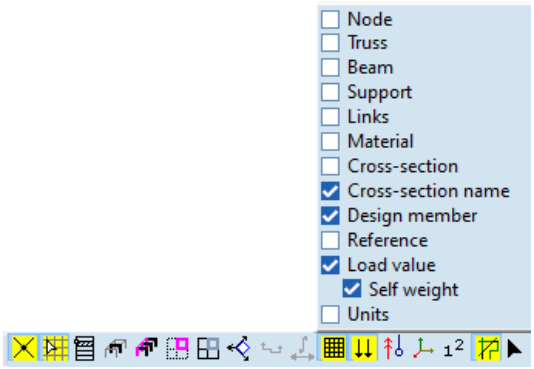


Selects all IPE 80 or IPE 120 columns matching the other criteria.

2.18 Speed buttons

Speed buttons allow you to change display settings or activating stories, parts, workplanes, etc. without entering the *Display option / Symbols*, *Model options* or any other dialog.

Speed buttons are located in the bottom right corner of the graphics area.




Auto intersection


Mouse snap to grid


Stories


Parts in tree view


Display parts containing the selected elements


Display selected element only


Hide selected elements


Workplanes


Section lines & planes & segments


Virtual beams/strips


Display mesh


Display load symbols


Display symbols


Display local systems


Numbering

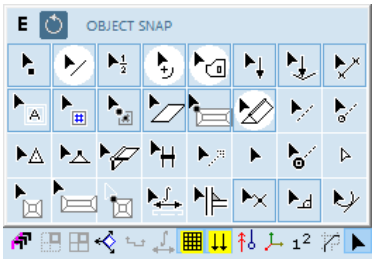

Background layers (visibility, detection, locking)


See **2.18.1 Object snap**

2.18.1 Object snap



Place the mouse over the icon to open the object snap switches. Object snaps activated in the current operation has a white circle in the background. For the list of cursor shapes see **4.7.1 Cursor identification.**
Switches allow deactivation / activation of individual snap functions. Deactivated objects will be ignored by the mouse cursor, activated objects will always be detected regardless the operation you perform.
The first click on a cursor shape deactivates the snap (red background), the second click activates it (green background), the third one restores the default state.





Deactivated snap function

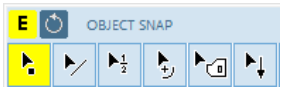


Activated snap function



Default snap depending on the current operation

Click on E to set an exclusive snap. Only the snap you click on will be detected. The background of the chosen snap function turns to yellow.



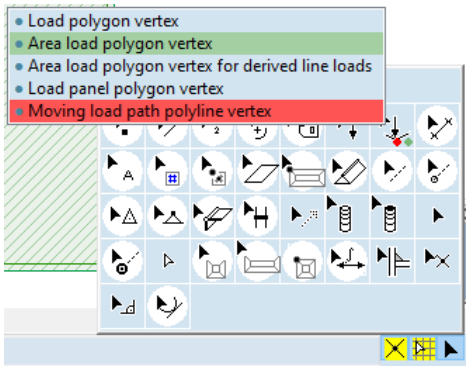


This button resets object snap to the default status (depends on the current operation).

The blue frame around the cursor shape means that the cursor is associated with multiple objects. The cursor shape for a line is can show the detection of a structural line, a line on the architectural model or background layer.

Fine-tune object snap

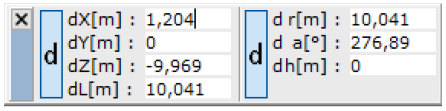
Right-click on an icon to fine-tune the object snap functions. E. g. if you right-click on the *Load polygon vertex* icon, you can activate or deactivate snap to different kind of load polygon vertices.
If only some subtypes are activated / deactivated, the background of the icon remains blue but small green / red dots appear at the bottom of the icon.



2.19 Information windows

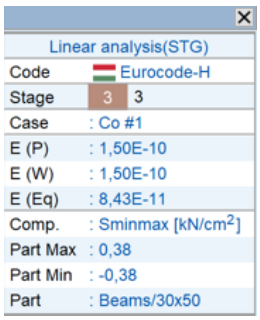
The information windows are situated in the graphics area. You can move these windows on the screen by clicking title bar, holding down the left mouse button, and dragging it to a new location on the screen.

2.19.1 Coordinate window



See 4.4 Coordinate window

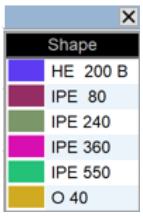
2.19.2 Info window



Shows information about the display of the results such as: active part(s), current perspective setting, type of analysis, current design code, current stage, current load case or load combination, solution errors, current result component.
For the explanation of E(U), E(P), E(W), E(EQ) parameters see 5. Analysis and 5.1 Static
In case of geometric nonlinear calculation, a warning message appears if a stability problem occurred in the current increment. See 5.1 Static analysis

In case of nonlinear calculation taking into account the reinforcement short forms of the reinforcement warning messages also appear. See **6.5.5 Nonlinear analysis of reinforced concrete beam and column elements** and **6.5.6 Nonlinear analysis of RC surfaces**. The warning message appears if the problem occurred in the current or in any previous increment.
If the *Construction stages* tab is active or the results of a construction stage is displayed the current stage is also listed here. See **4.13 Construction stages – STG module.**
If more than one part is activated a list of active parts is displayed provided that the number of parts does not exceed a limit. This limit can be set by right clicking the info window and clicking the *Settings* menu item.

2.19.3 Color coding



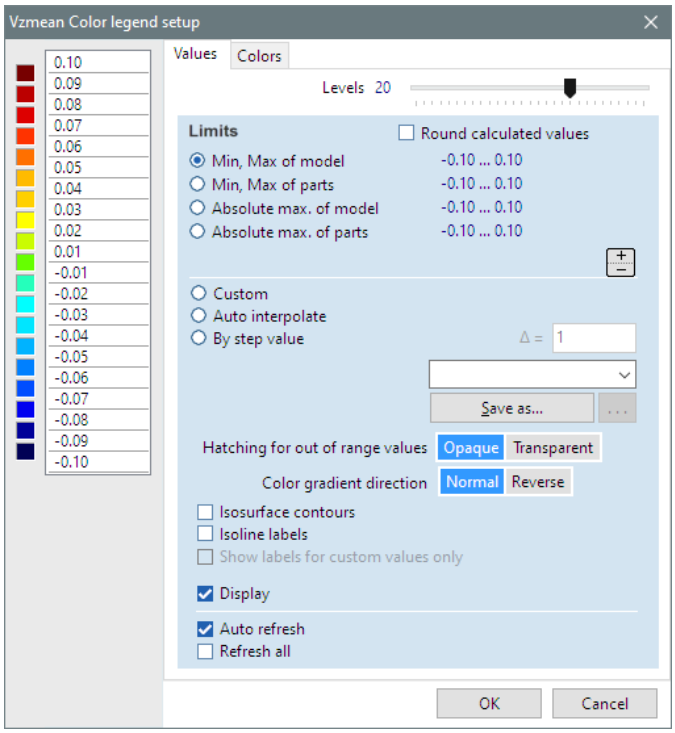
This info window appears after activating a color coding (see **2.16.5 Color coding**) unless it is turned off in the Window menu (see **3.6.2 Information Windows**).
The type of the coding is displayed within the black header line.

2.19.4 Color legend window

Color legend

Displays the color legend corresponding to the result component being displayed. You can resize the window and change the number of levels simply by dragging the handle beside the level number edit box or entering a new value. Colors will be updated immediately.

Color legend setup



You can set the color legend details in the color legend setup dialog box. To open this dialog box simply click the color legend window.
The first tab *(Values)* is to set color ranges and a few result display and labeling options,. The second tab *(Colors)* is to assign different color gradients to different result components.

Values

Limits Setting criteria for the interval limits:

Min/max of model
Sets the lower and upper limit values to the minimum and maximum values of the entire model. Intermediate values are interpolated.

Min/max of parts
Sets the lower and upper limit values to the minimum and maximum values of the active parts. Intermediate values are interpolated.

Abs. max of model
Sets the lower and upper limit values to the maximum absolute value of the entire model with the respective negative and positive signs.
The intermediate values are interpolated.

Abs. max of parts
Sets the lower and upper limit values to the maximum absolute value of the active parts with the respective positive and negative signs.
The intermediate values are interpolated.

Round calculated values
If the interval limits are interpolated between the minimum and maximum values (no custom values or step value has been introduced) the interpolated values can be rounded.



Separate positive / negative ranges
Pressing the button creates two ranges: one for the positive and one for the negative values. This bicolor mode may be useful when displaying reinforcement difference.

Custom
Click an item of the list on the left to edit its value. If you are in editing mode you can navigate through the list by ↑ and ↓ keys and edit the current item. When you click OK the series of interval values must be monotonically decreasing from top to bottom.

Auto Interpolate
If *Auto Interpolate* is checked the series will be recalculated each time you enter a new value. If you enter a new top or bottom value the recalculated series will be linear between top and bottom values. If you enter a new value at a middle interval the recalculated series will be bilinear, i.e., linear between the top and the new value and between the new and the bottom value but steps may differ.

By step value
Color values are determined by the given step Δ. When entering a new level value the other levels will be recalculated using the step. Switching from other crieria the array starts from the lowest value and using the latest step value.

You can save the settings of the scale using the *Save As* button. To review saved settings click the **...** button.

Hatching for out-of-range values
Hatching for out-of-range values can be set to *Opaque* or *Transparent*.

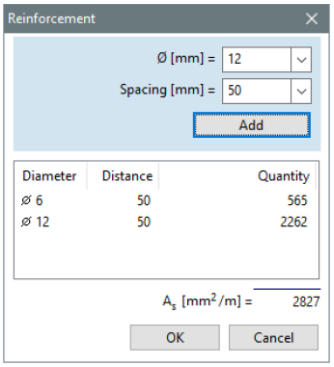
Color gradient direction
Allows swapping the direction of the color gradient.

Isosurface contours
If checked, a contour line appears at the outline of isosurfaces.

Isoline labels
If checked, isolines are labelled automatically.

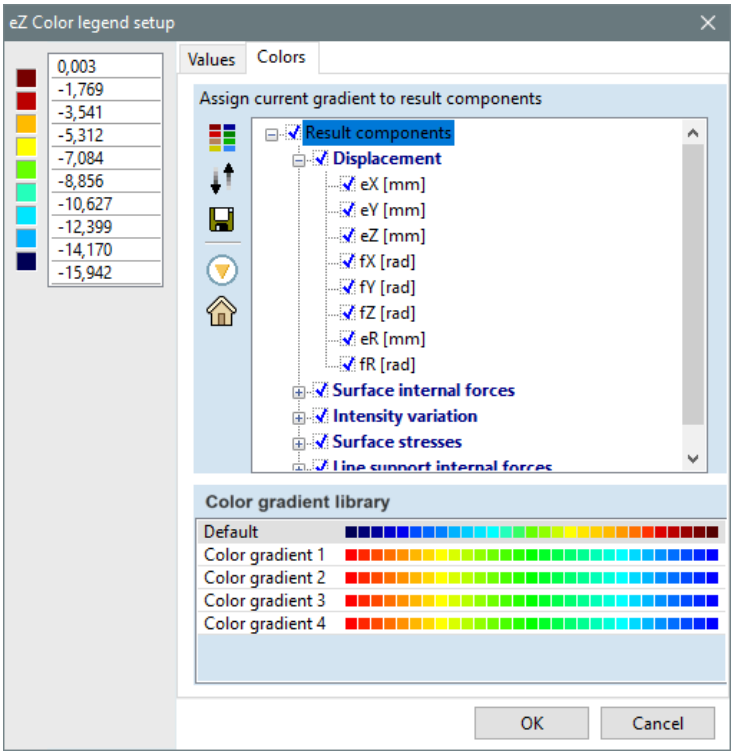
Standard interval limit settings are also available directly from the color legend window popup menu. To activate popup menu click right mouse button on the window.

Calculate
When displaying reinforcement values click *Custom* and *Calculate* to get the amount of reinforcement from rebar diameters and distances for the selected list item.



☞ **When displaying actual reinforcement schemes AXISVM does not assign color to numerical values but to different rebar configurations. It can be set to display all schemes or just those within the active (visible) parts.**

Colors



Colors can be modified by assigning a color gradient to the values. A gradient can be assigned to result components, e.g., displacements can be displayed in different colors than internal forces.
Setting a gradient from light to dark can help to resolve the ambiguity of a grayscale output.



New color gradient
A new color gradient can be defined by dragging the gradient end points to the desired position.



Revert color gradient
Swaps the start and end color of the gradient.



Save color gradient to the color gradient library
Gradients can be saved to a library for future use.



Save current color gradient assignments as default
Current result component assignments become the default setting for new models.



Restore default color gradient assignments
Sets the default color assignments for result components.

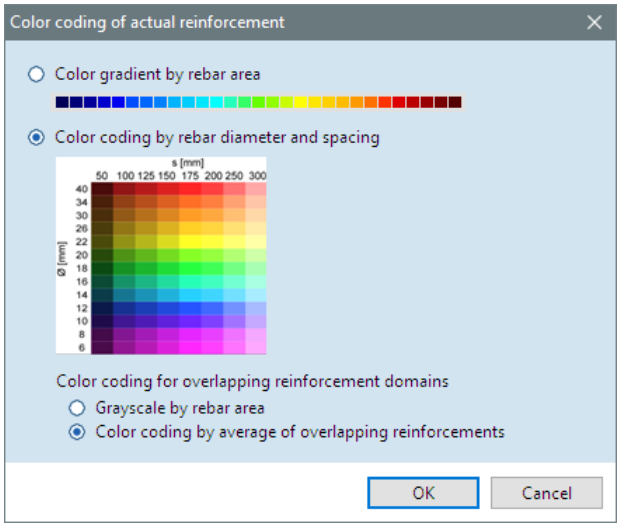
Clicking on an item of the *Color gradient library* applies the gradient.

Color coding for actual reinforcement schemes

If an actual reinforcement component *(xb, yb, xt, yb)* is displayed the color legend lists all reinforcement schemes defined in the model in the following format:

Ø16/200 (1005) [3.4],
i.e., *Ødiameter/spacing (rebar area) [concrete cover]*

Click on the color legend to display options of color coding for actual reinforcement schemes.



Color gradient by rebar area

A color is assigned to the scheme according to its specific rebar area.

Color coding by rebar diameter and spacing

A color is assigned to the scheme according to the rebar diameter and spacing. Hue depends on the diameter and brightness depends on the spacing.
If reinforcement domains overlap the color of the overlapping area can be determined by one of the following methods:

Grayscale by rebar area

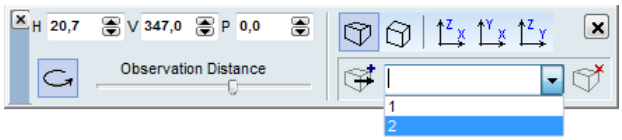
A grey color is assigned to the area according to the total specific rebar area.

Color coding by average of overlapping reinforcements

An average spacing and diameter is calculated as

$$s = \frac{1}{\sum_i \left(\frac{1}{s_i}\right)}; \quad d = s \cdot \sum_i \frac{d_i}{s_i}$$

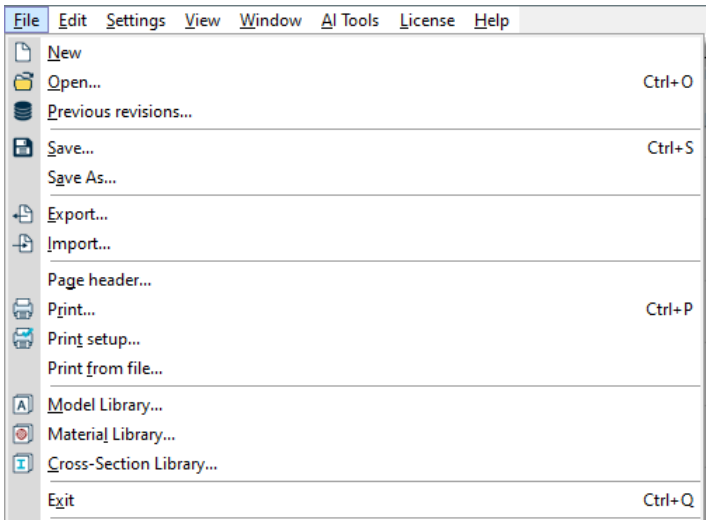
2.19.5 Perspective window tool



See 2.16.3 Views

3. The Main Menu

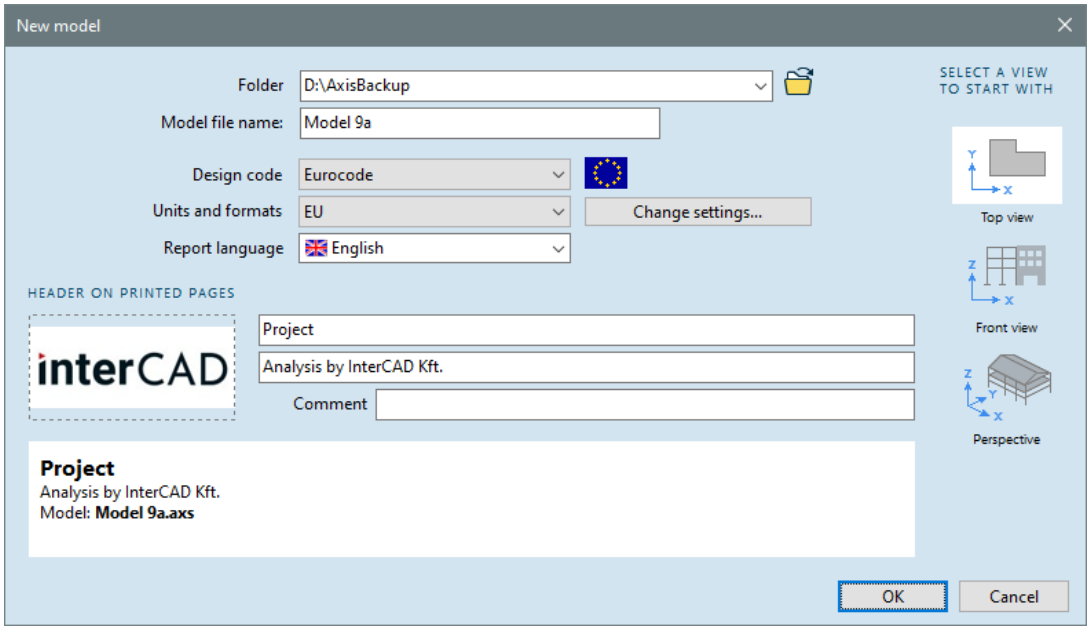
3.1 File



The menu commands are described below.

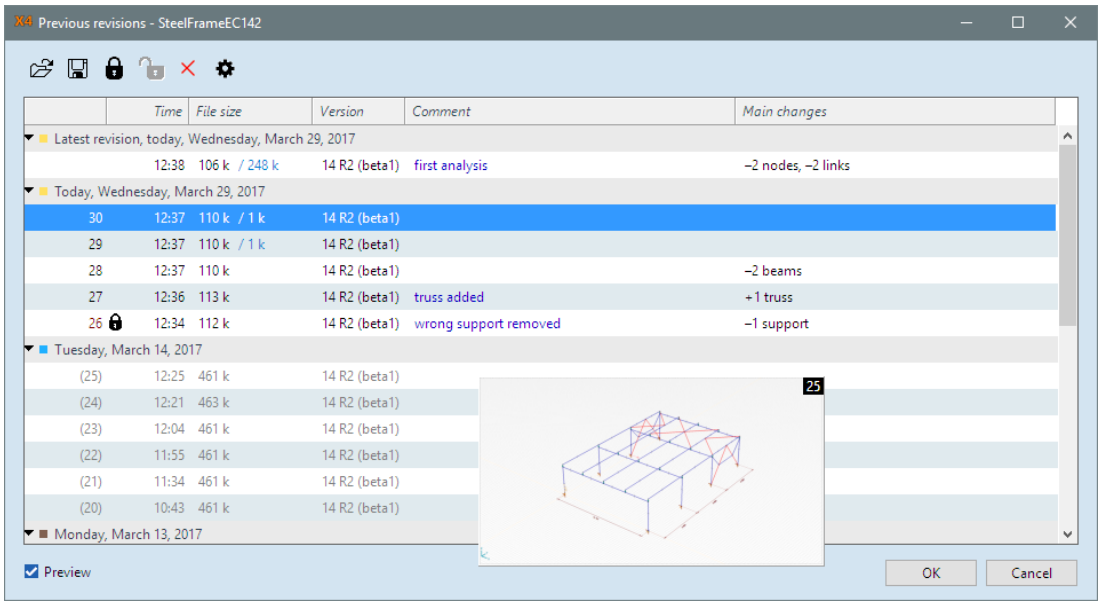
3.1.1 New model



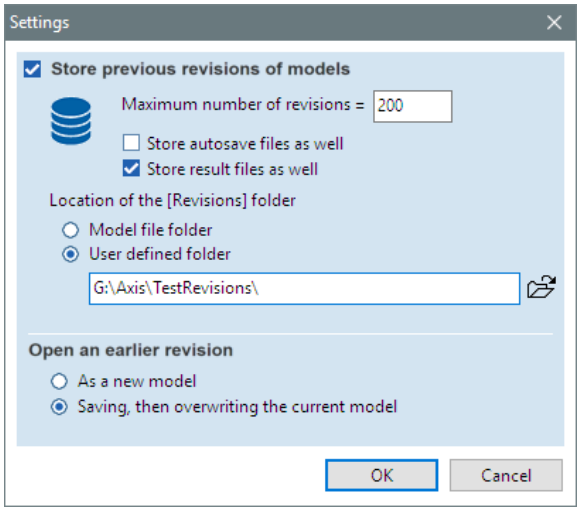


Creates a new model. Use this command to start a new modeling session. If you have not saved the current model, a prompt appears asking if you want to save it first. Refer to the *Save* and *Save as* commands for more information on how to save your current model.
You must specify a name for the new model. You can select the appropriate Standard and system of units. You can enter specific information in the Heading section, that will appear on each printed page. A new model uses the default program settings.
If a page header logo was selected for printing drawings, tables and reports, the header also appears here. Click on the logo to change it or try *Settings / Preferences / Report / Company logo / Settings*.
See 3.3.11.15 Report

3.1.2 Previous revisions...



AXISVM can preserve previous revisions of the model. This feature is activated by default. The *Previous revisions* dialog lists previous revisions in reverse chronological order (the topmost item is the latest one). The File size column shows the size of the model file (and the result file if it exists).
Revisions can be opened and locked. Comments can be entered and edited by clicking on a cell in the *Comment* column.



The maximum number of stored revisions can be in the *Settings* dialog. If the number of revisions has reached the maximum the oldest revision is deleted upon saving a new revision. It is also possible to store automatically saved revisions and result files (For the autosave settings see *Settings / Preferences / Data integrity*, **3.3.11 Preferences**).
Autosave files appear in grey and their revision number appears in parentheses.
The *Revisions* folder containing the subsequent revisions of the model is created under the model file folder as default, but the location of this folder can be changed.

There are two ways to open an earlier revision: opens it *As a new model* or *Saving, then overwriting the current model*. This latter option saves the current state of the model as a revision then the earlier revision overwrites the model.

This dialog is also available in *Settings / Preferences / Data integrity*,

Open. Saves the selected revision as a new model and opens it. It is possible to preserve earlier revisions.

Save a previous revision. Saves the selected revision under a new name. It is possible to preserve earlier revisions.

Lock revisions. Locks the selected revisions. These revisions are not deleted even if the number of saved revisions has reached the maximum defined in *Settings*.

Unlock revisions. Unlocks the selected revisions. If a revision is unlocked, it will be deleted if necessary.

Delete. Deletes the selected revisions (locked revisions will be excluded).

☞ **Previous revisions are stored in a *Revisions**model name* subfolder created below the folder of the model file. Copying or moving the file does not copy or move these subfolders. In order to copy or move the revisions the above subfolder must be copied or moved to the new location.**

3.1.3 Open



[Ctrl]+ [O]

Loads an existing model into AXISVM.If you have not saved the current model, a prompt appears asking if you want to save it first. Refer to the *Save* and *Save as* commands for more information on how to save your current model. Selecting this command will bring up the Open dialog box.
If the folder name appearing in the dialog box is what you want, simply enter the file name in the edit box or select it from the list box. If the directory is not what you want, select the drive and directory names along with the file name.

☞ **AXISVM saves your model data as modelname.axs (input data), and the results as modelname.axe. Both file includes the same identifier unique for each save which makes it possible to check if axs and axe files belong to the same version of the model. Results of construction stages are stored as modelname.stg*X* where *X* = 1,2, ..., *N* (N is the total number of stages).**

The open dialog looks different on Windows XP and Vista / Windows 7 / Windows 8 operating systems.

3.1.4 Save



[Ctrl]+ [S]

Saves the model under the name displayed at the top of the AXISVM screen. If you have not saved the model yet, the *Save as* dialog box automatically appears prompting you to enter a name. Use the *Save as* command if you are changing an existing model, but want to keep the original version.

If you select *Create backup copy* in the *Settings / Preferences / Data integrity / Auto save* a backup file of your previous model will be created.
If *Settings / Preferences / Data integrity / Compress AXS* is checked the model file will be saved in a compressed format. The average size of the compressed file is about 10% of the original. Compression is more efficient on large files.

3.1.5 Save as

Names and saves the model. Use this menu command to name and save a model if you have not saved the model yet, or if you are changing an existing model, but want to keep the original version.
Selecting this menu command will bring up the Save As dialog box.

Converting models

Models created with previous AXISVM versions (if applicable) will be converted into the current version file format when you open them for the first time.

☞ **The File / Save As / File Format command lets you save the model in earlier formats.**

The save dialog looks different on Windows XP and Vista / Windows 7 / Windows 8 operating systems.

3.1.6 Export



Export functions create different types of files to transfer data to other engineering applications.

3.1.6.1 Export to DXF

DXF file

Saves the geometry of the model to a `DXF` file format for use in other CAD programs. The geometry is saved with actual dimensions, in a `Modelname.`**`DXF`** file.
Selecting this menu command will bring up the Export DXF dialog box, which lets you specify the units of measurement in the exported file. The results set in the current view (in the diagram, contour, surface 2D, etc.. view mode) can also be exported.

The following formats are available for DXF output:

- *AutoCAD 2004 DXF file*
- *AutoCAD 2000 DXF file*
- *AutoCAD R12 DXF file*
- *AutoCAD reinforcement design file*
- *AutoCAD 2004 reinforcement design file*

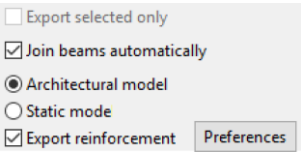
Reinforcement design files are isoline diagrams of required surface reinforcement exported with labels and a color legend. Each direction is on a separate layer. To export this type of DXF file, go the *R. C. Design* tab and select a result component related to required reinforcement.

It is recommended to use the AutoCAD 2004 format to avoid data loss. Earlier DXF formats support 256 palette colors and ASCII characters only.
For further DXF file export options and their description, see section **3.1.6.7 Export to other applications**.

3.1.6.2 Export to IFC

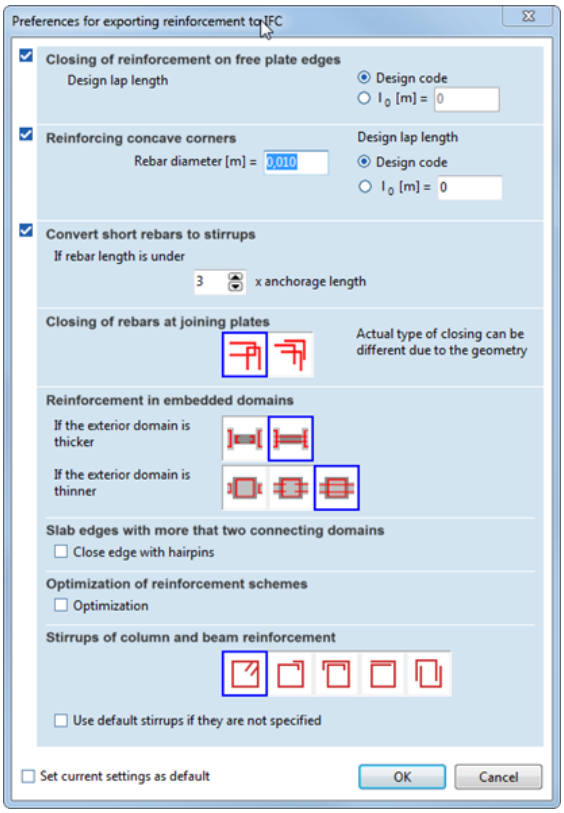
IFC 2x, 2x2, 2x3 file

Export of this file type requires an **IFC module.**
Exports an IFC file describing the model with achitectural objects (walls, slabs, columns, beams). IFC files can be imported in ArchiCAD, AutoDesk ADT, Revit, Nemetscheck Allplan, Tekla-Xsteel and other architectural programs.



Checking *Export reinforcement* on the save dialog the and clicking on *Preferences* displays a dialog to set how the actual reinforcement will be exported.

Since the release of 13R1, users have the possibility to export reinforcement to .ifc files.
The built-in algorithm of AXISVM helps to export not only the field reinforcement based on user given intensities, but shaped bars too, like hairpins, hooks etc.



- *Closing reinforcement on free plate edges*: free plate edges are closed by hairpins, whose lengths are calculated based on the lap length of bars, which is in turn calculated either by the standard set in the program, or the user given value of l0. This option can be turned on or off.
- *Reinforcing concave corners*: concave corners can be strenghtened by simple bars, whose lengths are calculated based on the lap length of bars, which is in turn calculated either by the standard set in the program, or the user given value of l0. This option can be turned on or off.
- *Convert short rebars to stirrups*: to avoid useless duplicating of rebars, the algorithm can be given a multiplicator value. Those simple bars, whose lengths are greater than the minimal lap length multiplied by this value, will be converted to a closed stirrup. This option can be turned on or off.
- *Closing of rebars at joining plates*: along the edges of plates that do not share the same plane, the algorithm can connect the reinforcements of the individual plates either by hairpins or by hooks. The actual geometry of these shaped bars depend on the width

and angle of the joining plates, and sometimes, the algorithm revises the user's choice. The sizes of the shaped bars are calculated with respect to the minimal lap length.

- *Reinforcement in embedded domains*: the method of closing, embedding rebars of the embedded domains can be chosen by the user. They can either be handled separately, or their rebars can be embedded into each other, or if the inner domain is small enough, the outer domain's bars may as well be driven through its body.
- *Slab edges with more than two connecting domains*: in such cases an automatic solution cannot be found, so the user can choose whether to completely skip those edges or tho close them independetly from each other.
- *Optimization of rebar positions* to distribute reinforcement evenly if reinforcement domains overlap
- calculated raw bar lengths are always rounded up to the nearest value of this array: 0.4, 0.5, 0.6, 0.8, 1.0, 1.2, 1.5, 2.0, 2.4, 3.0, 4.0, 6.0, 12.0 m
- *Stirrups of column and beam reinforcement*: exporting strirrups into the file will happen based on the strirrup properties adjusted by the user. If no such setting is available, user can select whether to use the standard's rules for calculating stirrup positions, or skip this completely. There are five different stirrup shapes available at the moment, form which users can select by clicking on the icon of the desired shape.

Export model data

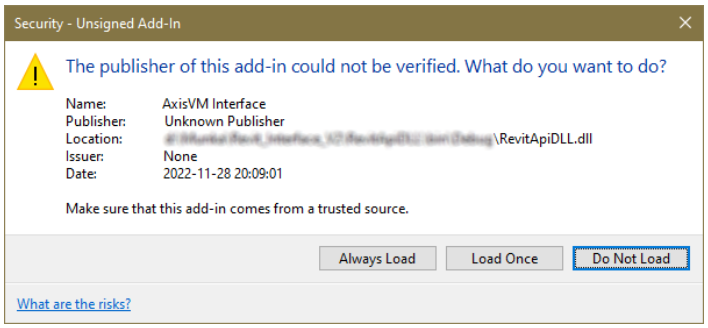
If the Architectural model option is selected only architectural objects will be exported. If Static model is selected, finite element meshes, loads, load cases, load groups and load combinations will be included in the IFC file. Dynamic, influence line or moving loads will be excluded.

3.1.6.3 Export to Revit

Export to Revit (.are)*

Export of this file type requires a **REV module**, and Autodesk Revit 2025 or newer.
AXISVM model (the entire model, active parts or selected elements) can be saved to a file with ARE extension, readable from Revit after installing an add-on.
Exported elements are trusses, beams, ribs, domains and supports. The import add-on for Revit allows filtering of elements, defining the default geometry for nodes and line supports and mapping of materials defined in AXISVM to Revit materials.
If Revit has been installed on the computer, the Revit add-on is installed automatically even if the REV module is not included in the configuration.
The **.addin* file needed for the collaboration between the two programs is placed in a folder called *c: \ Users \ <Username> \ AppData \ Roaming \ Autodesk \ Revit \ Addins \ yyyy,* where yyyy is 2025 or higher, depending on the Revit version.

☞ **The popup window shown below will appear at the initialization phase of every add-in or third party executable.**



☞ **In this case, please press "Load Once" if you wish to load the add-in, and use its abilities, or "Always Load" if you wish to suppress this dialog in the future.**

Exported data

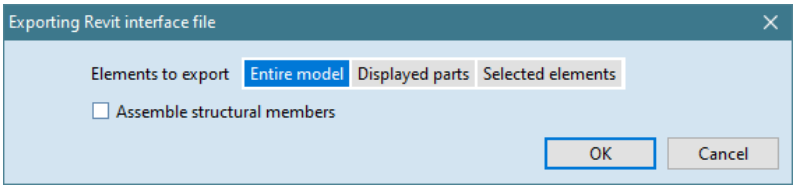
For the time being, only geomatrical data are exported to Revit, which are:

- linear elements with their materials and sections
- regions and their materials
- supports

Usage

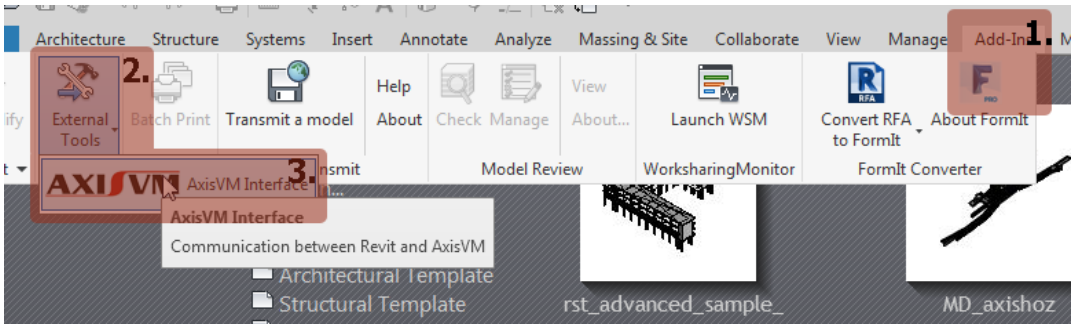
Export process must always be started from AXISVM, the way mentioned below:
File menu -> Export -> Export to revit (.are)*

By pressing the *Settings* button of the open dialog you can choose between several options:



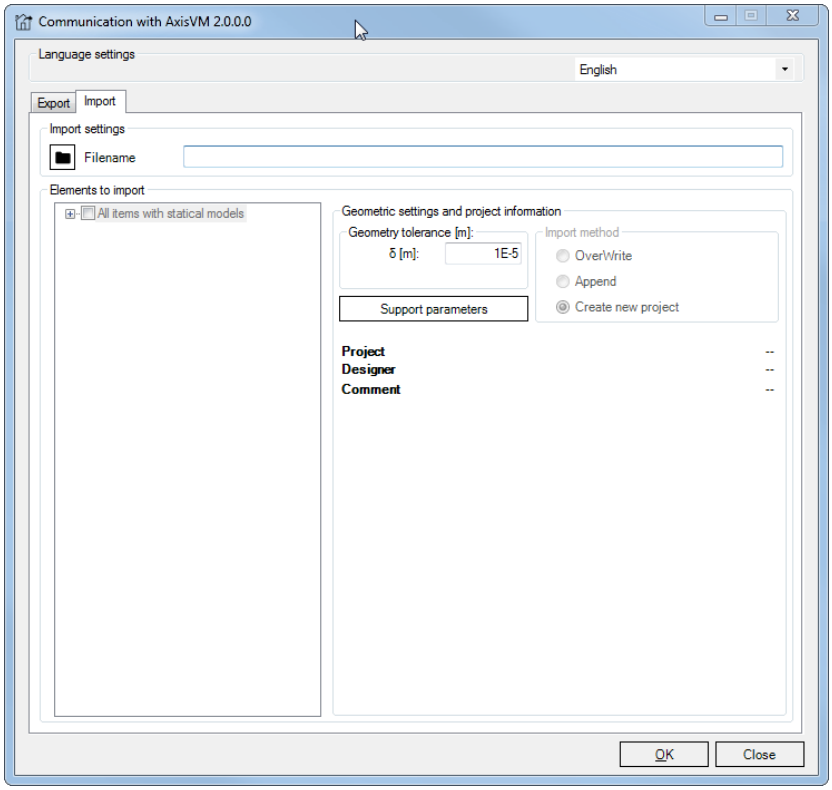
Entire model – all exportable data will be written into the file
Displayed parts – all parts visible on screen will be exported
Selected elements – only elements selected before will be exported

Having saved the .are extension file you can import it into Revit like on the attached picture:

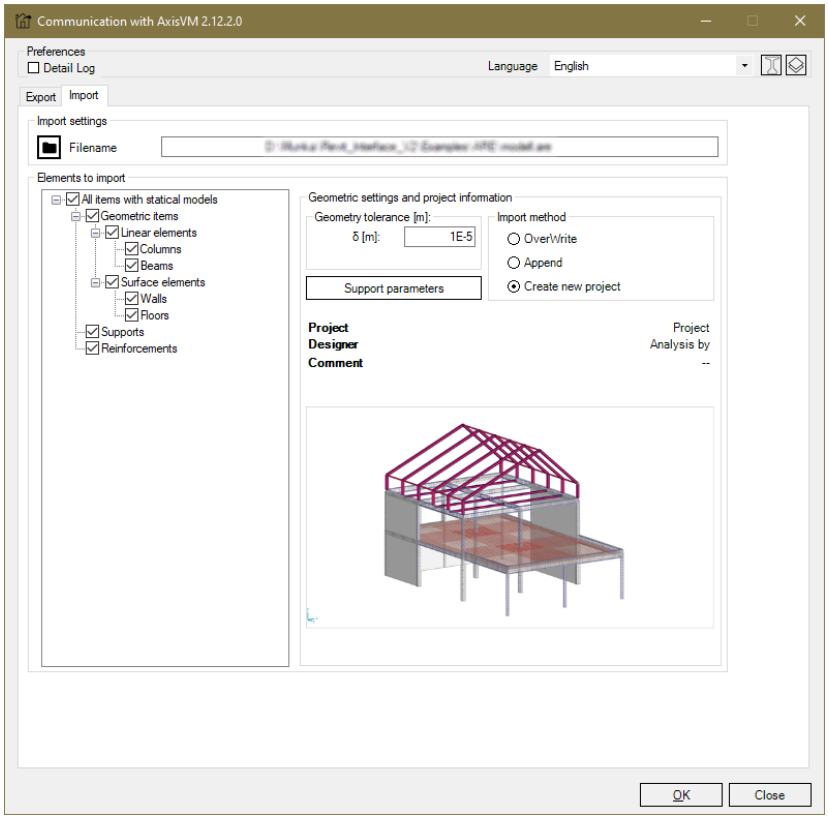


Add-Ins / External Tools / AXISVM Interface

After this, the form seen below will pop up:



In as much as there is an opened project in Revit, then both *Export* and *Import* tab sheets will be enabled, otherwise only *Import* tab sheet can be used.
To import.are extension files you have to press the button located left to the *Filename* label.
After this the tree on the left is refreshed by the content of the file, and a preview of the model to be imported is visible on the bottom right corner.



Elements that are actually imported can be filtered by checking the nodes of the left tree.

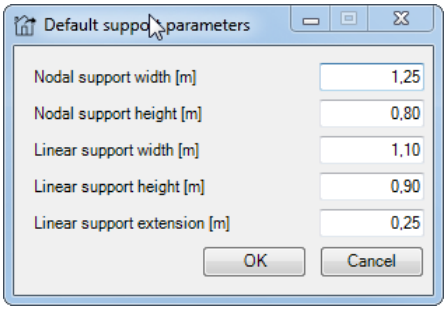
Import method

If you have an open project at the moment of import, you can select from several options:
Overwrite – all elements will be deleted from the project (except for the ones pinned before), and new elements will be imported into this purged project
Append – new elements are add the project preserving older items
Create new project – opened projects remain unchanged, and a brand new one will be created, into which all elements will be imported
On the other hand, if you have no opened project, then a new one will be created.
It is important to note, that if the project is read-only, then only the third option is enabled.

☞ **In case of a problem, a detailed log file can be created by selecting the checkbox at the top of the window. This will generate a large log file with the name of the import file but with the extension .log, that can be sent to the distributor with the support request.**

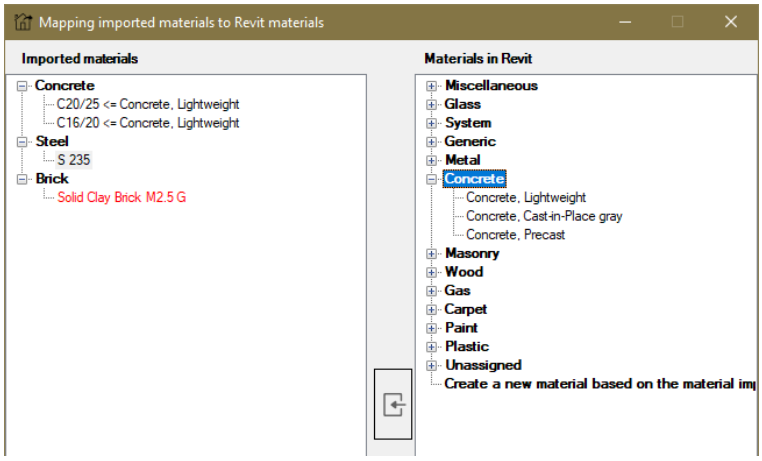
Support parameters

Supports are imported based on the following considerations:
If there has been geometry assigned to a certain support in AXISVM, then this property will be used. Otherwise geometry will be created based on default parameters, which can be adjusted by clicking the button *Support parameters*.



Mapping materials

As AXISVM material records cannot be directly mapped to Revit materials, user interaction is needed to help on this mapping. It can be done on the form seen below:



You can only continue the import process only if all mappings are done on this form. In order to map AXISVM material to Revit material, you have to select a material as material type on the left tree, and another one from the rigth tree. Then if you press the button between the two trees, the selected AXISVM material will be mapped onto the selected Revit material. Selection on the left tree will then automatically jump on the next unprocessed node. Unprocessed nodes are drawn red, everything else is drawn black. You do not have to do this mapping one by one. Instead, if you select a node with several children, then all the node's children will be assigned the selected Revit material.
If you select the last node of the right tree, a new Revit material will be inserted into the database with properties inherited from the selected AXISVM material.
Having accomplished all mappings, OK button will be enabled, and you can continue the import process.

3.1.6.4 Export to Tekla Structures

Tekla Structures file

Two different file formats are available:

Tekla (TS) Structures ASCII file (.asc)*
Saves the geometry of the model into a file format that is recognized by Tekla Structures. The file includes the coordinates of i and j-end nodes, the cross-sectional properties and the reference point of truss and beam elements.

Tekla (TS) DSTV file (.stp)*
Saves the data of the truss and beam elements (end points, material, cross section, reference) as a standard DSTV file. This file format is supported by several steel designer CAD software.

Tekla Structures 2021 (and the later versions) support two-way connection with external programs. The **TI module** of the AXISVM can establish such a connection so it is worth using this mutual data exchange instead of the above interface files (3.1.9.1 Two-way connection for Tekla Structures versions 2021 and later)

3.1.6.5 Export to Nemetschek Allplan

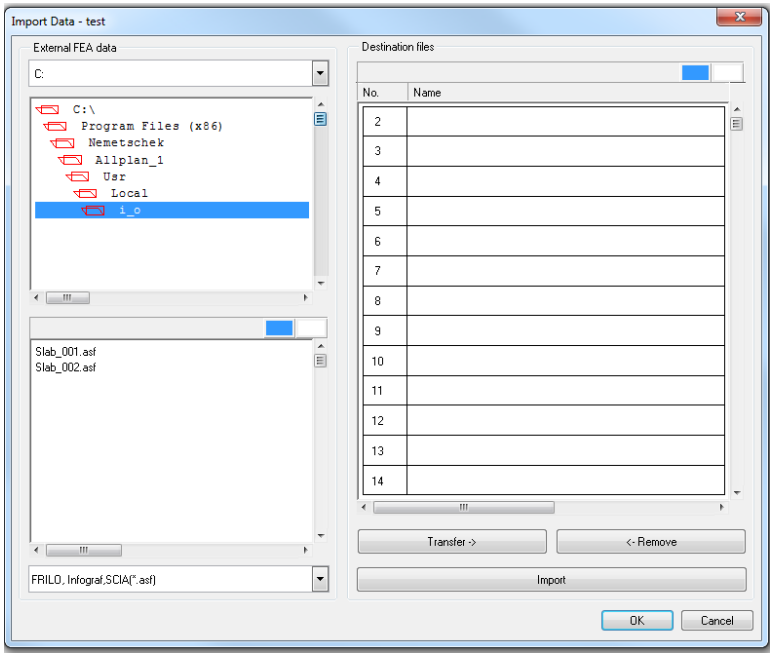
ASF file

Export of this file type requires an **ALP module.**
Exports reinforcement of all active domains into separate files (filename_001.asf...) which can be later imported to Nemetschek Allplan in batch mode (at once). Due to import limitations of Allplan, domains are transformed to XY or XZ plane whichever plane is closer to the exported domain.
Only reinforcement of domains are exported, results of individually defined surface elements will be ignored. Three reinforcement values are exported for triangular mesh elements and four values for quadrangles.
Select *Move To Origin* to move exported domains to the origin (outline polygon node closest to the origin will be justified to the origin).
Select what type of reinforcement to export: required (calculated), actual (applied) or the maximum of both.

☞ **Export to ASF file is available only on the R.C. Design tab**

Importing ASF file into AllPlan

- To import an asf file go to *Create/Interfaces/Import FEA files* menu item in Allplan menu.
- Select the folder with asf files from the directory structure in the top-left window. You can see the asf files in the bottom-left window. Select one or more asf files from this list and select the place of these files by clicking in the row of the list on the right side *(Destination Files)*.
- Click on the *Transfer* -> button, and the selected files will appear in the list.
- Click on *Import* button, and the selected files will be imported into Allplan program.
- Close the window with OK



Displaying imported reinforcement in AllPlan

- Click on *Create/Engineering/Bar Reinforcement/FEA reinforcement Color Image* menu item in the Allplan menu
- Click on the desired *asf* file
- Choose layer (top/bottom) and type (compression/tension) of the imported reinforcement values
- Click on OK

3.1.6.6 Export to SAF file format

Export of this file type requires an **SAF module.**

This Excel-based file format supports exchange of structural model data with other applications supporting this protocol. The file consists of standardized spreadsheets with the geometry, structural and load properties of the model, thus can be viewed and/or edited with Excel. SAF (Structural Analysis Format) protocol is supported by ArchiCAD, AllPlan, SCIA and RFEM.
Entities exported to the resulting file are

- trusses, beams, columns, ribs, supports, regions and connection elements (hinges, joints)
- cross-sections
- support stiffnesses
- load, load combinations

Usage

Select *File / Export / SAF export (*.xlsx)*, select the location and filename for the export and press *Save*. The following settings can be accessed by clicking the *SAF options* button.

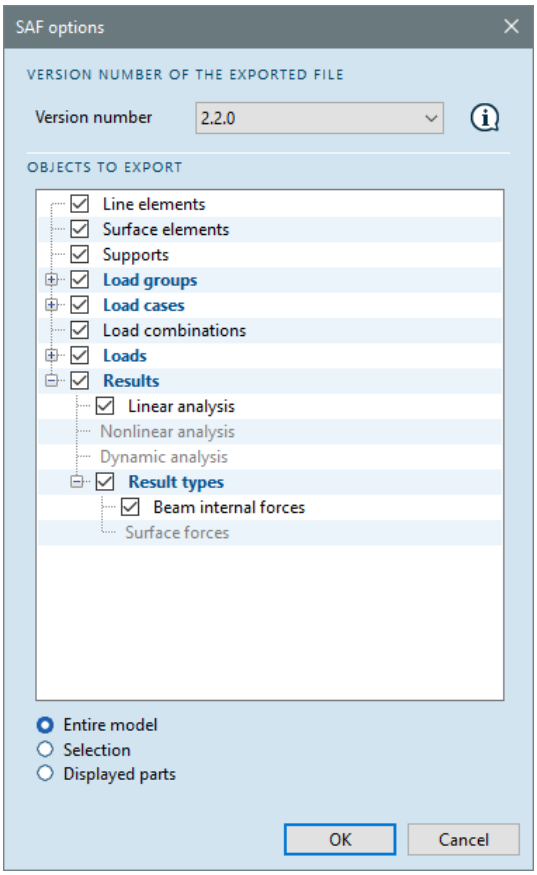
SAF options

Objects to be exported can be filtered.

First the exported SAF version must be chosen (1.0.5–2.2.0). More information can be obtained about the differences by clicking on the info icon. It is advised to use the latest possible version, but if the target application does not not support it, the latest appropriate version can be selected.

The tree of the available object types reads load groups and load cases from the model. In case of huge number of loads and load cases it may be necessary to uncheck them to keep the resulting file size reasonable.

The main filter at the bottom of the form allows export of
the entire model
only the selected parts
or the displayed parts only.



3.1.6.7 Export to other applications

Bocad file

Saves the geometry of the model into a file format that is recognized by the Bocad software. The file includes the coordinates of i and j-end nodes, the cross-sectional properties and the reference point of truss and beam elements.

StatikPlan file

For StatikPlan AXISVM exports a DXF file including the contour of the reinforced concrete plate, the calculated reinforcements as isolines and the result legends on different layers.

PianoCA file

Generates a *.pia interface file for PianoCA. It includes the data, supports, loads and the calculated results of the selected beam elements.

CADWork file

Creates a DXF file to use in CADWork reinforcement detailing software.
Only selected domains will be exported.
As CADWork works in 2D, selected domains must be in the same plane. Each domain in the DXF file is transformed to a local X-Y coordinate system, Z coordinate represents the calculated amount of reinforcement.

SDNF 2.0, 3.0 file

Saves the model in SDNF (Steel Detailing Neutral Format) file readable by steel detailing products (Advance Steel, SDS/2, Tekla Structures, PDMS).

Glaser -isb cad-

**.fem file*

Exports reinforcement domains to the Glaser -isb cad-program.

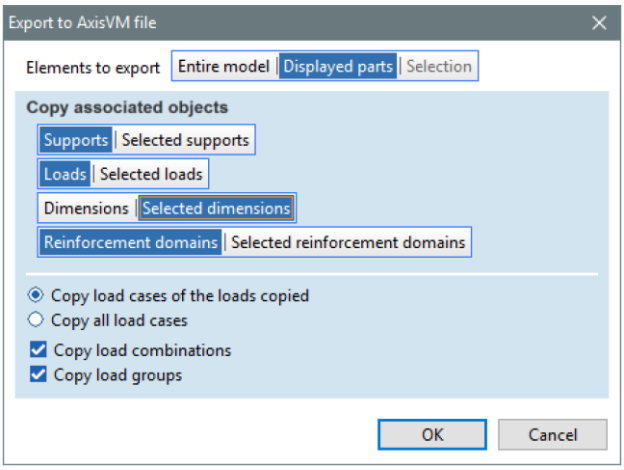
3.1.6.8 Export to AXISVM

AXISVM Viewer

Saves the model in AXISVM Viewer format (*.axv).
See **8 AXISVM Viewer and Viewer Expert**

AXS file

The following groups of elements can be exported: the entire structure, displayed parts or selected elements.
To select export options similar to those of the Copy options (see **3.2.8 Copy / paste options**) click the *Settings* button of the *Export* dialog.



Export selected only

Exports the selected elements only.

Coordinate units

The coordinate units of the exported file can be selected here.
The default unit is meter [m].

3.1.7 Import

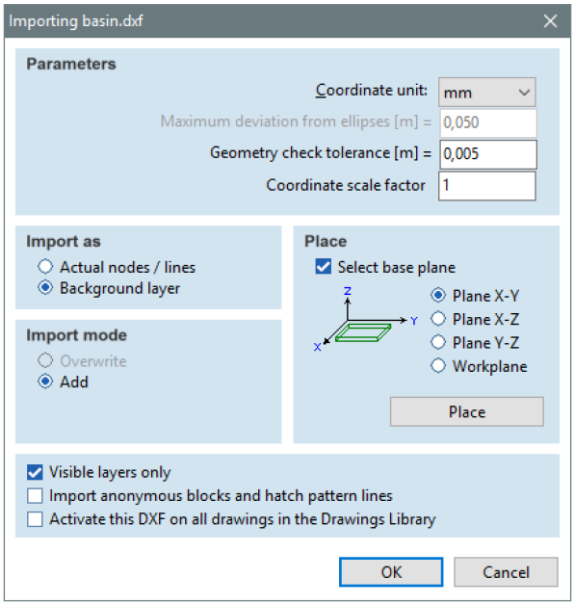


Import functions read and process different types of files to transfer data from other engineering applications and provide a link to BIM systems (Building Information Modeling).

3.1.7.1 Importing DXF files

*AutoCAD *.dxf*

Imports a geometry mesh from a DXF file exported in AutoCAD 12, 13, 14, 2000 or 2004 format into AXISVM. The layers of the imported file are loaded into the Layer Manager. See **3.3.3 Layer Manager**
If the file date of the imported file has changed, the Layer Manager will ask if you want to update the layers. Selecting this menu command will bring up the Import DXF dialog box.

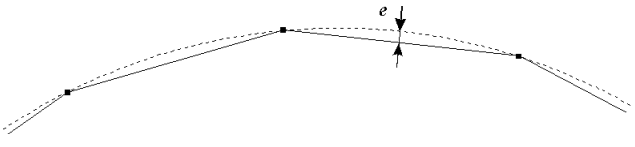


☞ **The ellipses will be converted to polygons only if you load them as active mesh otherwise they remain ellipses.**
In Results view, you can import only to the background layers.

Parameters

Coordinate unit
You need to specify the length unit used in the imported DXF file.

Maximum deviation from the arc [m]:
Importing a DXF file as an active mesh, ellipses will be converted to polygons based on this value.



Geometry check tolerance
When you import a DXF file as an active mesh, AXISVM checks for coinciding points (nodes) and lines in your model, and merges them.

You can specify the maximum distance to merge points. Points that are closer together than the specified distance are considered to be coinciding. The coordinates of the merged points (nodes) are averaged.
You must always set this to a small number relative to your model dimensions.

Import As — You must specify whether you wish to use the imported DXF file as an active mesh or as a background layer.

Active mesh (nodes & lines)
The imported geometry is considered as if it were created with AXISVM commands.
DXF layers can be used to create parts.

Background layer
The imported geometry is used as a background layer that is displayed but is inactive as a mesh. Import a DXF file as background layer when you want to create the model based on architectural plans or sections. You can use the entities in the background layer as a reference during editing your model.

Import Mode — You can choose between overwriting the former geometry or adding a new geometry to the former one

Place — Lets you specify the plane of the DXF layer (X-Y, X-Z, or Y-Z).
The Place button allows to graphically position the imported DXF drawing in your model space.

Visible layers only — With this option AXISVM imports only the layers set visible in the DXF file.

Import hatch pattern lines — Hatching is represented by individual lines in a DXF file so in most cases it is not recommended to import them. If you need the hatching check this option.

3.1.7.2 Importing IFC files

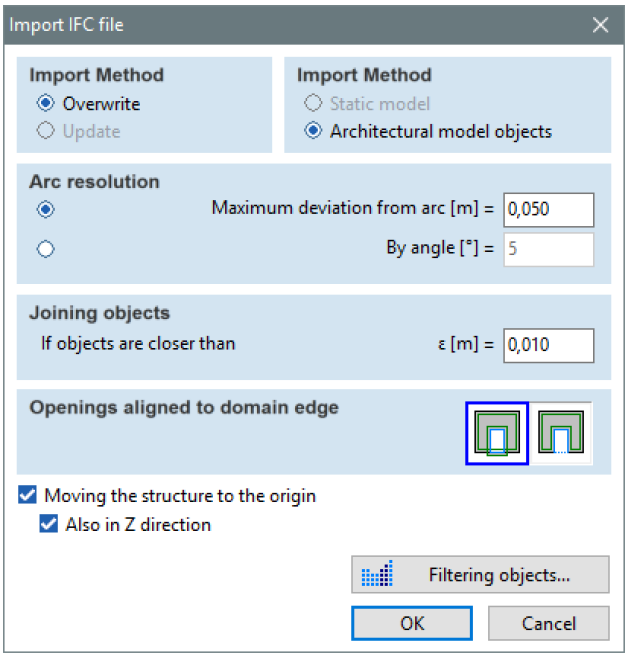
IFC 2x, 2x2, 2x3, IFC4 file

Importing requires an **IFC module**.

Imports objects from an architectural model saved as an IFC file. Imported objects can be displayed as a 3D background layer or can be converted to a native model by assigning materials, cross-sections etc. to them. Existing architectural models are always overwritten by the new one.

You can import object based architectural models from ArchiCAD, AutoDesk Architectural Desktop, Revit Structure, Revit Building, Nemetschek Allplan, Bocad, Tekla and Xsteel.

Importing IFC files can extract the static model (if available) or the architectural objects overwriting or updating the existing information within the AXISVM model.



This function is available only on *Geometry*, *Elements*, *Loads* and *Mesh* tabs.

Static model — From IFC version 2x3 it is possible to export details of the static model (nodes, topology, supports, loads, load combinations). The *Static model* option is available only if the file contains this information. If it describes architectural objects (columns, beams, walls, slabs, roofs) only the static model can be created automatically in AXISVM after importing the file.

Architectural model objects — This option can overwrite or update existing architectural model information in the AXISVM model. AXISVM can read columns, beams, walls, slabs, roofs.
See 4.9.26 Creating model framework from an architectural model

Opening aligned to domain edge

 *Import as opening*

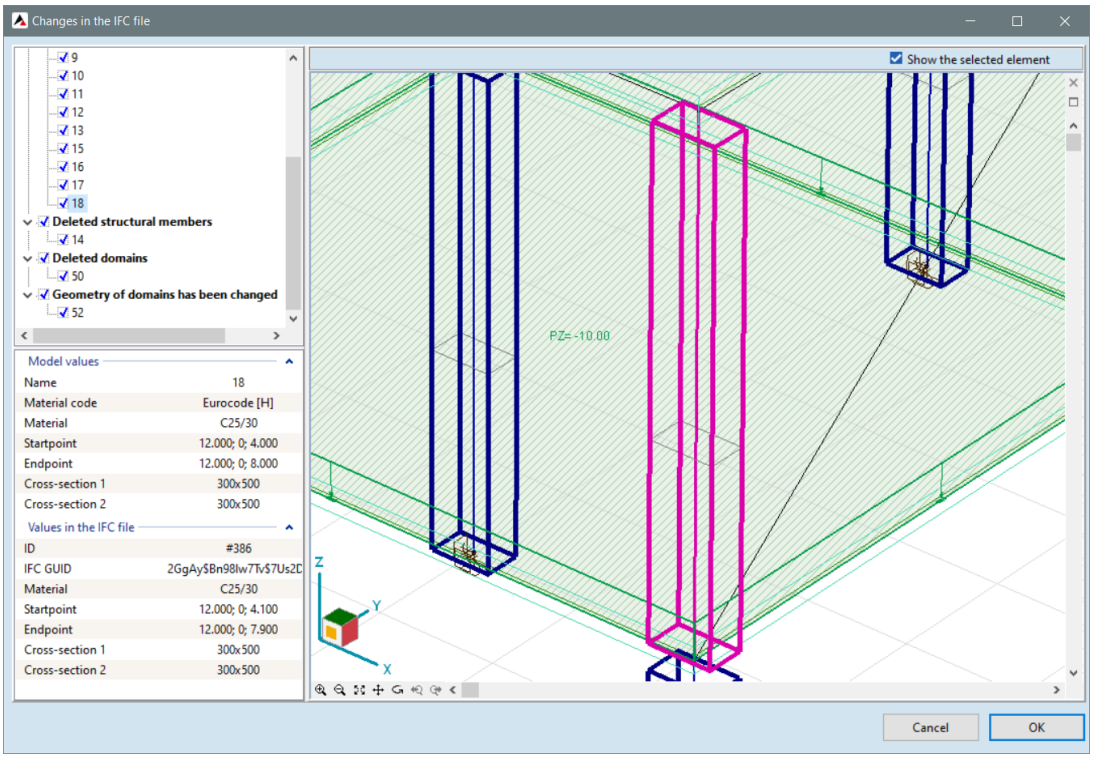
 *Adjust the domain outline*

Moving the structure to the origin

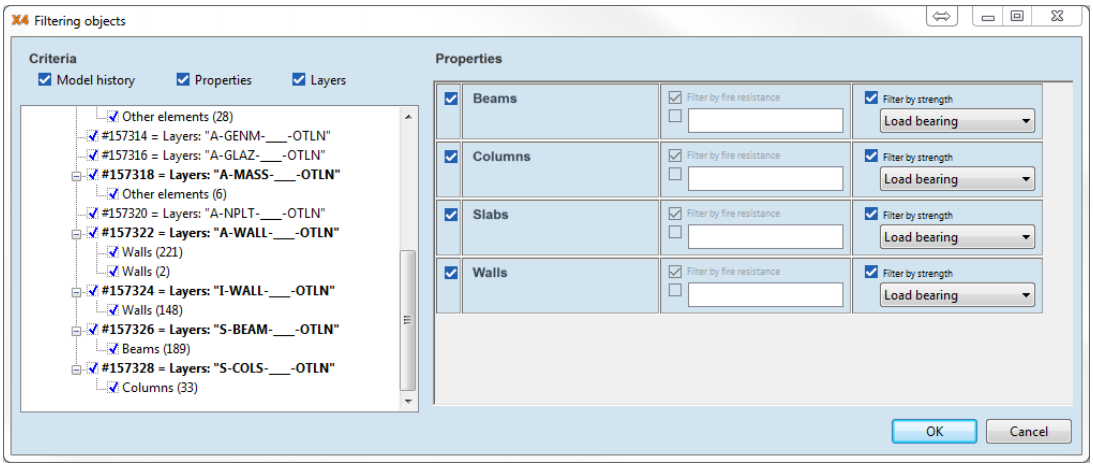
If checked, AXISVM calculates the centre of gravity (COG) of the imported model, translates the whole model, so that X and Y coordinates of the COG get to (0, 0) of the coordinate system. It can be useful when the imported model has large coordinate values.
If *Also in Z direction* checkbox is checked, a vertical translation will be also performed, moving the Z coordinate of the COG to zero too.

Automatic detection of changes

This function lists new, modified and deleted objects when reimporting IFC files. Each change can be approved or ignored. Selecting a change in the tree shows the position of selected element within the current model.



Filtering objects



Filtering by *Model history* – The user can choose to import new elements or modified elements
This type of filter works properly only on files previously exported from AXISVM and modified in another program.
Filtering by *Properties* – The user can choose elements with a load bearing capacity and/or by fire rating.
This type of filter works properly only if this information is included in the file.
Filtering by on *Layers* – if the IFC file contains layer information for the entities the user can select which layers to import and which layers to ignore.

☞ **When exporting a model from ADT (Architectural Desktop) turn off the automatic intersection of walls before creating the IFC file.**

3.1.7.3 Importing Revit RAE files

Import from Revit

AXISVM can import data through a direct connection between AXISVM and Autodesk Revit 2025 (or newer). It is based on the Revit API which lets other programs to query and modify its database.
At the end of AXISVM setup process the installer creates a folder called *RevitImport* under the AXISVM folder, and copies all the necessary files.
The **.addin* file is created in folder
c: \ *Users* \ *<UserName>* \ *AppData* \ *Roaming* \ *Autodesk* \ *Revit* \ *Addins* \ *yyyy* \, where "yyyy" represents an integer value greater or equal to 2025. Both this file and the whole content of "RevitImport" folder are necessary for the proper communication between the two programs.

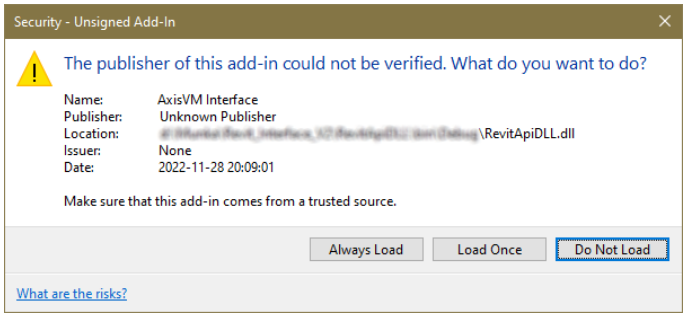
There are two methods of communication

1. **Using an intermediate file.**
2. **Through AXISVM COM interface.**

Revit 2023 and later versions use a new analytical model. According to this, AXISVM uses this model.

☞ **Keep in mind to create an analytical model before exporting the Revit project to rae file. Follow the Revit Knowledge Base (Physical to Analytical Automation).**

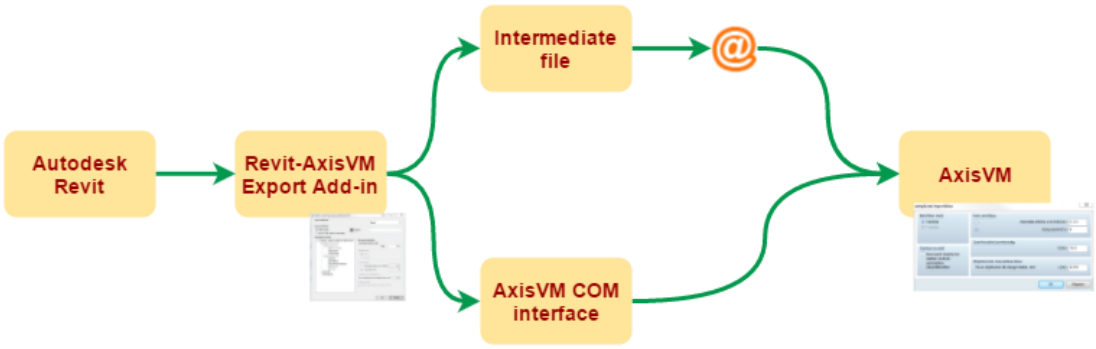
☞ **The popup window shown below will appear at the initialization phase of every add-in or third party executable.**



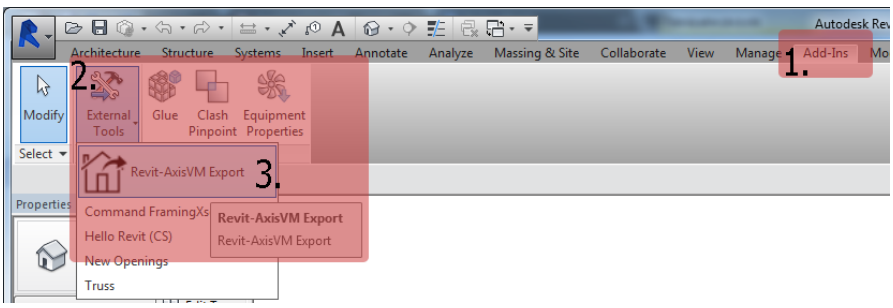
☞ **In this case, please press "Load Once" if you wish to load the add-in, and use its abilities, or "Always Load" if you wish to suppress this dialog in the future.**

Usage

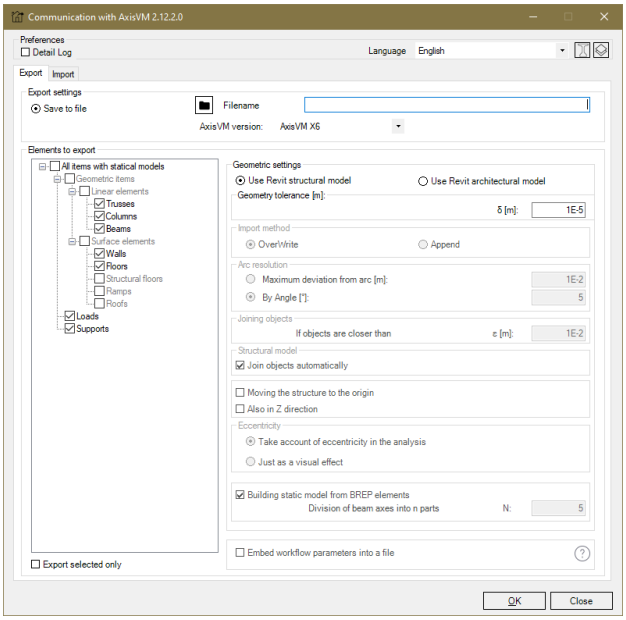
The scheme of dataflow is



The process must be started **from Revit** as shown below:
Add-Ins / External Tools / Revit-AXISVM Export



The *Export statical model to AXISVM* dialog let us to choose between the two methods. This choice also determines the available settings on the dialog.
Choosing the *Using AXISVM COM interface* offers setting certain geometry parameters. The language of the dialog can also be selected from a list.
If the radio button *Use Revit structural model* is checked, then the add-in exports the structural model defined in Revit as it is, otherwise it uses the architectural model to calculate the statical frame to be exported.

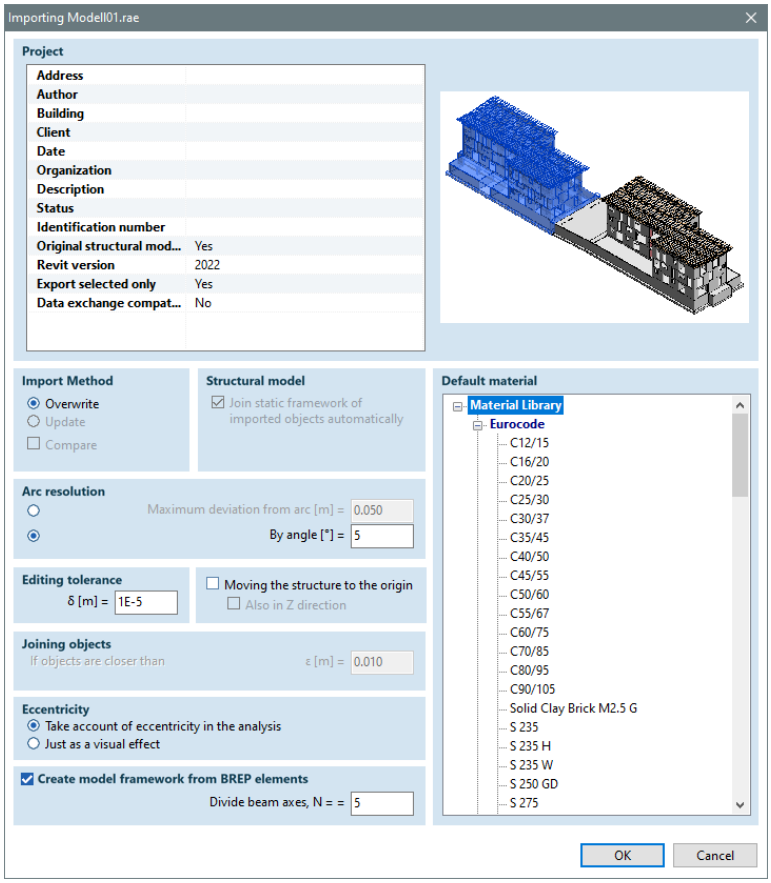


Import via an intermediate file

Users should choose this option, if they wish to send the data to another user.

First, they have to set the name of the file to be saved then select an AXISVM version as the target. After a successful save operation, the file created can be imported into this version of AXISVM.

When importing the file into AXISVM a dialog will pop up.



Project The project info declared in Revit is listed in the upper-left list, while the image to the right shows a screenshot of the project as could be seen in Revit.

Import method If the AXISVM model is not empty, two options are available to overwrite the model or update it.

Structural model If this option is activated AXISVM changes the raw geometry data to build a proper structural framework. Colliding beams are connected with a rigid body, neighbouring domains are joined, beams running on a domain are converted to ribs. Some of these calculations require considerable resources and may slow down the import process.

Arc resolution Certain geometry calculation requires arcs to be converted into polylines. The conversion criterion can be set here.

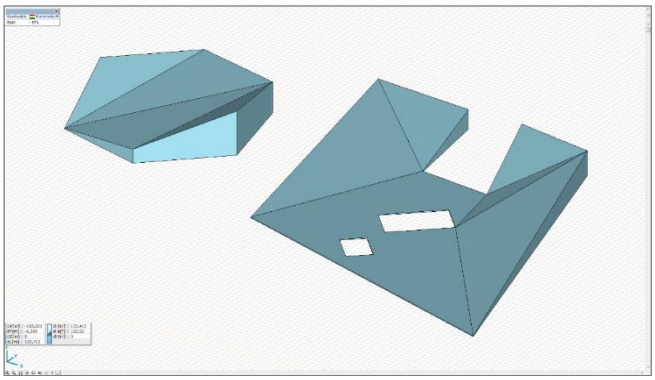
Editing tolerance Tolerance used when calculating geometry.

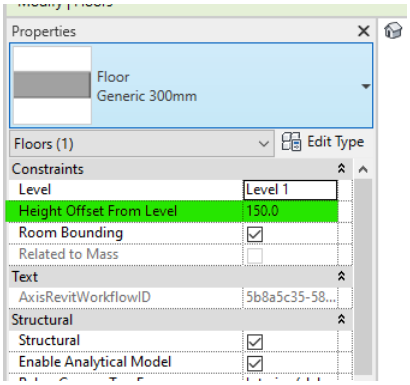
Joining objects To improve the efficiency of collision detection bounding boxes of objects are inflated a bit. This tolerance value can be set here. Active only if the checkbox on panel *Structural model* is checked.

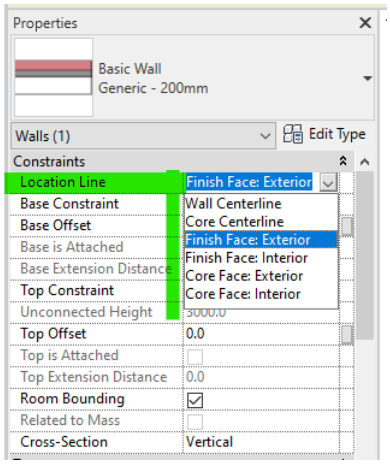
Import through COM interface Use this option if you process the data extracted from Revit yourself and you run Revit and AXISVM on the same machine. After a successful export through the AXISVM COM interface, AXISVM launches and you can continue editing the model there. The above geometry parameters must be set in the Revit export dialog.

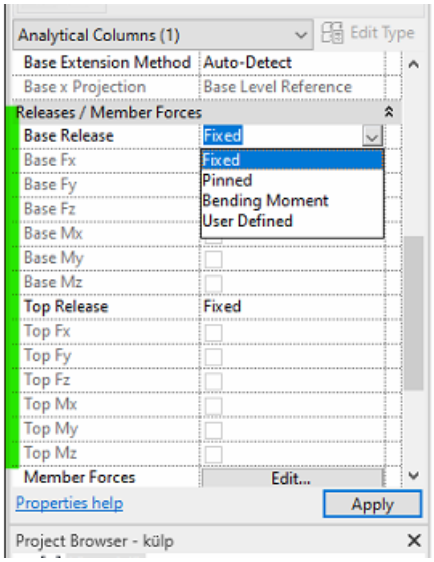
Moving the structure to the origin If checked, AXISVM calculates the centre of gravity (COG) of the imported model, and translates the whole model so that X and Y coordinates of the COG get to (0, 0) of the coordinate system. It can be useful when the imported model has large coordinate values.
If Also in Z direction checkbox is checked, a vertical translation will be also performed, moving the Z coordinate of the COG to zero too.

Elements to export Element types to be exported can be set by selecting tree items.



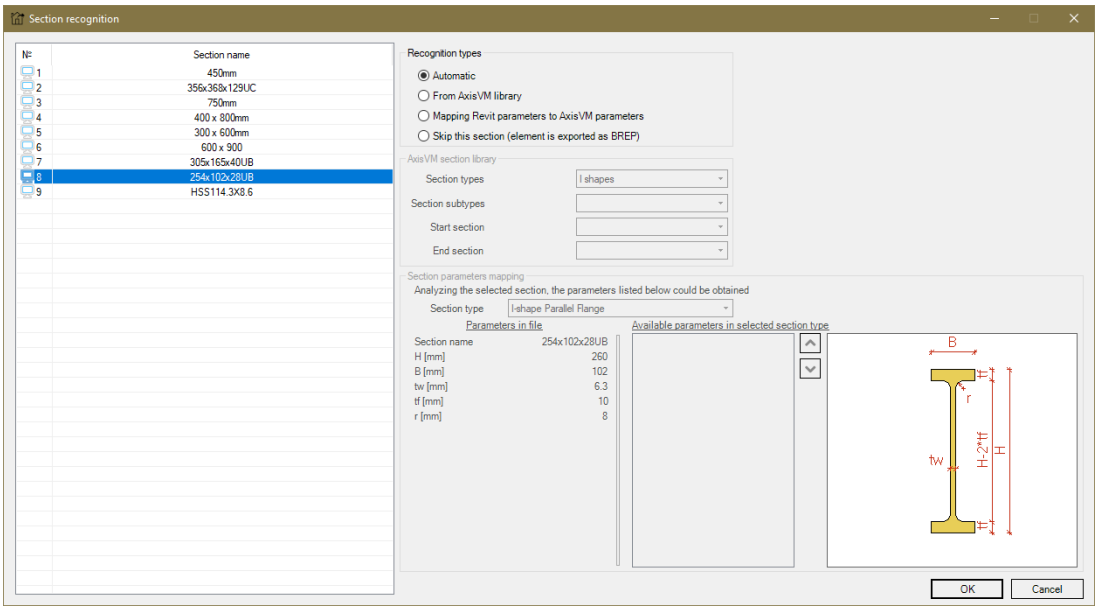






Section recognition

Sometimes the cross-section information read from the Revit database is limited so identification of sections requires user interaction. If so, a dialog is displayed.



Section recognition can happen in several ways:

Automatic

The add-in was able to get sufficient information about a certain type of cross section.

From AXISVM library

If the automatic recognition has failed, you can select the proper section from the AXISVM section library.

Mapping Revit parameters to AXISVM parameters

Storage of section parameters in Revit is not consistent so a unique parameter matching may be needed. You can select a property from the list and move it up or down into the proper position to associate parameters in AXISVM and Revit.

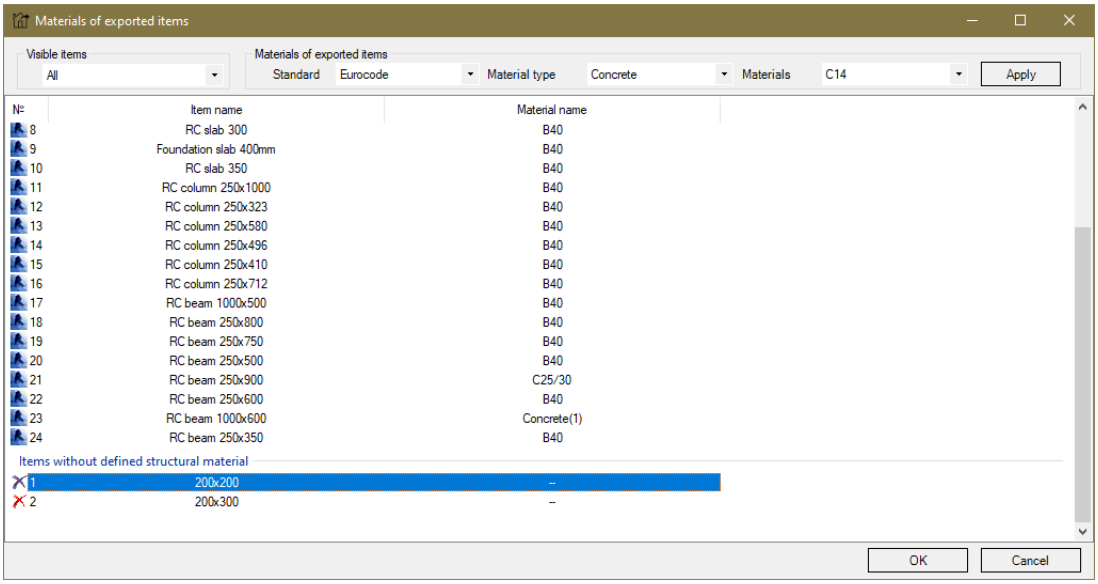
☞ **The add-in learns the parameter matching you defined so next time it will identify the section properties without user interaction.**

Skip this section

Skipping the section means that all elements having this section will be imported as a general object with its **boundary representation** but without a structural model.

Setting materials

Extracting proper material information from Revit poses similar problems. If the add-in fails, it will call your help to identify the materials that could not be recognized, and you can associate AXISVM material names to element names.



You can leave some elements unassigned. At the import phase, AXISVM will assign a default material to them.

If the installer fails...

If the installer fails due to any reason and Revit does not find the dll file of the add-in you can install it manually by running !REGISTER_Revit.BAT from the folder where AXISVM is installed.

☞ **It is important to know, that the communication between the two programs is based on the Revit API interface. The structural model of the elements is queried from the Revit database (if Revit does not provide this information, AXISVM can import only the wireframe of the element). This structural model still requires many minor adjustments to make it usable. AXISVM tries to join objects (if you check this option) but sometimes you have to edit the model to get a proper structural framework.**

Comparison based data exchange

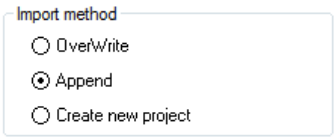
Since version X5, AXISVM and Revit can share a workflow between each other. This is actually a comparison based export-import.
With this, engineers working on the same project can import new or modified entities without effecting unchanged elements.
The workflow process can be started both from AXISVM and Revit.

When starting from Revit one must leave the checkbox on the bottom of the main window checked.



The reason is that in order to inject the necessary parameters into the project file, the application needs the permission of the user and to save the file afterwards. If it can not happen, the exported file will not contain workflow parameters. AXISVM will still be able to read the produced file, however comparion based import will not be possible.
When starting the workflow from AXISVM there is no need for such acts, necessary parameters will automatically be embedded in the file.
One can use comparison based import only if there is a modell previously opened in either application.

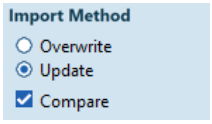
In Revit radio buttons must be switched like on the picture below:



While in AXISVM, you can see the lines shown below when importing a file previously saved with workflow parameters:

Revit version	2019
Export selected only	No
Data exchange compat...	Yes

For the comparison to actually happen you have to check the checkbox shown below:



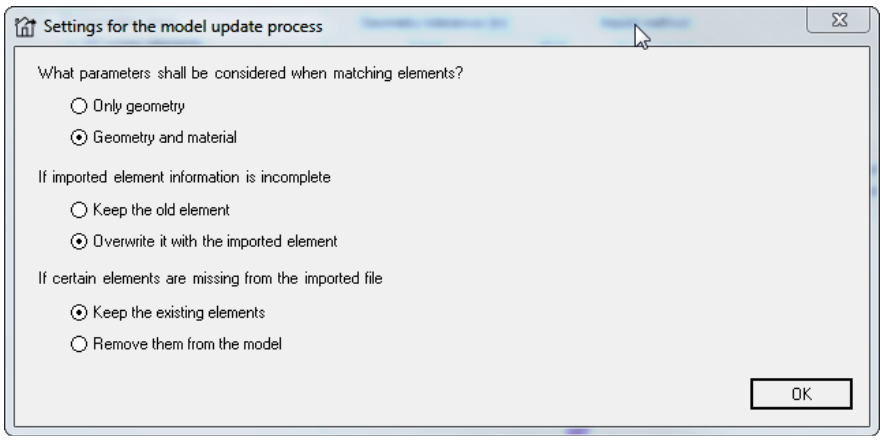
During the import, three basic scenarios are handled:

1. Element to be imported can not be found in the actual model, so it can be simply processed and imported.
2. One or more items of the actual model can not be found in the file to be imported. In this case, users can select to either delete these elements or to ignore and leave them intact.
3. The item to be imported can be found in the actual model. In this case, we perform a comparison based on
    a. the material of the elements
    b. the statical model of the elements
    c. the profile or thickness of the elements

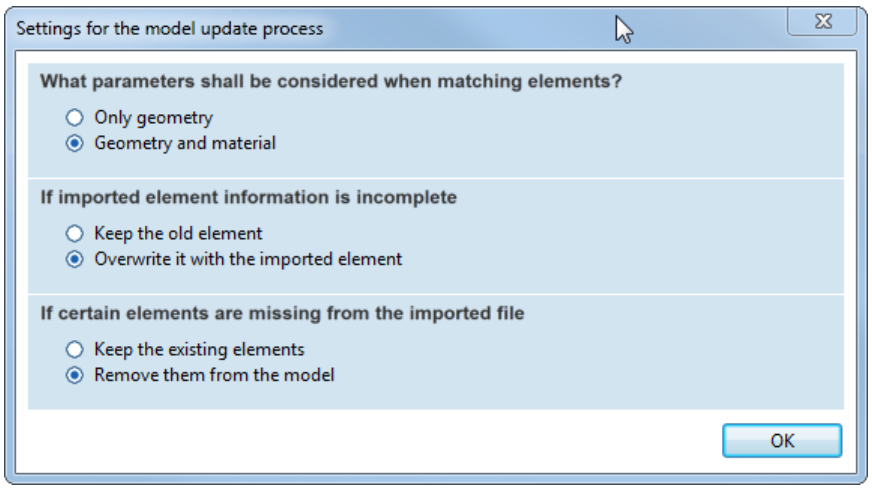
If after this we find them equal, the imported item is skipped. However, if they are found to be different, arriving item is processed, and the former one is deleted.
When -for any reason- we cannot perform a succesful comparison, users can select to keep the existing item or to import the new one.

These mentioned user preferences can be adjusted prior to the actual import process on the form pasted below:

Revit dialog



AXISVM dialog



3.1.7.4 Import from Tekla Structures

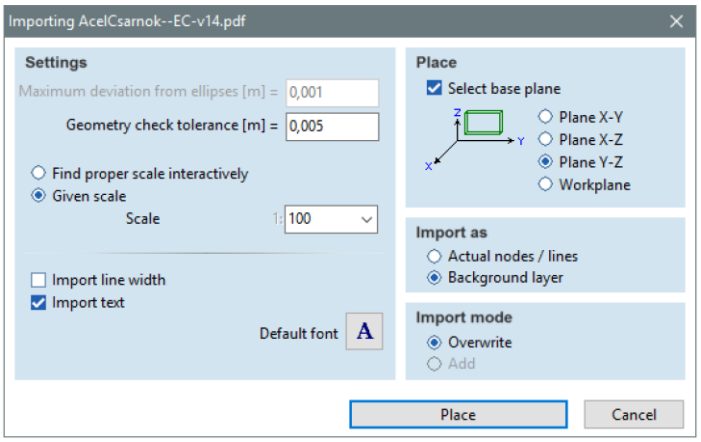
Versions before Tekla Structures 2021 support one-way communication only (**3.1.9.2 One-way connection for Tekla Structures versions before** 2021)

For Tekla Structures 2021 or later it is recommended to use the IT module to establish a two-way connection between AXISVM and Tekla (**3.1.9.1 Two-way connection for Tekla Structures versions 2021 and later**)

3.1.7.5 Importing PDF files

*PDF *.pdf*

This function imports drawings from PDF files as a background layer or AXISVM lines.
Only lines, curves and text objects are processed, images and other elements are ignored.
The final size of the imported drawing can be set interactively on the screen *(Find proper scale interactively)* or can be adjusted by specifying a *Scale*.
Importing text and line width is optional. Note: certain pdf creating engines export text as a series of graphical objects making it impossible to distinguish between text and graphics.



You can set the *Default font* to use when displaying text with an unavailable font. After clicking on the *Place* button, a page number must be entered for multipage documents. Only one page can be imported at a time.

For other settings and commands see the above part describing importing DXF files.

3.1.7.6 Importing SAF files

Import of this file type requires an **SAF module**.

This Excel-based file format supports exchange of structural model data with other applications supporting this protocol. The file consists of standardized spreadsheets for geometry, structural and load properties of the model, hence can be viewed and/or edited with Excel. SAF protocol is supported by ArchiCAD, AllPlan, SCIA and RFEM.
Entities imported from the incoming file are

- trusses, beams, columns, ribs, supports, regions and connection elements (hinges, joints)
- cross-sections
- support stiffnesses
- load, load combinations

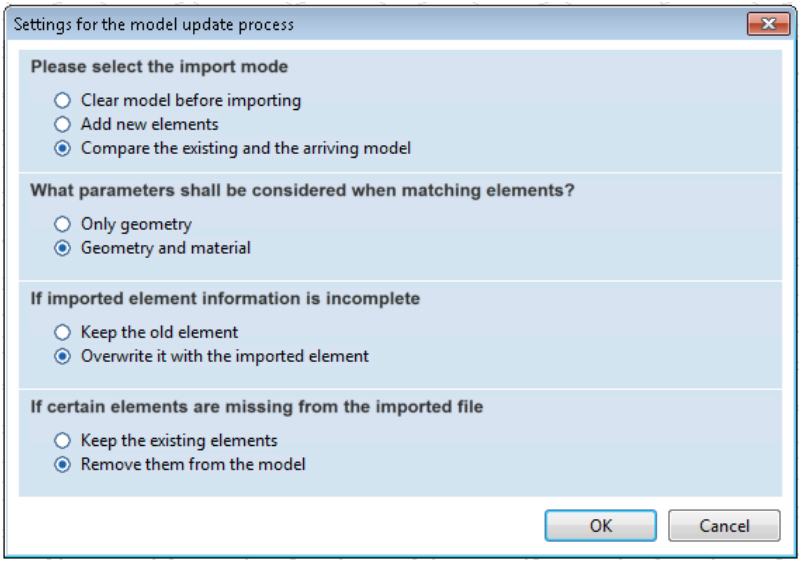
Usage

Select *File / Import / SAF import file (*.xlsx)* and select the file to be read and press *Open*.
Alternatively, you can drag and drop the imported file on the main window of the application.

Import

The import process differs if a model is already opened in AXISVM or not.

- If not, then AXISVM simply parses the file and imports entities.
- In case of an opened model, the following dialog is displayed



  - The first option clears the existing model and imports the incoming file
  - The second option adds all incoming entities to the existing model

- The third option compares the existing and arriving model, and imports only those elements which are either
  - new
  - or modified either in their material or in their geometry

If the last option is selected the users can choose to

- compare only the geometry of the two models
- or include the comparison of materials.

Geometry comparison consists of comparing the axes and cross-sections of linear elements, the polygons and thickness of regions.
To handle the situation when the imported data is incomplete, or any unexpected issue happens, users can choose either

- ignore the arriving element
- or replacing the existing item anyway

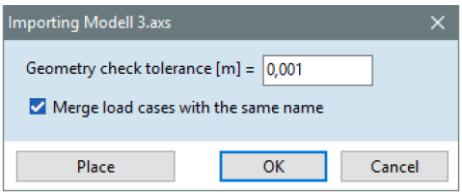
The user can control what happens if certain elements are missing from the arriving file. The two options

- Keep the original elements unchanged
- Remove them

For the time being comparison-based file exchange supports only linear elements, domains and supports.

3.1.7.7 Importing AXISVM AXS files

*AXISVM *.axs*



Imports a model from an existing AXISVM file into AXISVM, and merges it with the current model.
During the merging process, the Geometry Check (See section **4.8.20 Geometry check**) command is automatically applied.

If there are different properties assigned to the same merged elements, the properties of the current model will be retained. Load groups and combinations if any, are appended to the existing ones as new groups and combinations, and the load cases as new cases. If no load groups or combinations are defined in the imported model, the load cases will be appended to the existing ones as new cases.
If the same load case exists in both models, loads will be merged if the *Merge load cases with the same name* checkbox is checked.

If both models contains loads that are limited to one occurrence (e.g., thermal) in the same load case, the load in the current model will be retained.
The Section Lines/Planes Parts with the same name are merged, otherwise they are appended.

When importing an AXISVM file a dialog is displayed.
Use the *Place* button to graphically position the imported model in your model's space.

3.1.7.8 Importing files from other applications

*Stereo Lithography *.stl file*

Reads the triangular mesh describing the surface of a model from a file in STL format (binary or text). Multiple nodes and degenerated triangles are filtered out. Import can be transferred to a background layer as well.

*Bocad interface *.sc1 file*

Opens a data file created by Bocad steel construction software *(*.sc1)* and imports beam cross-sections and geometry.

*Glaser -isb cad- *.geo file*

Imports **.geo* files exported by Glaser -isb cad- describing beam or surface models.

SDNF file (Steel Detailing Neutral Format)

Imports a file exported in Steel Detailing Neutral Format used in data exchange between steel detailing programs.

DSTV format
**.stp file*

In its current version AXISVM is capable of importing stp files implementing "PSS_2000_04" file scheme based on "Standard Description for Product Interface Steel Construction 4/2000". This filetype is designed for storing steel structures' data, and can describe linear structures as well as planar regions. Even statical calculations can be imported through this interface, though with limited capabilities.
If static calculations and an architectural model are both embedded in the file, the user can choose importing either the calculation or the architectural model. When trying to import data onto an existing model, the user can select between overwriting or just supplementing it.

3.1.8 Parametric design

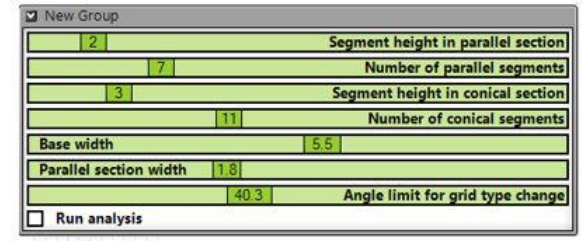
„Parametric design is based on the idea, that specific numbers, i.e., parameters control the geometry. To fully exploit its potentials, these parameters are linked to each other, and these links form a system. Thus, it is possible that by modifying only one parameter, the whole geometry changes. This idea is regularly used in structural design. Imagine that by modifying only the span of a truss, the height and the truss layout are automatically updated both in the 3D model and in the shop drawings. To build a system capable of doing this requires time, and therefore it is essential to think it over whether it is worth the time to create a parametric model. However, once we have built such a system of parameters, links, and a dynamically changing model, we have a highly flexible tool."

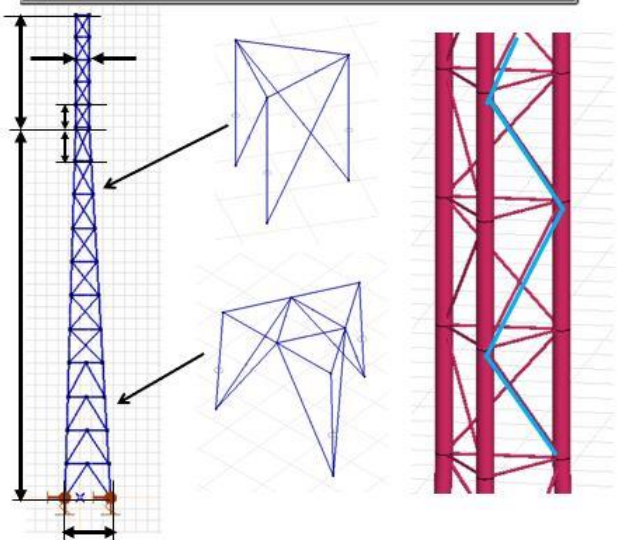
Based on the article of Péter Romvári: *Parametric design in practice* in epiteszforum.hu

Parametric add-ons

The most commonly used parametric design software in structural design are Grasshopper and Dynamo. These are often referred to as visual programming language and environment that can be used to create parametric models easily. By using the GrasshopperToAXISVM and DynamoToAXISVM add-ons, the parametric models created in Grasshopper/Dynamo can be exported to AXISVM.

First, the GrasshopperToAXISVM or DynamoToAXISVM add-on should be installed in Grasshopper/Dynamo. These add-ons communicate with AXISVM through its API (COM interface). It is required that AXISVM and a version of Grasshopper/Dynamo are all installed on the same computer. After starting the export, AXISVM automatically launches, showing the exported model.
One advantage of this interface is that it is easy to create parametric models and even templates in Grasshopper/Dynamo. For example, the parameters of a truss may be the span, height, layout and cross-section sizes. The other advantage is that many geometric functions are available in these environments enabling creation of complex spatial models.





3.1.8.1 Grasshopper/Rhinoceros add-on

**.gh, *.3dm files*

Grasshopper is the parametric design tool of the Rhinoceros CAD software. Grasshopper can be started from within Rhinoceros. To export the contents from a **.gh* file, the GrasshopperToAXISVM add-on shall be installed in Grasshopper in Rhinoceros version 6, 7 or 8. After installation, the modeling and exporting/importing functions (components) are available in the Grasshopper menu, in the AXISVM tab. Rhinoceros files (**.3dm*) shall be loaded into Grasshopper before exporting to AXISVM.

When the parameters change, the AXISVM model automatically tracks the changes. After exporting, the final AXISVM model may be edited by the user. Those modifications provide no feedback towards Grasshopper, thus they have no influence on the Grasshopper parameters.

Available features in **GrasshopperToAXISVM 5.x:**

- line elements (truss, beam, rib)
- spring, gap
- line element eccentricity, beam end release (eccentric as well)
- surface elements
- domains (normal, variable thickness, variable eccentricity, ribbed, composite rib, hollow core, CLT, trapezoid, custom stiffness matrix)
- load panels
- supports (nodal, line, surface)
- diaphragm, rigid elements, edge hinge
- loads (concentrated load on node, domain, load panel; distributed loads on lines, surfaces, domains, load panels, polyline load on domains and load panels, self-weight, thermal load)
- cross-section (select from library, add from text input, draw thick polygon)
- material (selected from library, add from text input)

Additional features from **GrasshopperToAXISVM 6.0** on:

- parametric cross-section
- CLT-panel, reference, spring characteristics
- node-node and line-line link elements
- new load components (seismic, wind, snow, fluid loads, support displacement etc.)
- meshing (for line elements and domains)
- analysis processes (linear static, nonlinear static, vibration, seismic, buckling analysis)
- steel design member
- timber design member
- cross-section optimization (for steel and timber design members)

some analysis results can be imported into Grasshopper

Functions available from **GrasshopperToAXISVM V7.0:**
(full use requires AXISVM X7 R3 or higher)

- cross-section optimalization (for steel and timber design members)
- the most important result components are readable from Grasshopper
- control of result display parameters in AXISVM from Grasshopper
- switch between adding/replacing data in the AXISVM model (from V7.1 on)
- items to distribute the load panel loads over may be selected (from V7.2 on)

New features in **GrasshopperToAXISVM V8.0:**
(requires AXISVM X8 or higher)

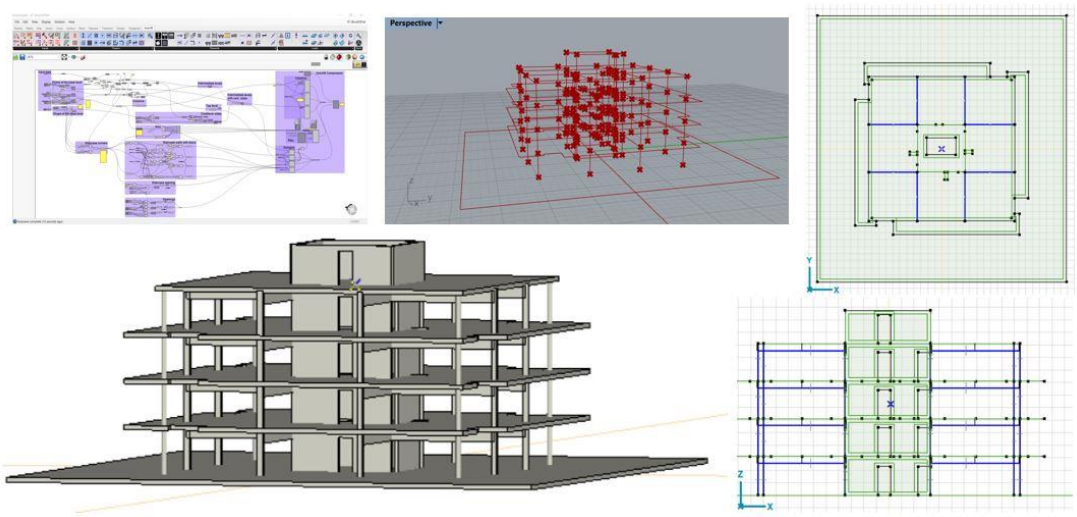
- new feature package related to the GH-AXISVM connection to enhance user experience:
    - Rhino preview for AXISVM components
    - user-defined part folders and parts
    - easier identification of parametric elements in AXISVM (select / deselect items)
    - access to the content of a saved .axs AXISVM file (import and use items, define loads)
- new feature package related to loads:
    - moving loads
    - nodal mass
    - fault in length
    - line tension/compression
    - influence line load
    - concentrated load on surface elements
    - domain or load panel polyline loads may be associated to structural members
- new general feature package:
    - selected material, cross-section and CLT panel name is displayed on the component
    - warping characteristics
    - updates related to result import

- intermediate updates in V8.1:
    - access to the result case list of existing .axe files
    - extract the extreme result values in envelope and critical cases
    - new result types can be imported: spring force, surface stress, CLT stress;
    - quicker assignment of cross-sections to the optimization groups (for CS optimization based on predefined shapes)
- intermediate updates in V8.2:
    - force can be imported for surfaces, surface supports, gaps, NN and LL link elements and edge hinges
    - import of CLT utilization
    - grouped import also for CLT stress, nodal and line support force
    - with these updates, both the unique and the grouped import is available for all result types (displacements, forces, stresses, utilizations)
- intermediate updates in V8.3:
    - import RC1 result components: reinforcement quantities and cracking values (for calculated and actual reinforcement), reinforcement design forces, shear resistance values
    - create custom materials
    - extract material properties
    - set reference directions of load panels

automatic AXISVM license selection in COM connection (from AXISVM X8R1i on)

New features in **GrasshopperToAXISVM V9.0**:
(requires AXISVM 27)

- aligning the orientation of structural elements with that of Rhino lines
- transfer and import of design parameters for reinforced concrete beams and columns (RC2)
- expanded capabilities of the Importeur component: restraints, load cases, load combinations, load panels and parts
- handling of AXISVM logical model parts directly from Grasshopper
- existing components are further refined to become more flexible, compact and efficient to use, alongside simplified load definition workflows
- selection of steel design member ULS classification method



The AXISVM add-on, installation guide, manual and examples can be downloaded from:
https://www.food4rhino.com/en/app/grasshoppertoaxisvm

These materials are also available via our website:
https://axisvm.eu/axisvm-downloads/#applications

3.1.8.2 Dynamo/Autodesk plugin

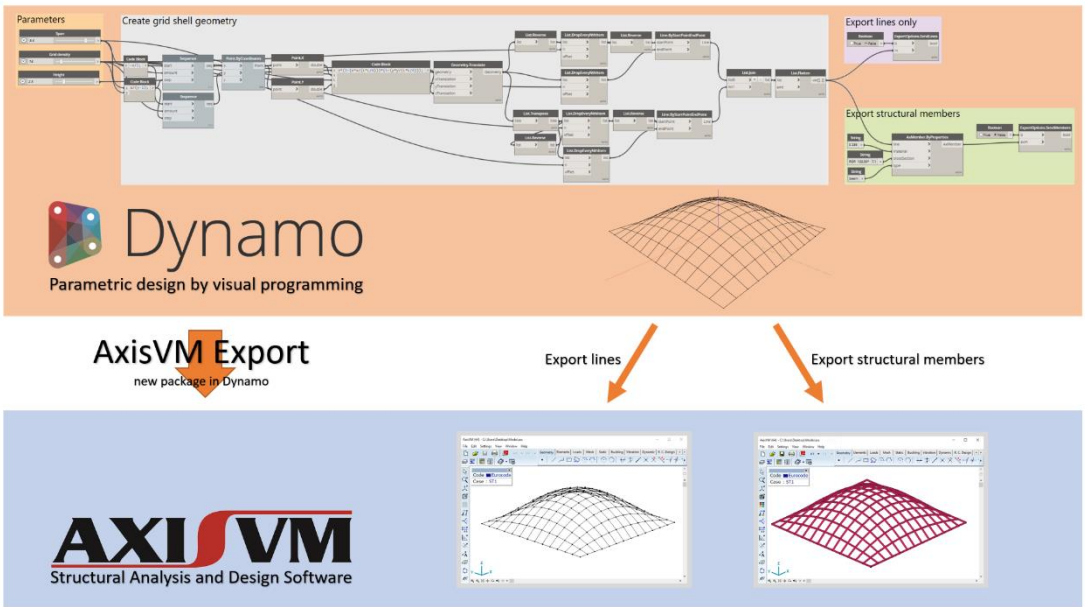
**.dyn, *.rvt files*

Dynamo is a parametric design tool of the Autodesk Revit software. Dynamo can either be started from within Revit or as an independent software (Dynamo Studio, Dynamo Sandbox). To export a file with *.dyn extension, the AXISVM Export plugin shall be installed in Dynamo. After installation, the modeling and exporting functions (nodes) are available on the left-hand side menu of Dynamo, at the AXISVM drop-down menu. Revit files (*.rvt) shall be loaded into Dynamo before exporting to AXISVM.

In Dynamo, before starting the export, first, a new model shall be created using the AxModel node. Self-weight, nodal load, and nodal supports can be added to the parametric structural model. After starting the export, a one-directional link is created between the parametric model and the AXISVM model. This means, that modifying a (geometrical or structural) parameter in Dynamo automatically updates the structural model in AXISVM. In AXISVM, further structural elements can be added, but these do not influence the Dynamo parametric model. In Dynamo, it is possible to start an AXISVM linear static analysis. The link between the models is destroyed by closing any of the models.

Available features in DynamoToAXISVM 3.0 (requires AXISVM X7 or higher):

- line elements (truss, beam, rib)
- spring, gap
- line element eccentricity, beam end release (eccentric as well)
- surface elements
- domains (normal, variable thickness, variable eccentricity, ribbed, composite rib, hollow core, CLT, trapezoid, custom stiffness matrix)
- load panels
- supports (nodal, line, surface)
- diaphragm, rigid elements, edge hinge
- loads (concentrated load on node, domain, load panel; distributed loads on lines, surfaces, domains, load panels, polyline load on domains and load panels, self-weight, thermal load)
- cross-section (select from library, add from text input, draw thick polygon)
- material (selected from library, add from text input)



To use the DynamoToAXISVM add-on in Revit, you need Revit version 2022 or newer.
Open Revit. Go to the *Packages* tab, select *Search for a Package* from the drop-down list. It may take a few seconds to load. Start typing "DynamoToAxisVM" in the search box. Click the *Install* button of the appropriate component. Copy *Interop.AxisVM.FW4.dll* file from the AXISVM directory to the installation folder of the add-on.
For further information see: **https://github.com/AxisVM/DynamoToAxisVM/wiki**

3.1.9 Tekla Structures – AXISVM connection – TI module

AXISVM is able to communicate with Tekla Structures software different ways in case of

- version 2021 and later,
- version before 2021.

3.1.9.1 Two-way connection for Tekla Structures versions 2021 and later

Setup

If the user chooses to install the Tekla plugin, the AXISVM Installer copies all necessary files including the AXISVM *Import &Export* plugin for Tekla. No other user activity is required. If **TI** module is included in the AXISVM configuration, the two-way connection is functional after the AXISVM installation.

The AXISVM *Import &Export* plugin for Tekla Structures can be installed separately. Download the separate free installer from the AXISVM website.

3.1.9.1.1 Transferring model from AXISVM to Tekla Structures

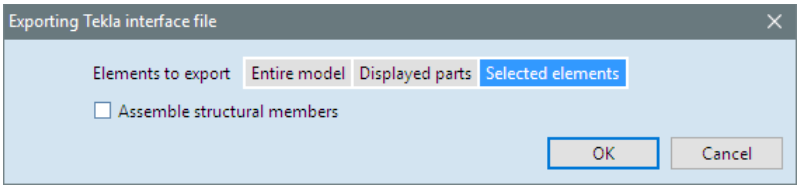
AXISVM with a **TI** module can export the model to Tekla Structures by creating a file with ATE extension. This ATE file can be read to Tekla using a dedicated Tekla plugin building the model using Tekla Open API commands.

Exported elements

AXISVM can export the following elements: trusses, beams, ribs, domains, supports and actual reinforcement. The AXISVM *Import & Export* plugin for Tekla Structures allows filtering of elements. Exported data includes element geometry, material and cross-section of line elements, material of domains and stiffness values for nodal supports. Loads and load combinations are not yet exported.

Export to ATE file

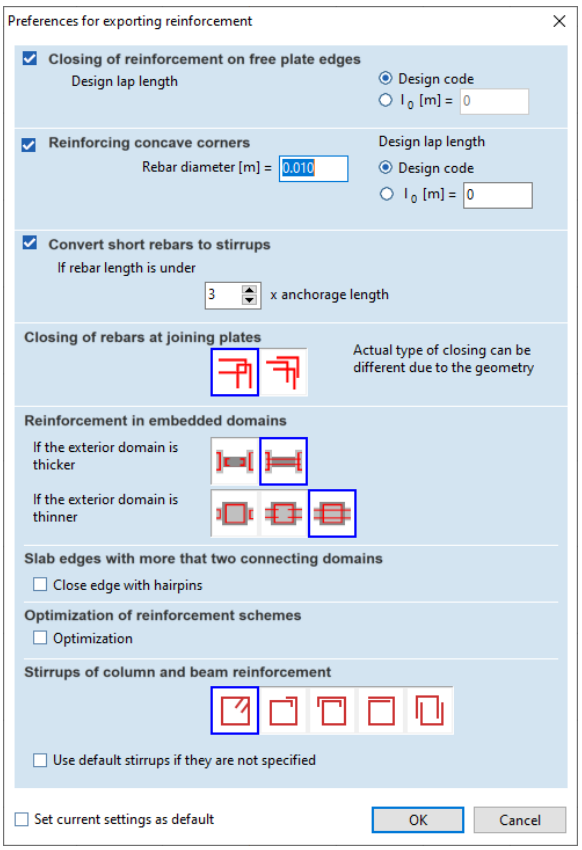
The export can be launched from AXISVM via the main menu: *File / Export / Export to Tekla (*.ate)*.
Click the Settings button to set the available options.



Entire model: all data is exported
Displayed parts: only the current active parts are exported
Selected elements: only selected elements are exported
Assemble structural members: creates structural members from elements on the same line and with the same properties

Export actual reinforcement

Since X5 Release 4, users can export reinforcement to .ate files.

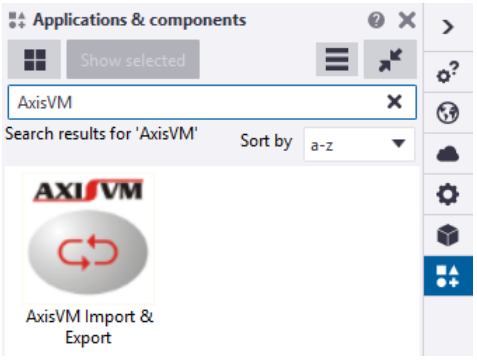


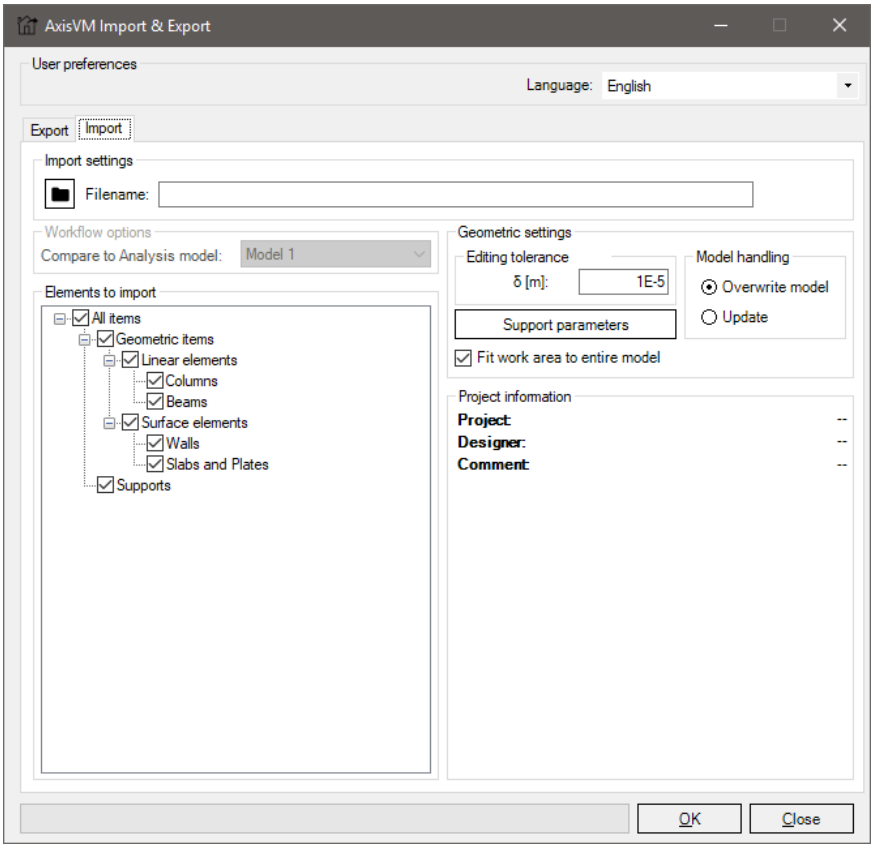
The algorithm of AXISVM helps to export not only the applied field reinforcement but shaped bars too, like hairpins, hooks etc.
Calculated raw bar lengths are always rounded up to the nearest value of this array: 0.4, 0.5, 0.6, 0.8, 1.0, 1.2, 1.5, 2.0, 2.4, 3.0, 4.0, 6.0, 12.0 m.

- *Closing reinforcement on free plate edges*: free plate edges are closed by hairpins, whose lengths are calculated based on the lap length of bars, which is calculated according to the design code, or can be entered manually as l_0. This option can be turned on or off.
- *Reinforcing concave corners:* concave corners can be strengthened by simple bars, whose lengths are calculated based on the lap length of bars, which is calculated according to the design code, or can be entered manually as l_0. This option can be turned on or off.
- *Convert short rebars to stirrups:* to avoid useless duplicating of rebars, this algorithm uses a multiplicator value. Simple bars having a length greater than the minimal lap length multiplied by this value, will be converted to a closed stirrup. This option can be turned on or off.
- *Closing of rebars at joining plates:* along the edges of plates that do not share the same plane, the algorithm can connect the reinforcements of the individual plates either by hairpins or by hooks. The actual geometry of these shaped bars depends on the width and angle of the joining plates. The algorithm may revise the user's choice. The sizes of the shaped bars are calculated with respect to the minimal lap length.
- *Reinforcement in embedded domains:* the method of closing or continuing rebars of embedded domains can be chosen. If the inner domain is small enough, the outer domain's bars may as well be driven through its body.
- *Slab edges with more than two connecting domains:* there is no automatic solution in such cases, so the user can choose whether to ignore those edges or close them one by one.
- *Optimization of reinforcement schemes:* Optimization of rebar positions to distribute reinforcement evenly if reinforcement domains overlap. May be time-consuming.
- *Stirrups of column and beam reinforcement*: stirrups of the applied column and beam reinforcement are exported. If stirrup spacing was not defined the default can be exported optionally.

Import ATE file to Tekla Structures

The file with .ate extension can be imported to Tekla Structures via a dedicated plugin. Make sure that the *Analysis & Design models* dialog box is closed during the import process because it slows it down significantly. Search for "AxisVM" in the *Applications & components* catalogue in Tekla Structures and double click on the AXISVM *Import & Export* to open it and switch to its *Import* tab. The following dialog will appear





Import settings

Press the button left to the *Filename* label. A preview of the model to be imported will be displayed in the bottom right corner.

Elements to import

Imported elements can be filtered by checking the nodes.

Workflow options

Active only if *Model handling* is set to *Update*. Select the analysis model to compare the imported model to.

Editing tolerance

Tolerance used when calculating geometry.

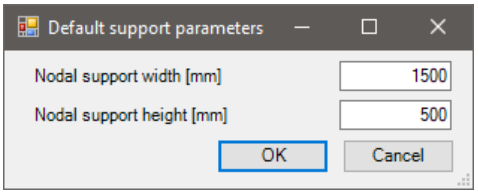
Model handling

Overwrite model: All the existing objects will be deleted before importing the model.

Update: Choose this method to compare the existing model to the imported one. Changed elements will be updated. After clicking on the OK button further options will be displayed.

Support parameters

Nodal supports geometry in Tekla will be created from default parameters. Click the *Support parameters* button to adjust these values.

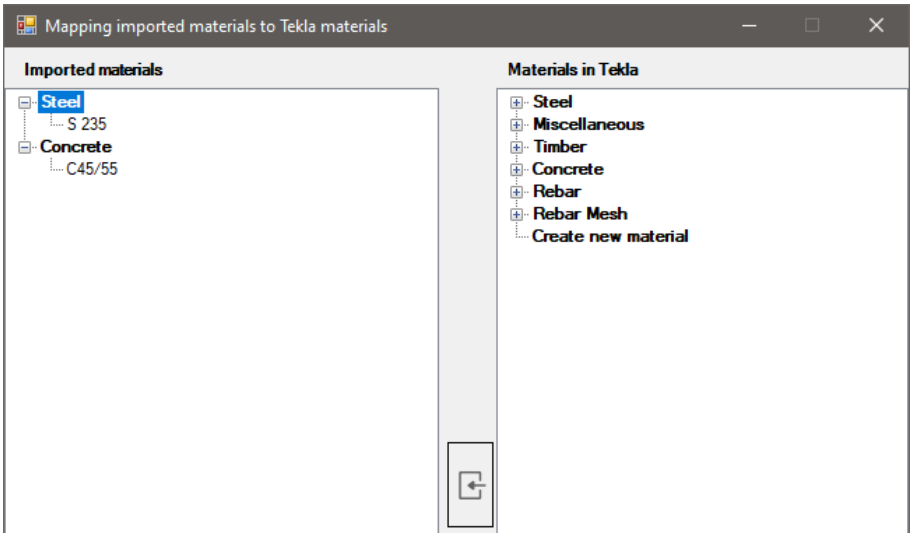


Project information

Information about the exported AXISVM model.

Mapping materials

As the Tekla equivalent of an AXISVM material cannot be found automatically, the mapping of materials must be performed manually in the following dialog.



Select an AXISVM material from the left tree, select its matching material in Tekla on the right then click on the button between the two views to map the imported material to the Tekla material. The selection in the left tree jumps to the next unmatched material. The name of the matched material turns red, unmatched materials are black.
If a material category is selected on the left (and not a material), all materials of this type will be matched to the same material in Tekla.
It is also possible to create a new Tekla material based on the properties of an AXISVM material by selecting *Create new material* from the right tree.
If all imported materials are matched the OK button becomes enabled and the import of the model can be continued.

3.1.9.1.2 Transferring analysis model from Tekla Structures to AXISVM

The AXISVM *Import & Export* plugin in Tekla Structures can export the model data to a file with TAE extension. This file can be imported to AXISVM.

Exported data

The plugin exports the following elements: line elements with their material and cross-section, domains with their material, nodal supports, loads and load combinations. Only elements present in the chosen analysis model in Tekla will be exported. Supports must be defined as separate point supports. Line elements with a custom cross-section are not converted to static model and exported only as a custom shape.

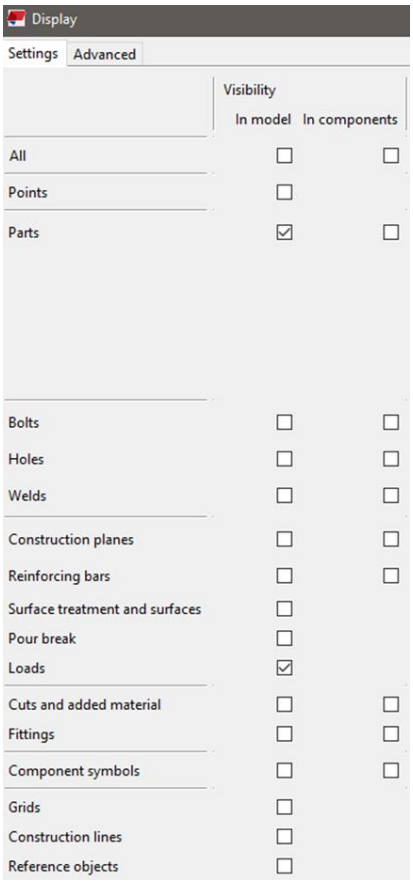
Preparation

It is important to create the analysis model correctly in Tekla so that AXISVM can work with no further adjustments. To generate a good analysis model set the properties as they appear on this image.

The optimal settings can vary depending on the model and you might have to configure the analysis properties of the parts individually.

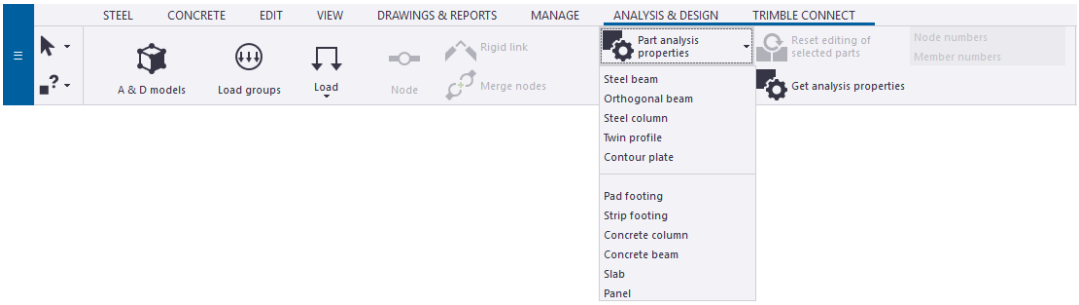
Before creating the analysis model, it is helpful to have only the relevant elements displayed. You can configure which elements are visible in the view properties.

With only the relevant elements displayed select the entire model and set the analysis properties of every part. This way the created analysis model will have the right default properties and it minimises the further adjustments needed later.

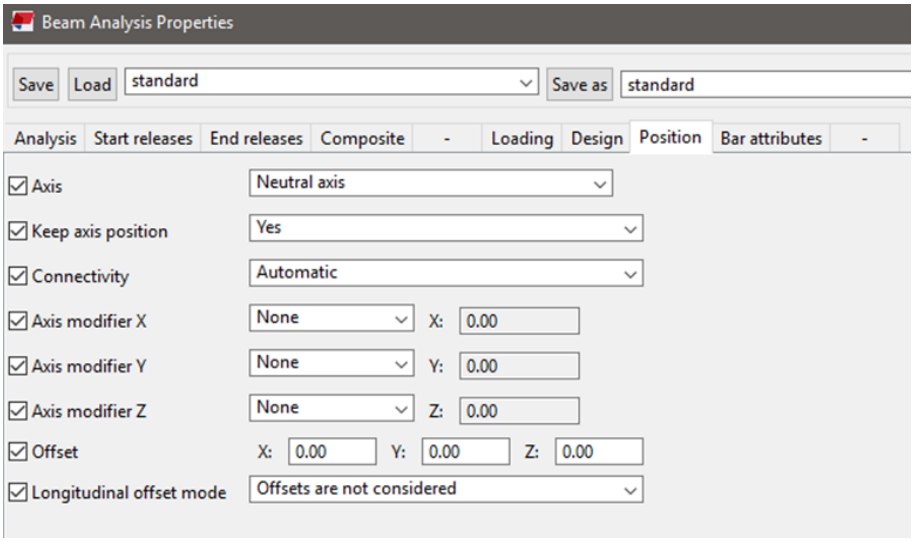


Setting analysis properties of parts

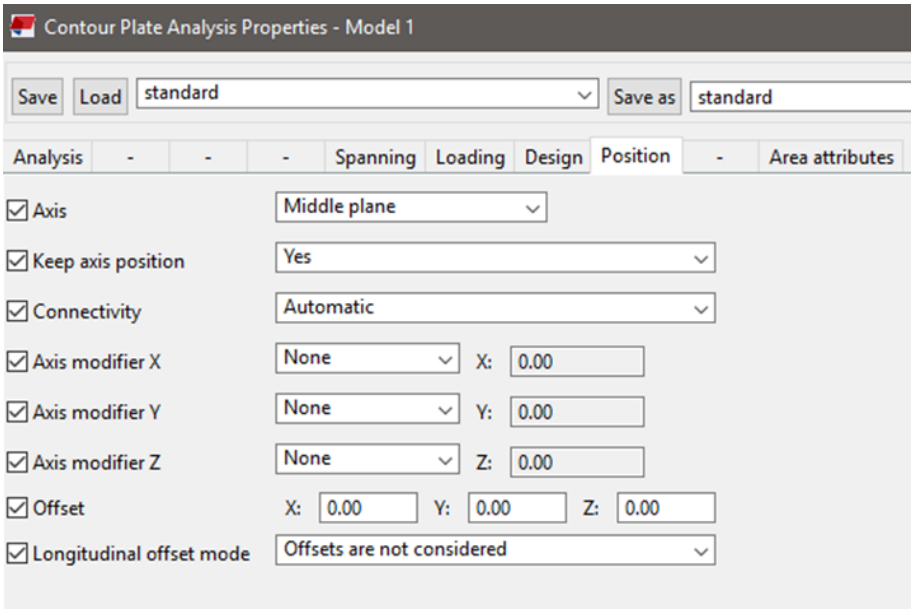
You can access the part analysis properties int the menu below:



For beams and columns set these parameters on the *Position* tab:



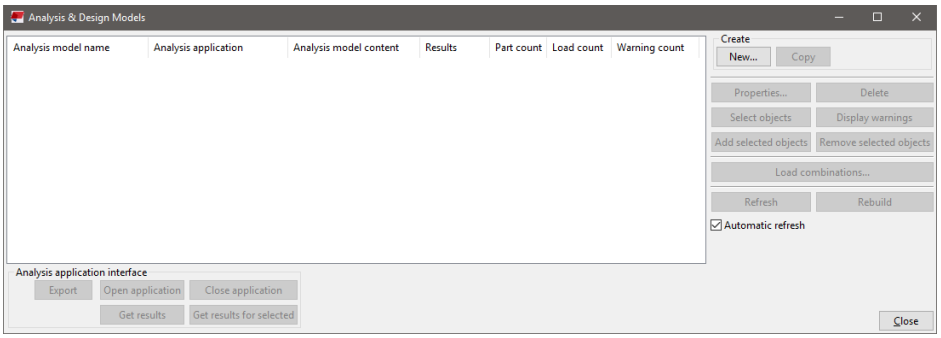
For contour plates, slabs and panels set these parameters on the *Position* tab:



Creating the Analysis Model

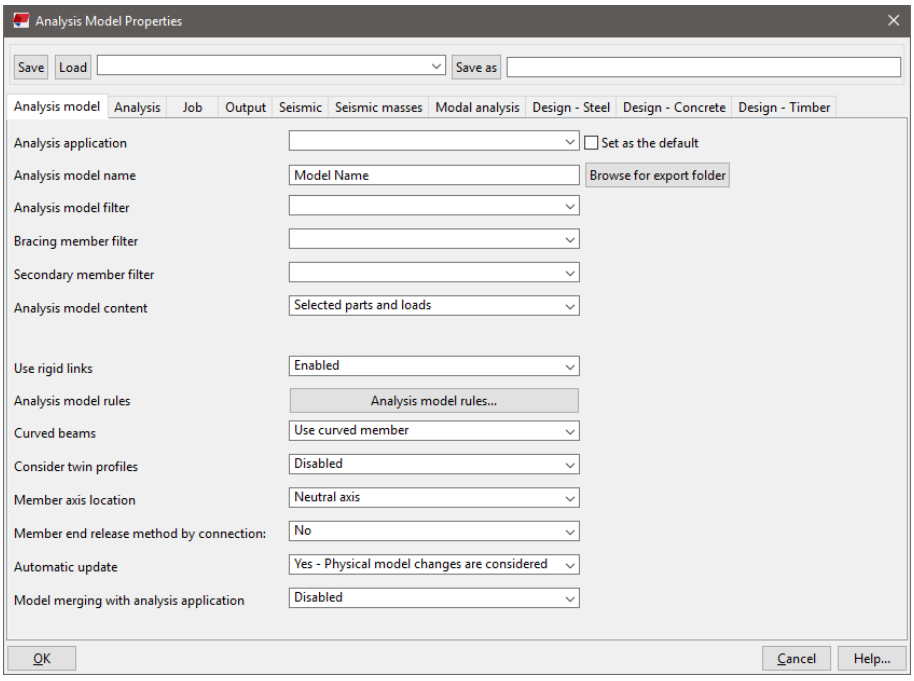
After setting the analysis properties of every part you can create the analysis model.

1. Open the Analysis & Design models dialog box.



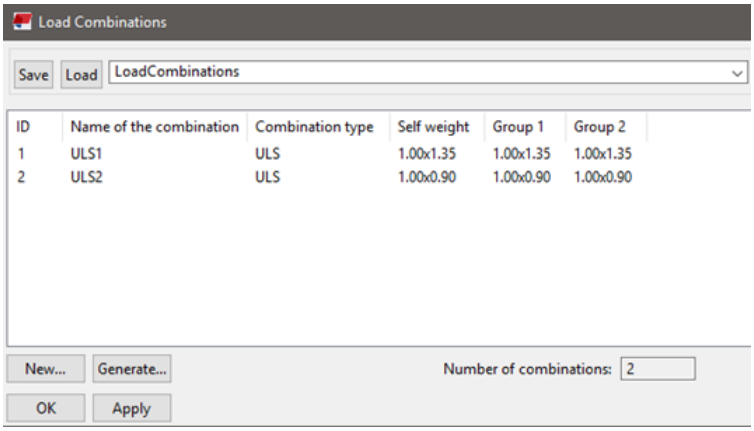
2. Press *New...* to create a new analysis model with the parameters below:

Analysis Model Properties

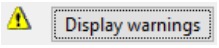


3. Select the parts and loads you would like to add to the analysis model and click on the *Add selected objects* button. To remove elements from the analysis model select them and click on *Remove selected objects*.

4. Press *Load combinations...* to configure the load combinations.

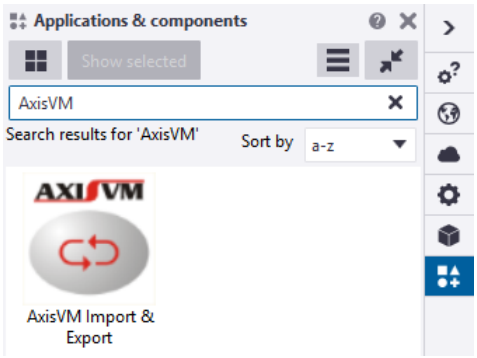


5. Check if there are any warnings indicated by Tekla Structures and try to resolve them.



Exporting the Analysis Model to TAE file

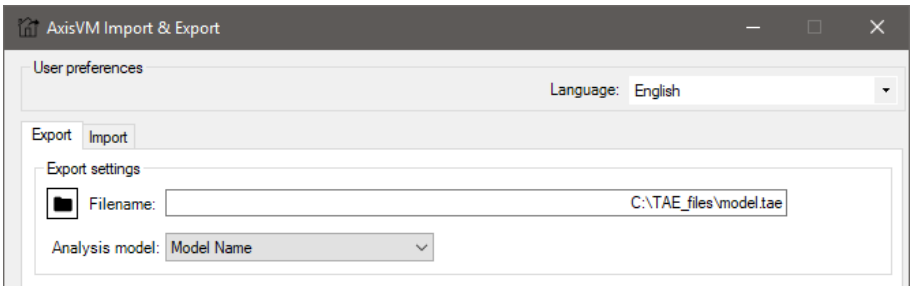
1. Search for "AxisVM" in the *Applications & components* catalogue and open the AXISVM *Import & Export* plugin by double clicking on it.



2. Select where to save the TAE file and choose a name for it. You also have to choose which analysis model you wish to export.

Press *OK* to start the export process.

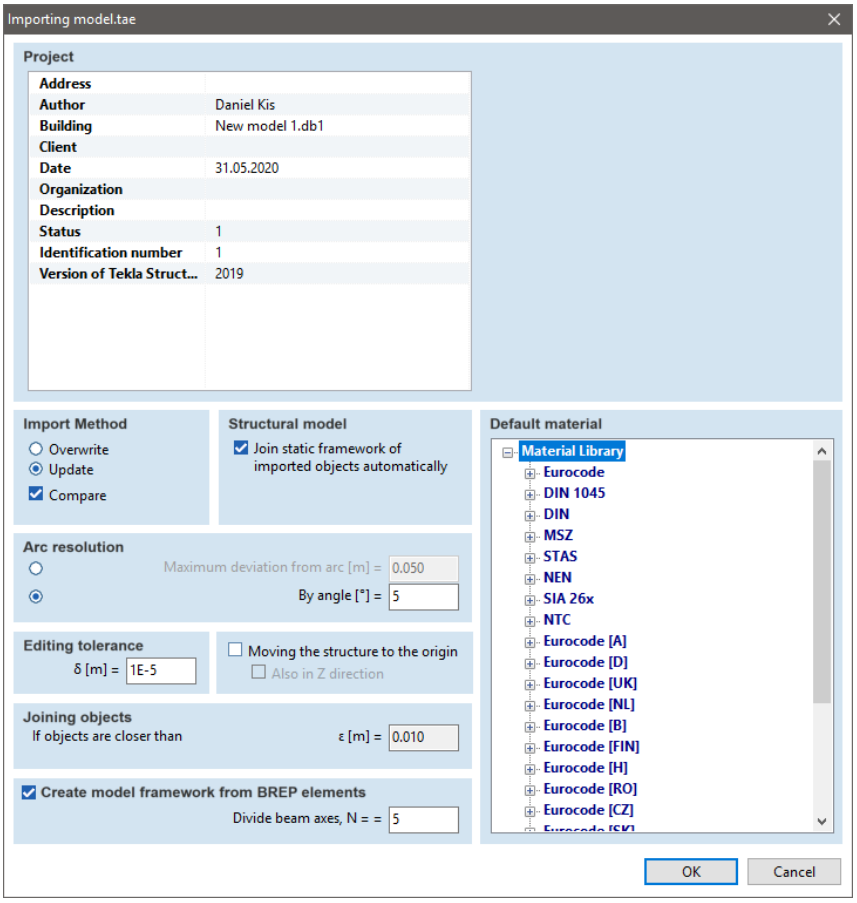
Export the model



If you see the message that *Exporting data succeeded*, the operation was successful.
The resulting file with *.tae* extension can be either dropped onto the user interface of AXISVM or imported via *File / Import*.

Importing TAE file to AXISVM

When importing the file into AXISVM a dialog will pop up:



Project The project info as defined in Tekla.

Import method If the AXISVM model is not empty, two options are available to overwrite the model or update it.

Structural model If this option is activated AXISVM changes the raw geometry data to build a proper structural framework. Colliding beams are connected with a rigid body, neighbouring domains are joined, beams running on a domain are converted to ribs. Some of these calculations require considerable resources and may slow down the import process.

Arc resolution Certain geometry calculations require arcs to be converted into polylines. The conversion criterion can be set here.

Editing tolerance Tolerance used when calculating geometry.

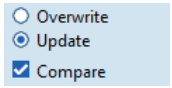
Joining objects To improve the efficiency of collision detection bounding boxes of objects are inflated a bit. This tolerance value can be set here. Active only if the checkbox on panel *Structural model* is checked.

Default material Users can select the material assigned to entities for which Tekla supplied no information.

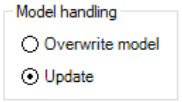
3.1.9.1.3 Workflow between AXISVM and Tekla Structures

AXISVM and Tekla Structures can share a workflow between each other. This is a comparison based export-import. With this, engineers working on the same project can import new or modified entities without affecting unchanged elements.
The workflow process can be started both from AXISVM and Tekla Structures.
For the comparison to happen you have to choose *Update* on the form and check the checkbox shown below:

AXISVM:



Tekla:



Importing to AXISVM

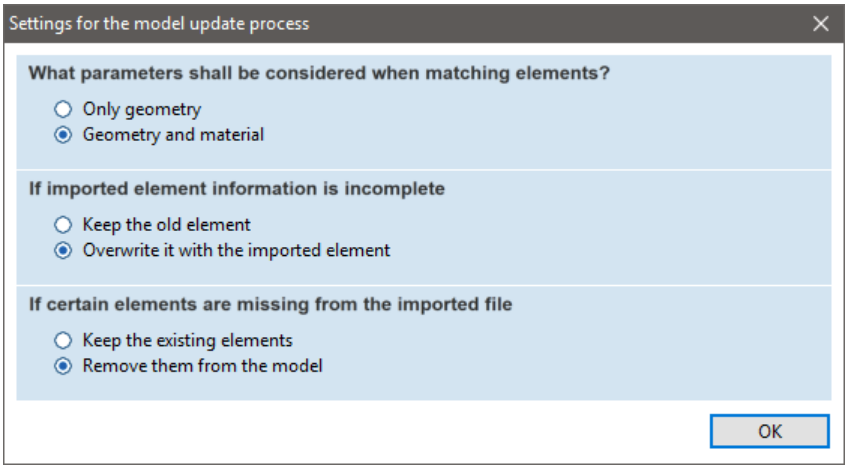
When importing a Tekla model to AXISVM, three basic scenarios are handled:

1. Element to be imported cannot be found in the analysis model, so it can be simply processed and imported.
2. One or more items of the actual model cannot be found in the file to be imported. In this case, users can select to either delete these elements or to ignore and leave them intact.
3. The item to be imported can be found in the actual model. In this case, we perform a comparison based on
    a) the material of the elements
    b) the statical model of the elements
    c) the profile or thickness of the elements.

If after this we find them equal, the imported item is skipped. However, if they are found to be different, arriving item is processed, and the former one is deleted. When - for any reason - we cannot perform a successful comparison, the user must decide whether to keep the existing item or import the new one.

These preferences can be adjusted prior to the actual import process on the form below:

AXISVM dialog

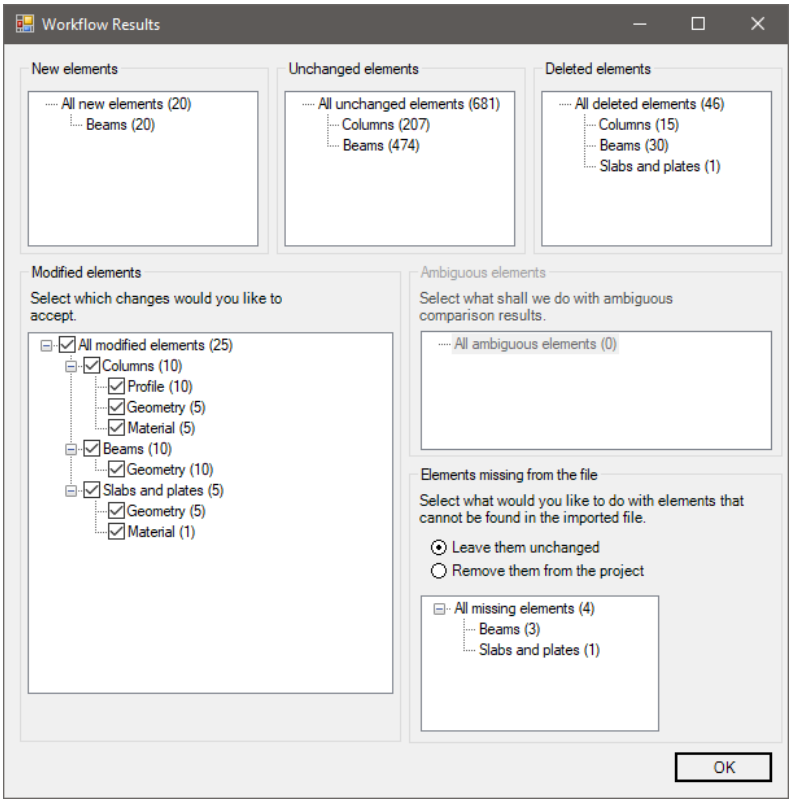


Importing to Tekla Structures

When importing an AXISVM model to Tekla Structures we perform a comparison based on:

a) the material of the elements
b) the statical model of the elements
c) the profile and thickness of the elements
d) spring characteristics

After the comparison you can review the comparison results and customize the actions to perform.



New elements This list shows the elements created in AXISVM since the last export. These elements will always get created after the import.

Unchanged elements No changes were done to these elements. We are leaving them as they are.

Modified elements The elements which got modified in AXISVM are listed here. Every change is accepted by default. Uncheck the changes you do not wish to perform.
Selecting a node will highlight the related elements.

Deleted elements These elements got deleted in AXISVM since the last export. They are getting deleted after the import process.

Missing elements These elements can't be found in the imported file but present in the Tekla model. You can select to either delete these elements or to ignore them and leave them intact.

Ambiguous elements If an element has incomplete information or we can't compare it for some reason, then it will appear here. You can decide if you want these elements to overwrite the current ones or you want to keep the old elements.

3.1.9.2 One-way connection for Tekla Structures versions before 2021

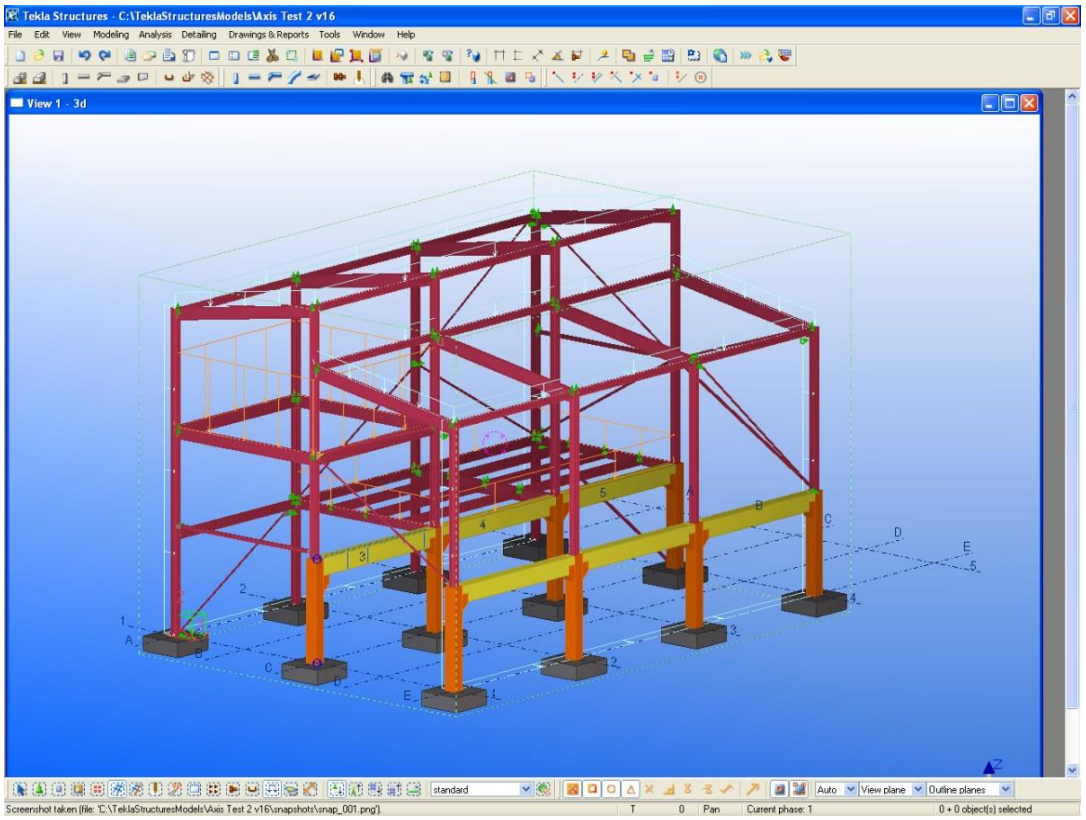
The connection between the two software is made through a COM server enabled to run AXISVM. To make the connection work first the COM server must be registered within the operating system (in the Registry) then Tekla Structures must be notified that a compatible server is available.
AXISVM setup automatically performs these registering operations, however if Tekla Structures is not installed the second registration cannot be completed. Therefore, after installing Tekla Structures the registration has to be started again by running two batch files from the AXISVM program folder:

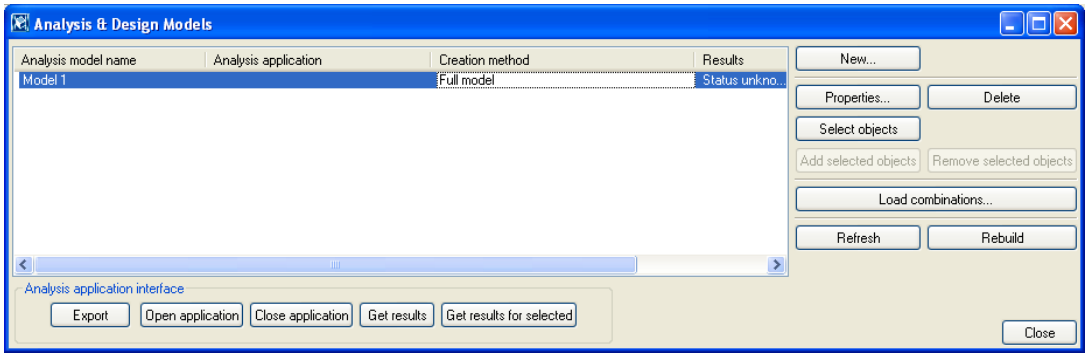
```
!REGISTER_AXISVM_X64.BAT
!REGISTER_TEKLA_X64.BAT
```

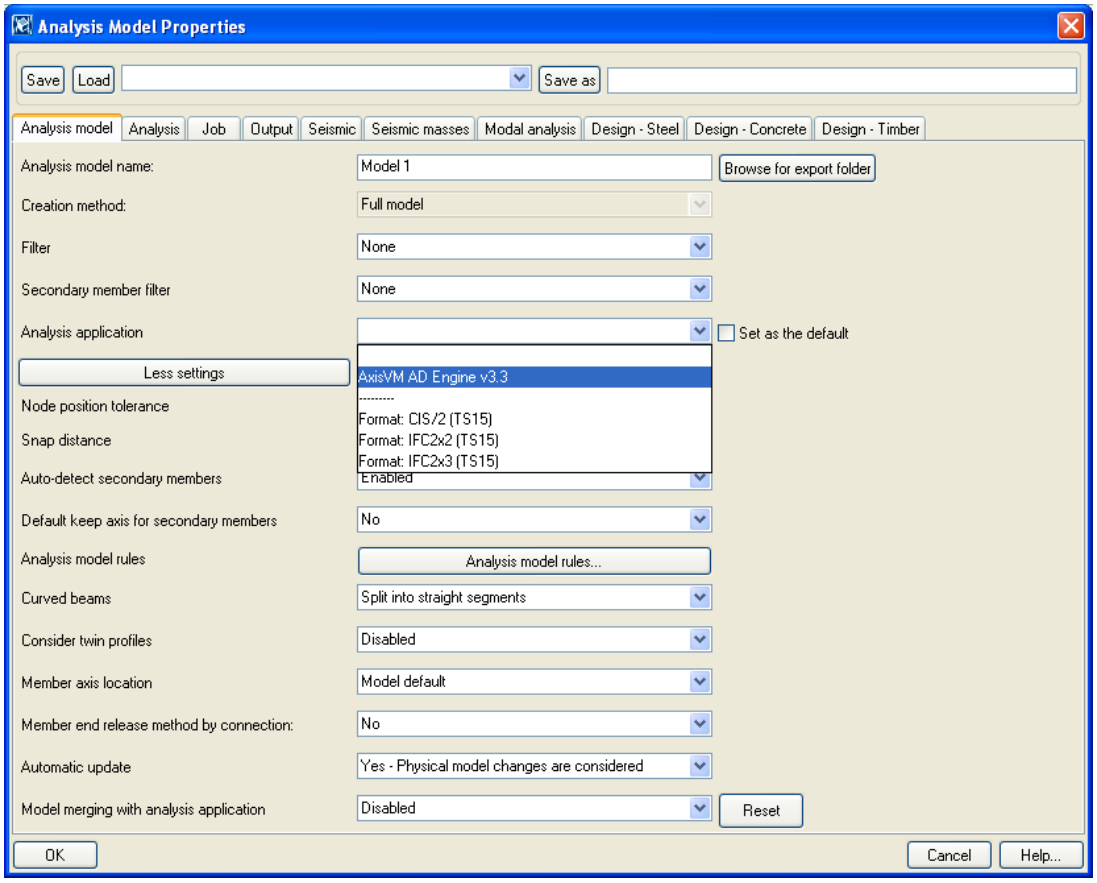
If connections fail any time it is recommended to run this registration again.

Connection

After a successful registration the model built in Tekla Structures can be transferred to AXISVM in the following way: click *Analysis & Design models...* in the *Analysis* menu then click the *Properties* button to set AXISVM AD Engine as the *Analysis engine*.

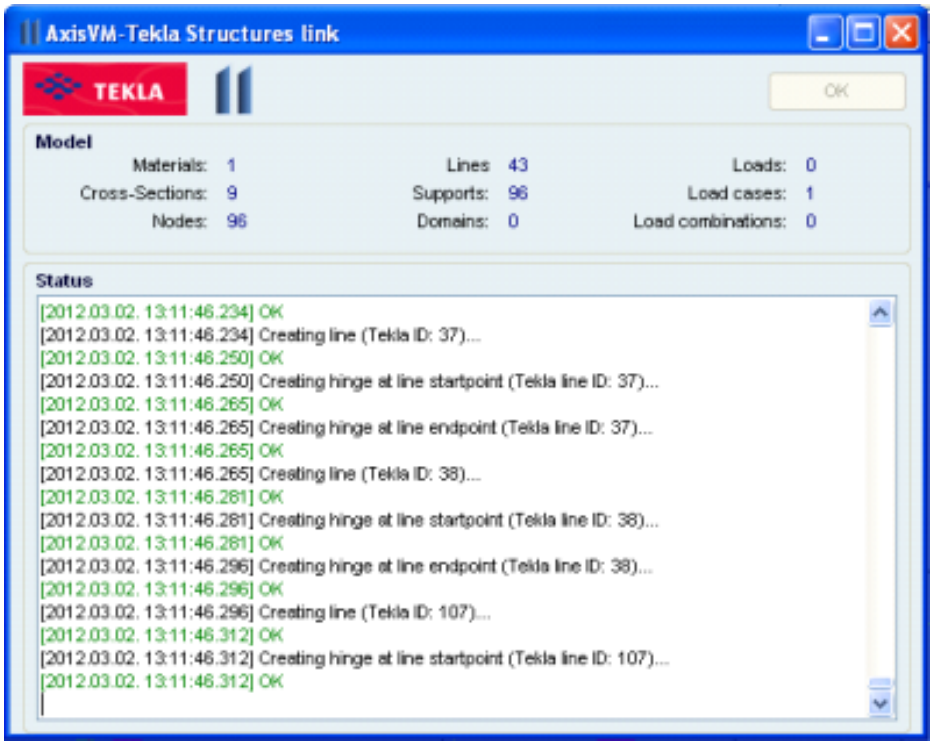




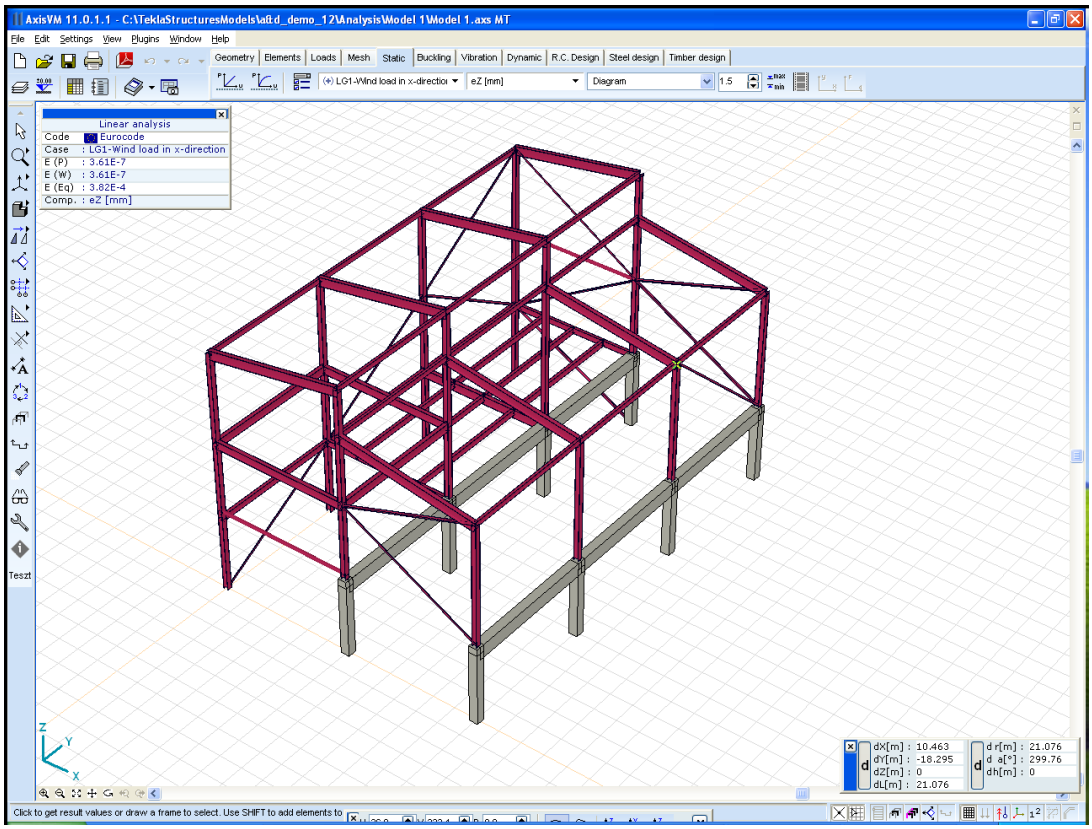


If AXISVM AD Engine does not appear in the dropdown list the registration was not successful and has to be repeated.

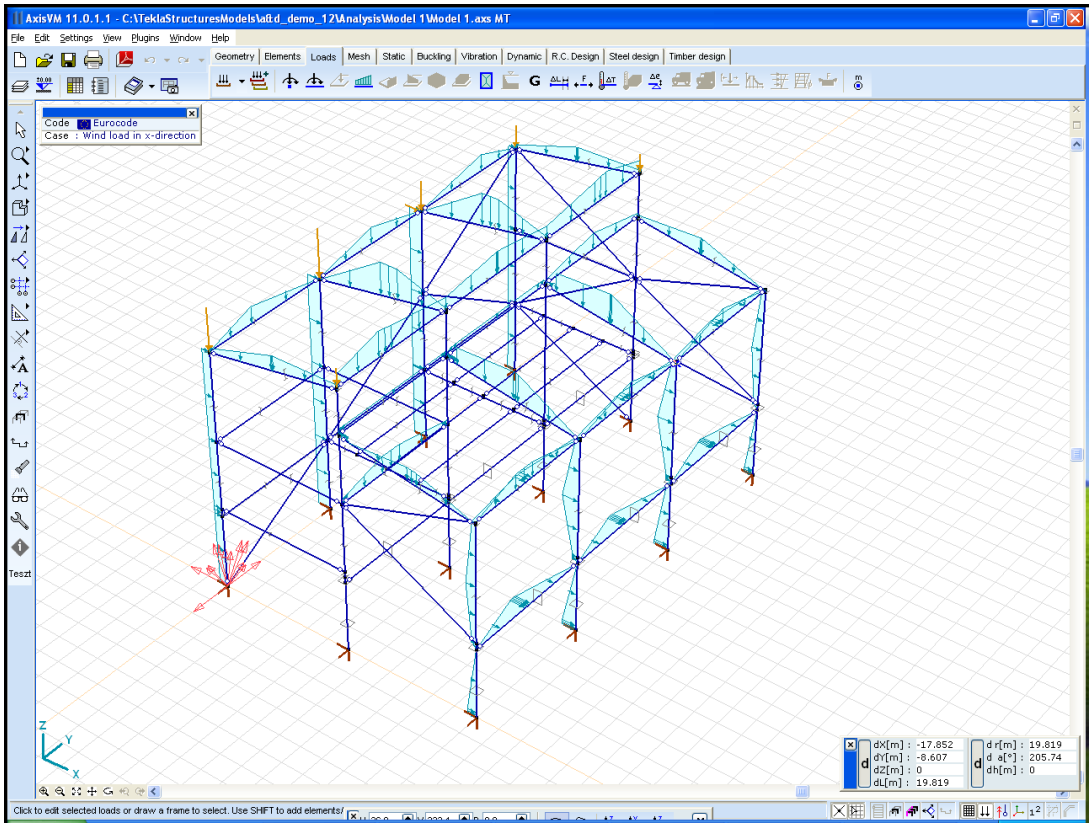
Getting back to the *Analysis & Design models* dialog click *Run* to start the transfer of the model. The process status is displayed in dialog. If the transfer is completed successfully click the OK button to see the model in AXISVM.



The model transferred to AXISVM:

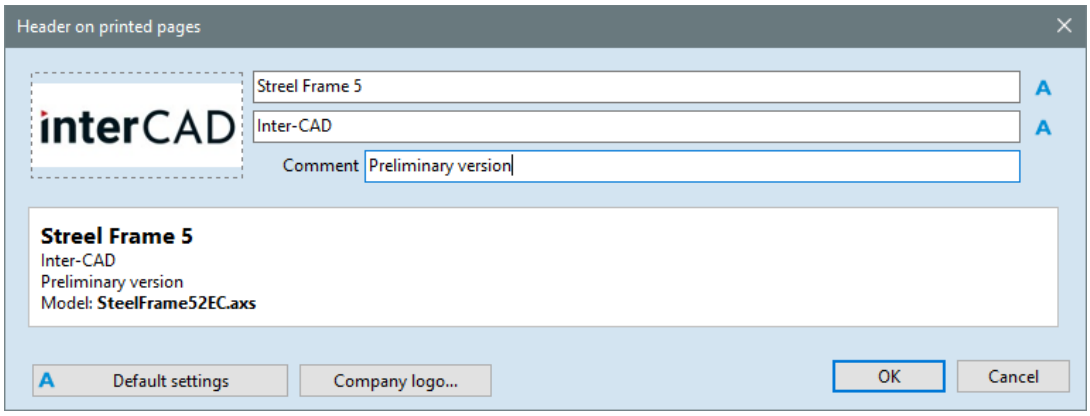


Loads and load cases specified in Tekla Structures are also converted.



3.1.10 Page header

Lets you specify a header text (two lines), which contains the name of the project and designer. It will appear on the top of every printed page. An additional comment line can be added.



If a page header logo was selected for printing drawings, tables and reports, the logo also appears here. Click on the logo to change it or try *Settings / Preferences / Report / Company logo / Settings...*
See 3.3.11.15 Report

3.1.11 Print setup

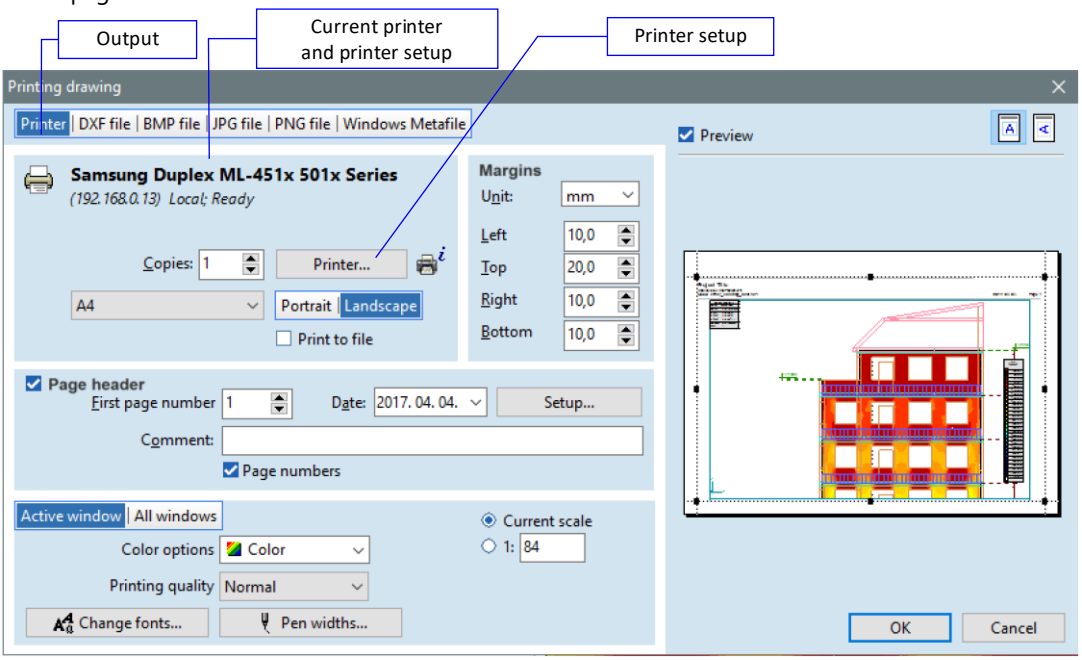
Allows setting the parameters of the default printer.
This is a standard Windows dialog therefore its language corresponds with the language of the installed operating system.

3.1.12 Print

[Ctrl]+ [P]

Lets you print the model according to the current display settings. Allows the setup of the printer, and of the page.

Printing drawing



Send To

Lets you send the output directly to the printer or to a graphics file (DXF, BMP, JPG, PNG or Windows Metafile [WMF/EMF]).

Printer

Lets you select and setup the printer.

If *Print to file* is selected, the data sent to the printer will be stored in the `Name.prn` file, where `Name` is a file name to be entered.

You can set the number of copies required.

Click on the printer name or on the *Printer...* button to invoke the standard Windows Printer Setup dialog where you can change printer and printer settings in detail.

Scale

Lets you set the scale of the drawing to print. In case of perspective or rendered view or if the output is sent to a Windows Metafile the scale cannot be set.

Margins (Printer/DXF)

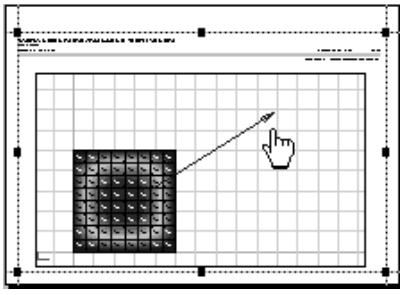
Lets you set the size and the units of the page margins. You can also drag margin lines within the preview area by their corner and midside handles.

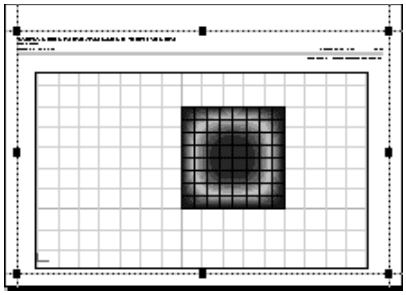
Bitmap size (BMP, JPG)

Lets you set the bitmap size in pixels, inch, mm or cm and bitmap resolution in dpi (dots per inch).

Preview

Lets you view the printed image prior printing. If you select *Printer* as a target the graphics cursor turns to a hand whenever it enters the preview area. By pressing the left mouse button and moving the mouse you can specify an additional panning which will affect the printed output only.





Page header

Lets you set the date and remark that will appear on each page, and the starting number for the page numbering. If the *Page numbers* checkbox is turned off a blank space will appear after *Page* allowing handwritten page numbers. *Setup...* displays the *Page header* dialog (see **3.1.10 Page header**) where you can change the company logo.

Orientation

Lets you set the orientation of the page.

Color options

Lets you select printing in grayscale, color, or black and white.

If your printer cannot print in color you may get different results in the first two cases. If you select *Grayscale* the output will be converted to grayscale using an internal grayscale palette of AXISVM. If you select *Color* the conversion to grayscale will be performed by the Windows printer driver. Try both to find which works better for you. When black and white printing is selected, all entities are printed in black.

Paper size

Lets you set the size of the paper.

Change fonts

Lets you select fonts to be used in printing and set the font size.

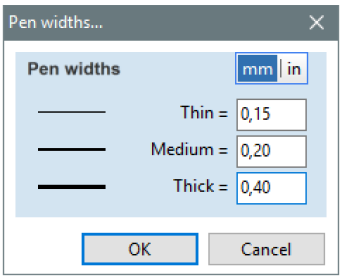
Pen widths

Sets the size of the pens for printing.

Thick lines are used for drawing supports and rigid elements.

Medium lines are used for isolines and section line.

Thin lines are used for elements and geometry and other entities.



Windows to Print

Lets you print either the active window or all windows displayed.

Printing to file

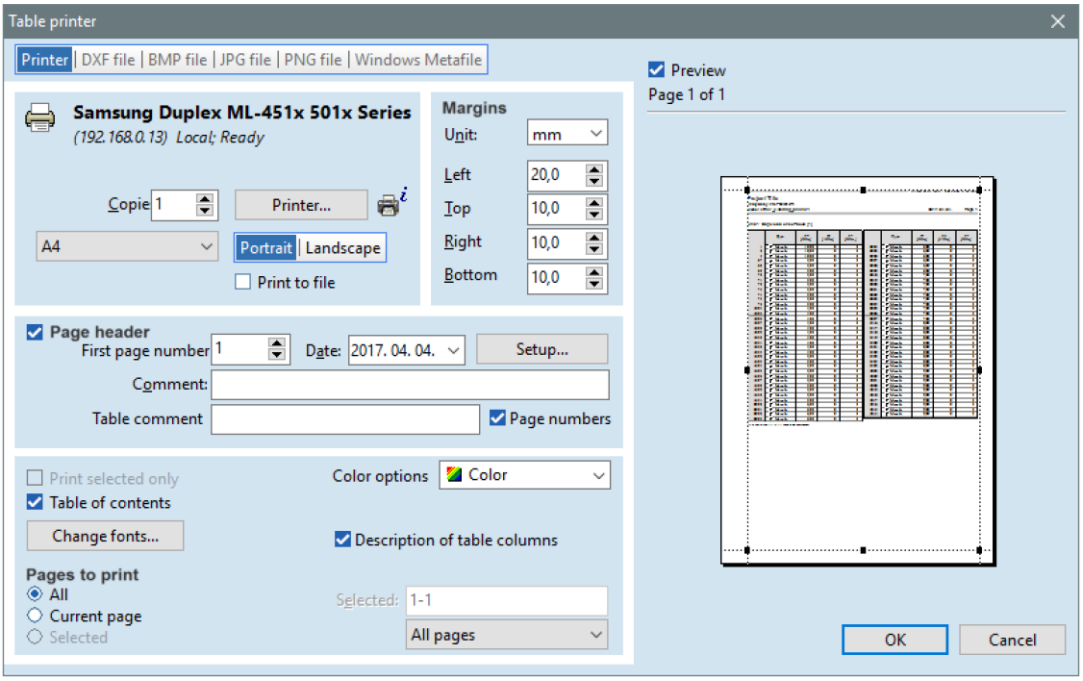
When *Print to File* is selected the printing is redirected to a file, `name.prn` that you can print anytime later.

If the file `name.prn` already exists, you can add your printing to it, or overwrite it.

Printing table

When printing from the table browser, you can set the pages (all / even / odd) of all / current / selected pages you want to print.

Example: Entering 1, 3, 7-10, 20-18 in the Selected field the 1st, 3rd, 7th, 8th, 9th, 10th, 20th, 19th, and 18th page will be printed in this order.



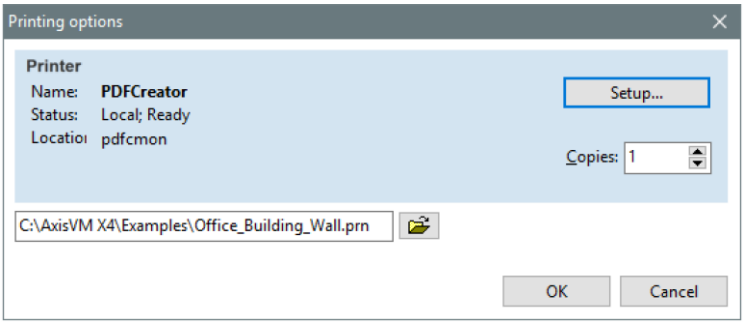
Print selected only
: Enabled only when printing a report. If this option is checked only report items selected in the Report Maker will be printed. If unchecked, the entire report will be printed.

Table of contents
: Printing table of contents at the beginning of the report is optional.

Description of table columns
: If this option is checked a summary appears at the end of each table, describing the meaning of column headers.

3.1.13 Printing from file

You can print the prn file you created from the following window.

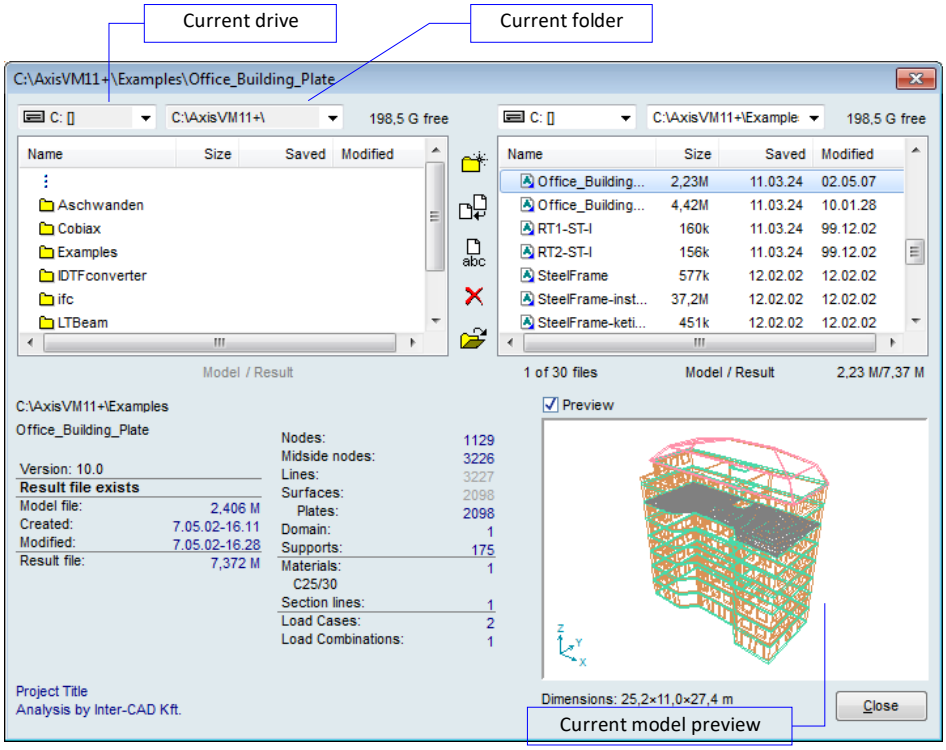


You can print more than one prn file at a time. You can set the printing order with the up/down arrows in the right of the file list box, or dragging the file names to a new position with the mouse.

3.1.14 Model Library

The *File/Model Library* command lets you preview, get information and manage your model files.
As in *Open* and *Save as* dialog windows the standard file access dialog box items are displayed, but in the list box you can select multiple files.

The AXISVM model files are marked with the symbol. If a model has a result file the symbol has a blue right-bottom corner,



New
Creates a new sub-folder in the current folder with the name you enter.

Copy
Copies the selected files to a different folder. You can specify whether to copy the result files or not.

Rename/Move
Renames the selected files in the current folders or moves them into a different folder.

Delete
Deletes the selected files from the current folders. You can specify to delete only the result files or all.

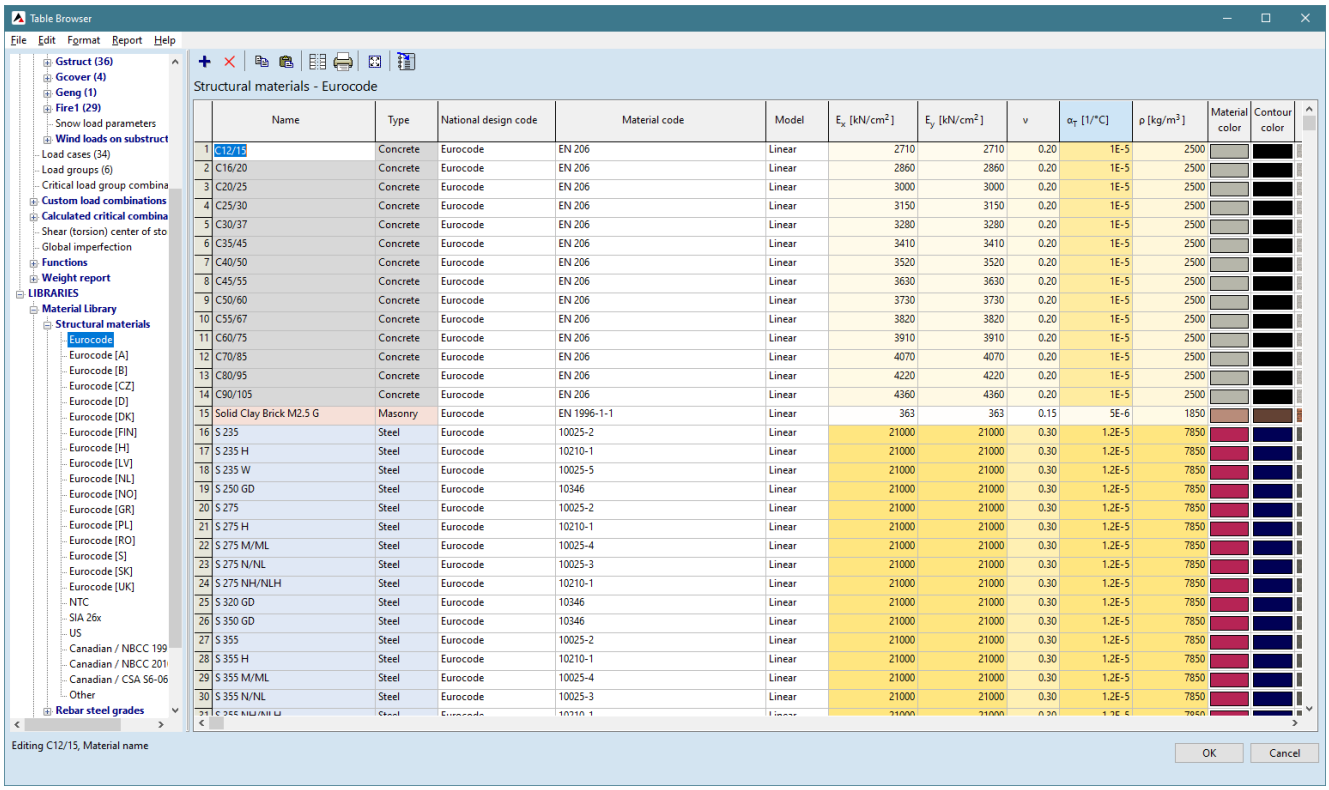
Open
Opens the selected file for editing.

AXISVM files are marked with . If a result file is available, the bottom right corner of the icon is blue.

Preview
Shows the model wireframe in front, side, top view or in perspective depending on the model dimensions. Model information is also displayed in a list.

Close
Quits the Model Library.

3.1.15 Material Library



The material library window can also be opened using the Table Browser icon and by selecting Libraries/Material Library. See **4.9.10 Line elements, 4.9.26 Creating model framework from an architectural model**

See the detailed description of the Table Browser in section **2.9.**

Properties of materials

This table contains the properties of materials often used in civil engineering to the Eurocode, SIA and Italian codes. You can add, modify, or delete existing material data. In case of entering a new material with an existing name it will be added as **materialname_number**. These materials can be used in any model.

☞ **Changes in the material library does not reflect in models using the modified material.**

When entering a new material, the following dialog is displayed:

Define new material

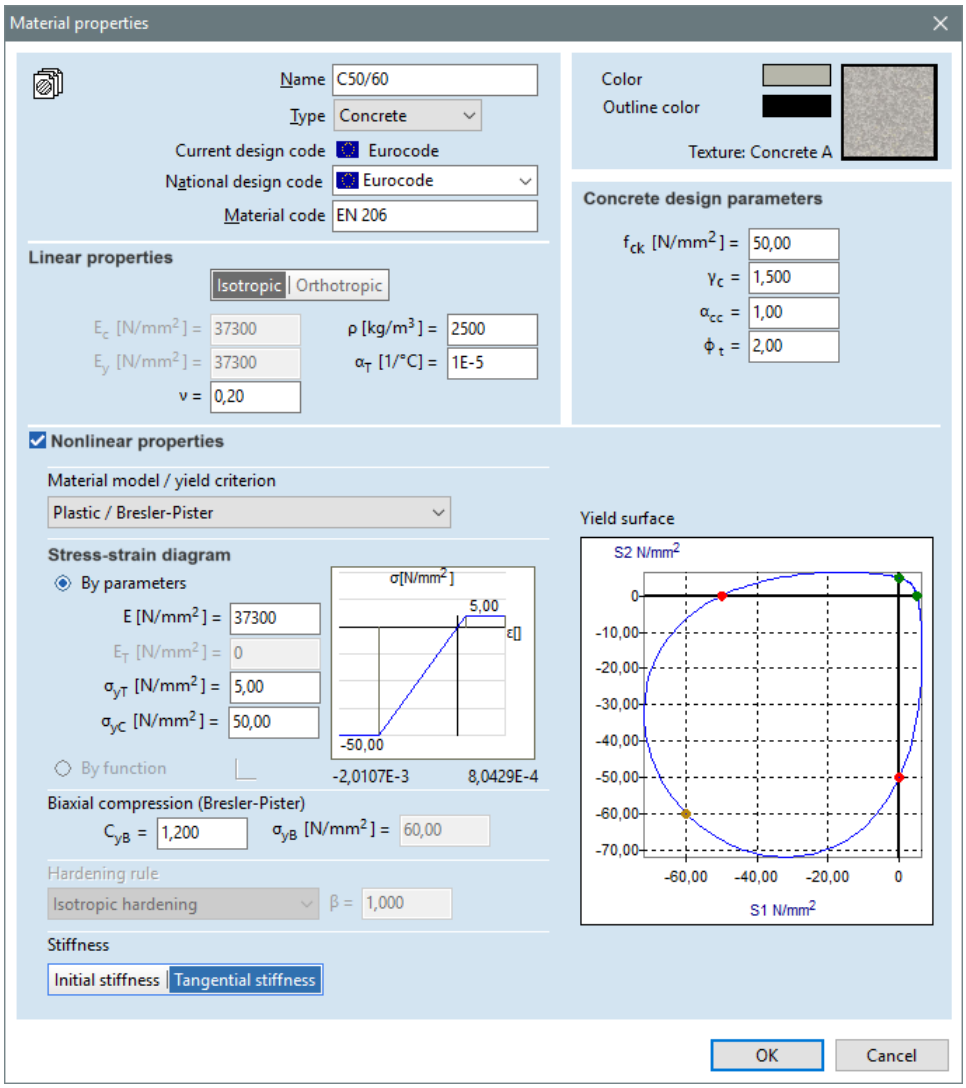
[Ctrl+Ins]

Change material properties

Definig new material or clicking to a non-editable column (eg. national design code, type) a dialog appears, in which all material properties, calculation and design parameters can be defined or changed. The fields containing the basic properties independent of the design code can be edited in the table.

When a material with a name identical to one existing is entered an index is attached to the name (*name_index*) to differentiate from the existing one.

If no texture was assigned to the material click the sample rectangle to select one from the library. See **2.16.4 Display mode**



Material Properties

For each material the following properties are stored:

Material type: [Steel, concrete, timber, aluminium, masonry, other]
Design code, material code
Material name
Fill color on the screen
Contour line color on the screen,
Texture

Linear properties

The material model can be isotropic or orthotropic.

E_x	[kN/cm2]	Young's modulus of elasticity in the local *x* direction
E_y	[kN/cm2]	Young's modulus of elasticity in the local *y* direction
ν	-	Poisson's ratio
α_T	[1/°C]	Thermal expansion coefficient
ρ	[kg/m3]	Mass density

In case of timber materials:

ρ is the air dry mass density (12% humidity) and the modulus of elasticity *E* is based on bending test results. The effect of time (relaxation) is not taken into account.

Calculation of further material properties:

$$E_z = \max\{E_x, E_y\}; \quad \nu_{ij} = \begin{cases} \nu & \text{if } E_i \geq E_j \\ \nu \frac{E_i}{E_j} & \text{if } E_i < E_j \end{cases}; \quad G_{ij} = \frac{E_i E_j}{E_i + E_j + 2\nu_{ij} E_j}$$

where $ij = \{xy, xz, yz\}$

Timber material properties

In case of Eurocode, the program modifies the moduli of elasticity of timber line and surface elements (if their material was selected from a Eurocode database) according to the following table:

Analysis type	*Moduli of elasticity*
Linear analysis Only materially nonlinear static Buckling	$E_x = \frac{E_{0,mean}}{1+k_{def}}$ $E_y = \frac{E_{90,mean}}{1+k_{def}}$ $E_z = \max\{E_x, E_y\}$ $G_{xy} = G_{xz} = G_{yz} = \frac{G_{mean}}{1+k_{def}}$
Only geometrically nonlinear static Materially and geometrically nonlinear static	$E_x = \frac{E_{0,mean}}{\gamma_M}$ $E_y = \frac{E_{90,mean}}{\gamma_M}$ $E_z = \max\{E_x, E_y\}$ $G_{xy} = G_{xz} = G_{yz} = \frac{G_{mean}}{\gamma_M}$
Vibration Dynamic analysis	$E_x = E_{0,mean}$ $E_y = E_{90,mean}$ $E_z = \max\{E_x, E_y\}$ $G_{xy} = G_{xz} = G_{yz} = G_{mean}$

k_{def} is a parameter assignable to line elements **(4.9.10 Line elements)** and domains **(4.9.6 Domain)**

In case of SIA, the program modifies the moduli of elasticity of timber line and surface elements (if their material was selected from a SIA database) according to the following:

$$E_x = E_{0,mean}\,\eta_w;\; E_y = E_{90,mean}\,\eta_w;\; E_z = \max\{E_x, E_y\}$$
$$G_{xy} = G_{xz} = G_{yz} = G_{mean}\,\eta_w$$

where η_w depends on the *moisture content* which is a parameter assignable to line elements and domains.

Nonlinear properties

Material model used in the software is valid only in case of small strains.

Material model / yield criterion	*Material type*		
	Steel	*Concrete, masonry*	*Other*
NL elastic / Strain energy based	✓	—	✓
Plastic / von Mises	✓	—	✓
Plastic / Bresler-Pister	—	✓	✓

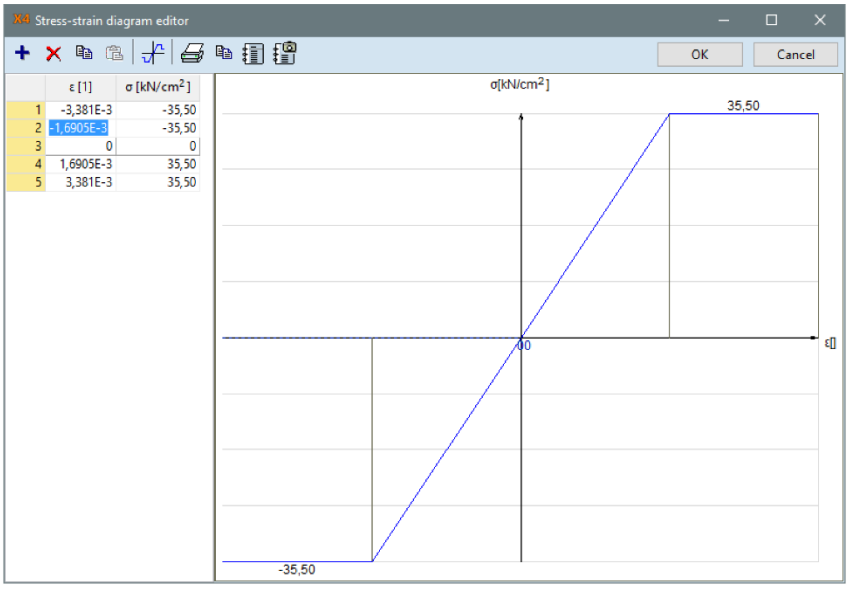
Stress-strain diagram

By parameters

E [kN/cm^2]	Young's modulus of elasticity for nonlinear analysis (rise of the initial section of the σ-ε diagram)
E_T [kN/cm^2]	Young's modulus of elasticity for nonlinear analysis (rise of the tangential section of the **σ-ε** diagram)
σ_{yT} [kN/cm^2]	Tensile yield stress
σ_{yC} [kN/cm^2]	Compressive yield stress

By function

Stress-strain diagram editor



Most of the toolbar functions are the same as in the time history diagram editor. See **4.10.31 Dynamic loads (for time-history analysis) – DYN module**

Symmetrical function

If this option is activated defining the positive part of the function also defines the negative part.

Biaxial compression

σ_{yB} [kN/cm^2] Biaxial compressive yield stress. $\sigma_{yB} = C_{yB} \cdot \sigma_{yC}$

C_{yB} [] Biaxial compression factor

Restrictions on parameters

Material model / yield criterion	E_T	σ_{yC}	C_{yB}	*σ-ε diagram*
NL elastic / Strain energy based	$E_T \geq 0$	$\sigma_{yC} \geq 0$	N.A.	arbitrary
Plastic / von Mises	$E_T \geq 0$	$\sigma_{yC} = \sigma_{yT}$	N.A.	symmetrical
Plastic / Bresler-Pister	$E_T = 0$	$\sigma_{yC} \geq \sigma_{yT}$	$1 \leq C_{yB} \leq 1.2$	N.A.

N.A. – not available

☞ **The σ-ε diagram must be at least bilinear both on the tension and compression side. The function must be monotonically increasing, and its slope must be monotonically decreasing.**

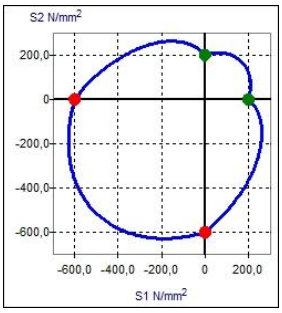
Yield surface visualization

The diagram shows the section cut of the yield surface by the plane of the principal stresses S1-S2. This boundary curve belongs to plane stress state.

NL elastic / Strain energy based

This material model has nonlinear elastic behaviour. It means that the point representing the state of the material moves along the σ-ε curve when load increase or decrease. No irreversible deformation.

Asymmetric stress strain behavior is available.



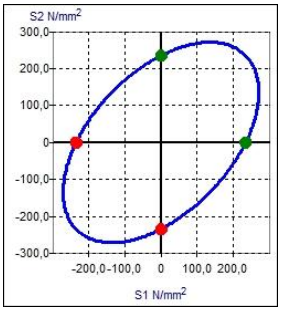
$\sigma_{yT} = 200$ MPa, $\sigma_{yC} = 600$ MPa

Plastic behaviour

Increasing load moves the point representing the state of the material along the σ-ε curve, decreasing load moves it parallel with the initial section of the curve.

Plastic / von Mises

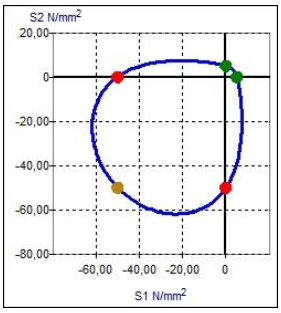
This material model is recommended for steel material. Only symmetric stress-strain behaviour is available for this material model. [35] **https://en.wikipedia.org/wiki/Von_Mises_yield_criterion**



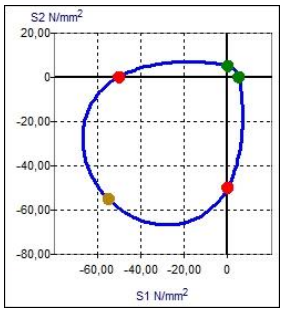
$\sigma_{yT} = 200$ MPa , $\sigma_{yC} = 200$ MPa

Plastic / Bresler-Pister

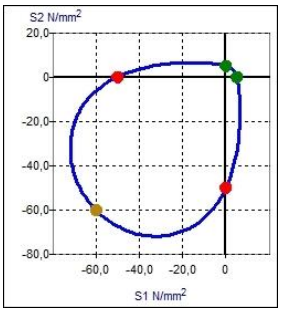
This material model is recommended for concrete and masonry materials, because it can model the low yield strength for tension and the high yield strength for uniaxial and biaxial compression simultaneously. [36] **https://en.wikipedia.org/wiki/Bresler_Pister_yield_criterion**



$\sigma_{yT} = 5$ MPa , $\sigma_{yC} = 50$ MPa
$C_{yB} = 1.0$
$\sigma_{yB} = C_{yB} \cdot \sigma_{yC} = 50$ MPa



$\sigma_{yT} = 5$ MPa , $\sigma_{yC} = 50$ MPa
$C_{yB} = 1.1$
$\sigma_{yB} = C_{yB} \cdot \sigma_{yC} = 55$ MPa



$\sigma_{yT} = 5$ MPa , $\sigma_{yC} = 50$ MPa
$C_{yB} = 1.2$
$\sigma_{yB} = C_{yB} \cdot \sigma_{yC} = 60$ MPa

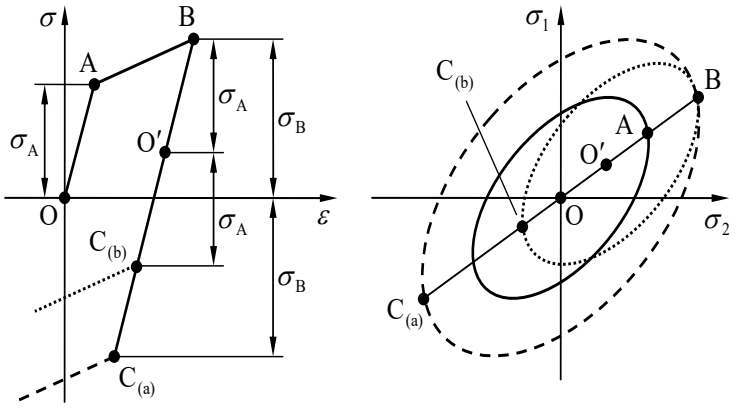
Stiffness

Initial stiffness — The materially linear initial stiffness matrix is used for the evaluation of the global stiffness matrix. The calculation is slower (more iterations are necessary) but the convergence is more stable.

Tangent stiffness — The materially nonlinear tangent stiffness matrix is used for the evaluation of the global stiffness matrix. The calculation is faster (less iterations are necessary) but if the increments are not small enough convergence problems may occur.

Strain hardening

The strain hardening could be:
(a) Isotropic: $\beta = 1$ (b) Kinematic: $\beta = 0$



In case of kinematic hardening the size of the yield surface does not change but its centre moves from the origin of the stress space to the O' location. In such a case the relative stresses $\sigma_{r,i} = \sigma_i - \sigma_{O',i}$ with respect to the current origin are substituted into the equation of the effective stress calculated on the basis of the yield criterion. See **6.1.23 Truss/beam/rib stresses** and **6.1.24 Surface element stresses** In case of isotropic hardening $\sigma_{O',i} = 0$ and $\sigma_{r,i} = \sigma_i$.

Design Parameters Design parameters depend on the material type and the design code.

Steel

EC, SIA 26x, Italian	f_y	Yield stress
	f_u	Ultimate stress
	f_y*	Yield stress 40mm < t < 100mm)
	f_u*	Ultimate stress (40mm < t < 100mm)

Concrete

EC, Italian	f_{ck}	Characteristic compressive cylinder strength at 28 days
	γ_c	Partial factor
	α_{cc}	Concrete strength reduction factor for sustained loading
	Φ_t	Creep factor
SIA 26x	f_{ck}	Characteristic compressive cylinder strength at 28 days
	γ_c	Partial factor
	Φ_t	Creep factor

Timber

EC	$f_{m,k}$	Characteristic bending strength
	$f_{t,0,k}$	Characteristic tensile strength parallel to grain
	$f_{t,90,k}$	Characteristic tensile strength perpendicular to grain
	$f_{c,0,k}$	Characteristic compression strength parallel to grain
	$f_{c,90,k,y}$	Characteristic compression strength perpendicular to grain *(y)* (for solid and Glulam timber $f_{c,90,k,y} = f_{c,90,k,z} = f_{c,90,k}$)
	$f_{c,90,k,z}$	Characteristic compression strength perpendicular to grain *(z)* (for solid and Glulam timber $f_{c,90,k,y} = f_{c,90,k,z} = f_{c,90,k}$)
	$f_{v,k,y}$	Characteristic shear strength *(y)* (for solid and Glulam timber $f_{v,k,y} = f_{v,k,z} = f_{v,k}$)
	$f_{v,k,z}$	Characteristic shear strength *(z)* (for solid and Glulam timber $f_{v,k,y} = f_{v,k,z} = f_{v,k}$)
	$E_{0,mean}$	Mean Young's modulus of elasticity parallel to grain *(x)*
	$E_{90,mean}$	Mean Young's modulus of elasticity perpendicular to grain *(y)*
	$E_{0.05}$	5% modulus of elasticity parallel to grain *(x)*
	G_{mean}	Mean shear modulus
	ρ_k	Characteristic density
	ρ_{mean}	Mean density
	γ_M	Partial factor of the material
	s	Size effect exponent (for LVL materials)

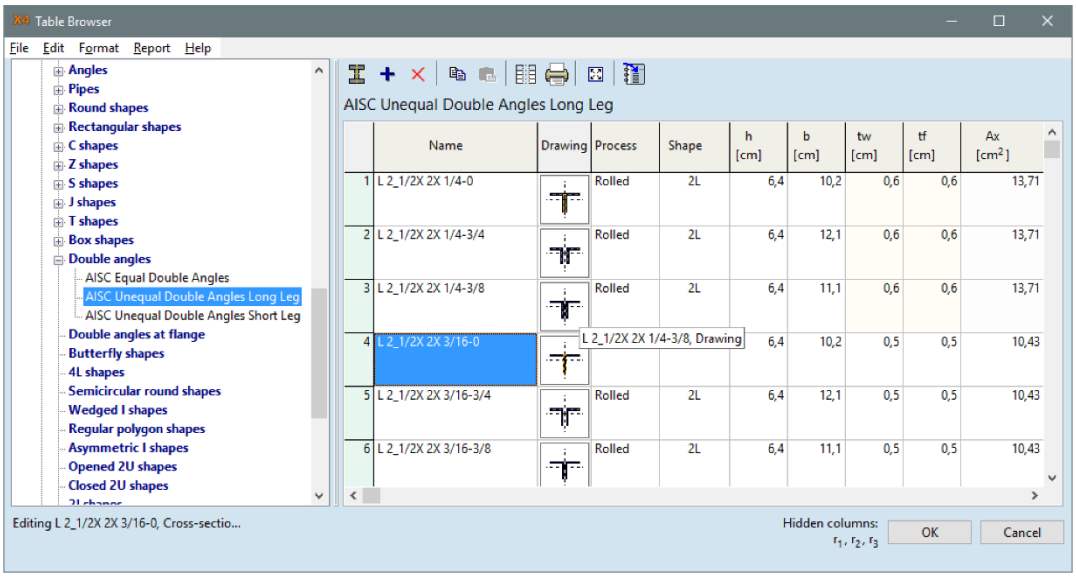
Masonry

EC	f_b	Normalized compressive strength of the masonry units
	f_k	Characteristic compressive strength of the masonry
	f_{vk0}	Initial characteristic shear strength
	f_{xk1}	Characteristic flexural strength of masonry bending about an axis parallel with bed joints
	f_{xk2}	Characteristic flexural strength of masonry bending about an axis perpendicular to bed joints

3.1.16 Cross-Section Library

AXISVM provides preloaded cross-section libraries, that contain the most frequently used steel shapes and concrete cross-sections, and allow you to create standard cross-section property sets that you can use over and over again in many different models. The libraries includes products of manufacturers worldwide.

For the description of the Table Browser see **2.9 Table Browser.**



☞ **The Undo function does not work when libraries are modified.**

Create a new library

You can create a custom cross-section library by the *File / New Cross-Section Table* command in the Table Browser. You have to specify a library name, a library file name and a cross-section type to categorize the library in the Table Browser. A *Geographical region* and a *Manufacturer* can also be entered. These fields can be used to filter cross-sections when importing from the library. See **4.9.2.1 Browse cross-section libraries**

Standard and custom cross-section library files (*.sec) are stored in the folder where the application is stored.

Assign a name to each cross-section, and specify the following properties:

Name
Process *Rolled, welded, cold-formed, other.*
Shape *I (H, W), U, L, Pipe, Round, Rectangle, C, Z, S, J, T, Box, Custom*

Cross-section properties

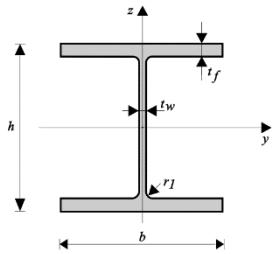
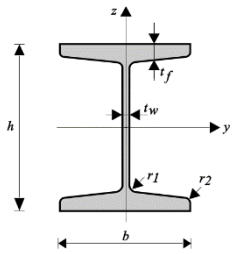
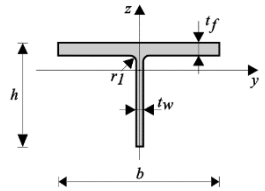
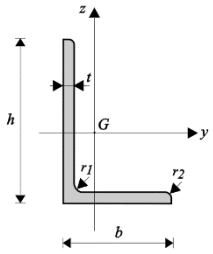
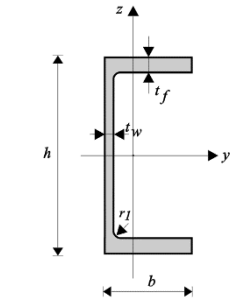
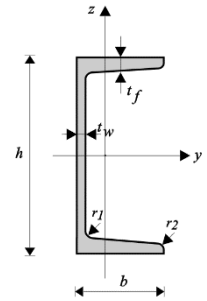
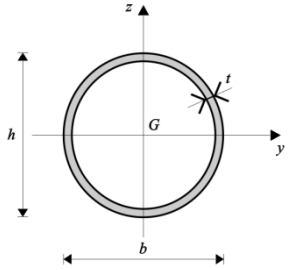
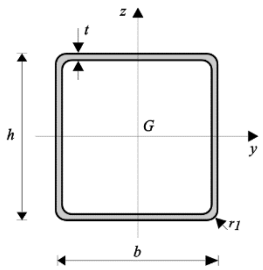
When creating a new cross-section in the table all property values have to be entered.

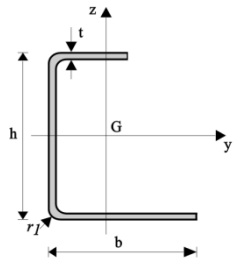
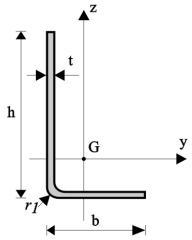
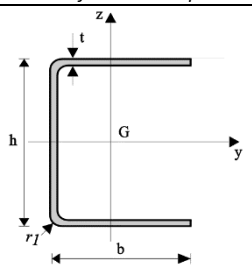
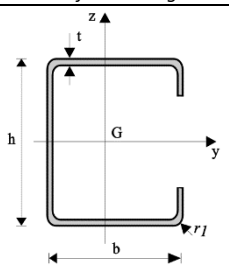
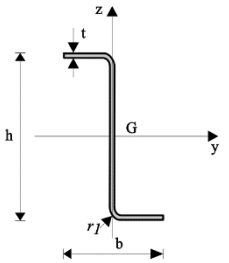
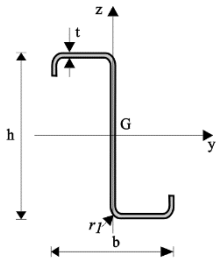
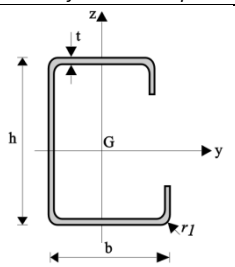
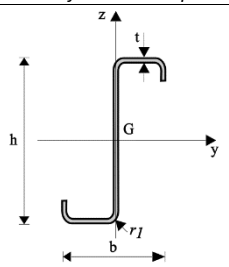
h	Dimension in the local ***z*** direction (height)
b	Dimension in the local ***y*** direction (width)
tw	Web thickness
tf	Flange thickness
Ax	Axial (cross-sectional) area
Ay	Shear area
Az	Shear area
Ix	Torsional inertia
Iy	Flexural inertia about local ***y*** axis
Iz	Flexural inertia about local ***z*** axis
Iyz	Centrifugal inertia (area product moment of inertia)
I_1, I_2	Principal inertia about local 1st / 2nd axis
Iω	Warping modulus (used for the design of steel shapes)
$W_{1,el,t}$	Elastic cross-section modulus, top = $I_1 / e2_max$ (see diagram below)
$W_{1,el,b}$	Elastic cross-section modulus, bottom = $I_1 / e2_min$
$W_{2,el,t}$	Elastic cross-section modulus, top = $I_2 / e1_max$
$W_{2,el,b}$	Elastic cross-section modulus, bottom = $I_2 / e1_min$

$W_{1,pl}$	Plastic cross-section modulus
$W_{2,pl}$	Plastic cross-section modulus
i_y, i_z	Radius of inertia about local 1st / 2nd axis
Hy, Hz	Bounding box dimensions in local ***y*** and ***z*** direction
y_G, z_G	Position of the center of gravity of the cross-section in local ***y, z*** direction relative to the lower-left corner of the circumscribed rectangle
y_s, z_s	Position of the shear center in local ***y*** and ***z*** directions relative to the center of gravity
r_1, r_2, r_3	Rounding (corner and fillet) radii
$\beta_y, \beta_z, \beta_\omega$	Wagner's coefficients describing the degree of symmetry [43]
S.p.	Stress calculation points

The Cross-section Library contains different type of cross-sections:

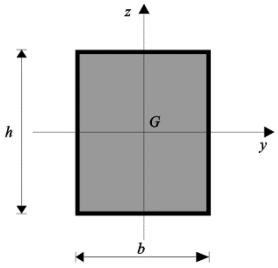
Steel cross-section

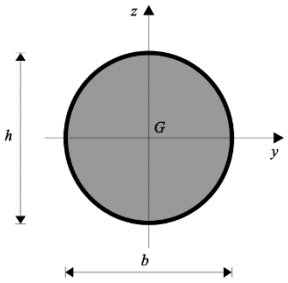
<table>
<tr><td>
</td><td>
</td></tr>
<tr><td>Hot rolled parallel flange I beam</td><td>Hot rolled tapered flange I beam</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>Hot rolled T shape</td><td>Hot rolled angle</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>Hot rolled parallel flange channel (U shape)</td><td>Hot rolled tapered flange channel (U shape)</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>Cold-formed pipe</td><td>Cold-formed and hot rolled (RHS) box shape</td></tr>
</table>

	
Cold-formed J shape	*Cold-formed angle*
	
Cold-formed channel (U shape)	*Cold-formed C shape*
	
Cold-formed Z shape	*Cold-formed S shape*
	
Cold-formed C shape (LINDAB)	*Cold-formed Z shape (LINDAB)*

Concrete cross-sections

The concrete cross-sections are listed starting from the size 20x20 to size 80x80 cm in steps of 2 and 5 cm.





3.1.16.1 Cross-Section Editor



Cross-section Editor allows you to edit thin / thick walled (solid) cross-sections or create certain parametric composite sections of two or three different materials.
You can use parametric circular, rectangular, ring and polygonal shapes, or any shape listed in the cross-section libraries to build complex cross-sections. The shapes used to build a new thin-walled or thin-walled cross-section are referred to as components, and have to be of the same material.
You can translate, rotate, mirror, copy or move the selected components at any time during the editing. When a component is placed to its location graphically, the principal axes and the cross-sectional properties of the composite cross-section are computed.
You can use keyboard commands the same way as in main editing windows.

The **OK** button exits and closes the cross-section editor window, and saves your current cross-section into the cross-section table of your model with a name you specify.

Cross-section editor is on the toolbar of the Cross-section Library and can also be launched from the line element dialog. See **4.9.10 Line elements**
The editor can be used when creating a native model from an architectural model through the IFC interface. See **4.9.26 Creating model framework from an architectural model**

Editor Keys

See **2.5 Using the cursor, the keyboard, the mouse**

Toolbar

Most important functions are available from the toolbar.



Prints the cross-section. See **3.1.12 Print**



Adds the image of the cross-section to the Gallery.
See **3.2.12 Saving drawings and design result tables**



Undoes the last operation.



Redoes the operation which was undone.



Copies the image of the cross-section to the Clipboard.

From Cross-section Library



Loads a cross-section from the Cross-section Library. Only thick or thin-walled cross-sections are available depending on the cross-section editor tab position. See **4.9.2.1 Browse cross-section libraries**

From DXF file



Contour of thick walled cross-sections can also be imported from a DXF file.

Stress-points



You can specify the points you want to calculate stresses for. The default stress-point is the center of gravity. You can specify up to 9 stress-points for each cross-section.
When applying a move command the stress-points can also be moved.
In case of thin-walled sections the shear flow is calculated only along the centre lines. The projected points, from where the shear stress values are assigned to the stress-points, are also displayed. If a stress-point is located at a corner or at a joint of several centerline segments, the highest value is taken into account.

☞ **Stress calculations are performed at the specified stress-points only. If you don't specify any stress-points, stress will be calculated in the center of gravity only. It means that no bending stress will appear.**

Icon bar

Editor functions and settings can be found on the Icon bar on the left. The behaviour of the Icon bar is the same as that of the main Icon bar. See **2.16 Icon bar.**

The only difference is that this Icon bar can be moved above the menus at the top or at the bottom but it is not dockable.



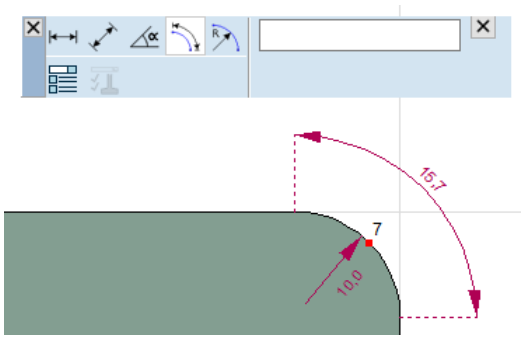
Geometry transformations



All standard geometry transformations (moving, rotating, mirroring, scaling) can be used. All operations are performed in the y-z plane of the cross-section editor.

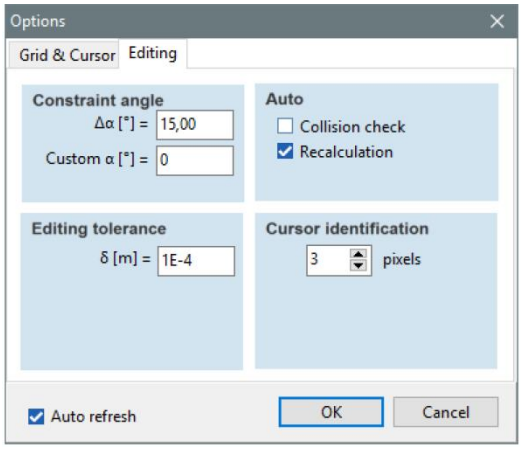
Dimensioning





Dimensioning tools can be selected from the upper toolbar of the palette. These are orthogonal and aligned dimension lines, angle, radius and arc length dimensions.
The bottom toolbar contains the buttons for fine tuning dimension line properties and choosing the smart dimension line option which places multiple dimension lines between the start point and the end point.





The Options dialog allows editing constraint angles and editing tolerance.

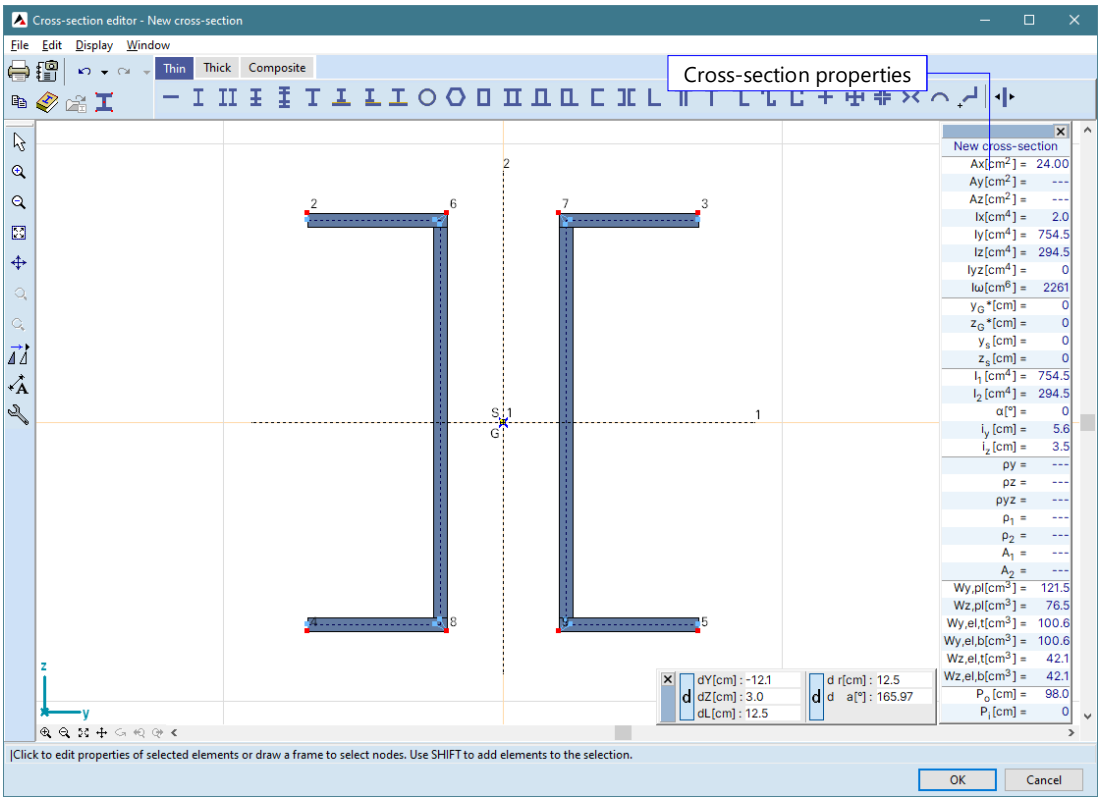
Automatic collision check turns the temporary outline of the shape to red if it touches or overlaps another shape in its current position.

Automatic recalculation recalculates cross-section parameters each time the shape changes.

Definition of a cross-section

The Editor can be used to create custom thin-walled or thick-walled (solid) or composite cross-sections. The components of a complex but non-composite cross-section will have the same material. Composite cross-sections are parametric and created from two or three materials.

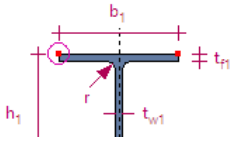
Thin-walled cross-sections



A component belonging to the thin-walled category can be added to your cross-section.

Base-point

You can select a base-point to each cross-section component, that allows you to position the component during editing, depending on its shape and final location within the composite cross-section.



Standard shapes can also be defined parametrically. In this case the following parameters has to be defined in the dialog:

Manufacturing process

There are three options (rolled, welded, cold formed.)

Dimensions

Values depending on the type of the cross-section (height, width, thickness, corner/fillet radius, diameter etc.).

Rotation

Lets you define a rotation by angle α. The default value is 0.

I shape



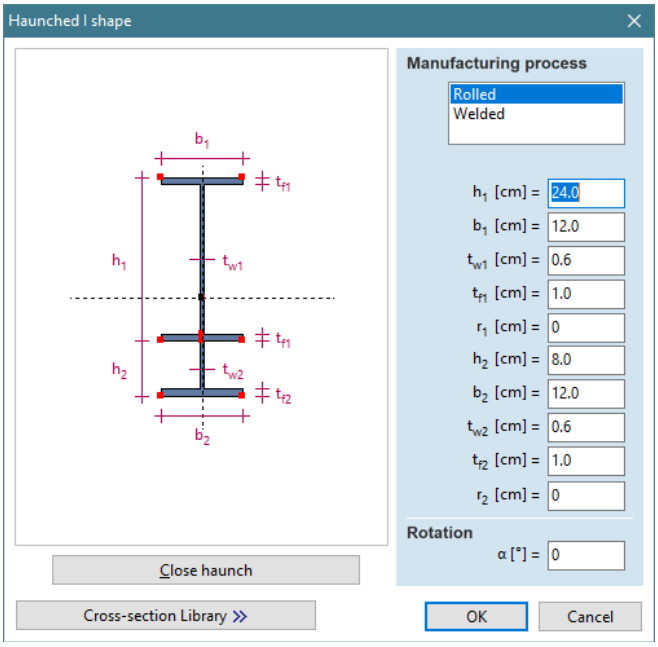
,

Haunched I shape



Double haunched I shape

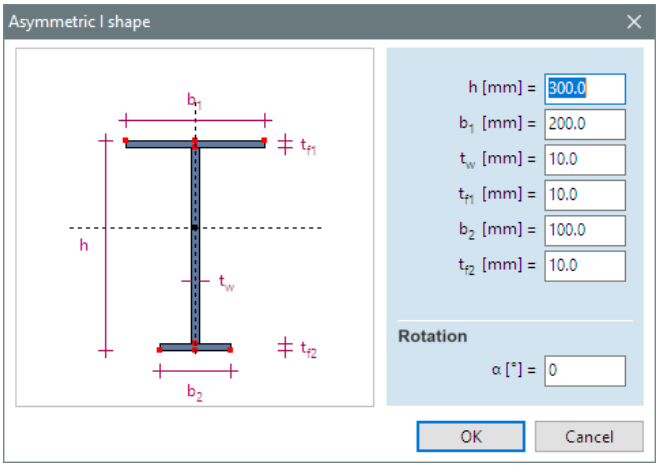




Definition of an I or haunched I shape by its height, width, web and flange thicknesses and a fillet radius.
The *Close haunch* button sets the minimum possible haunch size helping to define the end cross-section of a haunched beam with a variable cross-section.
The *Cross-section Library* button is to choose a component of the haunched I profile (the core cross-section or the I or T haunch profile) from the library.

Asymmetric I shape





Definition of an asymmetric I shape by its height, width, web and upper / lower flange dimensions.

L, U, C, Z, S shapes

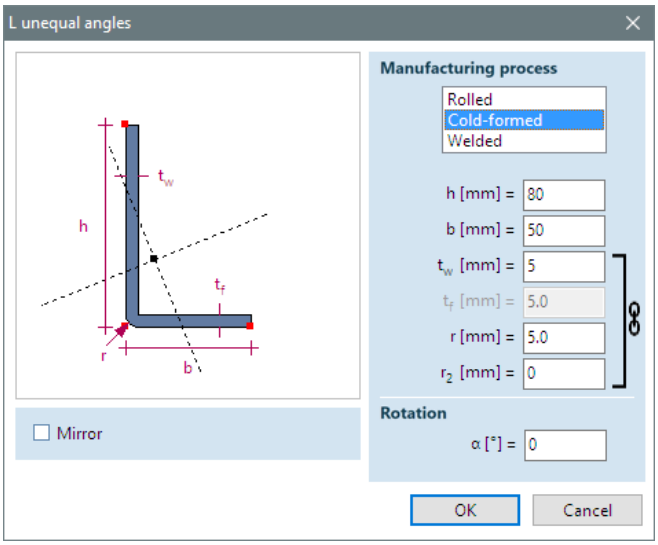










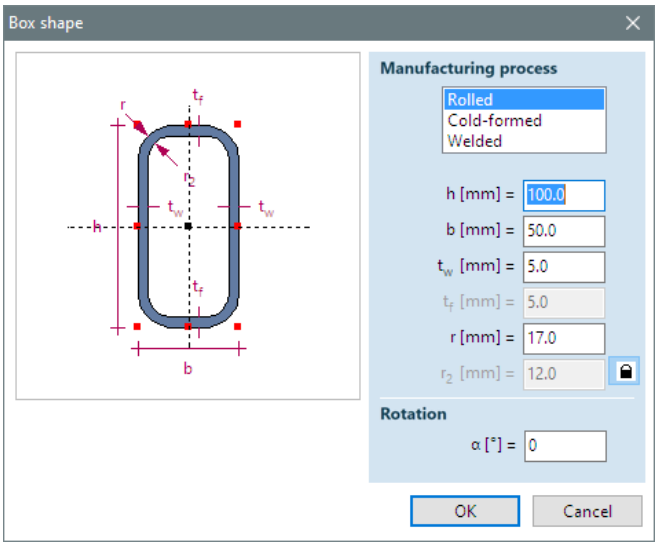


In the case of cold-formed L, U, C, Z, S sections, the rounding can be specified either with the inner or outer radius. However, the thickness and the values of the inner and outer rounding radii are not independent of each other. The following condition applies:

$$r_2 = \begin{cases} r - t_w\,, \text{if } r \geq t_w \\ 0\,, \text{if } r < t_w \end{cases}$$

After changing one of these parameters, the program ensures the compatibility by adapting the rounding radii.

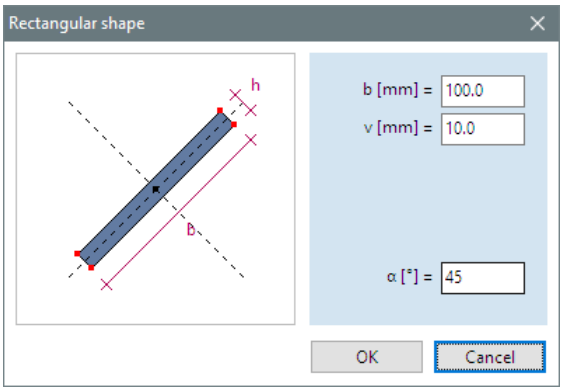
Box shape



The internal rounding radius of rolled and cold-formed box shapes can be specified independently of the external radius or fixed to it by clicking on the lock icon. In this latter case the following relationship applies:

$$r_2 = \begin{cases} r - t_w \text{ , if } r \geq t_w \\ 0 \text{ , if } r < t_w \end{cases}$$

Rectangular



Definition of a rectangle by its parameters *b* (width), *v* (thickness), and α, with *b* > *v*.

Pipe



Definition of a pipe by its parameters *d* (outside diameter), and *v* (thickness). The centerline is considered as the contour of a closed domain, which is displayed with a dashed line.

Other cross-sections

Definition of cross-sections by height, width, thickness and in the case of rolled or bended cross-sections by the corner/fillet radius.



Double shapes



The base cross-section can be defined parametrically (width, height, web and flange thickness) or taken from the Cross-section Library.
Special parameters for double shapes:

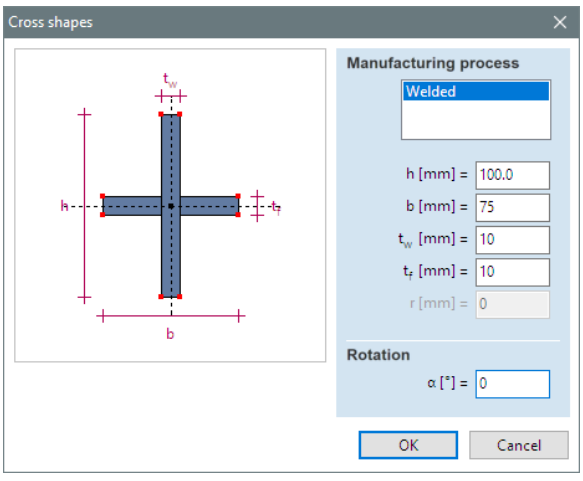
distance: *a*
orientation: facing or back-to-back (in case of 2U)

Cross shapes



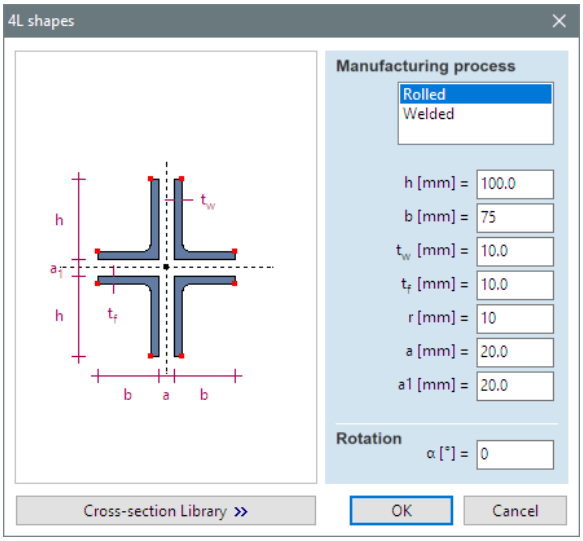
2I cross shapes





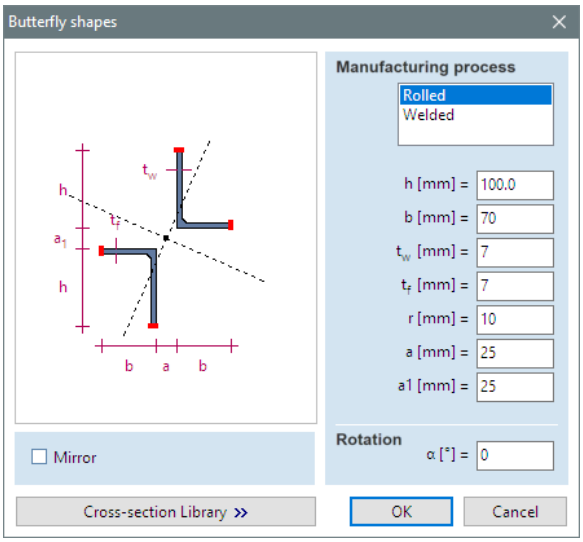
Definition of a cross shape by its parameters: *h* (height), *b* (width), *tw* (web thickness) and *tf* (flange thickness).

4L shapes



Definition of a 4L shape by its parameters *h* (height), *b* (width), *tw* (web thickness) and *tf* (flange thickness), *a, a1* (distances) and *r* (rounding radius).

Butterfly shapes



Definition of a butterfly shape by its parameters *h* (height), *b* (width), *tw* (web thickness) and *tf* (flange thickness), *a, a1* (distances) and *r* (rounding radius).

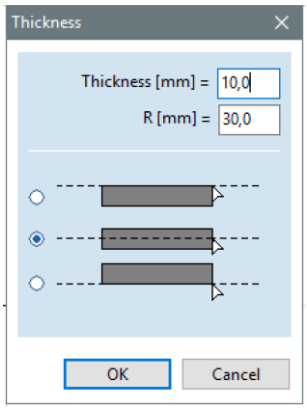
Polygonal

Definition of a polygonal shape.

Before the definition the position of the control line of the segment can be selected:

1. left side
2. center line
3. right side

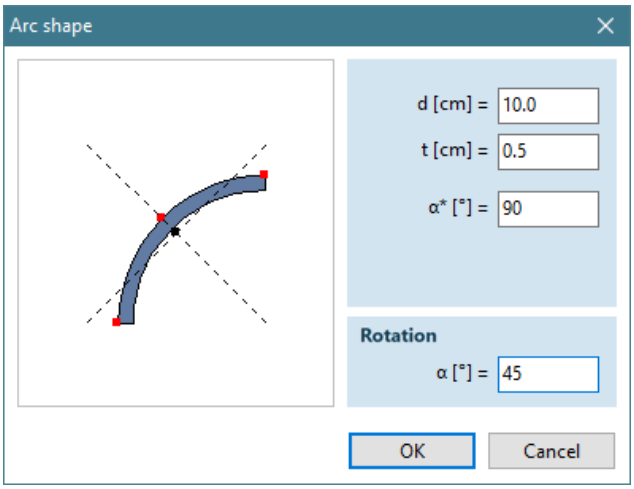
R: Rounding (corner and fillet) radii



Arc shape

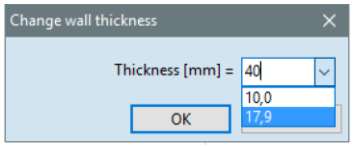


Definition of an arc shape by *d* (diameter), *α** (central angle) and *t* (thickness).



Changing wall thickness





For thin-walled cross-sections thickness of selected segments can be changed individually. For parametric shapes wall thickness can be changed through the parameters.

Delete

Using the **[Del]** key you can invoke the Selection Icon Bar, and select the components you want to delete. When deleting a component the stress-points will also be delete.

Stress-point

Deletes the selected stress-points.

☞ **You cannot delete the default stress-point (the center of gravity).**

Options

Lets you set the grid size, cursor step, and the zoom factors.

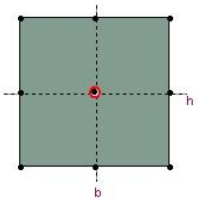
Thick-walled cross-sections



Parametric shapes:

Rectangle

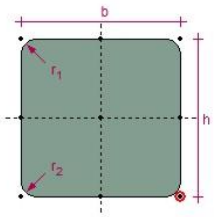




Rectangle with a width of *b* and a height of *h*.

Rounded rectangle

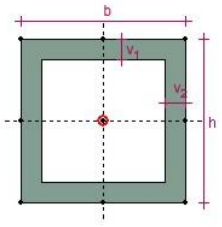




Rounded rectangle with a width of *b*, height of *h* and rounding radii *r1*, *r2*.

Hollow rectangle

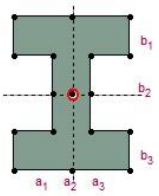




Hollow rectangle with a width of *b*, height of *h* and *v1*, *v2* wall thickness.

I shape

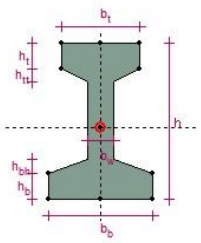




I shape defined by a_1, a_2, a_3, b_1, b_2, b_3 parameters.
One of the (a_1, a_3), (b_1, b_3) parameters can be set to zero to define T, U, L shapes

Haunched I shape

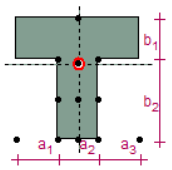




Haunched, I shape defined by the b_w web thickness, b_t, b_b top and bottom width, h total height and h_t, h_{th}, h_b, h_{bh} parameters.

T shape

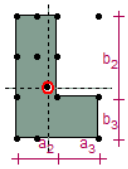




Defined like an I shape, but $b_3 = 0$.

L shape

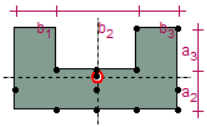




Defined like an I shape, but $a_1 = 0$ and $b_3 = 0$.

U shape

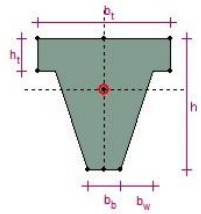




Defined like an I shape, but $a_1 = 0$.

T shape with haunched web

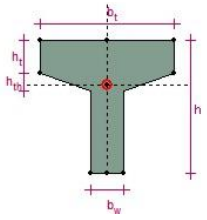




T shape with haunched web defined by b_w, b_b, b_t width, a h total height and h_t, flange thickness.

T shape with haunched flange

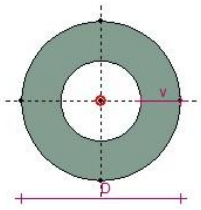




T shape with haunched web defined by b_w, web width b_t total width, h total height and h_t, h_{th} parameters

Hollow circle

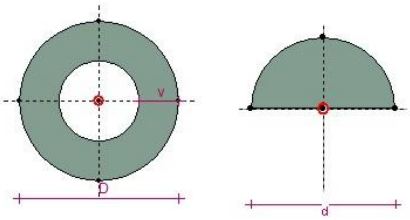




Hollow circle defined by the D outer diameter and the v wall thickness.

Round or sem-icir-cular shape

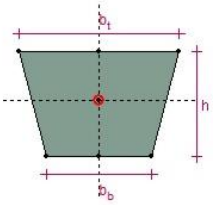




Round or semi-circular shape with a diameter of *d*.

Symmetric trapezoid





Symmetric trapezoid with b_t, b_b top and bottom width and a height of *h*.

Polygonal



Definition of a polygonal shape by drawing a a complex polygon. Press the **Esc** key, click the right mouse button or close the polygon to finish editing. During editing the following pet palette appears:




Line


Tangent


Arc with centerpoint


Arc by three points


Tangential arc


Arc with a given tangent

Insert a vertex



Insertion of a new vertex on the contour of the cross-section. Shape of the cross-section can be changed by dragging a vertex by the mouse.

Contour



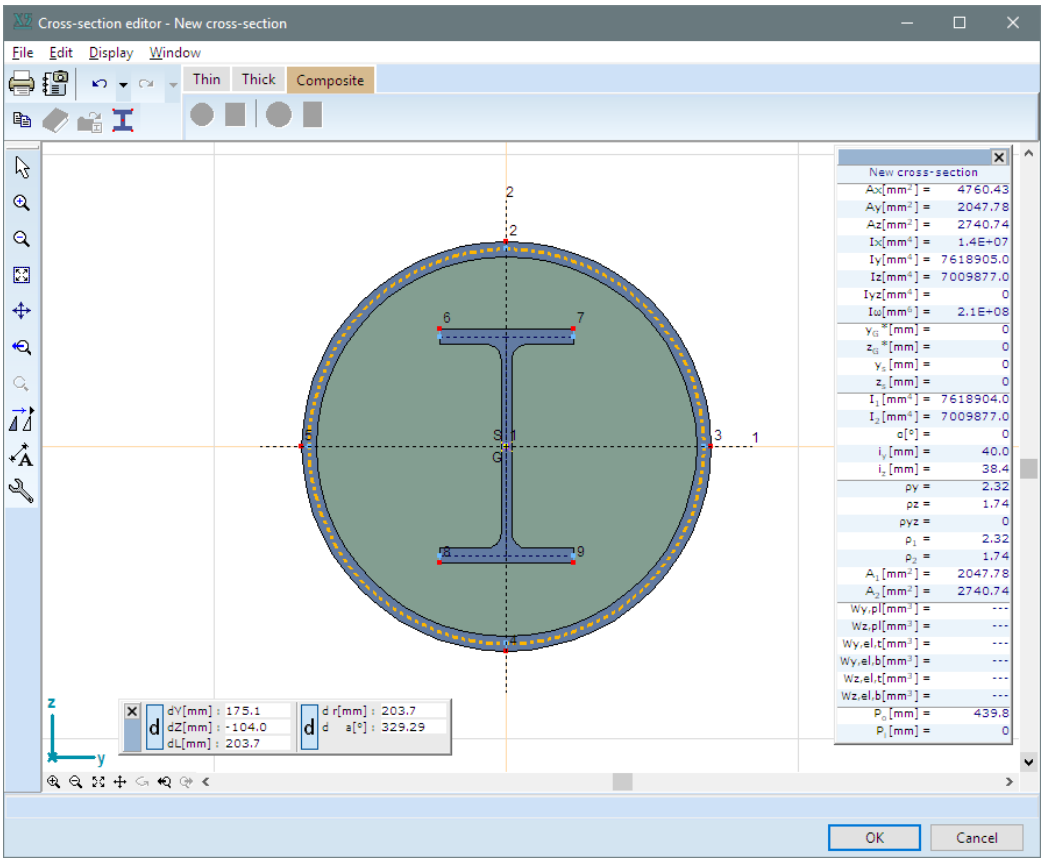
If the Contour button is down the cross-section can be defined. If the Hole button is down a hole can be specified.

Hole



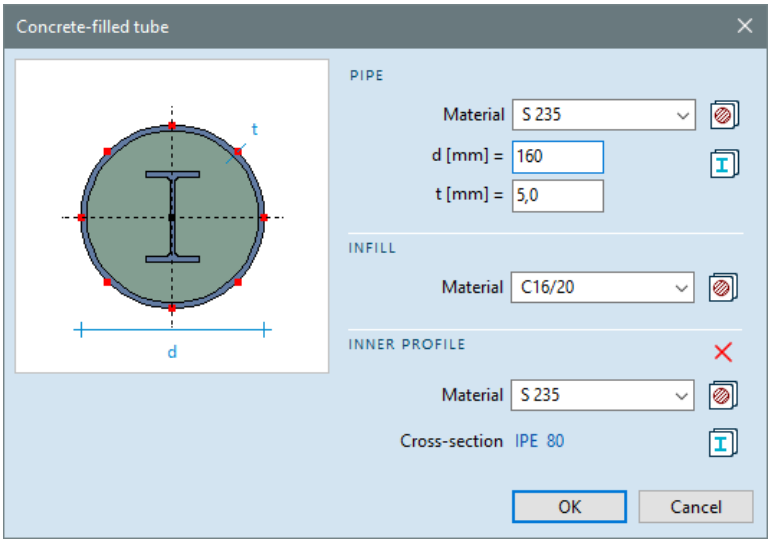
You can specify a hole in rectangular, circular, and closed polygonal shape components. The hole can be rectangular, circular, and closed polygonal.

Composite cross-sections



Composite sections are made of two or three different materials.
If the composite section contains an inner profile the center of gravity of the inner profile will get to the position of the center of gravity of the outer cross-section. If the inner profile is too big to fit into the outer cross-section the drawing of the definition dialog is replaced by a crossed-out rectangle.

Concrete-filled tube



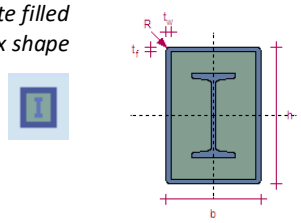
Definition of a concrete-filled tube requires the following parameters: the material, the d diameter and t thickness of the tube and the material of the infill. An optional inner profile can also be defined.

Materials can be imported from the Material Library, see **3.1.15 Material Library.**

The tube section can be entered parametrically or selected from the pipe cross-sections of Cross-section Library, see **3.1.16 Cross-Section Library**. Inner profile can be selected from the Cross-section Library.

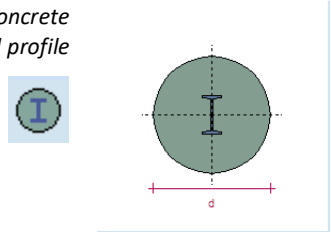
The selected inner profile can be removed from the section.

Concrete filled box shape



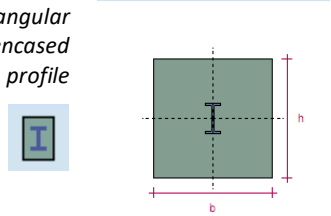
Definition of a concrete-filled box shape requires the following parameters: the material, the *b* width, *h* height, *tf* and *tw* thickness values of the box and the material of the infill. An optional inner profile can also be defined.

Circular concrete encased profile



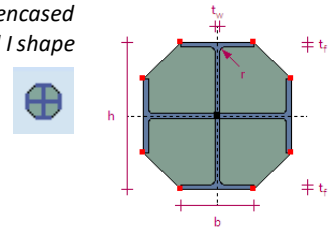
Definition of a circular concrete-encased profile requires the following parameters: the material and the d diameter of the round shape, an inner profile and its material.

Rectangular concrete encased profile



Definition of a circular concrete-encased profile requires the following parameters: the material, the *b* width and *h* height of the rectangle shape, an inner profile and its material.

Concrete encased crossed I shape



Definition of a concrete encased crossed I shape requires the following parameters: material of the crossed I shape and the infill, the *h* height, *b* flange width, and two wall thickness values: *tf* flange thickness and *tw* web thickness.

Delete

Using the **[Del]** key you can invoke the selection window, and select the components you want to delete. When deleting a component, the stress-points will also be deleted.

Polygon Deletes the selected components.

Stress-point Deletes the selected stress-points.

☞ **You can not delete the default stress-point (from the center of gravity).**

Options Lets you set the grid size, cursor step, and the zoom factors.

Compute properties Following cross-section properties are calculated:

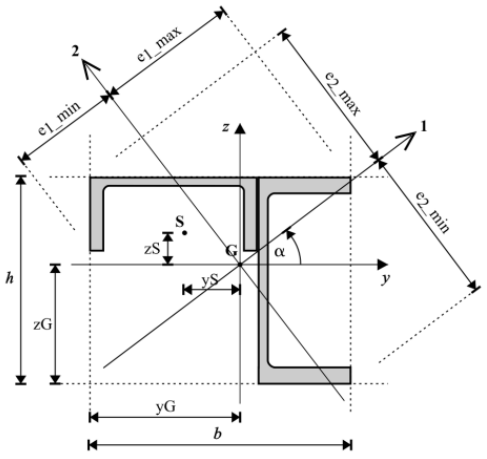
AXISVM calculates *Ax, Iy, Iz, Iyz, βy, βz* by integration, *Ay, Az, Ix, Iω, ρy, ρz, ρyz, ρ1, ρ2, A1, A2, βω* by performing a finite element analysis of the cross-section.

☞ **In case of a cross-section consisting of two or more independent parts, A_y, A_z, ρ_y, ρ_z, ρ_{yz}, ρ_1, ρ_2, A_1, A_2 are not determined.**

A_x	Axial (cross-sectional) area
A_y	Shear area in local *y* direction
A_z	Shear area in local *z* direction
I_x	Torsional inertia
I_y	Flexural inertia about local *y* axis
I_z	Flexural inertia about local *z* axis
I_{yz}	Centrifugal inertia
$I_1^{(*)}$	Principal inertia about local 1st axis
$I_2^{(*)}$	Principal inertia about local 2nd axis
α	Angle between local 1st axis and the local y axis.
I_ω	Warping modulus (used for the design of steel shapes)
ρ_y	shear factor in local *y* direction
ρ_z	shear factor in local *y* direction
ρ_{yz}	shear factor for local *yz* cross
ρ_1	shear factor for local 1st direction
ρ_2	shear factor for local 2nd direction
$A_1^{(*)}$	Shear area associated with shear forces in local 1st direction
$A_2^{(*)}$	Shear area associated with shear forces in local 2nd direction
$W_{1,el,t}^{(*)}$	Elastic cross-section modulus, top = I_1 / *e2_max* (see diagram below)
$W_{1,el,b}^{(*)}$	Elastic cross-section modulus, bottom = I_1 / *e2_min*
$W_{2,el,t}^{(*)}$	Elastic cross-section modulus, top = I_2 / *e1_max*
$W_{2,el,b}^{(*)}$	Elastic cross-section modulus, bottom = I_2 / *e1_min*
$W_{1,pl}^{(*)}$	Plastic cross-section modulus
$W_{2,pl}^{(*)}$	Plastic cross-section modulus
i_1	Radius of inertia about local 1st axis
i_2	Radius of inertia about local 2nd axis
y_G	Position of the center of gravity of the cross-section in local y direction relative to the lower-left corner of the circumscribed rectangle
z_G	Position of the center of gravity of the cross-section in local z direction relative to the lower-left corner of the circumscribed rectangle
y_S, z_S	Position of the shear center in local y and z directions relative to the center of gravity
β_y, β_z, β_ω	Wagner's coefficients describing the degree of symmetry [43]
P_o	Outer circumference (cross-section contour)
P_i	Inner circumference (holes)

(*) If first and second principal axes are the local *y* and *z* axes values with (*) appears with indices *y* and *z*.

Principal inertia



$$I_1 = \frac{I_y + I_z}{2} + \sqrt{\left(\frac{I_y - I_z}{2}\right)^2 + I_{yz}^2}$$

$$I_2 = \frac{I_y + I_z}{2} + \sqrt{\left(\frac{I_y - I_z}{2}\right)^2 + I_{yz}^2}$$

$$\tan(2\alpha) = \frac{2I_{yz}}{I_y - I_z}$$

$-90° < \alpha \leq +90°$, relative to the cross-section's local *y* axis.

Calculation of elastic cross-section modulus

$$W_{1,el,top} = \frac{I_1}{e_{2,max}} \qquad W_{1,el,bottom} = \frac{I_1}{e_{2,min}}$$

$$W_{2,el,top} = \frac{I_2}{e_{1,max}} \qquad W_{2,el,bottom} = \frac{I_2}{e_{1,min}}$$

Shear deformations

For beam elements the shear deformations are not taken into account even if a nonzero shear area was was entered.

The shear areas are used only in rib elements, if $A_y \neq 0$ or $A_z \neq 0$.

In the steel design module, the shear areas are calculated according to the corresponding design code, instead of using the values entered here.

$A_y = A_x/\rho_y \quad A_z = A_x/\rho_z$ where $\rho =$ shear factor

The concept of shear area and shear factor makes it possible to represent the continuously varying $\gamma_{xy}(y,z)$ and $\gamma_{xz}(y,z)$ shear deformations of the real cross-section properly by $\gamma_{xy,avg}$ and $\gamma_{xz,avg}$ average shear deformations. The proper representation ensures that the work done by the resultant shear forces V_y and V_z along the average shear deformation is equal to the work done by the continuously varying shear stress $\tau_{xy}(y,z)$ and $\tau_{xz}(y,z)$ of the real cross-section [16, Chapter 6.3; pp. 22, 219-221].
The equation for internal shear force in z direction:

$$\frac{1}{2}V_z\,\gamma_{xz,avg} = \frac{1}{2}\int\left[\tau_{xy}(y,z)\gamma_{xy}(y,z) + \tau_{xz}(y,z)\gamma_{xz}(y,z)\right]dA$$

The stress distribution in the cross-section due to the shear force V_z can be derived from the shear stress function $\Phi_z(y,z)$ calculated by finite element method:

$$\tau_{xy}(y,z) = \frac{V_z}{A_x}\frac{\partial\Phi_z}{\partial y} \qquad \tau_{xz}(y,z) = \frac{V_z}{A_x}\frac{\partial\Phi_z}{\partial z}$$

Hooke's law for the shear deformation and shear stress is:

$$\tau_{xy}(y,z) = G\gamma_{xy}(y,z) \qquad \tau_{xz}(y,z) = G\gamma_{xz}(y,z)$$

Substituting these into the integrand we obtain the following formula for the average shear deflection:

$$\gamma_{xz,avg} = \frac{V_z}{GA_x^2}\int\left[\left(\frac{\partial\Phi_z}{\partial y}\right)^2 + \left(\frac{\partial\Phi_z}{\partial z}\right)^2\right]dA$$

Introducing the cross-section shear factor $\rho_z = \frac{A_x}{\int\left[\left(\frac{\partial\Phi_z}{\partial y}\right)^2+\left(\frac{\partial\Phi_z}{\partial z}\right)^2\right]dA}$ we can write

$$\gamma_{xz,avg} = \frac{V_z}{G\frac{A_x}{\rho_z}} = \frac{V_z}{GA_z}$$

This formula shows that the average shear deformation can be calculated as if the cross-sectional area determining the shear stiffness of the finite element were A_z instead of the actual A_x.
The same applies to the shear force in *y* direction.

☞ **It is important to note that the reduced shear area can be used only to obtain the shear stiffness of the finite element. The average shear stress must be calculated using the actual area:**

$$\tau_{xy,avg} = \frac{V_y}{A_x} \qquad \tau_{xz,avg} = \frac{V_z}{A_x}$$

☞ **Simultaneous shear forces in *y* and *z* direction would require the introduction of a mixed shear cross-section area ρ_{yz} [16; Chapter 6.3]. However, the calculated value is just indicative as the elements describing the interaction of *y* and *z* direction shear forces are neglected when generating the stiffness matrix of rib finite elements.**

Compute properties of composite cross-sections

Properties of a composite cross-section are calculated as the modulus-weighted [16, Chapter 1.2; 21, Chapter 7.2.1] sum of the properties of its parts. The reference moduli E_{ref} and G_{ref} belong to the material of the outer part.

$$A_x = \frac{1}{E_{ref}}\sum E_i A_{x,i} \qquad I_x = \frac{1}{G_{ref}}\sum G_i I_{x,i} \qquad \rho_y = \frac{G_{ref}A_x}{\sum\frac{G_i A_{x,i}}{\rho_{y,i}}}$$

$$A_y = \frac{1}{G_{ref}}\sum G_i A_{y,i} \qquad I_y = \frac{1}{E_{ref}}\sum E_i I_{y,i} \qquad \rho_z = \frac{G_{ref}A_x}{\sum\frac{G_i A_{x,i}}{\rho_{z,i}}}$$

$$A_z = \frac{1}{G_{ref}} \sum G_i A_{z,i} \qquad I_z = \frac{1}{E_{ref}} \sum E_i I_{z,i} \qquad \rho_{yz} = \frac{G_{ref} A_x}{\sum \frac{G_i A_{x,i}}{\rho_{yz,i}}}$$

$$I_{yz} = \frac{1}{E_{ref}} \sum E_i I_{yz,i}$$

$$y_G = \frac{\sum y_{G,i} E_i A_i}{E_{ref} A_x}$$

$$z_G = \frac{\sum z_{G,i} E_i A_i}{E_{ref} A_x}$$

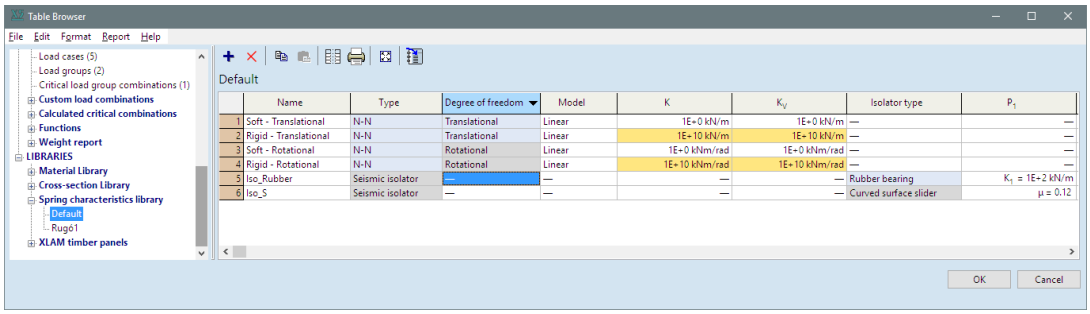
$$y_\omega = \frac{I_{yz} \sum E_i \left(z_{S,i} I_{z,i} - y_{S,i} I_{yz,i}\right) - I_z \sum E_i \left(z_{S,i} I_{yz,i} - y_{S,i} I_{y,i}\right)}{E_{ref} \left(I_y I_z - I_{yz}^2\right)}$$

$$z_\omega = \frac{I_y \sum E_i \left(z_{S,i} I_{z,i} - y_{S,i} I_{yz,i}\right) - I_{yz} \sum E_i \left(z_{S,i} I_{yz,i} - y_{S,i} I_{y,i}\right)}{E_{ref} \left(I_y I_z - I_{yz}^2\right)}$$

$$I_\omega = \frac{1}{E_{ref}} \sum E_i \left[I_{\omega,i} + \left(y_{S,i} - y_S\right)^2 I_{y,i} + \left(z_{S,i} - z_S\right)^2 I_{z,i} - 2\left(y_{S,i} - y_S\right)\left(z_{S,i} - z_S\right) I_{yz,i}\right]$$

☞ **The modulus-weighted computation method is based on the assumption that there is no shear connection between the parts. However, it has no effect for the parameters Ax, Iy, Iz, Iyz, yG, zG, because of the coincidence of the center of gravity of the parts. But the values of Ay, Az, Ix, Iω, yS, zS, ρy, ρz, ρyz would be different if there was shear connection.**

3.1.17 Spring characteristics library



The spring characteristics library window can also be opened using the *Table Browser* icon and by selecting *Libraries/Spring Characteristics Library*. In addition it is available in several dialog windows for easier selection. See **4.9.21 Spring**

The library has a Default library containing Rigid and Soft spring characteristics to translational and rotational degrees of freedom. Theese characteristics are automatically added for new models. You can add aa new library and you modify or expand existing spring characteristics libraries. In case of creating custom library, its elements will be added to the model only if they are added to the Spring characteristics (**2.9.1 Spring characteristics**) in Table Browser or be selected from the library when defining a new spring element.

This library allows definition of parametric seismic isolators as well.

Properties:

Name	Name of the characteristic
Type	Node to node spring (cannot be modified)
Degree of freedom	Translational/rotational degree of freedom
Model	Spring behaviour of the spring (*Linear / NL elastic / Plastic*)
K	Initial stiffness
K_v	Vibration stiffness
Damper type	*Kelvin-Voigt* or *Maxwell* model
C	Damping coefficient
Isolator type	*Rubber bearing, Curved surface slider, Other type*
E_{xy}	Effective horizontal stiffness
ξ	Damping ratio
E_z	Vertical stiffness

F_1 Yield force
K_T Horizontal stiffness in plastic state
μ Friction coefficient
R Equivalent curvature radius

Damping coefficient is shown if at least one spring characteristic has a value greater than zero. Seismic isolator parameter columns appear only if at least one isolator has been defined.

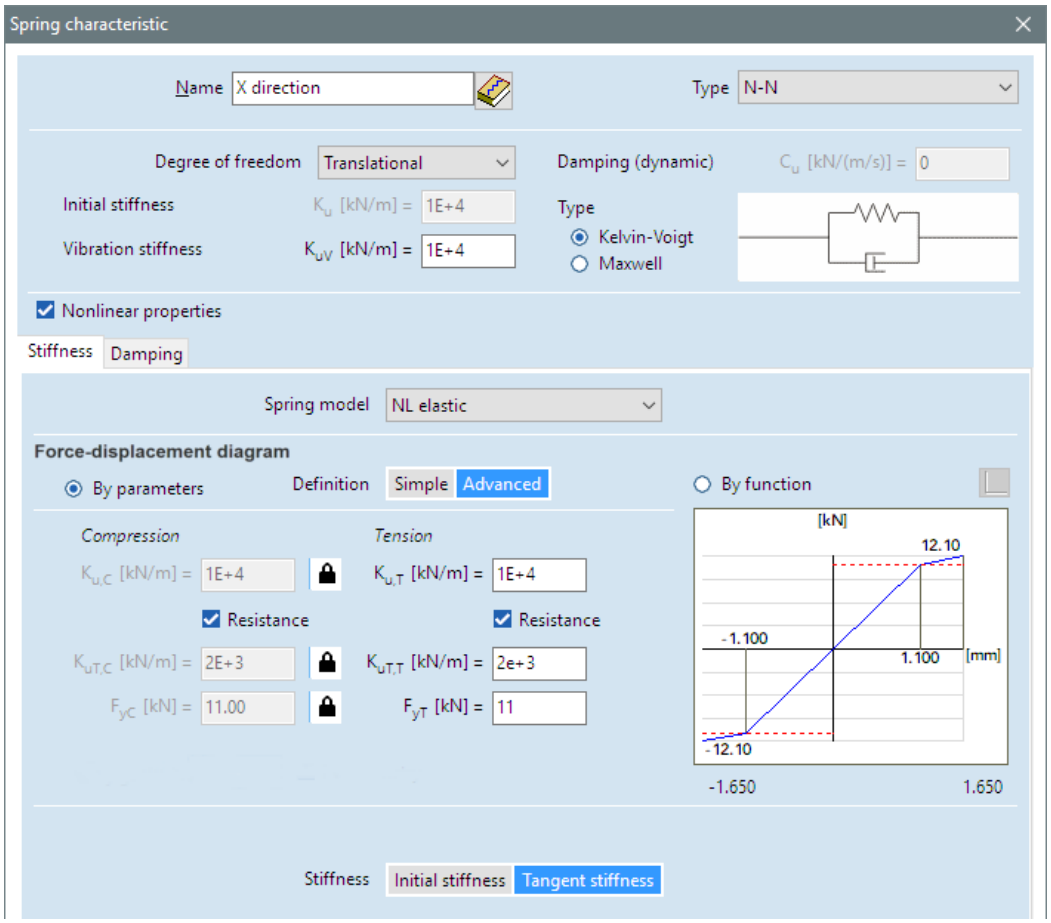
☞ **Changes in the Spring charateristics library does not reflect in models using the modified spring characteristic.**

Define new spring characteristic

[Ctrl+Ins]

Change properties

When entering a new spring characteristic, the following dialog is displayed.
Defining a new spring characteristic or clicking on a table column (except the *Name* column) a dialog appears, where all properties of spring charateristic can be defined or changed.
Characteristics with the same degree of freedom (translational or rotational) cannot have the same name. If an existing name is entered a number will be appended to the name.
A spring characteristic has the following properties: *Name, Type, Degree of freedom, Damping type*



Clicking on the icon the *Browse spring characteristics library* dialog is displayed.

3.1.17.1 Node-to-node spring

Stiffness / Damping

In default setting the springs operate on linear elastic principle. In the table the *Model* column contains the type of behaviour of the spring characteristic.

Translational degree of freedom

K_u	kN/m	Initial stiffness
K_{uV}	kN/m	Vibration stiffness
C_u	kN/(m/s)	Damping coefficient

Rotational degree of freedom

K_r	kNm/rad	Initial stiffness
K_{rV}	kNm/rad	Vibration stiffness

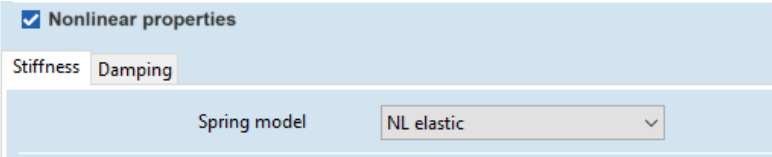
C_r	kNm/(rad/s)	Angular damping coefficinet

Nonlinear properties

A spring can have a nonlinear behaviour like materials. The parameters of the nonlinear behaviour are available if *Nonlinear properties* is checked.

The parameters for stiffness and damping can be defined on different tab.

Stiffness



Spring model

NL elastic Nonlinear characteristic without irreversibile deformation.

Plastic Irreversibile deformation occurs after reaching the limit force / limit moment.

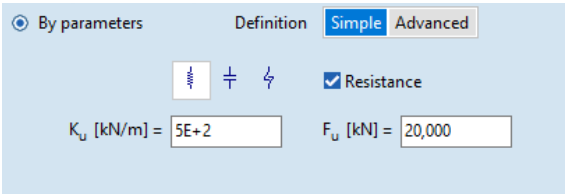
Force – displacement diagram

By parameters: The function of the spring characteristic can be defined in the same way as the nonlinear nodal support in the previous versions (simple) or as the stress-strain diagram of materials (advanced). The units depend on the degree of freedom. Plastic model requires advanced definition.

By function: A custom characteristic can be defined using the force-displacement function editor.

By parameters, Simple definition

In case of simple definition, the characterictic can be defined like a nonlinear support. There are options for symmetric, only compression, only tension characteristic and for resistance.



Both for tension / compression

Compression only

Tension only

Translational

Resistance	Characteristic with a force resistance
K_u *[kN/m]*	Initial stiffness
F_u *[kN]*	Resistance

Rotational

Moment resistance	Characteristic with a moment resistance
K_r *[kNm/rad]*	Initial stiffness
M_r *[kNm]*	Moment resistance

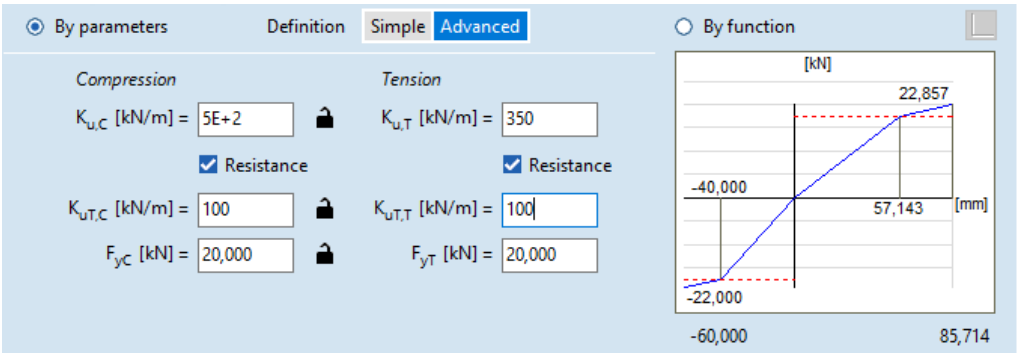
Setting a spring to *Compression only* or *Tension only* does not set the inactive stiffness to zero. To avoid numerical instability a negligibly small stiffness is introduced.

$$K_{u0} = \min\left\{10^{-3}\frac{\mathrm{kN}}{\mathrm{m}};\ K_u \cdot 10^{-8}\right\};\ K_{r0} = \min\left\{10^{-3}\frac{\mathrm{kNm}}{\mathrm{rad}};\ K_r \cdot 10^{-8}\right\}$$

If the deformation of the element is large even this stiffness can result in a significant force. In this case choose *Advanced* and define a characteristic with a resistance on the inactive side.

By parameters, Advanced definition

In case of advanced definition the tension and compression side can be different. If the spring model is plastic, the $K_{u,T}$ and $K_{u,C}$ have to be the same, the value of $K_{u,C}$ cannot be edited.
Using *Resistance* checkbox limit force/moment and tangent stiffness of the diagram after reaching the limit force can be specified. If the resistance is switched off a simple linear diagram is obtained.
The red dashed line represents the F_{yC}, F_{yT} and M_{yC}, M_{yT} limit values.



Translational

$K_{u,T}$ [kN/m]	Initial stiffness for tension
$K_{u,T}$ [kN/m]	Initial stiffness for compression
$K_{uT,T}$ [kN/m]	The slope of the diagram after reaching the limit force for tension
$K_{uT,C}$ [kN/m]	The slope of the diagram after reaching the limit force for compression
F_{yT} [kN]	Tension resistance
F_{yC} [kN]	Compression resistance

Rotational

$K_{r,T}$ [kNm/rad]	Initial stiffness for positive rotation
$K_{r,C}$ [kNm/rad]	Initial stiffness for negative rotation
$K_{rT,T}$ [kNm/rad]	The slope of the diagram after reaching the limit moment for positive rotation
$K_{rT,C}$ [kNm/rad]	The slope of the diagram after reaching the limit moment for negative rotation
M_{yT} [kNm]	Moment resistance for positive rotation
M_{yC} [kNm]	Moment resistance for negative rotation

Advanced definition of a characteristic active only in tension or compression

Advanced mode provides several ways to define a nonlinear elastic characteristic active on one side only. The simplest solution is to set the function to zero on the inactive side. However, this ideally one-sided characteristic is numerically the least stable near the active / inactive transition.



$K_{u,T} = 0\ ;\ F_{y,T} = 0$

Instead of the ideal one, it is better to include a characteristic that has the same initial stiffness and a negligibly small resistance on the inactive side. This ensures numerical stability with a definite appearance of the tension or compression state, while the resistance can be set so small that it will be displayed as zero with the given number of decimals. However, numerical stability problems can occur even with this characteristic. Unexpected behaviour of the model may make the active stiffness fail to ensure the equilibrium. In that case, the calculation will not converge.



$K_{u,T} = 0\ ;\ F_{y,T} > 0$

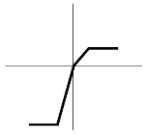
This stability problem can be avoided by setting a very small but not zero slope in the linear characteristic on the inactive side. This way a sufficiently large deformation can ensure the equilibrium on the inactive side as well and unrealistically large deformations will warn us of the unexpected behaviour of the model. However, it should also be considered that if a large-scale deformation in the inactive direction is the intended behaviour, an unrealistically significant force may develop on the inactive side.



$K_{u,T} > 0$

Advanced definition of a characteristic with a given resistance

A characteristic with a given resistance can also be defined as an ideal one where the stiffness becomes constant after reaching the resistance. However, a zero tangential stiffness may cause numerical instability and an unexpected behaviour of the model may make these springs fail to ensure the equilibrium. In that case, the calculation will not converge.



$K_{uT,TC} = K_{uT,T} = 0$

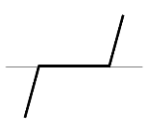
This problem can also be solved by setting a small but not zero slope after reaching the resistance.
This way a sufficiently large deformation can ensure the equilibrium and unrealistically large deformations will warn us of the unexpected behaviour of the model. However, it should also be considered that if a large-scale deformation is the intended behaviour, forces may considerably exceed the given resistance.



$K_{uT,C} > 0\ ;\ K_{uT,T} > 0$

Advanced definition of a characteristic with backlash

Backlash can be simulated by defining a small initial stiffness followed by a high stiffness nonlinear elastic state. It is recommended to replace the ideal zero stiffness with a small slope and small resistance. The ideal characteristic cannot be set parametrically only through then function editor.



$K_{u,C} = K_{u,T} = 0$



$K_{u,C} = K_{u,T} > 0$

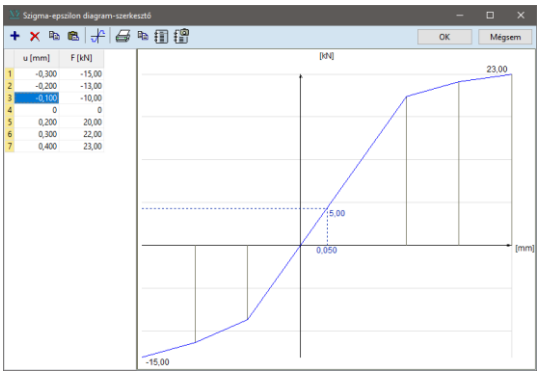
By function



Click the small button to edit the diagram.
The function editor is available for both nonlinear elastic and plastic spring models.

Force – displacement diagram editor

The elements of the toolbar are largely the same as in **4.10.31 Dynamic loads (for time-history analysis) – DYN module.**

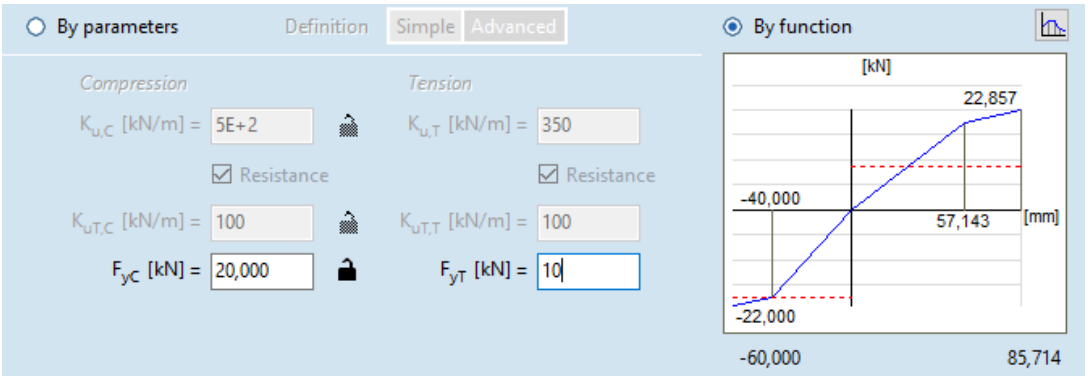


Using function editor the parameter values on advanced panel are set on the basis of the function in the following way (in case of translational components)

$K_{u,T}$ [kN/m]	The slope of the first segment on the tension side
$K_{u,C}$ [kN/m]	The slope of the first segment on the compression side
$K_{uT,T}$ [kN/m]	The slope of the last segment on the tension side
$K_{uT,C}$ [kN/m]	The slope of the first segment on the compression side
$F_{y,T}$ [kN]	*Plastic* – First breakpoint on the tension side *NL elastic* – Editable
$F_{y,C}$ [kN]	*Plastic* – First breakpoint on the compression side *NL elastic* – Editable

The user can enter the limit values (resistances) only in case of *NL elastic* spring model and the utilization will be calculated by comparing to this value.
For plastic spring model the parameter values cannot be edited and $K_{u,T} = K_{u,C}$

☞ **The force-displacement diagram must be monotonically increasing.**
In case of a plastic characteristic the slope of the diagram must be monotonically decreasing.



Hardening rule

Available only if *Plastic* spring model is selected.
Isotropic hardening means that both tension and compression resistances increase.
Kinematic hardening means that tension resistance increases and compression resistance decreases under tension, tension resistance decreases and compression resistance increases under compression while their differerence remains constant. See **3.1.15 Material Library**

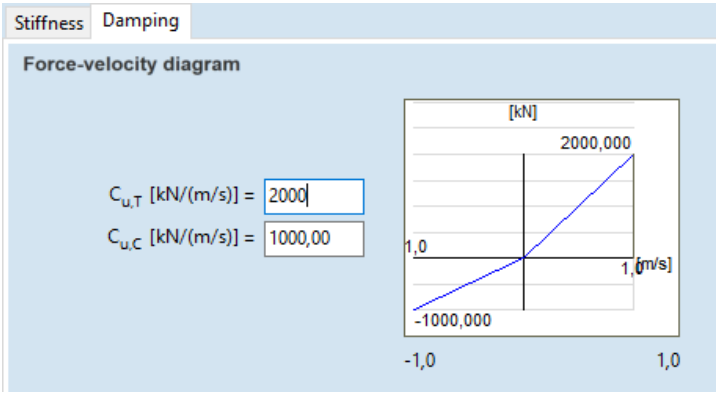
Stiffness

Initial stiffness — The materially linear initial stiffness matrix is used when evaluating the global stiffness matrix. The calculation is slower (more iterations are necessary) but the convergence is more stable.

Tangent stiffness — The materially nonlinear tangent stiffness matrix is used when evaluating the global stiffness matrix. The calculation is faster (less iterations are necessary) but if increments are not small enough convergence problems may occur.

Damping

A bilinear function can be defined, where the damping coefficients for tension and compression might differ.



$C_{u,T}$ *[kN/(m/s)]* Damping coefficient for tension
$C_{u,c}$ *[kN/(m/s)]* Damping coefficient for compression

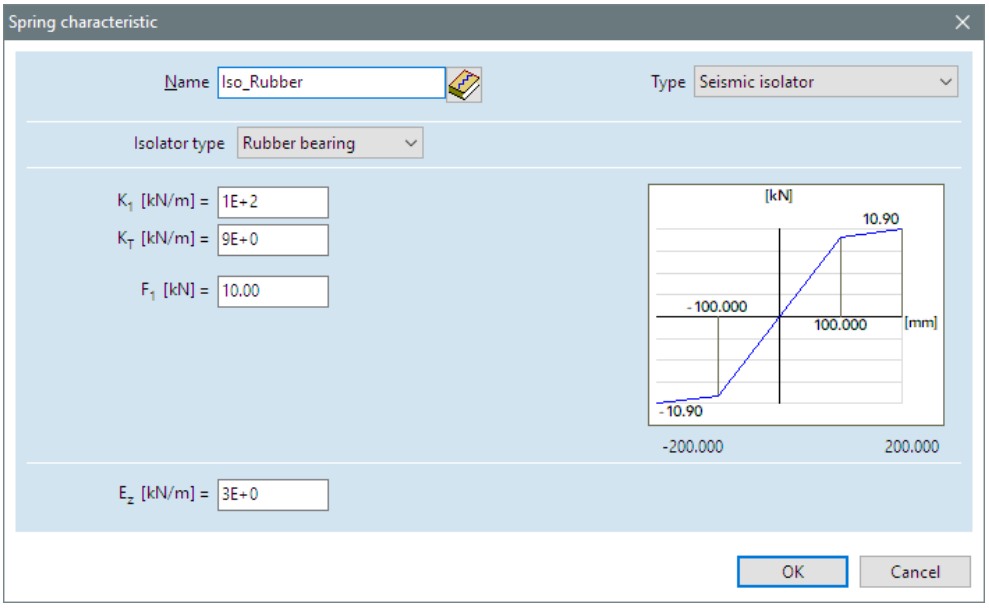
If the analysis uses linear material model the damping coefficient is the average of the two values:
$C_u = (C_{u,T} + C_{u,C})/2$

3.1.17.2 Seismic isolators

Type

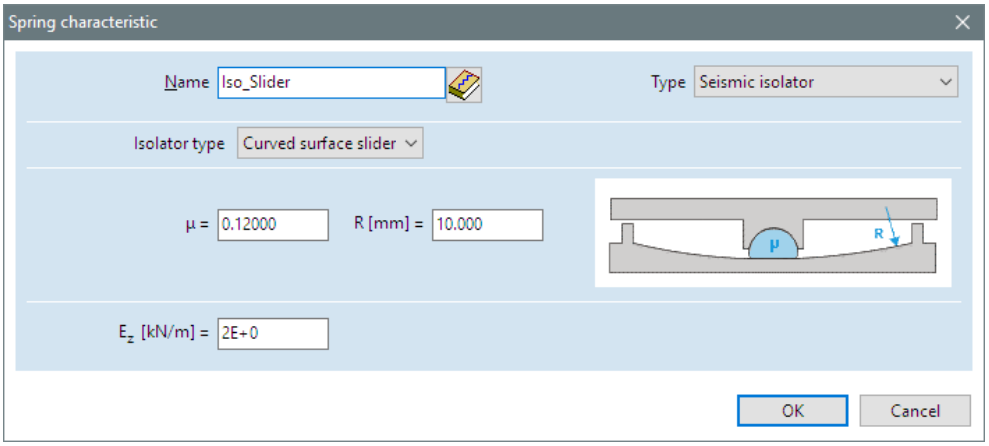
Three types of seismic isolators are available: *Rubber bearing, Curved surface slider* and *Other type.*

Rubber bearing



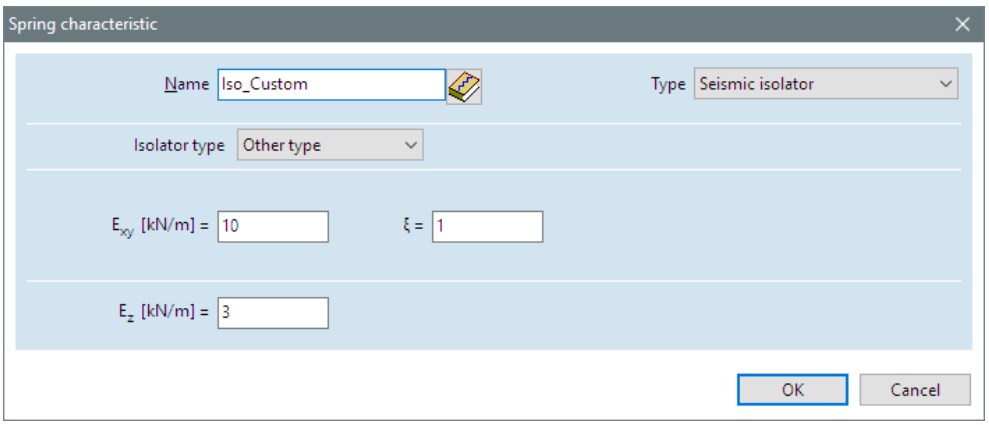
K_1 *[kN/m]* Initial value of the horizontal stiffness
K_T *[kN/m]* Horizontal stiffness in plastic state
F_1 *[kN]* Yield force
E_z *[kN/m]* Vertical stiffness

Curved surface slider



μ	Friction coefficient
R [mm]	Equivalent curvature radius
Ez [kN/m]	Vertical stiffness

Other type



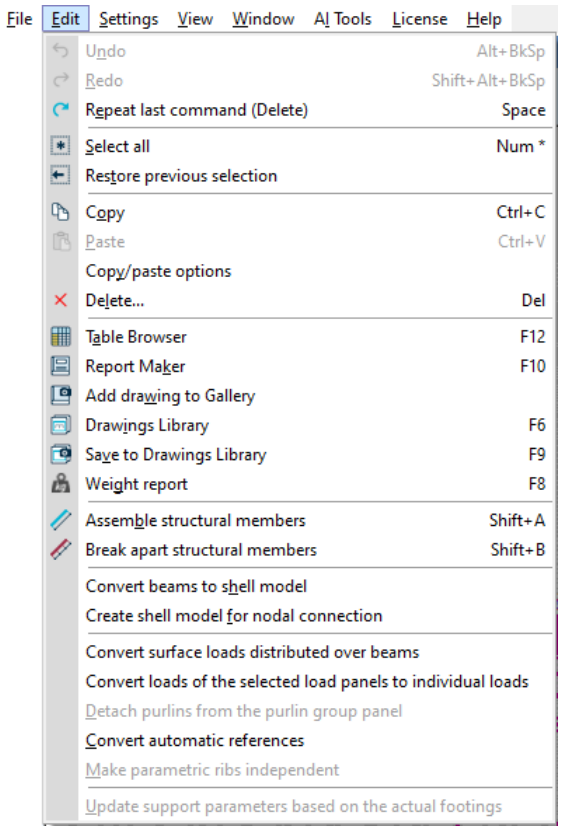
Exy [kN/m]	Effective horizontal stiffness
ξ	Damping ratio
Ez [kN/m]	Vertical stiffness

For details see **4.9.21 Spring**

3.1.18 Exit

[Ctrl]+ [Q] Exits the program.

3.2 Edit



3.2.1 Undo

Alt+BkSp

Undoes the effect of the previous commands. To undo a sequence of actions (more levels), click the down arrow next to the Undo icon, and then select the actions you want to undo based on the time or type of the commands.
You can set the number of undo/redo levels (maximum 99) in the Main menu *Settings / Preferences / Data integrity* dialog box.

3.2.2 Redo

Shift+Alt+BkSp

Undoes the undo command or goes forward to reverse one or more undo commands. You can select the actions you want to redo based on the time or type of the commands.

3.2.3 Repeat last command

[Space]

Launches the last command again if it was started by clicking on an icon or pressing a keyboard shortcut. The name of the command is also displayed.

3.2.4 Select All

**Num *

See 2.16.1 Selection

3.2.5 Restore previous selection



See 2.16.1 Selection

3.2.6 Copy



[Ctrl]+ [C]

Copies the selected elements of the model to the Clipboard. If nothing is selected but there are active parts, active parts are copied. If neither selection nor active parts are present the entire model is copied. This function copies the drawing of the current graphics window to the clipboard like in earlier versions but this operation can be deactivated.

3.2.7 Paste



[Ctrl]+ [V]

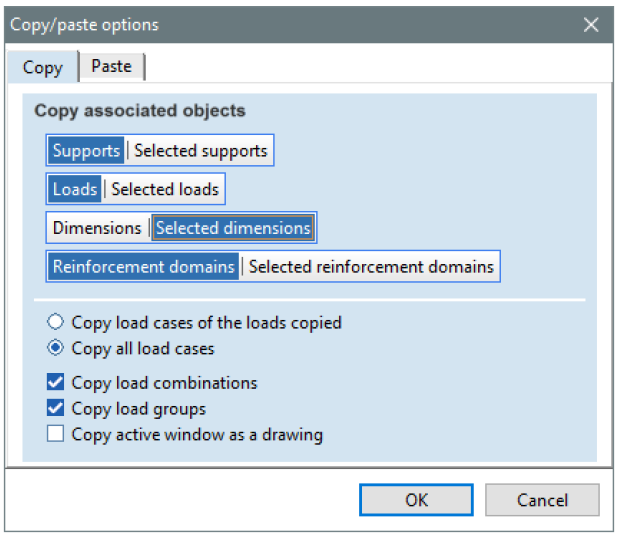
Pastes AXISVM elements from the Clipboard. For paste options see next chapter: Copy / paste options.

3.2.8 Copy / paste options

Copy options

Selected elements are always copied to the Clipboard. User-defined parts containing the selected elements are also copied.
If domains, beams, ribs, trusses are copied certain associated objects (supports, loads, dimension lines, reinforcement domains) are also copied.

If you want to control which associated objects should be copied select them and choose one or more of the following options: *Selected supports / Selected loads / Selected dimensions / Selected reinforcement domains.*
Load cases are copied with loads. If you want to copy all load cases choose *Copy all load cases* instead of *Copy load cases of the loads copied.*
Load combinations and load groups can also be copied.
Turn on *Copy active window as a drawing* to copy the active window as graphics as well (it was the only option in earlier versions).

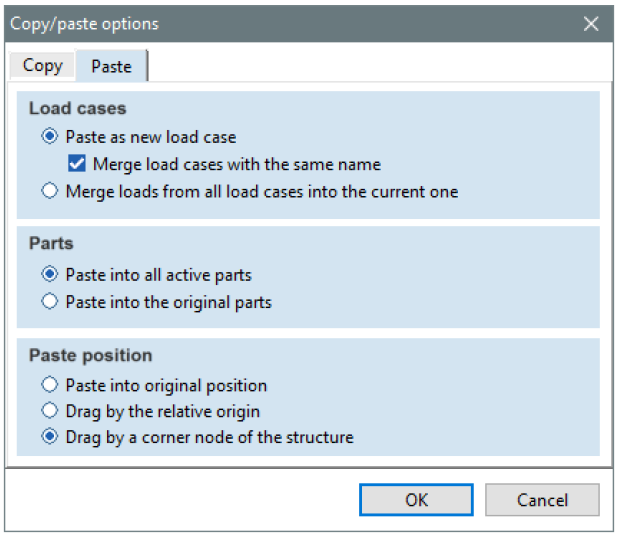


Paste options

Load cases

Pasting of load cases can be controlled with the following options:

Paste as new load case: load cases found on the Clipboard are copied as new load cases. If *Merge load cases with the same name* is turned on and the model has load cases with the same name as the clipboard load case these load cases will be merged (loads of the clipboard load case will be added to model load case). This option must be turned on when copying within the model to avoid creating unnecessary load cases.



Merge loads from all load cases into the current one. This option copies all loads from all clipboard load cases into the current load case of the model.

Parts
User-defined parts containing the selected elements are also copied to the clipboard. The first option is to paste elements of parts into all active parts of the model. The second option is to paste the parts themselves.

Paste position
There are three options.
Paste into original position: pasted elements will get into their original coordinate position. *Drag by the relative origin / Drag by a corner node of the structure:* If one of these options are selected paste position can be defined by clicking the left mouse button. In the first case the clicked position will become the position of the relative origin in the source model when the elements were copied. In the other case the clicked position will become the position of an automatically identified corner of the copied structure.

3.2.9 Delete

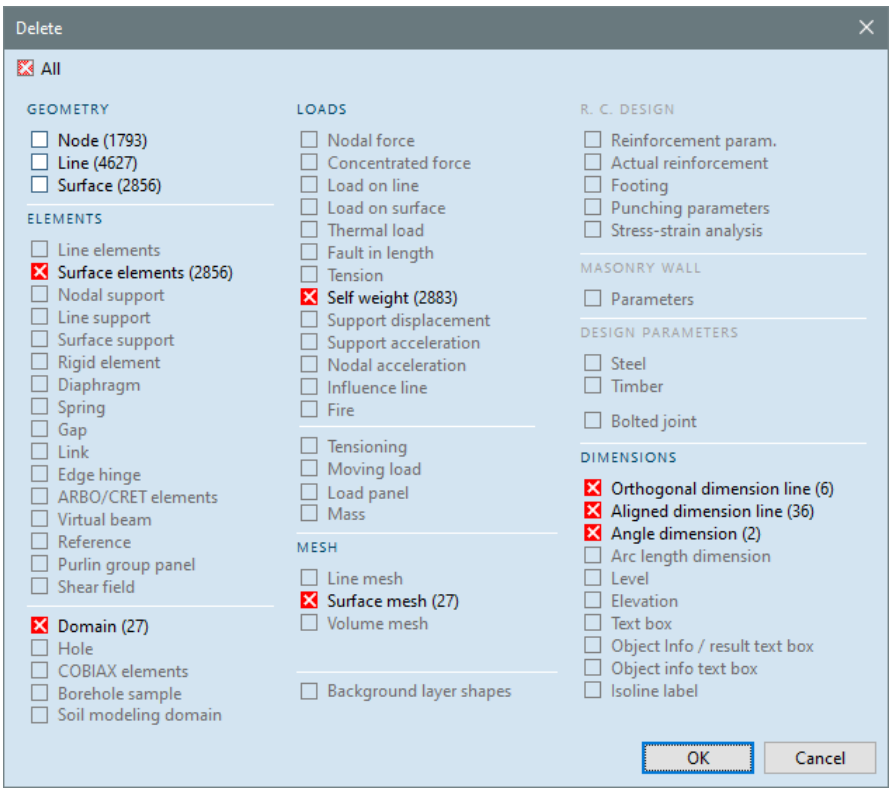


[Del]

Deletes the selected entities. If no elements are selected it brings up the Selection icon bar and then the Delete dialog window.
Lets you delete the selected geometric entities.
To delete:

1. Select the geometric entities to be deleted. You can select them by holding the **[Shift]** key pressed while you click on the entities with the left mouse button or use the Selection Icon Bar.
2. Press the **[Del]** key. If there is no selection, the selection toolbar appears and objects can be selected for deletion. See 2.16.1 Selection.
3. Enable the check-boxes of the entities you want to delete.
4. Press the **OK** button, to finish and close the dialog window.

In the dialog window the check-boxes are active or inactive according to the contents of the current selection set (intended for deletion).



Geometry

Lets you select geometric entities for deletion. Deleting geometric entities that have assigned finite elements, will result in the deletion of its finite elements and of the associated loads.

Elements	Lets you select finite elements and related objects for deletion. Deleting finite elements will not delete the respective geometric entity, but will delete the loads.
Loads	Lets you remove the selected load from elements.
Mesh	Lets you remove mesh from domains and/or eline elements.
R.C. Design	Lets you select the reinforcement parameters attached to the selected elements for deletion. Footing parameters are also deleted.
Masonry wall	Lets you select the masonry wall parameters.
Design parameters	Lets you select the steel / timber design parameters attached to the selected elements for deletion.
Dimensions	Lets you select the dimension lines, text boxes etc. for deletion.

3.2.10 Table Browser



[F12]

See 2.9 Table Browser

3.2.11 Report Maker



[F10]

See 2.10 Report Maker

3.2.12 Saving drawings and design result tables



Add drawing to Gallery **[F9]**

You can save drawings from AXISVM in many different contexts: you can save AXISVM main windows, beam displacement and internal forces diagrams, steel design results, nonlinear calculation results, reinforced concrete column and beam design diagrams, bolted joint diagrams. In case of a divided view you can select to save all windows or the active one only.

☞ **Drawings Library is another way to store diagrams. While Gallery contains static image files, the Drawings Library uses associative drawings following changes in the model.**
See 2.13 Drawings Library

Which file format to use?

Bitmap formats *(.BMP, .JPG)* store the pixels of the diagram, so Windows metafiles provide higher resolution when printed. JPG is a compressed format with a slight loss of quality but these files are much smaller than BMPs.

Windows metafiles *(.WMF, .EMF)* store a series of drawing commands so they can be scaled and printed in any size in the same quality. However if you choose hidden line removal or a rendered view drawn by OpenGL technology metafiles will contain only bitmaps.
To get a high resolution rendered view print the picture directly.

Drawings will be saved to a subfolder *Images_modelname* automatically created under the folder of the model file. These pictures can be inserted into a report. Do not modify the name of the subfolder *Images_modelname*.

3.2.13 Weight Report



[F8]

The weight of the entire model, selected elements or details can be listed as a table per material, per cross-section or surface type. *Weights per material* also displays weight of the footings.
Weight report / Rebar statistics displays the total amount of reinforcement bars placed in surface elements, beams and columns. *Detailed rebar statistics* breaks down the rebar data by type (slab, beam, column) position and diameter.
The *Purlin statistics* table contains the length, mass, surface area, and volume of purlin-type line elements (See 4.9.11 Modeling of cold-formed steel purlins). The values in the table also take into account the overlapping profiles and their lengths in the overlap zones.
The *Purlin fastener statistics for purlins* table is displayed if there are purlins or girts designed using the SD3 module for which the rafter-purlin connection parameters and the number of bolts have been specified. This fastener table summarizes the bolt quantities and provides an itemized list to the user to simplify the preparation of price quotations and the ordering of materials.

Detailed rebar statistics

Element	Type	Rebar position	Ø	Σ L [m]	m* [kg/m]	Σ m [kg]	Σ A_c [m^2]	Σ V_c [m^3]	Σ m/Σ V_c [kg/m^3]
1	Slab	axb/axb	Ø16	315.000	1.578	497.177	63.000	15.750	32
2		ayb/ayb	Ø16	315.000	1.578	497.177	63.000	15.750	32
3		axt/axt	Ø16	315.000	1.578	497.177	63.000	15.750	32
4		ayt/ayt	Ø16	315.000	1.578	497.177	63.000	15.750	32
5	Surface elements	axb+ayb/axb+ayb	Ø16	630.000	1.578	994.354	63.000	15.750	63
6		axt+ayt/axt+ayt	Ø16	630.000	1.578	994.354	63.000	15.750	63
8	Beam	Bottom	Ø16	18.000	1.578	28.410	---	1.080	26
9		Top	Ø14	18.000	1.208	21.751	---	1.080	20
10	Column	---	Ø16	192.000	1.578	303.041	---	1.440	210
11	Column stirrups		Ø8	65.280	0.395	25.758	---	1.440	18
			Total	**1553.280**		**2367.658**	**63.000**	**18.378**	**129**

3.2.14 Assemble structural members



Shift+A

AXISVM handles line elements as structural members. It means that *Meshing of line elements* on the *Mesh* tab creates finite elements but the line elements themselves are not divided. The *Find structural members* menu command joins adjacent line elements into a single element until a breaking point is found. A breaking point is defined by different local *x* or *z* directions, different material, cross-section or eccentricity, end release or a domain boundary. Line elements must be on the same line or on the same arc.

3.2.15 Break apart structural members

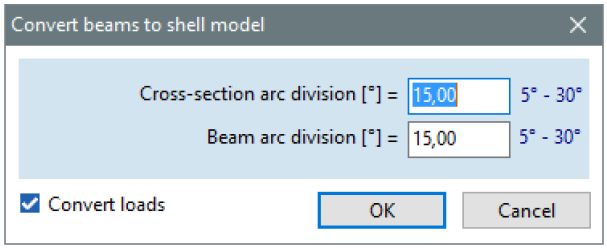


Shift+B

The *Break apart structural members* menu command breaks apart line elements created with the *Assemble structural members* command.

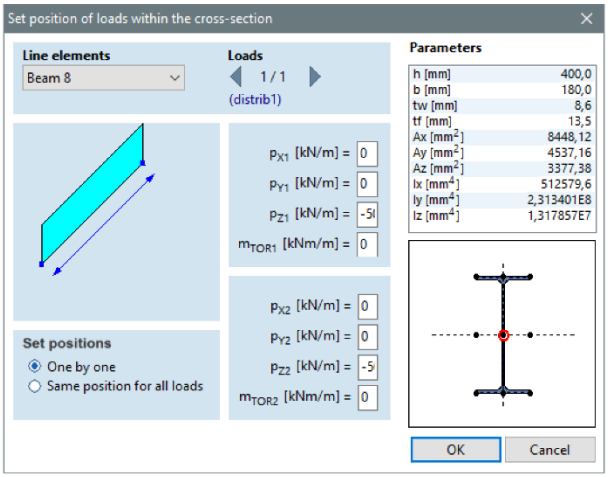
3.2.16 Convert beams to shell model

Selected beams can be converted to shell models. A shell model consists of shell elements created and connected according to the beam length and cross-section. Preferences for conversion of cross-section and beam arcs into polygons can be set in the parameter dialog.



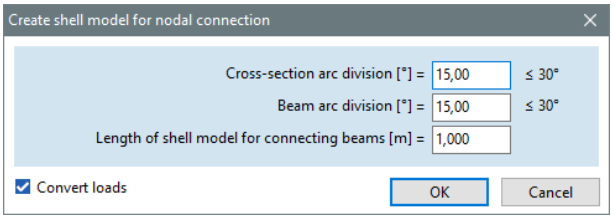
Convert loads

If *Convert loads* is selected beam loads can be converted to shell loads. To do this the user must specify the load position within the cross-section. Any of the nine points of the cross-section bounding rectangle can be selected but it is reasonable to choose a point actually on the section. Positions can be set one by one or in single step for all loads.
Select a line element from the list and select a load to convert. The load values can also be changed if necessary. If all loads are converted you can close the dialog.



3.2.17 Create shell model for nodal connection

Parts of beams connecting to selected nodes can be converted to shell model. The parameters are the same as above but the length of conversion can be set. The shell model is connected to the remaining part of the beam through rigid bodies.
Convert loads works the same way as described above.



3.2.18 Convert surface loads distributed over beams

This menu item converts selected surface loads distributed over beams into individual distributed beam loads.

3.2.19 Convert loads of the selected load panels to individual loads

Loads generated by the distribution algorithm from loads of load panels can be converted to individual loads. After the conversion they can be modified or deleted but cannot be updated form the load panel.

3.2.20 Convert automatic references

This menu item converts automatic references assigned to line or surface elements into reference vectors.

3.2.21 Make parametric ribs independent

If the model contains a parametric ribbed domain or a composite parametric ribbed domain (See **4.9.6.4 Parametric ribbed plates, 4.9.6.2 Composite ribbed domain**) created with actual ribs this command converts the parametric ribbed domain to a normal one. Ribs are not removed but become independent of the parametric grid.

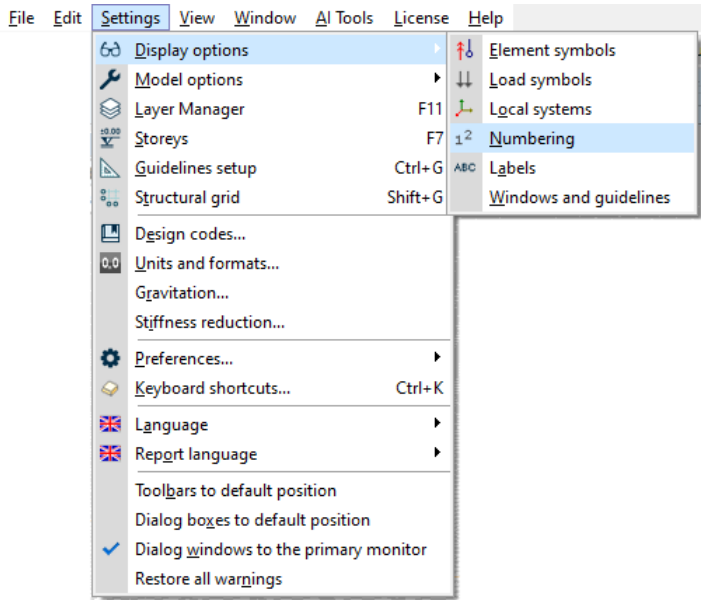
3.2.22 Update support parameters based on the actual footings

If support stiffnesses were estimated by specifying the soil layer structure and footing parameters, and the footing design was carried out based on the results of the structural analysis, it is possible to recalculate the support stiffnesses by adopting the footing parameters and soil properties used in footing design. To do this, select the nodal supports and line supports to be recalculated; then, by selecting this command from the menu the support stiffness values will be updated.
If stiffness calculation was carried out using an estimated load, this command will also import the force used in the settlement calculation during footing design.
See also
4.9.14.1.2 Estimating support stiffness from soil profile and footing parameters,
4.9.15.1.2 Estimating support stiffness from soil profile and footing parameters

3.3 Settings

3.3.1 Display options

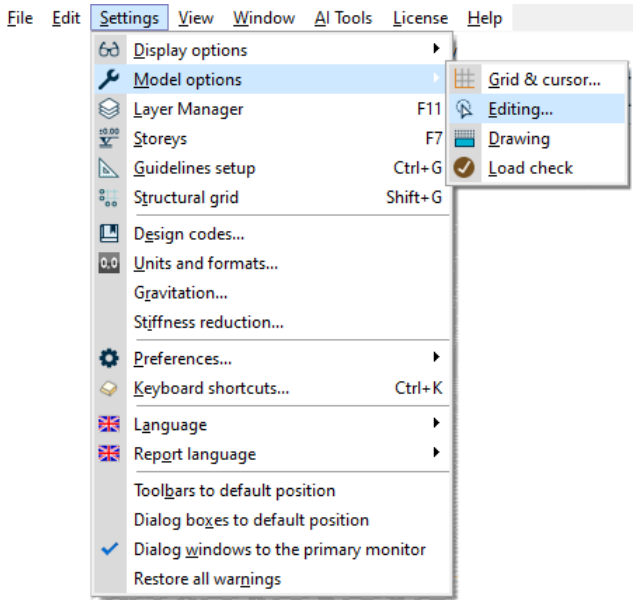




See 2.16.21 Display options

3.3.2 Model options



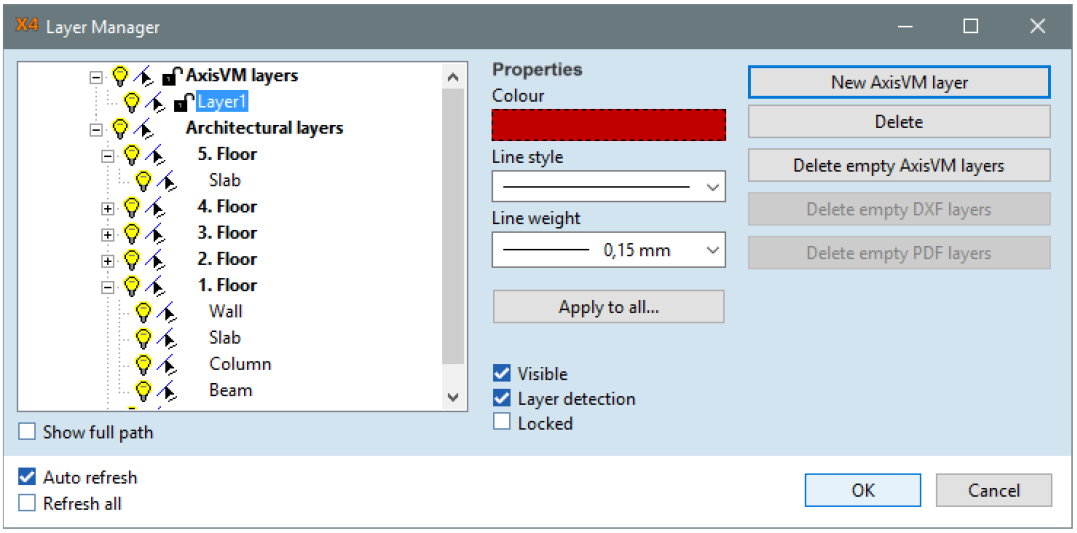


See 2.16.22 Model options

3.3.3 Layer Manager



[F11]



The Layer Manager allows you to manage AXISVM layers, imported DXF or ArchiCAD layers. While only one ArchiCAD layer can be imported, multiple DXF layers are allowed.
If no AXISVM layers are defined AXISVM automatically creates a new layer for dimension lines with the name *Dimensions*.
On the left side of the Layer Manager dialog a tree view of the available layers is displayed. If you select (highlight) a DXF layer in the tree, you can modify its properties in the right side (Name, Color, Style, Size). If you select the main DXF file entry of the tree, you can modify all the DXF layers at a time. Properties of AXISVM structural layers cannot be modified.

Apply to All: When using this button, a dialogue window will allow you to select the items in the DXF layers that will have their properties set based on the layer's settings.

The visibility of the layers or DXF files can also be set by clicking on the bulb or cursor symbol next to the layer or file name.

New AXISVM Layer Creates a new AXISVM layer. You can set the layer's name, color, line style, and width.

Delete More than one layer or group can be selected and deleted by the **[Del]** key.

Delete Empty AXISVM Layer Deletes all AXISVM layers that are empty (contain no entities).

Delete Empty DXF Layer Deletes all imported DXF layers that are empty (contain no entities).

Delete Empty PDF Layer Deletes all imported PDF layers that are empty (contain no entities).

...Apply to all Applies selected layer properties (color, line style and width) to all objects on the layer.

Visible Sets the layer visibility.

Equivalent to clicking on the light bulb icon.

Layer detection If activated the mouse will detect the underlying objects of the layer.

Equivalent to clicking on the arrow icon.

Locked Locked layers cannot be edited.

Equivalent to clicking on the lock icon.

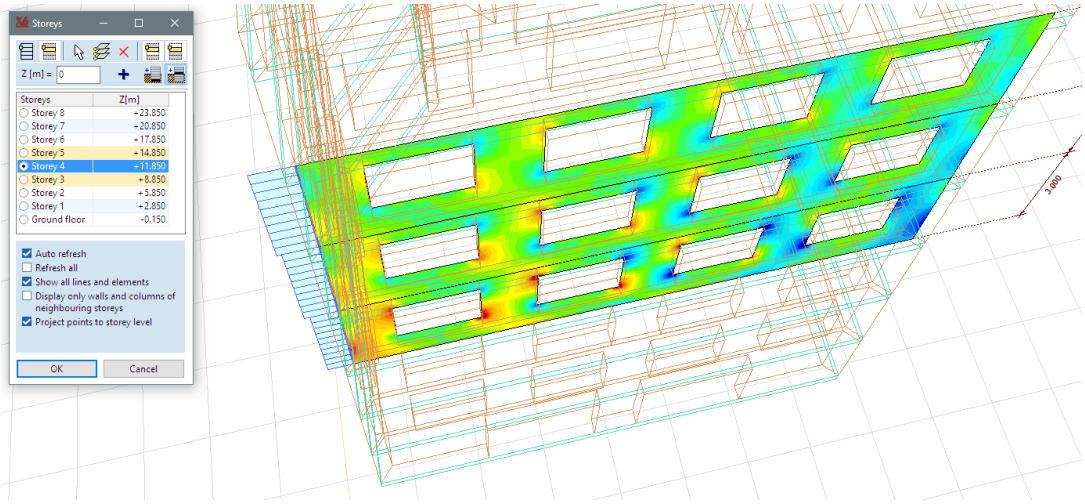
Show full path If activated, the tree view will display names of imported files with their full path.

Auto Refresh If activated, changing layer properties or their visibility immediately updates the main window.

Refresh All If activated, all views of the main window reflect changes, otherwise only the active view is affected.

3.3.4 Storeys

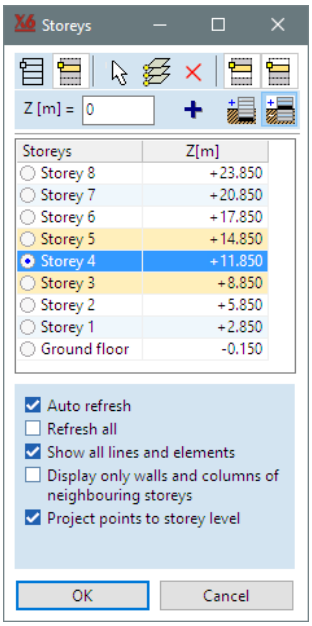
±0.00 [F7]



Storeys are to make it easier to overview and edit the model. They can be defined before drawing the model or assigned to an existing structure.
A storey is a workplane parallel to the global X-Y plane, with a given Z position. If a storey is selected mouse movements will be projected to the plane of the storey even if you find an element at a different Z position. Coordinates will always be projected to the storey plane to help tracing objects at different levels.
Storeys are always listed by decreasing Z position, having automatic names. Changing the report language makes storey names change.
Elements are considered to be part of a storey if their lowest Z coordinate is greater than or equal to the storey level but less than the next storey level. Therefore, if a multi-story column or wall was defined as a single element it will appear only at the lowest level. To change this behaviour the element has to be cut with storey planes.
New elements will be linked to their storey automatically.

☞ **If torsion effects have to be taken into account in seismic analysis, seismic storeys have to be defined separately in the seismic parameters dialog.**
If there are no seismic storeys defined, the program displays the shear center of storeys specified here.
See 3.3.4.1 Shear centers of storeys

Stories can be managed in the following dialog.



Turn off storeys 

If this button is down no storeys are displayed. Windows will show the entire structure or the active parts. storeys can be added or deleted in this state as well.

Display current storey 

If this button is down and an active storey is chosen the active storey will be displayed. The active storey can be chosen by clicking the radio button before its name.
Selection status of the list items is independent of this choice. More than one storey can be selected. Ctrl+click adds individual list items to the selection, Shift+click adds ranges to the selection. Delete operation works on the selected storeys and not on the active storey.
Active storey is yellow. If a neighbouring storey is also activated to help tracing objects on that storey, it appears as light yellow.

☞ **There can be only one active storey. However, display of neighbouring storeys is also possible. Editing will be constrained to the active storey.**

Pick up Click this icon to get back to the model and click one or more nodes to pick up Z coordinates. Close the process by clicking on an empty area. Z coordinates will be added to the list of storeys.



Enter a new storey Enter the Z coordinate into the edit field and click the + button. A new storey will be added to the list.

Find



If you have an existing multi-storey structure with slabs, you can find and add Z coordinates of horizontal domains to the list with one click. If not the all horizontal domains refer to a real storey you can delete unnecessary storeys later.

☞ **Storey position cannot be changed. Delete the storey and define a new one.**

Delete



Deletes selected storeys. Remaining storeys will be renamed and storey assignments of the elements will be updated automatically.

☞ **Deleting a storey does not delete any element.**

Display the storey below the current storey



Display the storey above the current storey



If any of these buttons is down elements of the storey below/above the active storey is also displayed to help tracing other objects.

☞ **To display further storeys, open the Parts dialog instead where logical parts of any storey can be turned on. Choosing a new active storey overrides the parts settings.**



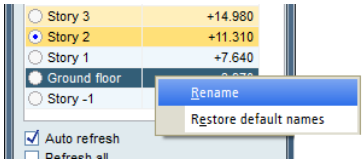
Define a new storey

Numbering of storeys



Numbering of storeys can be controlled with these buttons. If the left one is down (*Numbering of storeys from the bottom*) the lowest floor will be considered as ground floor and other storeys will have a positive number. If *Signed numbering of storeys* is selected the storey closest to the zero level will be the ground floor. Underground storeys will get a negative number, others will get positive numbers.

Renaming storeys



Click the right mouse button over a list item to get to the popup menu. Storeys can be renamed or their default names restored.

Several further options can be selected with checkboxes.

Auto refresh Drawing will be instantly updated.

Refresh all Storey settings will be applied on all windows.

Show all lines and elements If a storey is activated all slabs, walls, columns and beams associated to the storey will be displayed automatically. If other elements (geometry lines, springs, gap or link elements) on the storey are also to be seen, check this option.

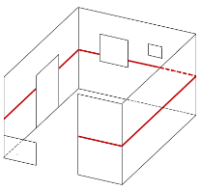
Display only walls and columns of neighbouring storey The storey above/below the active storey is usually displayed to find the supporting walls and columns. This options hides slabs and beams of the neighbouring storey.

If a storey is activated direct drawing of a column or a wall automatically starts with the storey height.

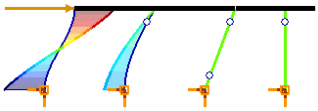
Project points to storey level This feature is to project all detected coordinates to the base level of the storey. It is useful when wall and column positions below the slab must be transferred the floor.

3.3.4.1 Shear centers of storeys

Shear center of a storey is calculated as the shear center of the cross-section created by cutting walls and columns below the storey. The ground floor has no shear center. The section of walls and columns is created at half height between the subsequent storeys. The position of the shear center depends only on this section. Holes and wall segments out of the section have no influence.
AXISVM takes into account end releases of columns and vertical edge hinges between wall segments.



Columns where end releases prevent bending in local *y* and *z* directions are ignored in the calculation. If a vertical edge hinge breaks the shear connection ($K_x = 0$) between wall segments, AXISVM also considers them disconnected.





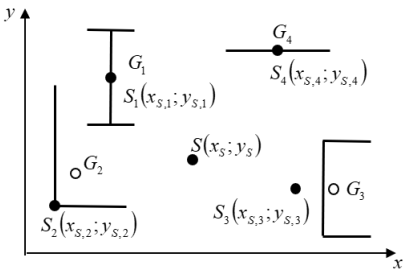
☞ **Stiffening effect of inclined elements is not considered.**

If there are inclined elements on a storey, the coordinates of the shear center are displayed in gray in the table. If there are no elements on a storey that can accommodate the shear force, a strike-through appears in the table cell.

The coordinates of the shear center [21; Chapter 7.2.1]:

$$x_S = \frac{J_{xy}\left(\sum J_{y,i}\, y_{S,i} - \sum J_{xy,i}\, x_{S,i}\right) - J_y\left(\sum J_{xy,i}\, y_{S,i} - \sum J_{x,i}\, x_{S,i}\right)}{J_x J_y - J_{xy}^2}$$

$$y_S = \frac{J_x\left(\sum J_{y,i}\, y_{S,i} - \sum J_{xy,i}\, x_{S,i}\right) - J_{xy}\left(\sum J_{xy,i}\, y_{S,i} - \sum J_{x,i}\, x_{S,i}\right)}{J_x J_y - J_{xy}^2}$$



where $J_{x,i}$, $J_{y,i}$, $J_{xy,i}$ are the weighted local flexural inertias of the elements in the section. These are the centroidal inertias weighted by the elastic moduli: $J_{x,i} = E_i I_{x,i}$, $J_{y,i} = E_i I_{y,i}$, $J_{xy,i} = E_i I_{xy,i}$
J_x , J_y , J_{xy} are quantities summed up for the whole section: $J_x = \sum J_{x,i}$, $J_y = \sum J_{y,i}$, $J_{xy} = \sum J_{xy,i}$
Important to note that these are simple sums instead of Steiner's parallel axis formulae.
$x_{S,i}$ and $y_{S,i}$ are the global coordinates of the shear center of the elements in the section.

☞ **If seismic storeys are defined, shear centers are calculated and displayed for seismic storeys. If there are no seismic storeys in the model but logical storeys (3.3.4 Storeys) are defined, shear centers are calculated and displayed for logical storeys.**

☞ **If a storey section goes through meshed and unmeshed domains, only the meshed domains are taken into account. Unmeshed walls are considered only if all wall segments in the section are unmeshed.**

3.3.5 Guidelines



[Ctrl+G]

Guidelines
See 2.16.9 Guidelines

3.3.6 Structural Grid



[Shift+G]

Structural Grid
See 2.16.8 Structural grid

3.3.7 Design codes



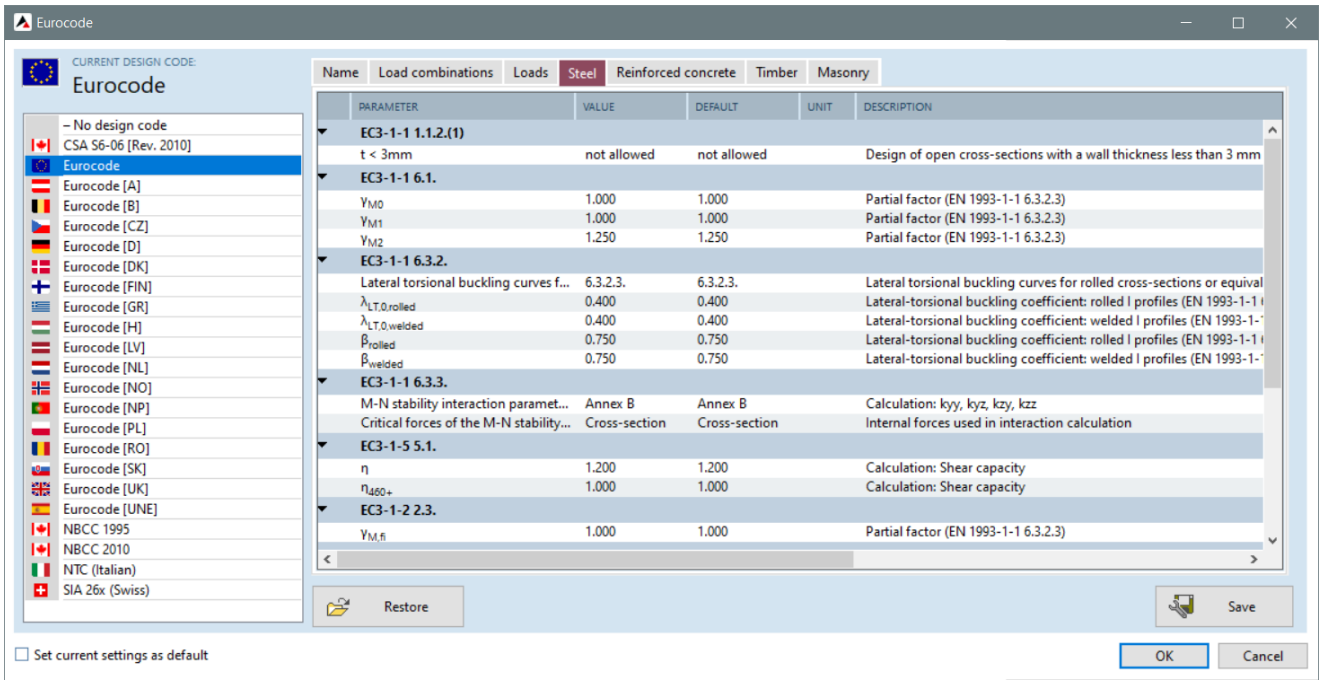
Sets the design code to be used in case of code specific tasks. Changing design code changes the method of calculating critical load combinations therefore all load group parameters but partial factors will be deleted. Seismic analysis parameters and seismic load cases will also be deleted. As material properties and certain reinforcement parameters are not the same in different codes it is recommended to revise the values you have specified.
If *Set current settings as default* is checked, new models will be created with the current design code.

In case of Eurocode-based design codes, a number of parameters prescribed in National Annexes (NAs) can be modified in AXISVM. Structural design according to NAs not included in the software can be performed by the modification of NA parameters of the basic Eurocode. The name of applied design codes are listed among the design code parameters in the *Name* tab.

The table displays the user value, default value, unit, description and code reference for each parameter. User values are editable. Click OK, to apply them. The current set of user values will be saved/loaded with the model.
Changing the design code causes the current set of user values to be lost unless it has been saved as the default set of user values by clicking on the *Save* button. New models will use these default values. The *Default* column of the table displays the value as specified in the NA. Clicking on the *Restore* button replaces all user values with the NA defaults. The unit of measurement is shown in the *Unit* column.

According to the rules and regulations in NAs, the parameters can be classified in three groups:

- the parameter is given by a number (e.g., partial factors): the value can be edited directly;
- the parameter is given by a formula; in most cases the constants of the formula are specified in the NA (e.g., slenderness limit for RC columns): in this case, the constants can be edited (note: constants are often represented by letter c);
- the parameter is given by neither a number nor by a formula; (e.g., method for calculation of second order eccentricities for RC columns, buckling curves for LTB of steel beams): in this case, the method or buckling curves can be selected from a dropdown list.



Supported design codes and annexes

The following European design codes and national annexes are implemented in the software presented according to topics and nations, respectively.

3.3.7.1 Basis of structural design EN 1990

EN 1990:2021

ÖNORM EN 1990
ÖNORM B 1990-1

NBN EN 1990+A1 NL:2015
NBN EN 1990 ANB:2021

ČSN EN 1990-ed.2 (730002)
ČSN EN 1990:2004/NA-ed.A (730002)

DIN EN 1990:2021-10
DIN EN 1990/NA:2010-12
DIN EN 1990/NA/A1:2012-08

DS/EN 1990 DK/NA:2021

SFS-EN 1990+A1+AC

ELOT EN 1990/A1:2006/NA:2010

MSZ EN 1990:2021

LVS EN 1990:2003/NA:2010

NEN-EN 1990+A1+A1/C2:2019/NB:2019
NEN-EN 1990+A1+A1/C2:2019

NS-EN 1990:2002+A1:2005+AC:2010+NA:2016

NP EN 1990:2009
NP EN 1990:2009/A1:2019

PN-EN 1990:2004 /A1:2008/AC:2010 /NA:2010/Ap1/Ap2

SR EN 1990:2004/A1:2006/NA:2009

STN EN 1990/NA:2009

BS EN 1990:2002+A1:2005

UNE-EN 1990:2019

NTC 2018

SIA 260:2013
SIA 260-C1:2020

3.3.7.2 Loads and combinations EN 1991

EN 1991-1-1:2002
EN 1991-1-1:2002/AC:2009
EN 1991-1-2:2002
EN 1991-1-2:2002/AC:2009
EN 1991-1-3:2003
EN 1991-1-3:2003/AC:2009
EN 1991-1-3:2003/A1:2015
EN 1991-1-4:2005
EN 1991-1-4:2005/AC:2010
EN 1991-1-4:2005/A1:2010
EN 1991-1-5:2003
EN 1991-1-5:2003/AC:2009
EN 1991-1-6:2005
EN 1991-1-6:2005/AC:2013
EN 1991-1-7:2006
EN 1991-1-7:2006/AC:2010
EN 1991-1-7:2006/A1:2014

ÖNORM EN 1991-1-1:2011
ÖNORM B 1991-1-1:2020
ÖNORM EN 1991-1-2:2013
ÖNORM B 1991-1-2:2003
ÖNORM EN 1991-1-3:2016
ÖNORM B 1991-1-3:2022
ÖNORM EN 1991-1-4:2011
ÖNORM B 1991-1-4:2023
ÖNORM EN 1991-1-5:2012
ÖNORM B 1991-1-5:2012
ÖNORM EN 1991-1-6:2013
ÖNORM B 1991-1-6:2006
ÖNORM EN 1991-1-7:2014
ÖNORM B 1991-1-7:2007

NBN EN 1991-1-1:2002
NBN EN 1991-1-1 ANB:2007
NBN EN 1991-1-2:2003
NBN EN 1991-1-2 ANB:2008
NBN EN 1991-1-3:2003
NBN EN 1991-1-3 ANB:2007
NBN EN 1991-1-3/A1:2015
NBN EN 1991-1-4:2005
NBN EN 1991-1-4/A1:2010
NBN EN 1991-1-4 ANB:2010
NBN EN 1991-1-5:2004



NBN EN 1991-1-5 ANB:2009
NBN EN 1991-1-6:2005
NBN EN 1991-1-6 ANB:2010
NBN EN 1991-1-7:2006
NBN EN 1991-1-7 ANB:2012
NBN EN 1991-1-7 ANB:2012/AC:2017
NBN EN 1991-1-7/A1:2014



ČSN EN 1991-1-1 (730035)
ČSN EN 1991-1-1:2004/NA-ed.A (730035)
ČSN EN 1991-1-2 (730035)
ČSN EN 1991-1-3-ed.2 (730035)
ČSN EN 1991-1-3:2005/NA-ed.A (730035)
ČSN EN 1991-1-4-ed.2 (730035)
ČSN EN 1991-1-4:2007/NA-ed.A (730035)
ČSN EN 1991-1-5 (730035)
ČSN EN 1991-1-5:2005/NA-ed.A (730035)
ČSN EN 1991-1-6 (730035)
ČSN EN 1991-1-6:2006/NA-ed.A (730035)
ČSN EN 1991-1-7 (730035)
ČSN EN 1991-1-7:2007/NA-ed.A (730035)



DIN EN 1991-1-1 : 2010-12
DIN EN 1991-1-1/NA : 2010-12
DIN EN 1991-1-1/NA/A1 : 2015-05
DIN EN 1991-1-2 : 2010-12
DIN EN 1991-1-2/NA : 2015-09
DIN EN 1991-1-3 : 2010-12
DIN EN 1991-1-3/A1 : 2015-12
DIN EN 1991-1-3/NA : 2019-04
DIN EN 1991-1-4 : 2010-12
DIN EN 1991-1-4/NA : 2010-12
DIN EN 1991-1-5 : 2010-12
DIN EN 1991-1-5/NA : 2010-12
DIN EN 1991-1-6 : 2010-12
DIN EN 1991-1-6/NA : 2010-12
DIN EN 1991-1-7 : 2010-12
DIN EN 1991-1-7/A1 : 2014-08
DIN EN 1991-1-7/NA : 2019-09



DS/EN 1991-1-1 DK/NA:2013
DS/EN 1991-1-2 DK/NA:2014
DS/EN 1991-1-3 DK/NA:2015 Version 2
DS/EN 1991-1-4 DK/NA:2015
DS/EN 1991-1-5 DK/NA:2012
DS/EN 1991-1-6 DK/NA:2007
DS/EN 1991-1-7 DK/NA:2013



SFS-EN 1991-1-1 + AC
SFS-EN 1991-1-2 + AC
SFS-EN 1991-1-3 + AC + A1
SFS-EN 1991-1-4 + AC + A1
SFS-EN 1991-1-5 + AC
SFS-EN 1991-1-6 + AC
SFS-EN 1991-1-7 + A1 + AC



ΕΛΟΤ EN 1991-1-1:2002/NA:2010
ΕΛΟΤ EN 1991-1-2:2002/NA:2010
ΕΛΟΤ EN 1991-1-3:2003/NA:2010
ΕΛΟΤ EN 1991-1-4:2005/NA:2010
ΕΛΟΤ EN 1991-1-5:2004/NA:2010
ΕΛΟΤ EN 1991-1-6:2005/NA:2010
ΕΛΟΤ EN 1991-1-7:2007/NA:2010



MSZ EN 1991-1-1:2005
MSZ EN 1991-1-2:2005
MSZ EN 1991-1-3:2016
MSZ EN 1991-1-4:2007
MSZ EN 1991-1-4:2005/A1:2011
MSZ EN 1991-1-5:2005
MSZ EN 1991-1-6:2007
MSZ EN 1991-1-7:2015

LVS EN 1991-1-1+AC:2014 L
LVS EN 1991-1-1:2003/NA:2022
LVS EN 1991-1-2+AC:2014 L
LVS EN 1991-1-2:2003/NA:2015
LVS EN 1991-1-3+AC:2014 L



LVS EN 1991-1-3:2006/A1:2019
LVS EN 1991-1-3:2003/NA:2019
LVS EN 1991-1-4+A1+AC:2014 L
LVS EN 1991-1-4:2005/NA:2011
LVS EN 1991-1-5+AC:2014 L
LVS EN 1991-1-5:2004/NA:2014
LVS EN 1991-1-6+AC:2014 L
LVS EN 1991-1-6:2005 A /NA:2014
LVS EN 1991-1-7+AC:2014 L
LVS EN 1991-1-7:2007/A1:2016
LVS EN 1991-1-7:2006/NA:2015



NEN-EN 1991-1-1+C1+C11:2019
NEN-EN 1991-1-1+C1+C11:2019/NB:2019
NEN-EN 1991-1-2+C3:2019/NB:2019
NEN-EN 1991-1-2+C3:2019
NEN-EN 1991-1-3+C1+A1:2019/NB:2019
NEN-EN 1991-1-3+C1+A1:2019
NEN-EN 1991-1-4:2011+A1+C2:2011/NB:2019+C2:2023
NEN-EN 1991-1-4+A1+C2:2011
NEN-EN 1991-1-5+C1:2011
NEN-EN 1991-1-5+C1:2011/NB:2019
NEN-EN 1991-1-6:2005
NEN-EN 1991-1-6:2005/NB:2013
NEN-EN 1991-1-7+C1+A1:2015
NEN-EN 1991-1-7:2015/NB:2019



NS-EN 1991-1-1:2002+NA:2019
NS-EN 1991-1-2:2002+NA:2008
NS-EN 1991-1-3:2003+A1:2015+NA:2018
NS-EN 1991-1-4:2005+NA:2009
NS-EN 1991-1-5:2003+NA:2008
NS-EN 1991-1-6:2005+NA:2008
NS-EN 1991-1-7:2006+NA:2008



NP EN 1991-1-1:2009
NP EN 1991-1-2:2010
NP EN 1991-1-2:2010/AC:2013
NP EN 1991-1-3:2009
NP EN 1991-1-3:2009/A1:2017
NP EN 1991-1-4:2010
NP EN 1991-1-4:2010/A1:2010
NP EN 1991-1-5:2009
NP EN 1991-1-6:2021
NP EN 1991-1-7:2021



PN-EN 1991-1-1:2004 /AC:2009 /NA:2010/Ap1/Ap2
PN-EN 1991-1-2:2006 /AC:2013-07 /NA:2010/Ap1/Ap2
PN-EN 1991-1-3:2005 /A1:2015-10/AC:2009 /NA:2010/Ap1
PN-EN 1991-1-4:2008 /A1:2010/AC:2009 /NA:2010/Ap1/Ap2/Ap3
PN-EN 1991-1-5:2005 /AC:2009 /NA:2010/Ap1
PN-EN 1991-1-6:2007 /AC:2013-07 /NA:2010/Ap1
PN-EN 1991-1-7:2008 /A1:2014-07/AC:2010 /NA:2015-02/Ap1/Ap2



SR EN 1991-1-1:2004/NA:2006
SR EN 1991-1-1:2004/AC:2009
SR EN 1991-1-2:2004/NA:2006
SR EN 1991-1-2:2004/AC:2013
SR EN 1991-1-3:2005/NA:2017
SR EN 1991-1-3:2005/AC:2009
SR EN 1991-1-3:2005/A1:2016
SR EN 1991-1-4:2006/NB:2017
SR EN 1991-1-4:2006/A1:2010
SR EN 1991-1-4:2006/AC:2010
SR EN 1991-1-5:2004/NA:2008
SR EN 1991-1-5:2004/AC:2009
SR EN 1991-1-6:2005/NB:2008
SR EN 1991-1-6:2005/AC:2013
SR EN 1991-1-7:2007/NB:2017
SR EN 1991-1-7:2007/A1:2015
SR EN 1991-1-7:2007/AC:2010

STN EN 1991-1-1/NA:2004
STN EN 1991-1-2/NA:2007
STN EN 1991-1-3/NA:2004
STN EN 1991-1-3/NA/Z1:2010
STN EN 1991-1-4/NA:2008
STN EN 1991-1-4/NA/Z1:2010



STN EN 1991-1-5/NA:2008
STN EN 1991-1-6/NA:2008
STN EN 1991-1-6/NA/Z1:2010
STN EN 1991-1-7/NA:2008



BS EN 1991-1-1:2002
BS EN 1991-1-2:2002
BS EN 1991-1-3:2003+A1:2015
BS EN 1991-1-4:2005+A1:2010
BS EN 1991-1-5:2003
BS EN 1991-1-6:2005
BS EN 1991-1-7:2006+A1:2014



UNE-EN 1991-1-1:2019
UNE-EN 1991-1-2:2019
UNE-EN 1991-1-3:2018
UNE-EN 1991-1-4:2018
UNE-EN 1991-1-5:2018
UNE-EN 1991-1-6:2018
UNE-EN 1991-1-7:2018



NTC 2018



SIA 261:2020
SIA 261-C1:2023

3.3.7.3 Design of reinforced concrete structures EN 1992



EN 1992-1-1:2004
EN 1992-1-1:2004/AC:2008
EN 1992-1-1:2004/AC:2010
EN 1992-1-1:2004/A1:2014
EN 1992-1-2:2004
EN 1992-1-2:2004/AC:2008
EN 1992-1-2:2004/A1:2019
EN 1992-2:2005



ÖNORM EN 1992-1-1:2015
ÖNORM B 1992-1-1:2018
ÖNORM EN 1992-1-2:2019
ÖNORM B 1992-1-2:2011
ÖNORM EN 1992-2:2012
ÖNORM B 1992-2: 2019



NBN EN 1992-1-1:2005
NBN EN 1992-1-1 ANB:2010
NBN EN 1992-1-1/A1:2015
NBN EN 1992-1-2:2005
NBN EN 1992-1-2:2004/A1:2019
NBN EN 1992-1-2 ANB:2021
NBN EN 1992-2:2005
NBN EN 1992-2 ANB:2014



ČSN EN 1992-1-1-ed.2 (731201)
ČSN EN 1992-1-1:2006/NA-ed.A (731201)
ČSN EN 1992-1-2 (731201)
ČSN EN 1992-1-2:2006/NA-ed.A (731201)
ČSN EN 1992-2 (736208)
ČSN EN 1992-2:2007/NA-ed.A (736208)



DIN EN 1992-1-1 : 2011-01
DIN EN 1992-1-1/A1 : 2015-03
DIN EN 1992-1-1/NA : 2013-04
DIN EN 1992-1-1/NA/A1 : 2015-12
DIN EN 1992-1-2 : 2010-12
DIN EN 1992-1-2/A1 : 2019-11
DIN EN 1992-1-2/NA : 2010-12
DIN EN 1992-1-2/NA/A1 : 2015-09
DIN EN 1992-1-2/NA/A2 : 2021-04
DIN EN 1992-2 : 2010-12
DIN EN 1992-2/NA : 2013-04



DS/EN 1992-1-1 DK/NA:2021
DS/EN 1992-1-2 DK/NA:2011
DS/EN 1992-2 DK/NA:2015



SFS-EN 1992-1-1 + A1 + AC
SFS-EN 1992-1-2 + AC
SFS-EN 1992-2 + AC



ΕΛΟΤ EN 1992-1-1:2005/NA:2010
ΕΛΟΤ EN 1992-1-2:2005/NA:2010
ΕΛΟΤ EN 1992-2:2006/NA:2010



MSZ EN 1992-1-1:2010
MSZ EN 1992-1-1:2004/A1:2016
MSZ EN 1992-1-2:2013
MSZ EN 1992-1-2:2004/A1:2019
MSZ EN 1992-2:2009



LVS EN 1992-1-1+AC:2014 L
LVS EN 1992-1-1+AC:2014/A1:2016
LVS EN 1992-1-1:2005/NA:2020
LVS EN 1992-1-2+AC:2014
LVS EN 1992-1-2:2007/A1:2020
LVS EN 1992-1-2:2005 A /NA:2014
LVS EN 1992-2:2008 L
LVS EN 1992-2:2008/AC:2013 L
LVS EN 1992-2:2006 /NA:2009



NEN-EN 1992-1-1+C2:2011
NEN-EN 1992-1-1:2005+C2:2011/A1:2015
NEN-EN 1992-1-1+C2:2011/NB:2016+A1:2020
NEN-EN 1992-1-2+C1:2011
NEN-EN 1992-1-2+C1:2011/NB
NEN-EN 1992-1-2+C1:2011/C11:2017
NEN-EN 1992-2+C1:2011
NEN-EN 1992-2+C1:2011/NB:2016



NS-EN 1992-1-1:2004+A1:2014+NA:2021
NS-EN 1992-1-2:2004+AC:2008+NA:2010
NS-EN 1992-2:2005+NA:2010



NP EN 1992-1-1:2010
NP EN 1992-1-1:2010/A1:2019
NP EN 1992-1-1:2010/AC:2012
NP EN 1992-1-2:2010
NP EN 1992-1-2:2010/A1:2019
NP EN 1992-2:2018



PN-EN 1992-1-1:2008 /A1:2015-03/AC:2011 /NA:2018-11/Ap1/Ap2/Ap3
PN-EN 1992-1-2:2008 /A1:2019-07/AC:2008 /NA:2010/Ap1/Ap2
PN-EN 1992-2:2010 /NA:2022-11/Ap1/Ap2/Ap3



SR EN 1992-1-1:2004/NB:2008
SR EN 1992-1-1:2004/NB:2008/A91:2009
SR EN 1992-1-1:2004/AC:2012
SR EN 1992-1-1:2004/A1:2015
SR EN 1992-1-2:2006/NA:2009
SR EN 1992-1-2:2006/A1:2019
SR EN 1992-1-2:2006/AC:2008
SR EN 1992-2:2006/NA:2009
SR EN 1992-2:2006/AC:2008



STN EN 1992-1-1/NA:2008
STN EN 1992-1-2/NA:2008
STN EN 1992-2/NA:2008



BS EN 1992-1-1:2004+A1:2014
BS EN 1992-1-2:2004+A1:2019
BS EN 1992-2:2005



UNE-EN 1992-1-1:2013
UNE-EN 1992-1-1:2013/A1:2015
UNE-EN 1992-1-2:2011
UNE-EN 1992-1-2:2011/A1:2021
UNE-EN 1992-2:2013



NTC 2018



SIA 262:2025

3.3.7.4 Design of steel structures EN 1993



EN 1993-1-1:2005
EN 1993-1-2:2005
EN 1993-1-2:2005/AC:2009
EN 1993-1-3:2006
EN 1993-1-3:2006/AC:2009
EN 1993-1-5:2006
EN 1993-1-5:2006/AC:2009
EN 1993-1-5:2006/A1:2017
EN 1993-1-5:2006/A2:2019
EN 1993-1-8:2005
EN 1993-1-8:2005/AC:2009
EN 1993-1-9:2005
EN 1993-1-9:2005/AC:2009
EN 1993-1-10:2005
EN 1993-1-10:2005/AC:2009



ÖNORM EN 1993-1-1:2014
ÖNORM B 1993-1-1:2017
ÖNORM EN 1993-1-2:2012
ÖNORM B 1993-1-2:2022
ÖNORM EN 1993-1-3:2010
ÖNORM B 1993-1-3:2007
ÖNORM EN 1993-1-4:2021
ÖNORM B 1993-1-4:2007
ÖNORM EN 1993-1-5:2020
ÖNORM B 1993-1-5:2008
ÖNORM EN 1993-1-6:2017
ÖNORM B 1993-1-6:2008
ÖNORM EN 1993-1-7:2007
ÖNORM B 1993-1-7:2008
ÖNORM EN 1993-1-8:2012
ÖNORM B 1993-1-8:2015
ÖNORM EN 1993-1-9:2013
ÖNORM B 1993-1-9:2017
ÖNORM EN 1993-1-10:2010
ÖNORM B 1993-1-10:2006



NBN EN 1993-1-1:2005
NBN EN 1993-1-1/A1:2014
NBN EN 1993-1-1 ANB:2018
NBN EN 1993-1-2:2005
NBN EN 1993-1-2 ANB:2010
NBN EN 1993-1-3:2007
NBN EN 1993-1-3 ANB:2011
NBN EN 1993-1-5:2007
NBN EN 1993-1-5 ANB:2011
NBN EN 1993-1-5:2006/A1:2017
NBN EN 1993-1-5:2006/A2:2019
NBN EN 1993-1-8:2005
NBN EN 1993-1-8 ANB:2010
NBN EN 1993-1-9:2005
NBN EN 1993-1-9 ANB:2010
NBN EN 1993-1-10:2005
NBN EN 1993-1-10 ANB:2010



ČSN EN 1993-1-1-ed.2 (731401)
ČSN EN 1993-1-1:2006/NA-ed.A (731401)
ČSN EN 1993-1-2 (731401)
ČSN EN 1993-1-2:2006/NA-ed.A (731401)
ČSN EN 1993-1-3 (731401)
ČSN EN 1993-1-3:2008/NA-ed.A (731401)
ČSN EN 1993-1-5-ed.3 (731401)
ČSN EN 1993-1-5:2008/NA-ed.A (731401)
ČSN EN 1993-1-8-ed.2 (731401)
ČSN EN 1993-1-8:2006/NA-ed.A (731401)
ČSN EN 1993-1-9-ed.2 (731401)
ČSN EN 1993-1-9:2006/NA-ed.A (731401)
ČSN EN 1993-1-10-ed.2 (731401)
ČSN EN 1993-1-10:2006/NA-ed.A (731401)



DIN EN 1993-1-1 : 2010-12
DIN EN 1993-1-1/A1 : 2014-07
DIN EN 1993-1-1/NA : 2022-10
DIN EN 1993-1-2 : 2010-12
DIN EN 1993-1-2/NA : 2010-12

DIN EN 1993-1-3 : 2010-12
DIN EN 1993-1-3/NA : 2017-05
DIN EN 1993-1-4 : 2015-10
DIN EN 1993-1-4/A2 : 2021-02
DIN EN 1993-1-4/NA : 2020-11
DIN EN 1993-1-5 : 2020-07
DIN EN 1993-1-5/NA : 2018-11
DIN EN 1993-1-6 : 2017-07
DIN EN 1993-1-6/NA : 2021-06
DIN EN 1993-1-7 : 2010-12
DIN EN 1993-1-7/NA : 2010-12
DIN EN 1993-1-8 : 2010-12
DIN EN 1993-1-8/NA : 2020-11
DIN EN 1993-1-9 : 2010-12
DIN EN 1993-1-9/NA : 2010-12
DIN EN 1993-1-10 : 2010-12
DIN EN 1993-1-10/NA : 2016-04



DS/EN 1993-1-1 DK/NA:2019
DS/EN 1993-1-2 DK/NA:2007
DS/EN 1993-1-3 DK/NA:2019
DS/EN 1993-1-4 DK/NA:2019
DS/EN 1993-1-5 DK/NA:2007
DS/EN 1993-1-6 DK/NA:2021
DS/EN 1993-1-7 DK/NA:2007
DS/EN 1993-1-8 DK/NA:2019
DS/EN 1993-1-9 DK/NA:2007
DS/EN 1993-1-10 DK/NA:2019



SFS-EN 1993-1-1
SFS-EN 1993-1-1/A1
SFS-EN 1993-1-2
SFS-EN 1993-1-3 + AC
SFS-EN 1993-1-5:2006 + A1:2017
SFS-EN 1993-1-8
SFS-EN 1993-1-9
SFS-EN 1993-1-10



ΕΛΟΤ EN 1993-1-1:2005/NA:2010
ΕΛΟΤ EN 1993-1-1:2005/A1:2014/NA:2017
ΕΛΟΤ EN 1993-1-2:2005/NA:2010
ΕΛΟΤ EN 1993-1-3:2006/NA:2010
ΕΛΟΤ EN 1993-1-5:2006/NA:2010
ΕΛΟΤ EN 1993-1-8:2005/NA:2010
ΕΛΟΤ EN 1993-1-9:2005/NA:2010
ΕΛΟΤ EN 1993-1-10:2005/NA:2010



MSZ EN 1993-1-1:2009
MSZ EN 1993-1-1:2005/A1:2015
MSZ EN 1993-1-2:2013
MSZ EN 1993-1-3:2007
MSZ EN 1993-1-5:2012
MSZ EN 1993-1-5:2006/A1:2018
MSZ EN 1993-1-5:2006/A2:2019
MSZ EN 1993-1-8:2012
MSZ EN 1993-1-9:2011
MSZ EN 1993-1-10:2011



LVS EN 1993-1-1:2005/NA:2015
LVS EN 1993-1-2:2006 +AC L
LVS EN 1993-1-2:2006/AC:2016
LVS EN 1993-1-2:2005 A /NA:2013
LVS EN 1993-1-3+AC:2014 L
LVS EN 1993-1-3:2007 /NA:2014
LVS EN 1993-1-5+AC:2013 L
LVS EN 1993-1-5+AC:2013/A1:2019
LVS EN 1993-1-5:2007/A2:2019 L
LVS EN 1993-1-5:2007 /NA:2013
LVS EN 1993-1-8+AC:2015
LVS EN 1993-1-8:2006/AC:2016
LVS EN 1993-1-8:2005 /NA:2014
LVS EN 1993-1-9+AC:2012 L
LVS EN 1993-1-9:2005 /NA:2012
LVS EN 1993-1-10+AC:2012 L
LVS EN 1993-1-10:2005 A /NA:2012
NEN-EN 1993-1-1+C2+A1/NB:2016
NEN-EN 1993-1-1+C2+A1:2016
NEN-EN 1993-1-1+C2/A1:2014



NEN-EN 1993-1-2+C2:2011/NB:2015
NEN-EN 1993-1-2+C2:2011
NEN-EN 1993-1-3:2006/NB:2011
NEN-EN 1993-1-3:2006
NEN-EN 1993-1-3:2006/C3:2009
NEN-EN 1993-1-5:2006/NB:2011
NEN-EN 1993-1-5:2006+C1:2012
NEN-EN 1993-1-8+C2:2011
NEN-EN 1993-1-8+C2:2011/NB:2011
NEN-EN 1993-1-8+C2:2011/C11:2016
NEN-EN 1993-1-9+C2:2012
NEN-EN 1993-1-9:2006/NB:2011
NEN-EN 1993-1-10+C2:2011
NEN-EN 1993-1-10:2006/NB:2007
NEN-EN 1993-1-10+C2:2011/C11:2015



NS-EN 1993-1-1:2005+A1:2014+NA:2015
NS-EN 1993-1-2:2005+NA:2009
NS-EN 1993-1-3:2006+NA:2015
NS-EN 1993-1-5:2006+AC:2009+A2:2019+NA:2019
NS-EN 1993-1-8:2005+NA:2009
NS-EN 1993-1-9:2005+NA:2010
NS-EN 1993-1-10:2005+NA:2009



NP EN 1993-1-1:2010
NP EN 1993-1-1:2010/A1:2017
NP EN 1993-1-2:2010
NP EN 1993-1-5:2012
NP EN 1993-1-5:2012/A1:2019
NP EN 1993-1-5:2012/A2:2019
NP EN 1993-1-8:2010
NP EN 1993-1-9:2010
NP EN 1993-1-10:2010



PN-EN 1993-1-1:2006 /A1:2014-07/AC:2009 /NA:2010/Ap1
PN-EN 1993-1-2:2007 /AC:2009 /NA:2010/Ap1
PN-EN 1993-1-3:2008 /AC:2009 /NA:2010/Ap1
PN-EN 1993-1-5:2008 /A1:2017-07/A2:2019-09/AC:2009 /NA:2010/Ap1
PN-EN 1993-1-8:2006 /AC:2009 /NA:2011/Ap1/Ap2
PN-EN 1993-1-9:2007 /AC:2009 /NA:2010/Ap1
PN-EN 1993-1-10:2007 /AC:2009 /NA:2010/Ap1



SR EN 1993-1-1:2006/NA:2016
SR EN 1993-1-2:2006/NB:2008
SR EN 1993-1-2:2006/AC:2009
SR EN 1993-1-3:2007/NB:2008
SR EN 1993-1-3:2007/AC:2013
SR EN 1993-1-5:2007/NA:2008
SR EN 1993-1-5:2007/AC:2009
SR EN 1993-1-5:2007/A1:2018
SR EN 1993-1-5:2007/A2:2020
SR EN 1993-1-8:2006/NB:2008
SR EN 1993-1-8:2006/AC:2010
SR EN 1993-1-9:2006/NA:2008
SR EN 1993-1-9:2006/AC:2009
SR EN 1993-1-10:2006/NA:2008
SR EN 1993-1-10:2006/AC:2009



STN EN 1993-1-1/NA:2015
STN EN 1993-1-2/NA:2008
STN EN 1993-1-3/NA:2010
STN EN 1993-1-5/NA:2010
STN EN 1993-1-8/NA:2008
STN EN 1993-1-9/NA:2007
STN EN 1993-1-10/NA:2007



BS EN 1993-1-1:2005+A1:2014
BS EN 1993-1-2:2005
BS EN 1993-1-3:2006
BS EN 1993-1-5:2006+A2:2019
BS EN 1993-1-8:2005
BS EN 1993-1-9:2005
BS EN 1993-1-10:2005



UNE-EN 1993-1-1:2013
UNE-EN 1993-1-1:2013/A1:2014
UNE-EN 1993-1-2:2016
UNE-EN 1993-1-3:2012
UNE-EN 1993-1-4:2012
UNE-EN 1993-1-5:2013
UNE-EN 1993-1-5:2013/A1:2019
UNE-EN 1993-1-5:2013/A2:2020
UNE-EN 1993-1-6:2013
UNE-EN 1993-1-7:2013
UNE-EN 1993-1-8.2013
UNE-EN 1993-1-9:2013
UNE-EN 1993-1-10:2013



NTC 2018



SIA 263:2013
SIA 263-C2:2016
SIA 263-C3:2022

3.3.7.5 Design of composite structures EN 1994



EN 1994-1-1:2004
EN 1994-1-1:2004/AC:2009



ÖNORM EN 1994-1-1:2009
ÖNORM B 1994-1-1:2007



NBN EN 1994-1-1:2005
NBN EN 1994-1-1 ANB:2010



ČSN EN 1994-1-1-ed.2 (731470)
ČSN EN 1994-1-1:2006/NA-ed.A (731470)



DIN EN 1994-1-1 : 2010-12
DIN EN 1994-1-1/NA : 2010-12



DS/EN 1994-1-1 DK/NA:2019



SFS-EN 1994-1-1 + AC



ΕΛΟΤ EN 1994-1-1:2005/NA:2010



MSZ EN 1994-1-1:2010



LVS EN 1994-1-1+AC:2015
LVS EN 1994-1-1:2005/NA:2015



NEN-EN 1994-1-1+C1:2011
NEN-EN 1994-1-1+C1:2011/NB:2012



NS-EN 1994-1-1:2004+NA:2009



NP EN 1994-1-1:2011



PN-EN 1994-1-1:2008 /AC:2009 /NA:2010/Ap1



SR EN 1994-1-1:2004/NB:2008
SR EN 1994-1-1:2004/AC:2009



STN EN 1994-1-1/NA



BS EN 1994-1-1:2004



UNE-EN 1994-1-1:2013
https://www.mystandards.biz/standard/csnen-1994-1-1-2006-na-ed-a-1.1.2012.html



NTC 2018



SIA 264:2014

3.3.7.6 Design of timber structures EN 1995



EN 1995-1-1:2004
EN 1995-1-1:2004/AC:2006
EN 1995-1-1:2004/A1:2008
EN 1995-1-1:2004/A2:2014
EN 1995-1-2:2004
EN 1995-1-2:2004/AC:2009



ÖNORM EN 1995-1-1:2019
ÖNORM B 1995-1-1:2023
ÖNORM EN 1995-1-2:2011
ÖNORM B 1995-1-2:2011



NBN EN 1995-1-1:2005
NBN EN 1995-1-1/A1:2008
NBN EN 1995-1-1/A2:2014
NBN EN 1995-1-1 ANB:2012
NBN EN 1995-1-2:2005
NBN EN 1995-1-2 ANB:2012



ČSN EN 1995-1-1 (731701)
ČSN EN 1995-1-1:2006/NA-ed.A (731701)
ČSN EN 1995-1-2 (731701)
ČSN EN 1995-1-2:2006/NA-ed.A (731701)



DIN EN 1995-1-1 : 2010-12
DIN EN 1995-1-1/A2 : 2014-07
DIN EN 1995-1-1/NA : 2013-08
DIN EN 1995-1-2 : 2010-12
DIN EN 1995-1-2/NA : 2010-12



DS/EN 1995-1-1 DK/NA:2019
DS/EN 1995-1-2 DK/NA:2007



SFS-EN 1995-1-1 + A1 + A2 + AC
SFS-EN 1995-1-2 + AC



ΕΛΟΤ EN 1995-1-1:2005/NA:2010
ΕΛΟΤ EN 1995-1-2:2005/NA:2010



MSZ EN 1995-1-1:2010
MSZ EN 1995-1-1:2004/A2:2015
MSZ EN 1995-1-2:2013



LVS EN 1995-1-1+A1+A2+AC:2014
LVS EN 1995-1-1:2005 /NA:2012
LVS EN 1995-1-2+AC:2007 L
LVS EN 1995-1-2+AC:2007/AC:2016
LVS EN 1995-1-2:2005 /NA:2009



NEN-EN 1995-1-1+C1+A1:2011
NEN-EN 1995-1-1+C1+A1:2011/NB:2013
NEN-EN 1995-1-1+C1+A1:2011/C1:2012
NEN-EN 1995-1-1+C1+A1/A2:2014
NEN-EN 1995-1-2:2005/NB:2011
NEN-EN 1995-1-2+C2:2011



NS-EN 1995-1-1:2004+A1:2008+NA:2010
NS-EN 1995-1-2:2004+NA:2010



NP EN 1995-1-1:2022
NP EN 1995-1-2:2022



PN-EN 1995-1-1:2010 /A2:2014-07 /NA:2010/
PN-EN 1995-1-2:2008 /AC:2009 /NA:2010



SR EN 1995-1-1:2004
SR EN 1995-1-1:2004/AC:2006
SR EN 1995-1-1:2004/A1:2008
SR EN 1995-1-1:2004/A2:2014
SR EN 1995-1-1:2004/NB:2008
SR EN 1995-1-2:2004
SR EN 1995-1-2:2004/AC:2009
SR EN 1995-1-2:2004/NB:2008


STN EN 1995-1-1/NA:2019
STN EN 1995-1-2/NA:2011


BS EN 1995-1-1:2004+A2:2014
BS EN 1995-1-2:2004


UNE-EN 1995-1-1:2016
UNE-EN 1995-1-2:2016


NTC 2018


SIA 265: 2021

3.3.7.7 Design of masonry structures EN 1996


EN 1996-1-1:2005


ÖNORM EN 1996-1-1:2013
ÖNORM B 1996-1-1:2016


NBN EN 1996-1-1+A1:2013
NBN EN 1996-1-1+A1 ANB:2016


ČSN EN 1996-1-1 (731101)
ČSN EN 1996-1-1+A1:2013/NA-ed.A (731101)
ČSN EN 1996-1-1+A1:2013/NA-ed.A/Z1-NA (731101)


DIN EN 1996-1-1:2013
DIN EN 1996-1-1/NA:2019


DS/EN 1996-1-1 DK/NA:2019


SFS-EN 1996-1-1 + A1


ΕΛΟΤ EN 1996-1-1:2006/NA:2010


MSZ EN 1996-1-1:2005+A1:2013


LVS EN 1996-1-1:2023


NEN-EN 1996-1-1:2006+A1:2013
NEN-EN 1996-1-1:2006+A1:2013/NB:2018


NS-EN 1996-1-1:2005+A1:2012+NA:2013


NP EN 1996-1-1:2005+A1:2015


PN-EN 1996-1-1 /A1:2013-05 /NA:2014-10/Ap1/Ap2/Ap3


SR EN 1996-1-1:2006
SR EN 1996-1-1:2006/AC:2010
SR EN 1996-1-1:2006/NB:2008


STN EN 1996-1-1+A1/NA1:2013


BS EN 1996-1-1:2005+A1:2012


UNE-EN 1996-1-1:2011/A1:2013


NTC 2018


SIA266:2015

3.3.7.8 Geotechnical design EN 1997


EN 1997-1:2004


ÖNORM EN 1997-1:2014
ÖNORM B 1997-1-1
ÖNORM B 1997-1-2
ÖNORM B 1997-1-3



NBN EN 1997-1:2005
NBN EN 1997-1/A1:2014
NBN EN 1997-1 ANB:2022



ČSN EN 1997-1 (731000)
ČSN EN 1997-1:2006/NA-ed.A (731000)



DIN EN 1997-1:2014
DIN EN 1997-1/NA:2010



DS/EN 1997-1 DK/NA:2021



SFS-EN 1997-1 + A1 + AC



ΕΛΟΤ EN 1997-1:2005/NA:2010



MSZ EN 1997-1:2006
MSZ EN 1997-1:2004/A1:2015



LVS EN 1997-1+AC:2009



NEN-EN 1997-1+C1+A1:2016
NEN-EN 1997-1+C1+A1:2016/NB:2019



NS-EN 1997-1:2004+A1:2013+NA:2020



NP EN 1997-1:2010
NP EN 1997-1:2010/A1:2016
NP EN 1997-1:2010/A1:2016/Errata1:2019



PN-EN 1997-1:2008 /A1:2014-05/AC:2009 /NA:2011/Ap1/Ap2



SR EN 1997-1-1:2004
SR EN 1997-1-1:2004/A1:2014
SR EN 1997-1-1:2004/AC:2009
SR EN 1997-1-1:2004/NB:2016



STN EN 1997-1/NA:2010
STN EN 1997-1/A1:2014



BS EN 1997-1:2004+A1:2013



UNE-EN 1997-1:2016



NTC 2018



SIA 267:2013
SIA 267-C1:2016
SIA 267-C2:2018

3.3.7.9 Seismic design EN 1998



EN 1998-1:2004/A1:2013
EN 1998-3:2005
EN 1998-3:2005/AC:2013
EN 1998-5:2004



ÖNORM EN 1998-1:2013
ÖNORM B 1998-1:2017
ÖNORM EN 1998-3:2013
ÖNORM B 1998-3:2018
ÖNORM EN 1998-5:2005
ÖNORM B 1998-5:2005



NBN EN 1998-1:2005
NBN EN 1998-1 ANB:2011
NBN EN 1998-1/A1:2013NBN EN 1998-3:2005
NBN EN 1998-3:2005
NBN EN 1998-3/AC:2013
NBN EN 1998-3 ANB:2011
NBN EN 1998-5:2005
NBN EN 1998-5 ANB:2011



ČSN EN 1998-1-ed.2 (730036)
ČSN EN 1998-1:2006/NA-ed.A (730036)
ČSN EN 1998-3-ed.2 (730036)
ČSN EN 1998-3:2007/NA-ed.A (730036)
ČSN EN 1998-5 (730036)
ČSN EN 1998-5:2006/NA-ed.A (730036)



DIN EN 1998-1/NA:2021-07



DS/EN 1998-1 DK/NA:2020



SFS-EN 1998-1
SFS-EN 1998-1/A1
SFS-EN 1998-1/AC
SFS-EN 1998-3
SFS-EN 1998-3/AC
SFS-EN 1998-5



ΕΛΟΤ EN 1998-1:2005/NA:2010
ΕΛΟΤ EN 1998-3:2005/NA:2010
ΕΛΟΤ EN 1998-5:2005/NA:2010



MSZ EN 1998-1:2008
MSZ EN 1998-1:2004/A1:2013
MSZ EN 1998-3:2011
MSZ EN 1998-5:2009



LVS EN 1998-1:2005/NA:2015



NPR 9998:2018
NPR 9998+C1:2020/A1:2020



NS-EN 1998-1:2004+A1:2013+NA:2021
NS-EN 1998-3:2005+NA:2013
NS-EN 1998-5:2004+NA:2014



NP EN 1998-1:2010
NP EN 1998-1:2010/A1:2013
NP EN 1998-1:2010/Errata1:2022
NP EN 1998-3:2017
NP EN 1998-5:2010



PN-EN 1998-1:2005 /A1:2014-01/AC:2009
PN-EN 1998-3:2005 /AC:2014-09
PN-EN 1998-5:2005



P100-1-2013



STN EN 1998-1/NA/Z1:2012
STN EN 1998-3/NA:2009
STN EN 1998-5/NA:2010



BS EN 1998-1:2004+A1:2013
BS EN 1998-3:2005
BS EN 1998-5:2004



UNE-EN 1998-1:2018
UNE-EN 1998-3:2018
UNE-EN 1998-5:2018

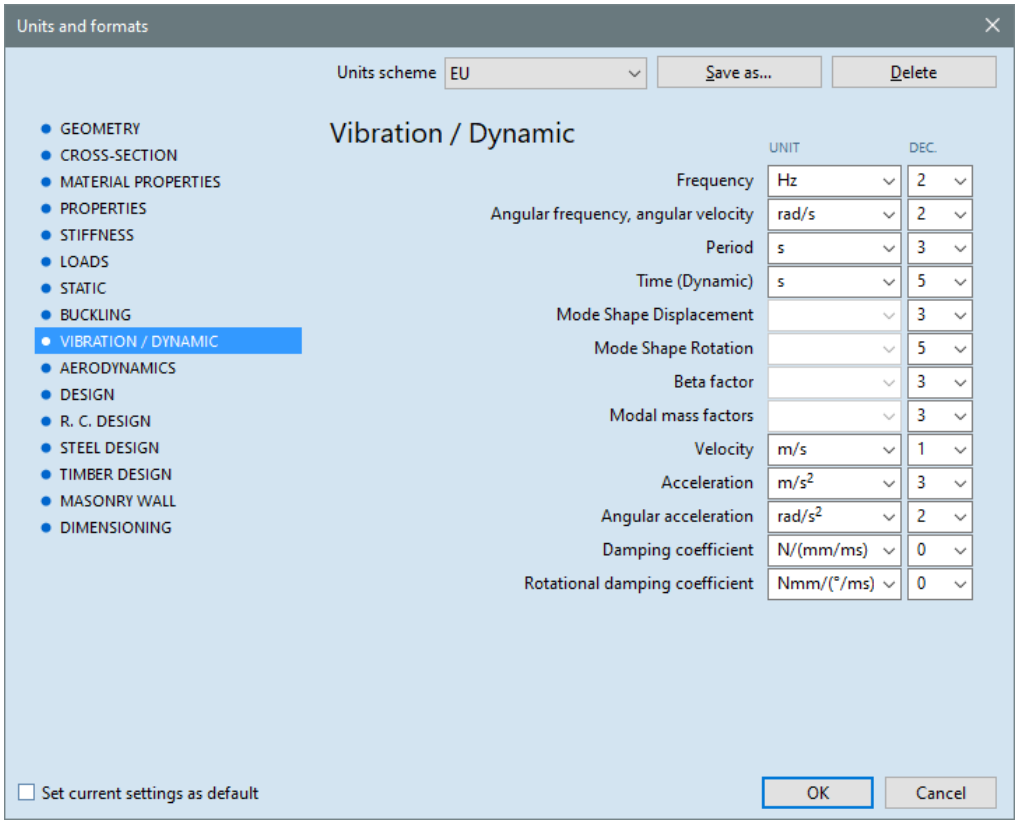


NTC 2018



SIA 261:2020

3.3.8 Units and Formats

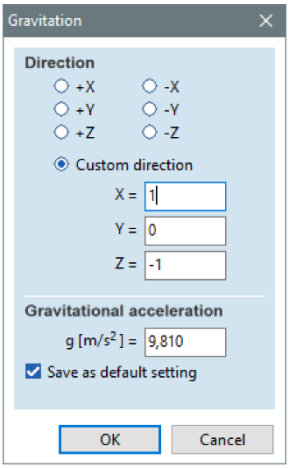


Lets you configure the units (SI and/or Imperial) and formats of variables used throughout the program (number of decimals used for displaying or exponential format). You can use predefined sets as the SI set, or create and save your own custom sets.

Storing unit settings

The software stores the defined units and formats settings in the *Axunits.ini* data file which can be found in the folder *c:\Users\'username'\AppData\Roaming\AxisVM\15*.

3.3.9 Gravitation



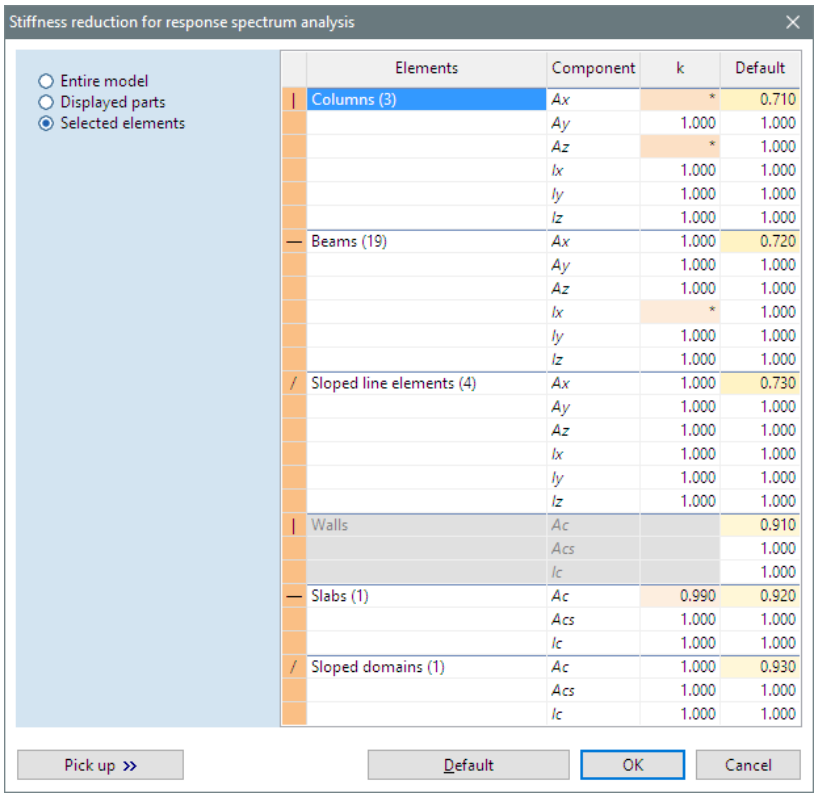
Lets you set the gravitational acceleration constant and the direction of gravitation as one of the global coordinate directions or a custom direction.

If *Custom direction* is selected the X, Y, Z components of the direction vector in the global coordinate system must be specified.

If *Save as default setting* is checked any new model will start with the entered value of gravitational acceleration.

3.3.10 Stiffness reduction

Seismic analysis based on response spectrum analysis according to Eurocode allows using stiffness reduction factors *(k)*. These reduction factors can be defined for individual structural members independently according to their architectural element type (column, beam, wall, slab, other element).
The radio buttons on the left are to choose the range of definition *(Entire model / Displayed parts / Selected elements)*. Reduction factors for these elements are listed and can be modified in the table. If elements with the same type have different reduction factors a * is displayed in the respective cell. Changing it will assign a common value for the specified range of elements.



Line elements

k(Ax)	Reduction factor for the *Ax* of the cross-section.
k(Ay)	Reduction factor for the *Ay* of the cross-section.
k(Az)	Reduction factor for the *Az* of the cross-section.
k(Ix)	Reduction factor for the *Ix* of the cross-section.
k(Iy)	Reduction factor for the *Iy* of the cross-section.
k(Iz)	Reduction factor for the *Iz* of the cross-section.

Domains

k(Ac)	Reduction factor for the cross-section area
k(Acs)	Reduction factor for the shear cross-section area
k(Ic)	Reduction factor for the bending inertia of the cross-section

Pick up

Click on the *Pick up* button then click on an element to copy its *k* values to the respective rows of the table according to the architectural type of the element.

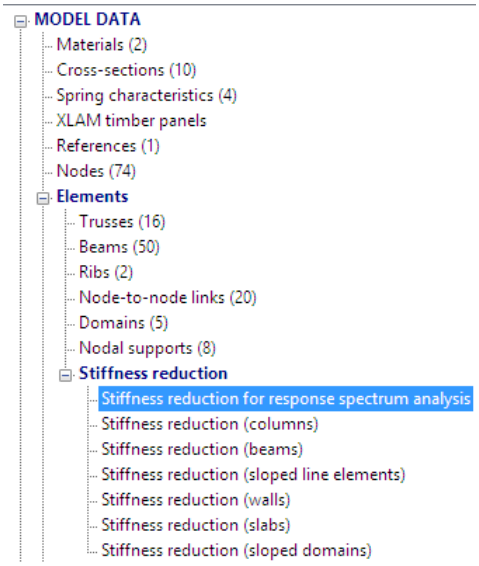
Default

The last column displays the default stiffness reduction values for each type of element (these values are also editable). If new elements are defined these default values will be automatically assigned to them. Click on the *Default* button at the bottom to set all *k* values to their default.

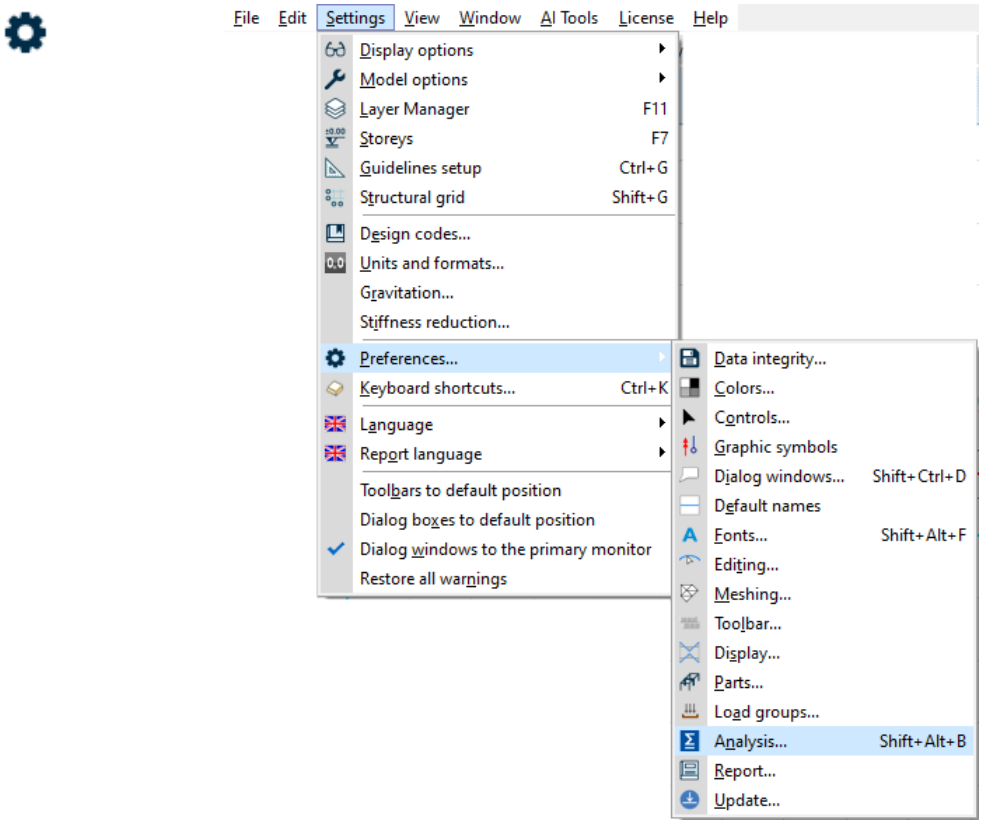
Stiffness reduction is optional in a vibration analysis (**5.2 Vibration**). If the vibration analysis has been performed with stiffness reduction and the response spectrum analysis uses these modal shapes, AXISVM performs two linear analyses: one without reduced stiffness and one with reduced stiffness. In seismic design situations (in seismic load cases or in combinations including seismic load cases) results obtained with reduced stiffness will be displayed, in other situations results without stiffness reduction will be used. (**4.10.25 Seismic loads – SE1 module**)

Reporting stiffness reduction

Stiffness reduction values are available in the Table Browser (see **2.9 Table Browser**) in tables within the folder *MODEL DATA / Elements / Stiffness reduction*. *Stiffness reduction for response spectrum analysis* contains the above table. Other tables list the stiffness reduction values by elements.

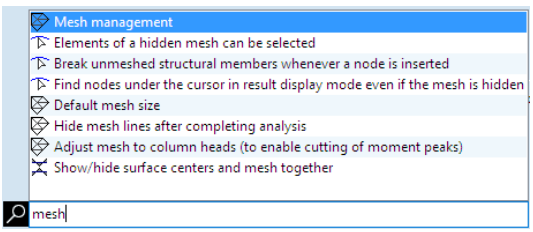


3.3.11 Preferences



Search

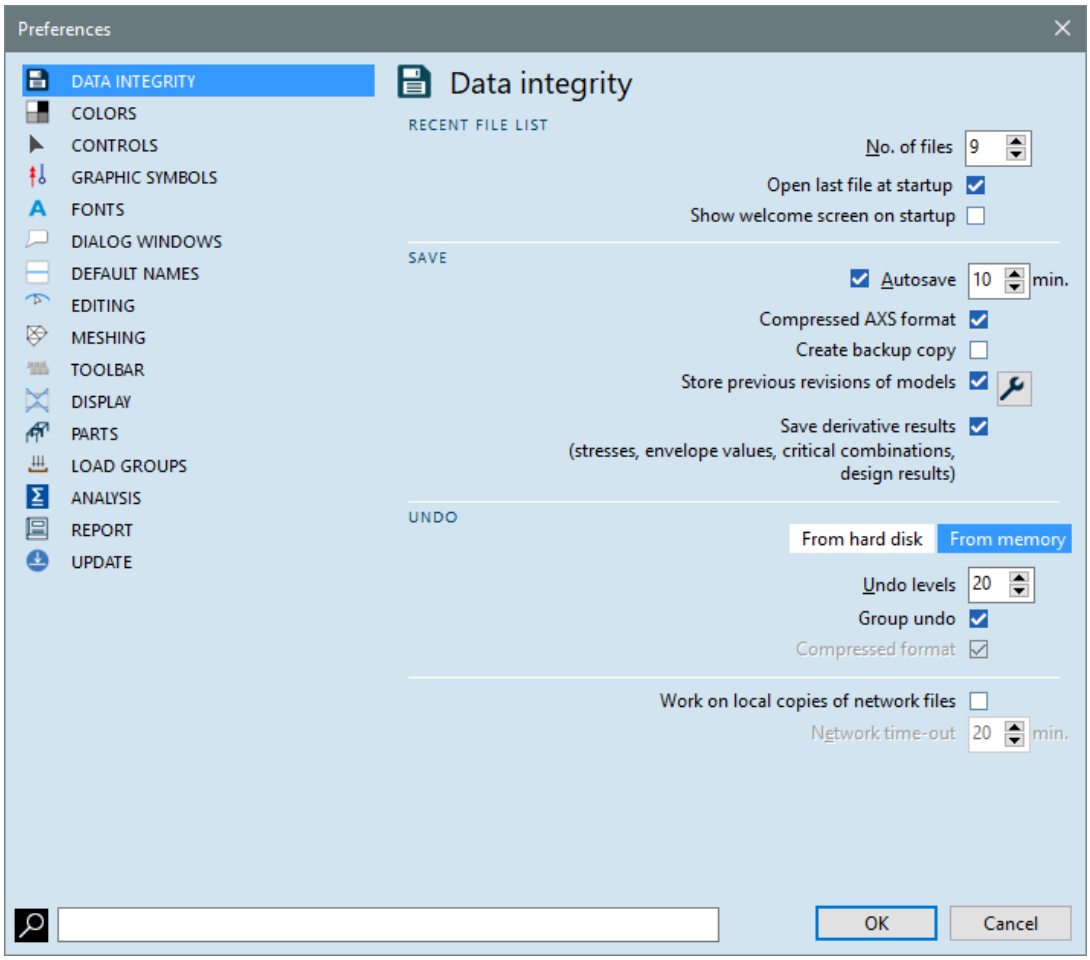
Settings can be found easier by using the search field in the bottom left of the dialog.
It works like the smart command line (see **2.17 Smart command line**), where these matches also appear.



Storing preferences settings

The software stores the settings described below in the data files under the *c: \ Users \ 'username' \ AppData \ Roaming \ AXISVM \ 27* folder.

3.3.11.1 Data integrity



Recent file list

Lets you set the number of recently opened AXISVM model files listed in the bottom of the File menu, and set if you want the last edited file to be opened at startup. The welcome screen (See **2.2 Protection and installation**) will be shown on startup if the *Show welcome screen on startup* checkbox is checked.

Save

To make sure that you do not lose your work, select the *Autosave* option by the check box. In the *min.* box, enter the interval at which you want to automatically save the opened model (1-99 minutes). You must still save the model when you exit. A model that is saved automatically is stored in the default temporary folder of the operating system (by default it is `C:\Users\username\AppData\Local\ Temp\`) as `~modelname.avm` until you perform a save command. When you have to restart AXISVM after a power failure or due to any other problem that occurred before you saved your work, AXISVM can recover it from the temporary file stored in the above folder under the name `$modelname.avm`.

Compressed AXS format
If this checkbox is checked the AXS model file will be saved in a compressed format. The average size of the compressed file size is about 10% of the original. The larger the model file the more efficient the compression is. Result files (*.AXE) are not compressed.

Create Backup Copy
If this checkbox is checked and a model is saved after making changes a backup copy is automatically created from the previous state of the axs file. Name of the backup file is `modelname.~AX`.

Store previous revisions of the model
If this option is checked subsequent revisions of the model are saved.

Click on the icon to get to the Settings dialog
See **3.1.2 Previous revisions…**

Save derivative results
If this checkbox is checked stresses, envelopes, critical combinations and design results will be saved as well.

Undo

The previous state of the model can be retrieved *From hard disk* or *From memory*. If you work on big models and/or your computer is low on memory it is recommended to use the first option (which is a bit slower).
You can undo your last actions. You have to specify the maximum number of actions you want to undo. This number must be between 1 and 99.
The *Group Undo* option allows you to undo the effects of complex commands in a single step. Undo data can be stored in memory or on hard disk. The first option is faster, the second option leaves more memory for the program (it may be important if a huge model is calculated).

Work on local copies of network files

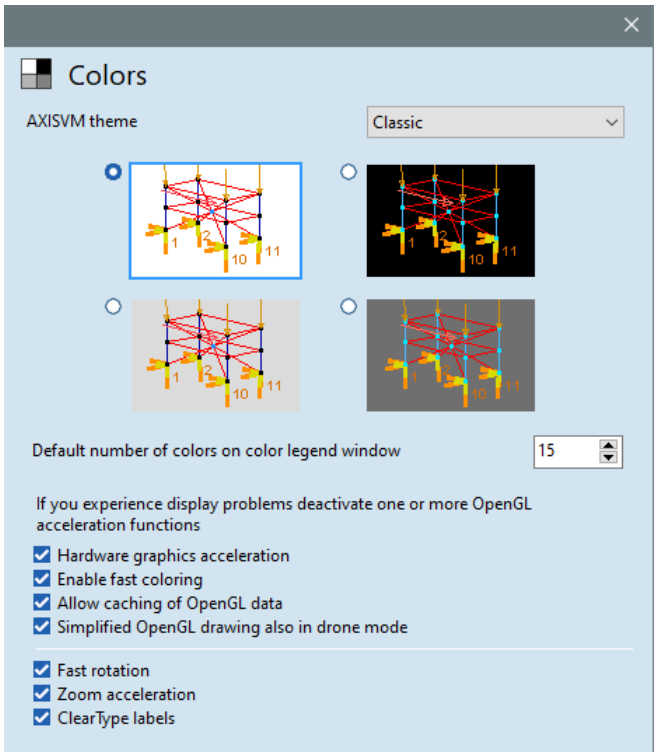
If models are opened through a network, the speed of data transfer may reduce the performance of AXISVM. This effect can be eliminated by allowing making local copies of network files. Local copies will be placed into the folder where the temporary files are stored during the analysis – except when this folder is set to the model folder. In this case the files are saved to the default folder for temporary files. The original files will be updated at each save operation.

Network time-out

In case off network hardware protection keys, if in a time period set here there is no activity (checks) with the key, the current AXISVM session is closed.
Disconnecting may also happen in a situation when you get a phone call and you do not use the program for a time longer than the network time-out. If another user asks for access to the key the server gives a license to him/her and when you try to continue your work the program displays an error message and halts at the next key check.

3.3.11.2 Colors

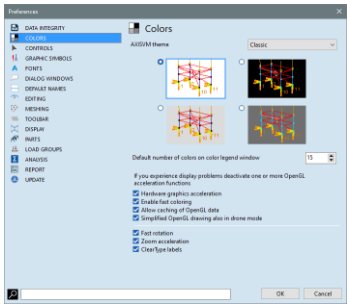




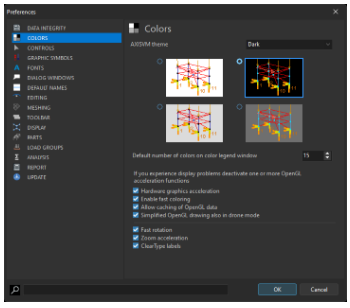
AXISVM theme

Selecting a theme changes the appearance of the entire user interface, even the edit fields and scroll bars.

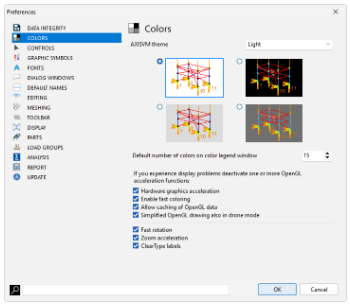
Classic	Light blue dialogs like in X8, white background color in the graphics area
Dark	Dark grey dialogs, black background color in the graphics area
Light	Light grey dialogs, white background color in the graphics area



Classic



Dark



Light

Although it is advisable to use the background colour associated with the theme, you can also choose one of the four options shown here (black, dark grey, light grey or white) regardless of the theme. Labels, numbers, symbols and elements will automatically change their colors to remain visible.

Default number of colors on color legend window: Set the number of colors displayed in the color legend window. See **2.19.4 Color legend window**

OpenGL settings

Turn on *Hardware graphics acceleration*, *Allow caching of OpenGL data* and *Enable fast coloring* for faster and smoother rotation of the model – if the video card and driver supports it. The results heavily depend on the configuration details of the operating system, the driver and the graphics card so sometimes turning of all or some of these settings may improve performance.

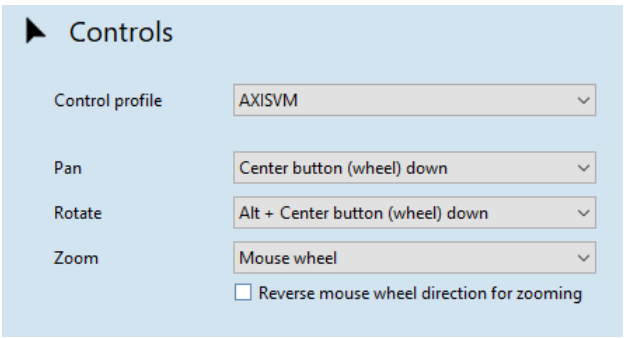
Zoom acceleration hides labels and other non-scaled items during zooming.

Reverse mouse wheel direction for zooming. If unchecked the mouse wheel has the default behaviour: turning the wheel up zooms in, turning the wheel down zooms out the view. If checked these functions will be reversed.

Enabling *ClearType labels* will improve the quality of on-screen text (this is the default setting). If this causes any rendering issues with your graphics card, you can disable it.

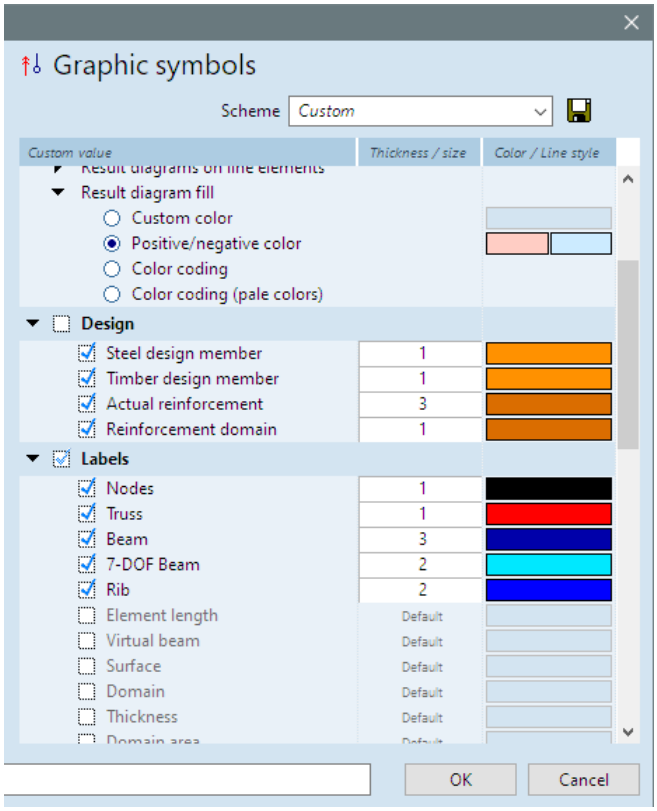
3.3.11.3 Controls



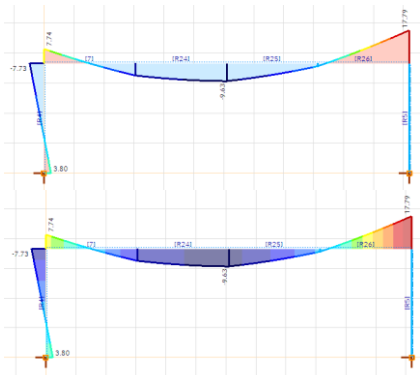


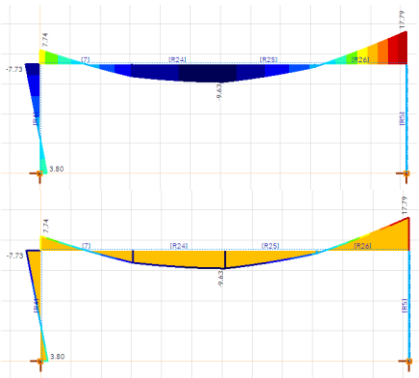
Here you can set which mouse actions you wish to use to control the pan, rotate and zoom functions in the main window. From the *Control profile* drop-down list, you can select a software (AXISVM, Archicad, Consteel, FEM-Design, Grasshopper, IDEA StatiCa, Autodesk Revit, RFEM, SCIA Engineer, Tekla Structures) to import a set of settings you wish to apply.

3.3.11.4 Graphic symbols

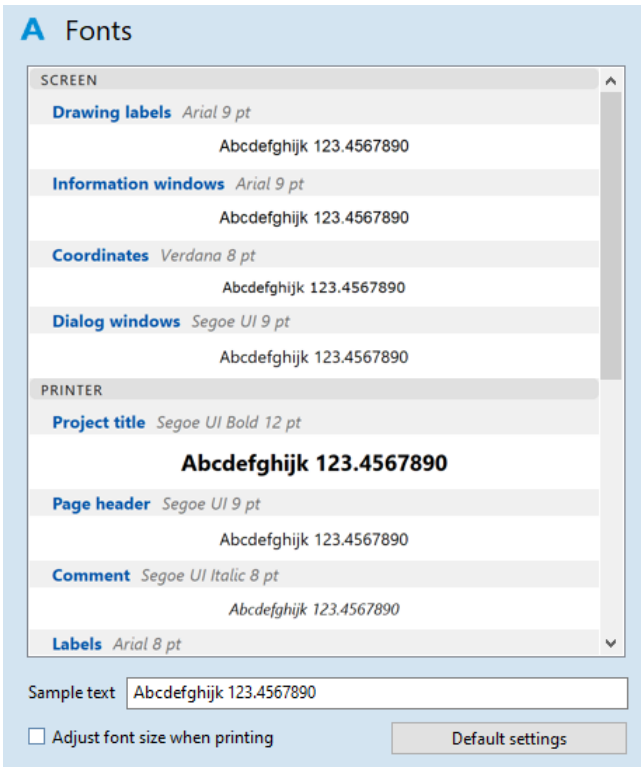


Color and line thickness of graphic symbols, color and size of labels can be customized. Color and thickness of eccentric axes of beams and ribs can be set.
If the checkbox in the *Custom value* column is left unchecked the symbol is set to default. If it is checked click on the *Thickness / size* value or the *Color* rectangle to edit the values. The new settings can be saved to a scheme clicking the *Save* icon. Schemes can be loaded by selecting from the dropdown list.
From Version X4 colors and line width of the local system axes, result diagrams and applied reinforcement can also be changed.





3.3.11.5 Fonts



This list allows changing fonts used on screen and in printed output. Move in the list with cursor keys or by clicking on items. Double-click on the selected line or press *Enter* to change the font.
To change the sample, edit *Sample text*.
Right click on an item to display the popup menu.

Modify...	change the font.
Default font	reset the selected font to its default.

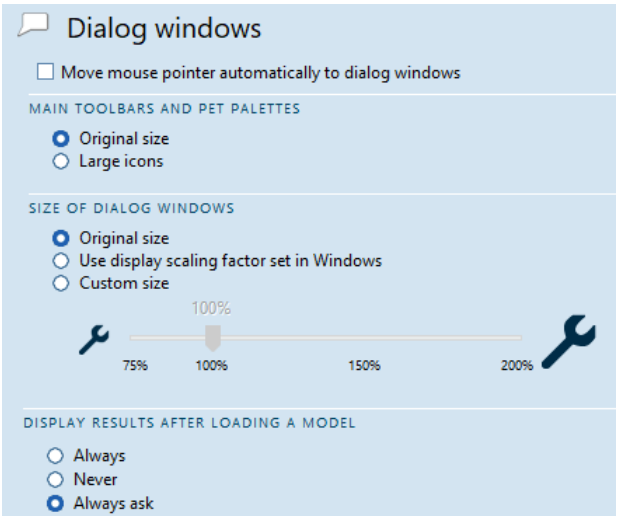
Click on *Default settings* to reset all fonts to their default.

Screen	*Drawing labels*	Font used for numbering and labeling on model drawings. Dimension lines and text boxes have their own font, see **2.16.11 Dimension lines, symbols and labels.**
	Information windows	Font used for information windows on the screen (color legend window, info window, coordinate window), see **2.19 Information windows.**
	Coordinates	Font used in the coordinate window, see **2.19.1 Coordinate window**
	Dialog windows	Font used for dialogs.
Printer	*Project title* *Page header* *Comment*	Fonts used for the printed page header. See **3.1.10 Page header**
	Labels	Font used for numbering and labeling on printed model drawings. Dimension lines and textboxes have their own font.
	Information windows	Font used for information windows (color legend window, info window) when printed on drawings.
Report	*Table title*	Font used for titles of printed tables.
	Table header row	Font used for the first (header) row of tables.
	Table rows	Font used for the rest of the table.

Due to different resolution of the screen and the printer the ratio of label size and drawing is different on the two devices, especially in case of textboxes. Check / uncheck *Adjust font size when printing* if you are not satisfied with the result.

3.3.11.6 Dialog windows





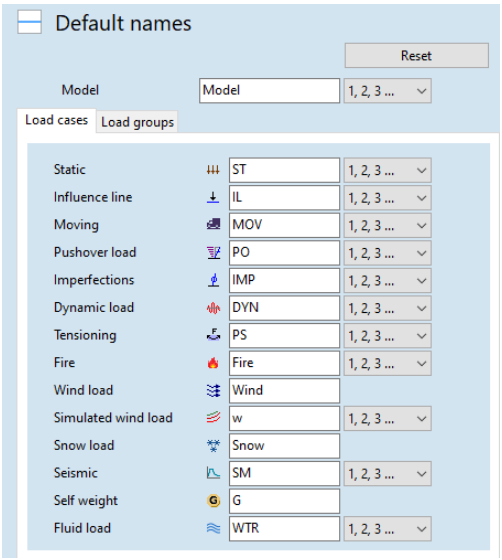
You can control the behavior of the dialog boxes that appear in the program.

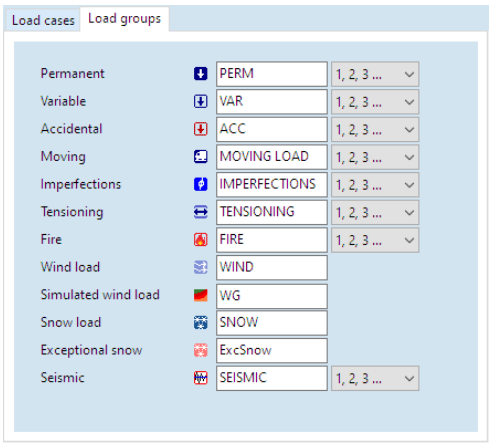
Move mouse pointer automatically to dialog windows positions the mouse pointer over the OK button of dialog windows. Certain mouse drivers provide this functionality without using this option.

Main toolbars and pet palettes offer two sizes for toolbar buttons. Choosing *Large icons* also scales up certain cursors. Size of dialog windows makes it possible to scale dialogs between 75 and 200%. Windows settings defined in *Control Panel / Display* can also be applied.
Opening large models with a huge result file can slow down loading considerably. The *Display results after loading a model* option controls what happens when a model with results has been opened. Results will be displayed *Always / Never / Always ask* (a dialog is displayed to confirm displaying the results).

3.3.11.7 Default names

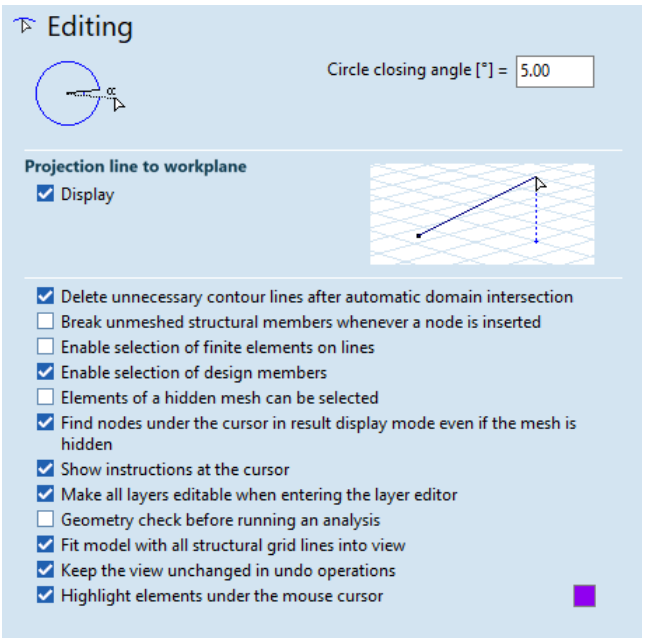






Change *Model* to change the default name set for a new model. Select the style of the postfix (1, 2, 3 ... or A, B, C, ...) appended to generate a unique name.
Default names and postfix styles of new load cases or load groups can be edited the same way.
Clicking *Reset* resets all names and styles to their defaults.

3.3.11.8 Editing



Circle Closing Angle Parameter for drawing arcs. If the center angle of the arc is smaller than this angle or it is closer to 360° than this angle then a whole circle will be drawn.

Projection line to workplane Display of projection lines can be turned on/off. Its shows the distance of the cursor from the current workplane.

Delete unnecessary contour lines after automatic domain intersection
Controls if contour lines are automatically deleted after domain intersection. If this function is turned off contour lines became internal lines of the union.

Break unmeshed structural members whenever a node is inserted
By default AXISVM uses structural members. These are line elements consisting of one or more finite elements. Unmeshed members contain one finite element only. If a new node is inserted on an unmeshed member or the line is divided the structural member remains unaffected but will contain more than one finite element. Clicking on a structural member selects all finite elements belonging to the member. This behaviour can be changed by checking this option. Then new nodes inserted on structural members will break apart the member. To break apart existing structural members use *Edit / Break apart structural members*.

Enable selection of finite elements on lines
If activated, finite elements of a structural member can be selected individually. Otherwise only the whole structural member can be selected.

Enable selection of design members
If activated, design members can be selected instead of structural members. Design members consists of a group of lines with the same design parameters handled as one entity for steel or timber design purposes.

Elements of a hidden mesh can be selected
If display of mesh is turned off this field controls if the hidden nodes / lines / surface elements can be selected or not. This switch also controls if these nodes and elements appear in tables or not.

Find nodes under the cursor in result display mode even if the mesh is hidden
As results are calculated on mesh nodes checking the results may require mouse snap to nodes even if the mesh is hidden. This behaviour can be controlled here.

Show instructions at the cursor
Controls the display of a small tooltip window at the cursor with instructions for the next step of the current task. If unchecked, instruction messages appear only at the bottom status line.

Make all layers editable when entering the layer editor
If activated, all locked layers will be unlocked when entering the background layer editor.
See **2.16.12 Editing background layers.** Otherwise locked layers must be unlocked manually.
See **2.12 Layer Manager** and **2.18 Speed buttons.**

Geometry checking before running an analysis
If activated a geometry checking is automatically performed before analysis.
See **4.8.20 Geometry check**

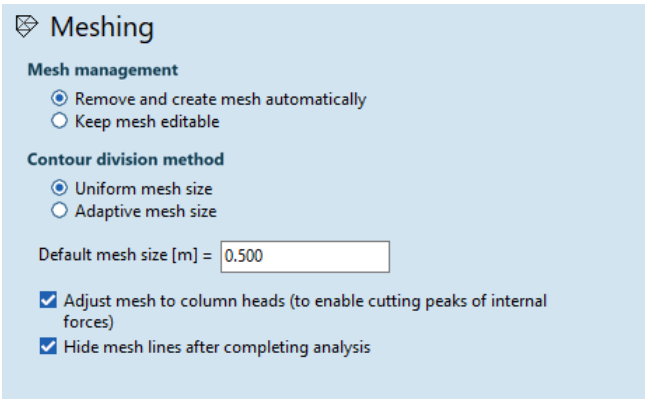
Fit model with structural gridlines into view
If activated and the model is zoomed to fit in view, structural gridlines are also taken into account when determining the zoom ratio.

Keep the view unchanged in undo operations
If activated undo operations does not affect the view of the model. Otherwise undoing a command sets the the model as it was displayed before the command.

Highlight elements under the mouse cursor
Element(s) under the mouse cursor will be highlighted (both in wireframe and rendered view). Click on the color sample to customize highlight color.

3.3.11.9 Meshing





Mesh management
One of the following mesh management methods can be chosen.

Remove and create mesh automatically
Any editing performed on a domain deletes its mesh. When launching the analysis missing meshes will be recreated based on the meshing parameters of the domain.

Keep mesh editable
Meshes can be edited manually.

Contour division method

Uniform mesh size
Meshes will be generated according to the user defined element size regardless of the shape of the domain (least number of finite elements).

Adaptive mesh size
Takes the shape of the domain into consideration and creates a better mesh by increasing mesh density wherever it is necessary.

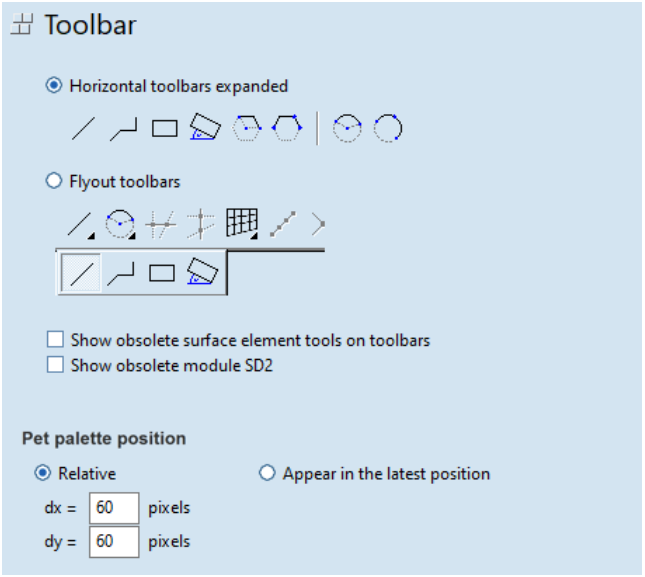
Default mesh size
When defining meshing parameters for a domain for the first time this value will appear by default.

Adjust mesh to column heads
Turning on/off this option will set the default status of the mesh parameters dialog.
See **4.12.1.2 Meshing of domains.**

Hide mesh lines after completing analysis
Checking *Hide mesh lines after completing analysis* automatically turns off mesh lines after completing analysis.

3.3.11.10 Toolbar





Displaying toolbar

If *Horizontal toolbars expanded* is chosen, all icons appear in a row. Vertical separator lines indicate different groups of functions.
If *Flyout toolbars* is chosen, different functional groups will be represented by a single icon. Clicking the arrow in the right bottom another toolbar flies out showing different tools.

Using domains and automatic meshes make it unnecessary to create individual surface elements and place loads on them. For compatibility reasons the related functions are not removed but kept hidden. Their visibility is controlled by the option *Show obsolete surface element tools on toolbars*.

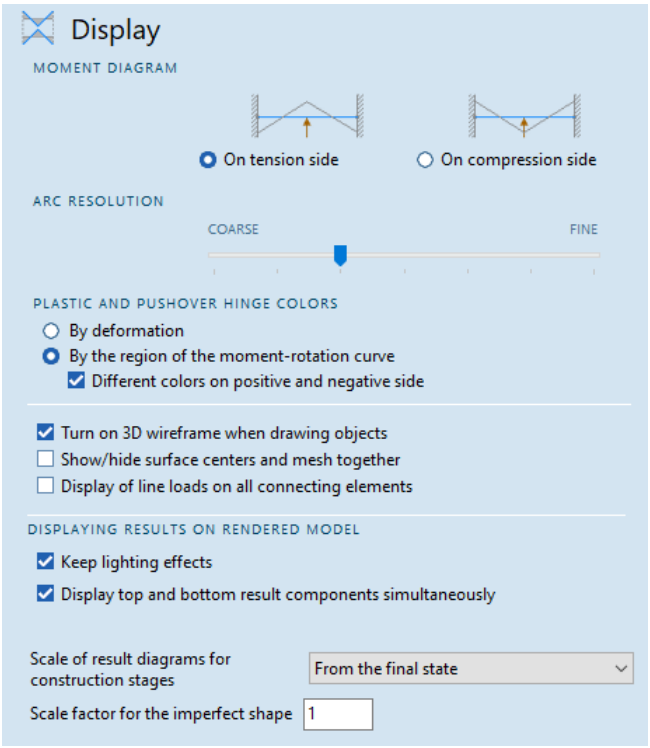
SC1 module is recommended to design bolted steel joints. The *obsolete SD2 module* still can be activated here, regardless the module configuration.

Pet palette position

Pet palette position can be:
Relative
Specify the horizontal *(dx)* and vertical *(dy)* distance from the operation in pixels.
Appear in the latest position
Pet palette appears in its latest position.

3.3.11.11 Display





Moment diagram Placement rule for moment diagrams can be set.

Arc resolution Arcs are displayed as polygons. Set the display resolution here. The finer the resolution the closer the polygon will get to the arc. This parameter affects drawing only and is not related to the precision of the analysis.

Plastic hinge colors These settings determine the color coding of plastic hinges. The first option is to color hinges according to the rotation. The second option is to color hinges according to the section of the moment-rotation curve where the hinge state point is located. Different colors can be assigned to the positive and negative side of the curve.

Switches

Turn on 3D wireframe when drawing models
Displays 3D wireframe of objects while drawing (see **4.9.4 Direct drawing of objects**) even if the active view is not in rendered mode.

Show/hide surface centers and mesh together
If checked turning on/off display of the mesh automatically turn on/off surface center symbols.

Display of line loads on all connecting elements
If an edge load is applied where a wall and two plates meet and parts are turned on (see **2.16.17 Parts**) the load will be displayed according to this setting. If this option is turned on the load will be displayed if an active part contains any of the three elements. If this option is turned off the load will be displayed only if an active part contains the elements the loads were originally assigned to. This is useful to check the local system of the load components.

Displaying results on rendered model If the *Solid model* result display option is selected (**6.1 Static**), surface of rendered elements will be colored according to the local value of the current result component.

Keep lighting effects
Rendering includes shading, therefore colors on surfaces may appear darker or lighter than expected. Turn this option off to display colors perfectly matching the color legend.

Display top and bottom result components simultaneously
The *Solid model* result display option renders elements with their physical thickness so it is possible to display otherwise separated result components for top and bottom values simultaneously on surface elements (e.g., surface stress or required amount of reinforcement).

Scale of result diagrams for construction stages

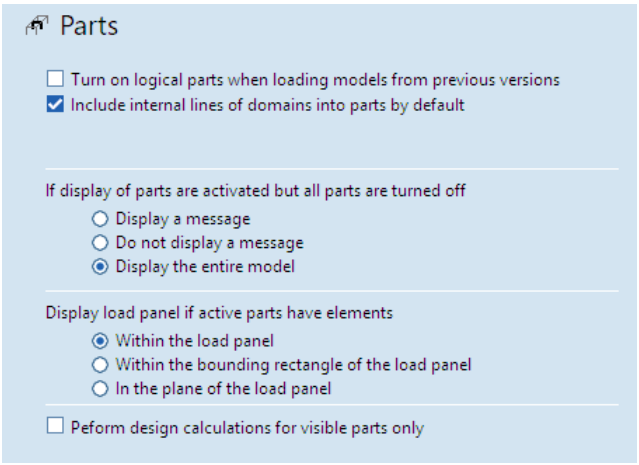
To make result diagrams comparable, results are displayed on the same scale at every construction stage. You can set this common scale to be determined either by the final state or by all phases. The latter option should only be selected if, for some reason, higher values are expected during construction.

Scale factor for the imperfect shape

In a load case representing global imperfection or an imperfection calculated from a buckling shape, the imperfect structure is displayed. The scale for displaying imperfection can be set here.

3.3.11.12 Parts





Turn on logical parts when loading models from previous versions
If activated, opening a model created with a version not supporting logical parts activates logical parts automatically.

Include internal lines of domains into parts by default
If activated, internal lines of domains will be present in parts where the domain is included.

If the user turns on display of parts and unchecks all parts AXISVM will behave according to the selected radio button.

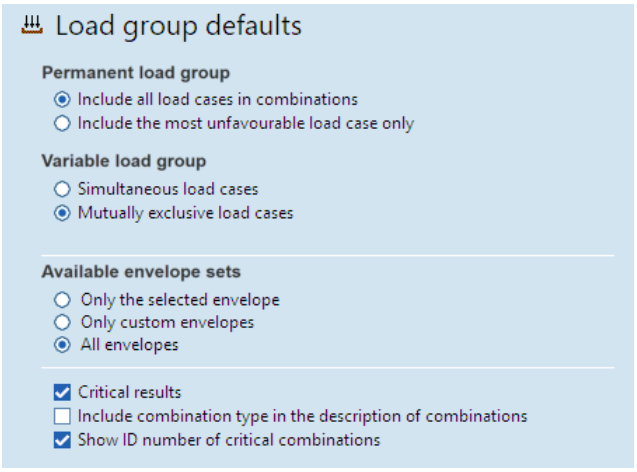
Display of load panels when only parts are visible can also be controlled here.

Perform design calculations for visible parts only
If the checkbox is active, the design calculations are performed for the visible parts only. In case of large models, this can result more efficient calculation process. However, if another part is set to visible, it takes some time to perform design calculation of the elements that do not have design results yet. If the checkbox is inactive, the program performs design calculations for every element in the model at once. In this case, if another part is selected, the design results are already available and the user does not have to wait for the calculation in every case.
The checkbox only affects the calculation of the following result components:

- required reinforcement
- crack width
- steel structural member utilization
- timber structural member utilization

3.3.11.13 Load group defaults





Here the default values of load group parameters can be set.
Settings for envelopes and combinations are also placed here. The content of load cases and combinations dropdown lists (on the result and design tabs) can be controlled here. AXISVM allows creating different envelope sets (see *Result display options* in **6.1 Static**). If the first option is selected only the selected envelope will appear in lists. If the second option is selected only custom envelopes will be listed. If the third option is selected all standard and custom envelopes will be listed.

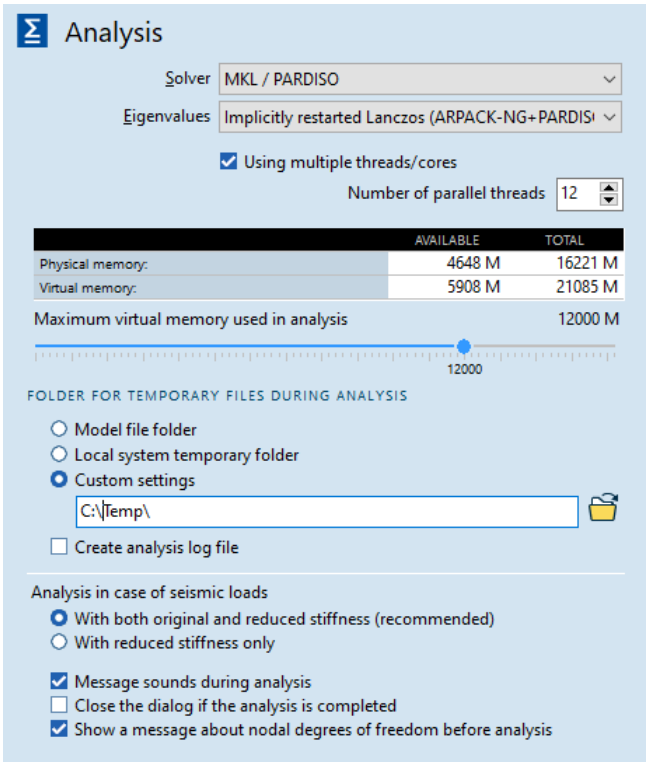
Critical results: check to enable finding of critical combinations. If unchecked results will be available of only for load cases / combinations and their envelopes.

Include combination type in the description of combinations: description of combinations can be extended to show the combination type (ULS or SLS types).

Show ID number of critical combinations: to make it easier to compare critical combinations the ID of the critical combination can be displayed as a prefix.

3.3.11.14 Analysis





Solver In addition to the *Classic solver* used up to version X8, you can select Intel's highly optimised MKL / PARDISO solver which significantly speeds up analysis of large models.

Eigenvalues

In addition to the *Subspace iteration* used up to version X8 for eigenvalue calculation, you can select the ARPACK-NG / PARDISO solver, particularly effective in structural analysis and employs the extremely fast *Implicitly restarted Lanczos* method to determine eigenvalues and eigenvectors (natural frequencies and critical load factors, as well as vibration and deflection modes).

Maximum virtual memory used in analysis

At the beginning of the analysis AXISVM divides the system of equations into blocks according to the available physical and virtual memory. It makes analysis more efficient but can considerably slow down other applications. Set the amount of virtual memory you let AXISVM use during the analysis here.

Using a single thread / Using multiple threads

Using multiple threads/cores makes AXISVM run analysis on multiple threads. Multi-threading improves speed of calculation. Improvement depends on the available memory and the model size. Linear analysis will be 1.5 times faster, while vibration analysis can be 4 times faster.
Setting *Number of parallel threads* allows adjusting the software to the capabilities of the hardware it is running on.
This switch also controls multi-threaded execution of other functions (calculation of critical combinations, geometry checks, certain dimensioning procedures). If you happen to encounter an unexpected error or AXISVM freezes, it is worth running the function again after disabling multi-threaded operation.

Folder for temporary files during analysis

You can specify the location of temporary files during analysis.
Select any of these options : *Model file folder, Local system temporary folder, Custom*
Create analysis log file: If this option is turned on technical details of the analysis will be logged and saved to a text file `modelname_log.txt`.

Analysis in case of seismic loads

With both original and reduced stiffness (recommended)
If this option is enabled AXISVM performs two linear analyses: one without reduced stiffness and one with reduced stiffness and in seismic load cases or combinations it displays results from the latter analysis.
With reduced stiffness only
If this option is enabled AXISVM performs only one linear analysis, with reduced stiffness and displays these results in all load cases and combinations.
See also **3.3.10 Stiffness reduction**.

Message sounds during analysis

If this option is activated system sounds will be played after completing an analysis or getting an error message. Sound card and speakers must be present.

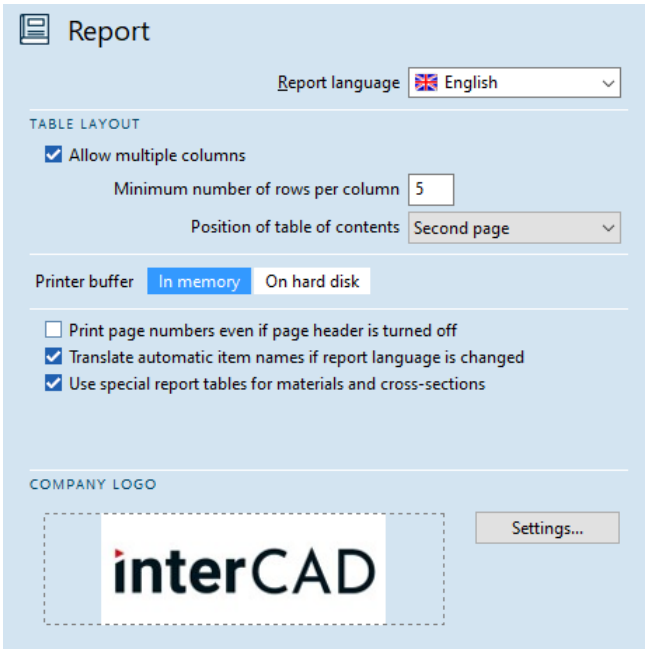
Close the dialog if the analysis is completed

Leave this option unchecked to see analysis parameters, size of the equations or running time after the analysis.

Show a message about nodal degrees of freedom before analysis

AXISVM checks if the nodal degrees of freedom are consistent with the geometry and loads of the structure before analysis, and if it is necessary, sends a message with an option to modify the degrees of freedom. See **4.9.24 Nodal DOF (degrees of freedom).** The message also displays the checkbox *Do not display this message for this model.* The program will remember the current answers and will not pop up the question again.
If you want the message window to reappear, you can activate it here.
If you turn off the message here, the program will check and change the degrees of freedom without notice and saves the change to the model.

3.3.11.15 Report



Report language You can select from the following languages: *English, German, French, Italian, Polish, Dutch, Hungarian, Romanian, Spanish, Portuguese, Czech, Slovakian, Serbian, Croatian, Slovenian, Greek*.

Table layout If the *Allow multiple columns* is checked, narrow report tables will be printed in a multi-column layout to reduce the space required. *Minimum number of rows per column* can be specified to avoid column breaks for short tables.
The position of table of contents in reports can be selected from a list *(Front page, Second page, Third page)*.

Printer buffer If a report includes many pictures building the entire report in memory may consume too much system resources and cause printing problems. In this case set printer buffer to hard disk.

Print page numbers even if page header is turned off
If this option is turned on page numbers appear on printed pages even if headers are disabled in the printing dialog.

Translate automatic item names if report language is changed
If this option is turned on AXISVM-generated names of Drawings Library or report items will be translated automatically.

Use special report tables for materials and cross-sections
If this option is checked reports will generate special tables for materials and cross-sections organizing properties in groups. This option is not available when exporting to RTF.

Materials

1 **S 235**

Type: Steel Eurocode-H, 10025-2 Linear

Material

E = 21000 kN/cm²
ν = 0.30
α_T = 1.2E-5 1/°C
ρ = 7850 kg/m³

f_y = 23.50 kN/cm²
f_u = 36.00 kN/cm²
f'_y = 21.50 kN/cm²
f'_u = 36.00 kN/cm²

2 **C16/20**

Type: Concrete Eurocode-H, EN 206 Linear

Material

E = 2860 kN/cm²
ν = 0.20
α_T = 1E-5 1/°C
ρ = 2500 kg/m³

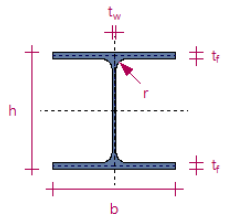
f_{ck} = 1.60 kN/cm²
γ_c = 1.500
α_{cc} = 1.00
ϕ_t = 2.00

Name: Material name; **Type:** Type of material; **Model:** Material model; **E_x:** Young's modulus of elasticity in local x direction; **E_y:** Young's modulus of elasticity in local y direction; **ν:** Poisson's ratio; **α_t:** Thermal expansion coefficient; **ρ:** Density; **Material:** Material color;

Cross-sections

1 **HEA180**

Process: Rolled Shape: I S.p.: 9



h = 17.1 cm
b = 18.0 cm
tw = 0.6 cm
tf = 0.9 cm
r = 1.5 cm

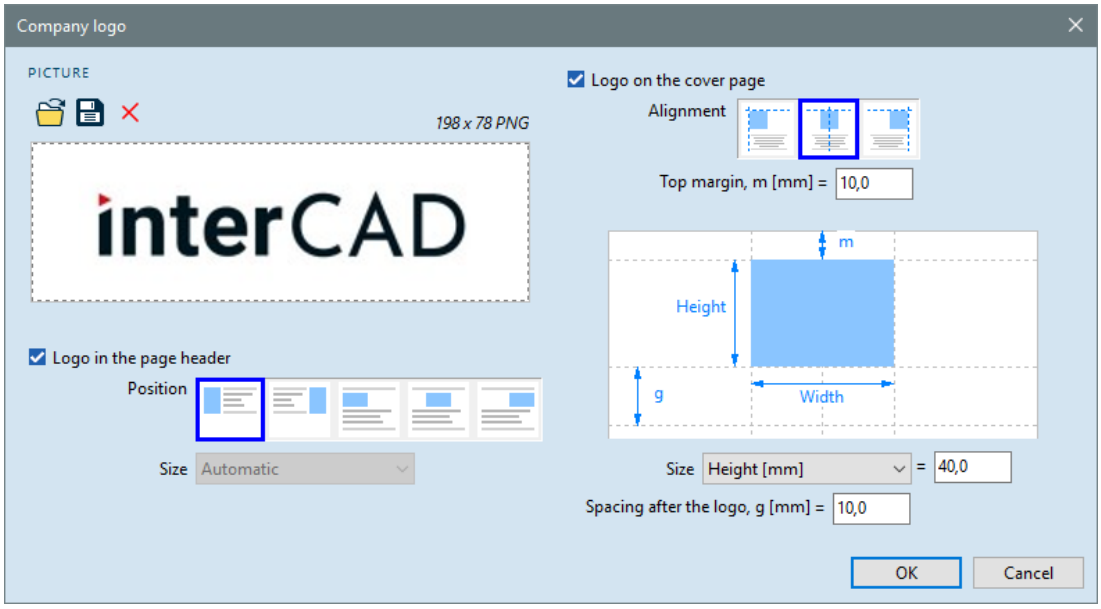
Ax = 45.26 cm²
Ay = 30.86 cm²
Az = 9.95 cm²
Ix = 15.0 cm⁴
Iy = 2510.7 cm⁴
Iz = 924.6 cm⁴
Iyz = 0 cm⁴
I_1 = 2510.7 cm⁴
I_2 = 924.6 cm⁴
α = 0 °
Iω = 58932 cm⁶

$W_{1,el,t}$ = 293.7 cm³
$W_{1,el,b}$ = 293.7 cm³
$W_{2,el,t}$ = 102.7 cm³
$W_{2,el,b}$ = 102.7 cm³
$W_{1,pl}$ = 324.9 cm³
$W_{2,pl}$ = 156.5 cm³

i_y = 7.4 cm
i_z = 4.5 cm
Hy = 18.0 cm
Hz = 17.1 cm
y_G = 9.0 cm
z_G = 8.6 cm

Name: Cross-section name; **Process:** Manufacturing process; **h:** Cross-section height; **b:** Cross-section width; **tw:** Web thickness; **tf:** Flange thickness; **r_1, r_2, r_3:** Rounding radius; **Ax:** Cross-section area; **Ay, Az:** Shear area; **Ix:** Torsional inertia; **Iy, Iz:** Flexural inertia; **Iyz:** Centrifugal inertia; **I_1, I_2:** Principal flexural inertia; **α:** Principal directions; **Iω:** Warping constant; **$W_{1,el,t}$, $W_{1,el,b}$, $W_{2,el,t}$, $W_{2,el,b}$:** Elastic section modulus; **$W_{1,pl}$, $W_{2,pl}$, $W_{1,pl}$:** Plastic section modulus; **i_y, i_z:** Radius of inertia; **Hy:** Dimension in local y direction; **Hz:** Dimension in local z direction; **y_G:** y coordinate of the center of gravity; **z_G:** z coordinate of the center of gravity; **y_s:** y coordinate of the shear (torsion) center relative to the center of gravity; **z_s:** z coordinate of the shear (torsion) center relative to the center of gravity; **β_y, β_z, β_w:** Wagner's coefficient; **S.p.:** Stress calculation points;

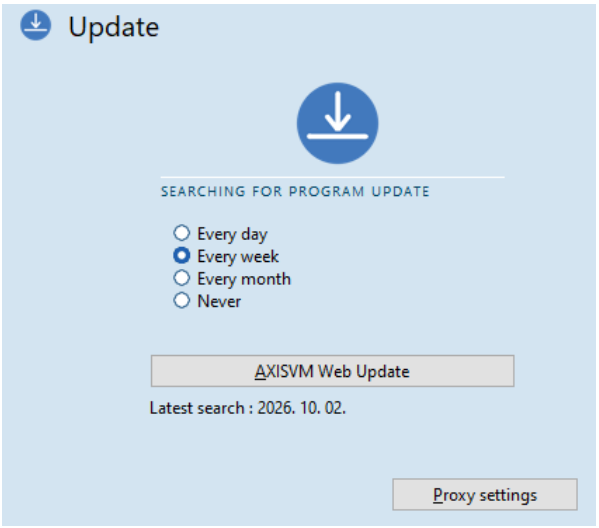
Company logo



A company logo can be loaded, saved, deleted using the three toolbar buttons. This image will appear in the page header of printed drawings, tables, reports and/or on the cover page of the report according to the settings for position, size, margins. These settings are applied when printing a report or exporting it to a DOCX file.

These settings have no effect on the RTF output as that is based on a separate RTF template file.
See 2.10.2 Report

3.3.11.16 Update



Searching for program update

AXISVM checks regularly if there is an update available on the web. The frequency of update checks can be controlled. If the *Never* is chosen an update process can be launched by clicking AXISVM *Web Update*. The date of the latest search is displayed. If internet connection goes through a proxy server, proxy settings have to be defined after clicking *Proxy settings*.

AXISVM Web Update

Click the button to get to the AXISVM Web Update Wizard which is a guide to the download process. If download is complete and the *Update the program* option is checked on the last page, the program quits and start the installation of the new release.

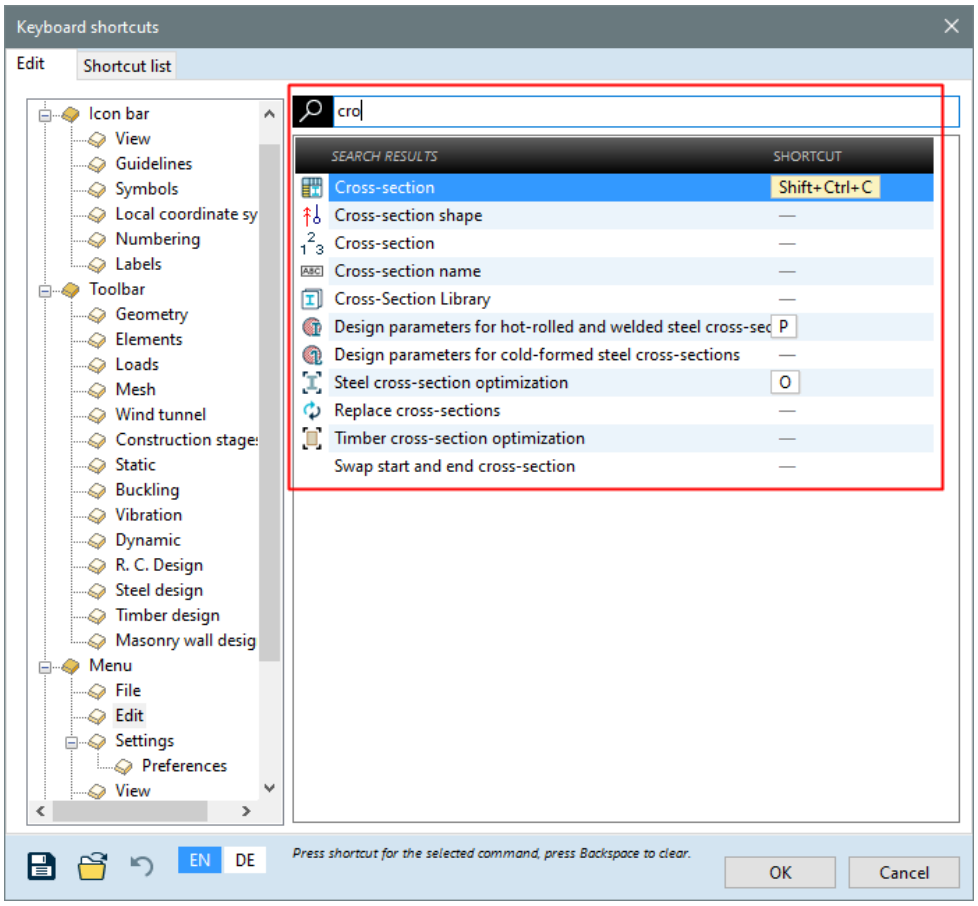
Proxy settings

If the network reaches the web through a proxy server, the configuration data (proxy name, port, user name and password) can be entered here.

3.3.12 Keyboard shortcuts

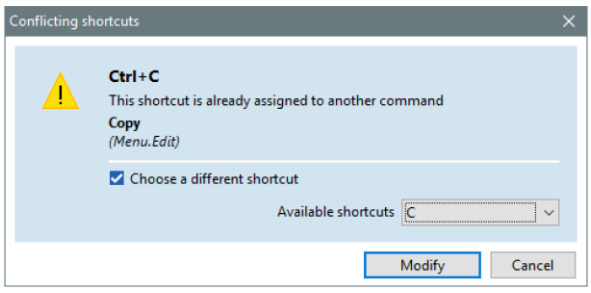
Keyboard shortcuts assigned to the menu items, toolbar buttons can be customized.

Edit



The tree on the left side displays the available command groups. Use the Search field to search for a command. Click a command in the list on the right side, then press the desired shortcut. **Backspace** clears the assignment.

If the desired shortcut is already in use a *Conflicting shortcuts* warning is displayed.



The message shows the command using the shortcut.

If the *Choose a different shortcut* is checked the shortcut can be selected from a list of available (unassigned) shortcuts.

If this option is left unchecked, clicking the *Modify* button clears the previous assignment and assigns the shortcut to the given command.

☞ **Shortcuts assigned to toolbar buttons on different tabs (Geometry, Elements, etc.) are not in conflict. So, the same shortcut can be assigned to commands on different tabs. Only shortcuts for the current tab will be activated.**

Saves the current shortcut configuration to an **.axsc* file.

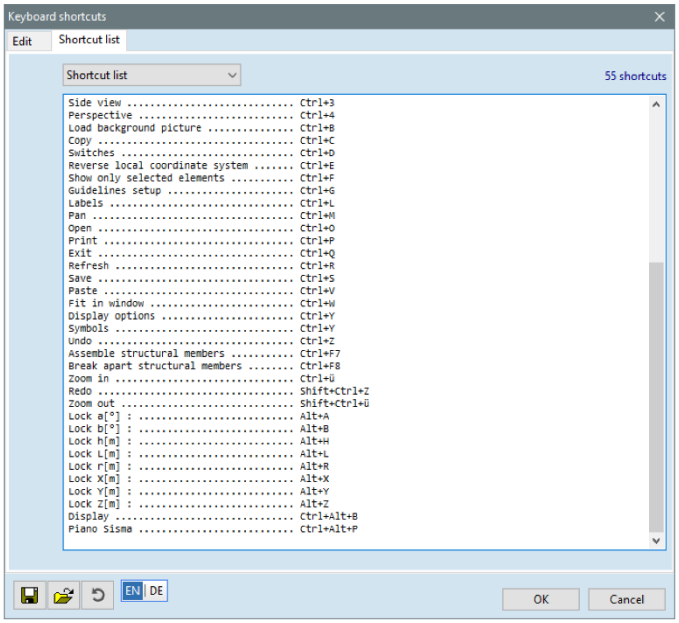
Loads a previously saved shortcut configuration from a **.axsc* file.

By default, shortcuts are stored in the `C:\Users\UserName\AppData\Roaming\AxisVM\27\Shortcuts` folder.

Resets the shortcut configuration to its default.

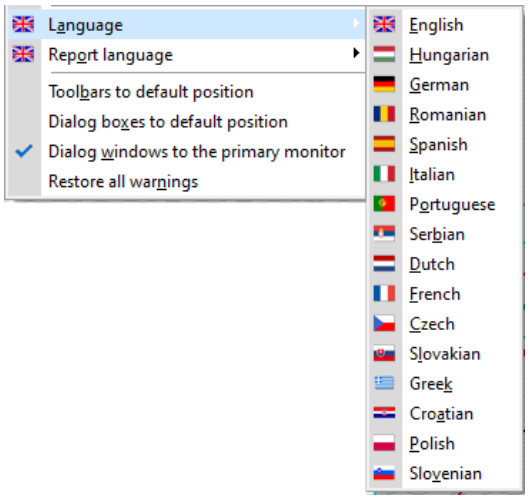
A switch to display English or German key names in the shortcut descriptions.

Shortcut list



The *Shortcut list* tab displays a list of all available shortcuts. It can be a *Command list* or a *Shortcut list* (the first is ordered according to the command groups, the second by the shortcuts)
For the entire list for the default configuration see **2.6 Keyboard shortcuts.**

3.3.13 Language



Switching between UI languages is part of AXISVM.
Language sets the language of the user interface.

3.3.14 Report language

Switching between report languages is part of AXISVM.
Report language controls the language of items in the printed output.

3.3.15 Toolbars to default position

The moveable Icon bar will get back to the left side. All flyout toolbars undocked and dragged to a new position will get back to the Icon bar. Custom combo box and edit field width settings on postprocessing tabs will be reset to default. See **6.1 Static.**

3.3.16 Dialog boxes to default position

AXISVM remembers the last position of dialogs and display them there. If any problem is detected on systems with multiple monitors setting dialog boxes to default position can solve it.

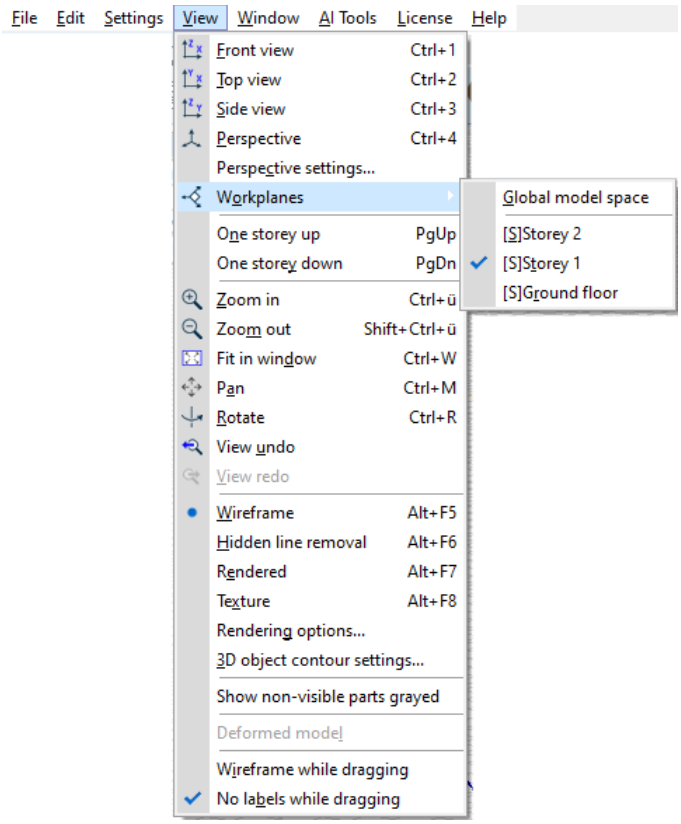
3.3.17 Dialog windows to the primary monitor

Somewhat similar to the previous function: it displays all dialog windows on the primary monitor. For example, if you have connected your laptop to an external monitor, dragged certain dialog boxes over to that monitor, and then disconnected it, this function ensures that all dialog boxes appear on the primary monitor, but does not overwrite their saved positions. So, when you reconnect the external monitor, these windows will reappear in their last positions.

3.3.18 Restore all warnings

Several warning windows appear with an additional checkbox: *Do not display this message*. Selecting this option suppress the warning. This menu item restores the display of all warnings.

3.4 View



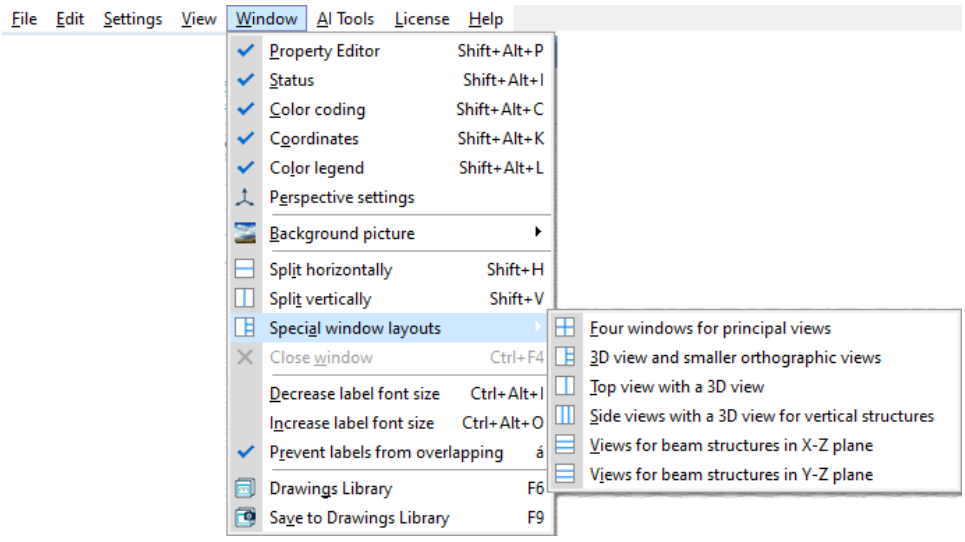
	Front view	**[Ctrl + 1]**	See **2.16.3 Views**
	Top view	**[Ctrl + 2]**	See **2.16.3 Views**
	Side view	**[Ctrl + 3]**	See **2.16.3 Views**
	Perspective view	**[Ctrl + 3]**	See **2.16.3 Views**
	Perspective settings		See **2.16.3 Views**
	Workplanes		See **2.16.7 Workplanes**
	One storey up / One storey down		If displaying of a storey is activated, this is the fastest way to go one storey up or down.
	Zoom in	**[Ctrl + /], [+]**	See **2.16.2 Zoom**
	Zoom out	**[Ctrl + Shift+ /], [–]**	See **2.16.2 Zoom**

Zoom to fit **[Ctrl + W]**	See **2.16.2 Zoom**
Pan	See **2.16.2 Zoom**
Rotate	See **2.16.2 Zoom**
View undo **[Ctrl + [ ]**	See **2.16.2 Zoom**
View redo **[Ctrl +]]**	See **2.16.2 Zoom**
Wireframe	See **2.16.4 Display mode**
Hidden line removal	See **2.16.4 Display mode**
Rendered	See **2.16.4 Display mode**
Texture	See **2.16.4 Display mode**
Rendering options...	See **2.16.4 Display mode**
3D object contour settings...	See **2.16.4 Display mode**
Show non-visible parts grayed	See **2.16.17 Parts**
Deformed model	Same as setting *Display shape* to *Undeformed* or *Deformed* in the *Display parameters* dialog. See **6.1 Static**
Wireframe cross-sections	In rendered mode thin walled cross-sections will be displayed only with mid-planes.
Actual cross-sections	In rendered mode thin walled cross-sections will be displayed as solid objects with their actual shape.
Wireframe while dragging	If it is switch on, the program display the wireframe of the model during the rotation or pan.
No labels while dragging	If this option is turned on, labels are not drawn during rotation or panning.

3.5 Plugins

If third party applications are installed into the *Plugins* folder they appear under the *Plugins* menu. Add-ons installed into the AddOns folder does not appear as menu items but create additional buttons on toolbars.

3.6 Window



3.6.1 Property Editor

Property Editor provides the fastest way to change properties of the selected nodes, elements or loads. All changes are made immediately. If the selection contains different elements it is possible to change their common properties (e.g., after selecting trusses, beams and ribs their material and cross-section will be editable). If result or design tabs are active the values are read only.
In certain fields regular mathematical expressions are also accepted.
Available operators and functions are:

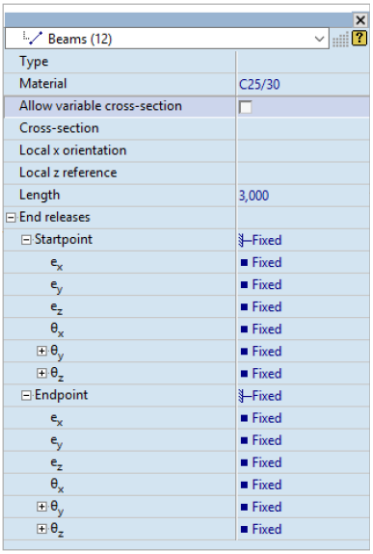
(,), SIN, COS, TAN, EXP, LN, LOG10, LOG2, SINH, COSH, TANH, ARCSIN, ARCCOS, ARCTAN, ARCSINH, ARCCOSH, ARCTANH, INT, ROUND, FRAC, SQR, SQRT, ABS, SGN.
Few fast operators:

++8	adds 8 to the actual value
- - 8	deducts 8 from the actual value

Negative numbers within operation have to be in brackets.
In these expressions # substitutes the actual value (For instance *#/3* divide it by 3). When entering a value of nodal coordinates, load values, surface thicknesses you can refer to global coordinates as *X, Y, Z* or *x, y, z*. In case of certain load types variables refer to other load components as well.

For nodal loads or point loads on beams variables *Fx, Fy, Fz, Mx, My, Mz* refer to force and moment components. For distributed beam loads *px1, py1, pz1, m1, px2, py2, pz2, m2* refer to load components. Variable names are not case sensitive.
Example 1: If you want to turn selected distributed wind loads with different *X* components to *Y* direction enter '*px1*' into field *pY1* and '*px2*' into field *pY2* then enter zero into fields *pX1* and *pX2*.
Example 2: to scale the structure in direction *X* by 200%, first select all nodes then click the line first line and enter *X*2* as X.
The question mark button turns on/off the help information.

Properties are displayed in a tree-like structure. Clicking a [+] or [–] symbol before the property name expands or collapses a list of sub-properties.
If the (...) button appears in a line the property can be changed using a separate dialog.
If the (>>) button appears in a line the property can be picked up from another element by clicking it.
Property Editor can be used to modify data but also to select and filter elements with the same property.



Filter



Selecting a property and clicking the filter button you can select all the elements having the same property value.
Example: changing an existing cross-section in the whole structure.
Select the cross-section property of a rib element, click the *Filter* button to select all rib elements with this cross-section then change their cross-section property.

Load components can be filtered the similar way. Select a load, select a load component in the Property Editor and click on the *Filter* button. This way you can select all loads with the same component value.

3.6.2 Information Windows



Lets you set the display of the Status, Color coding, Coordinate, and Color legend windows to on or off.
See 2.19 Information windows

3.6.3 Background picture



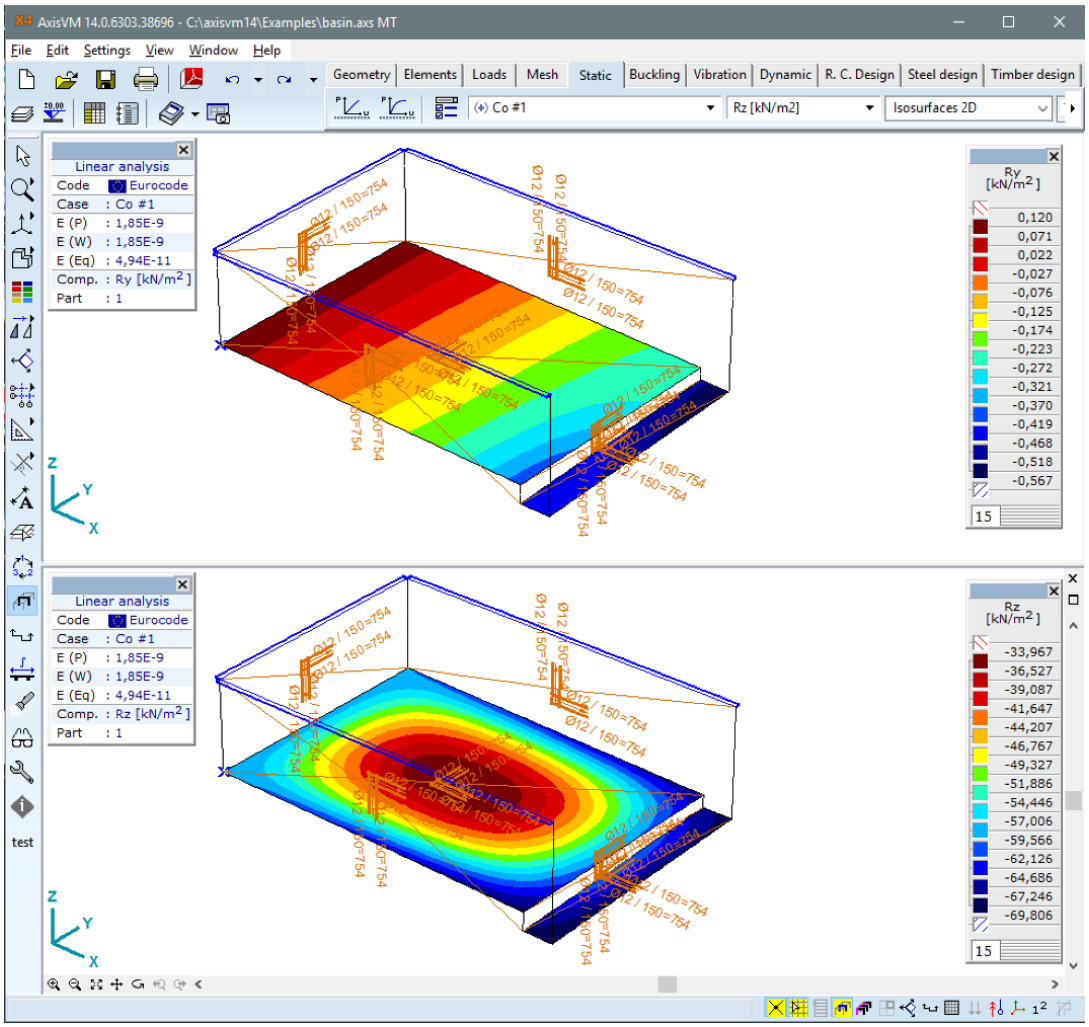
The submenu makes several options available. An automatically fitted background picture can be loaded to the main window of AXISVM to show the model in its future environment. *Load Background Picture...* submenu item or **[Ctrl+B]** opens a file browser dialog, *Reload Background Picture* shows the most recently used picture files. In multi-window mode each window can have its own background picture.

Picture in the active window can be turned on and off by clicking *Display* or by **[Ctrl+Alt+B].**
Save Background Picture saves the picture in the active window into a file. If the aspect of the picture differs from the window aspect *Shift Background Picture* makes it possible to drag the background to a new position. *Remove Background Picture* removes the picture in the active window.
Background pictures are saved into the AXS file.

After loading a background picture the model can be set to an appropriate view by zooming out, zooming in, panning, rotating and setting the perspective.

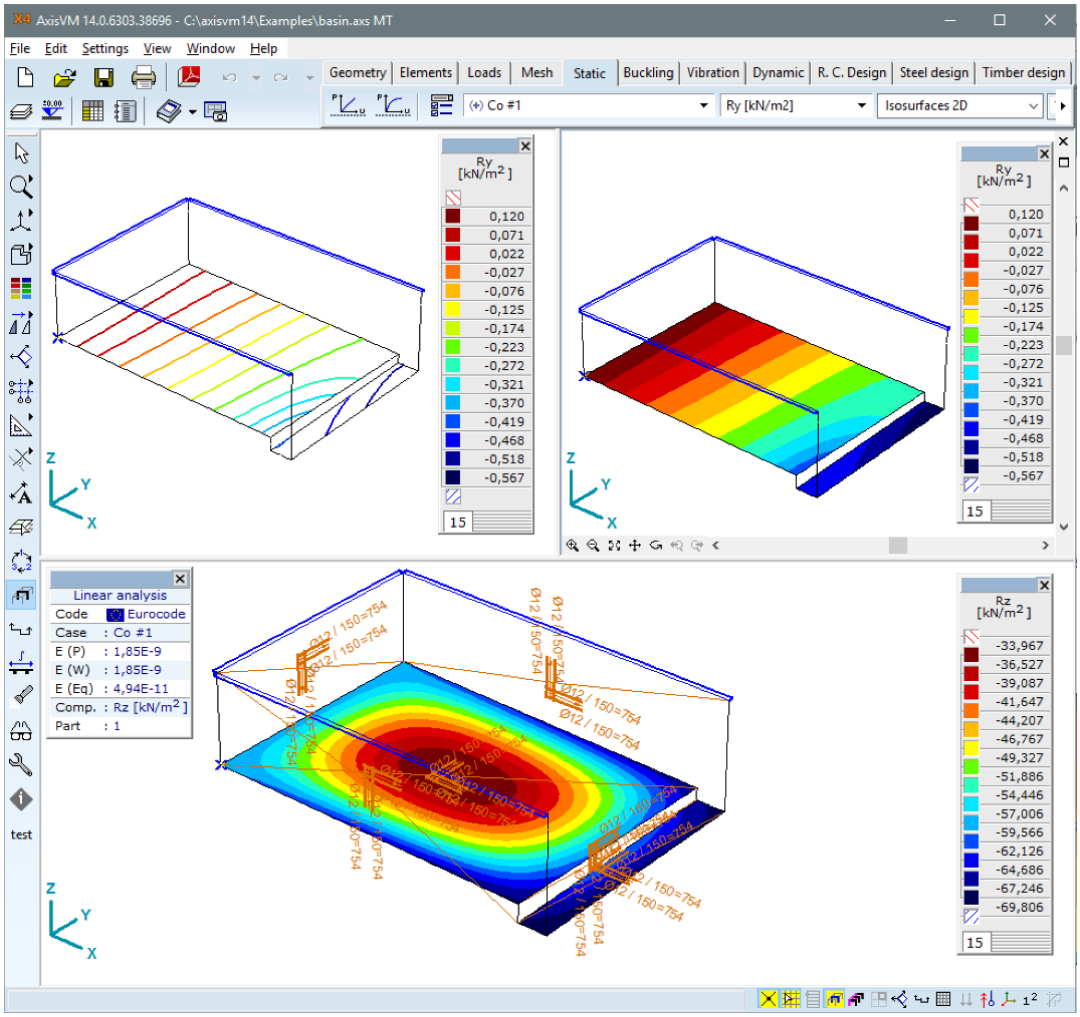
3.6.4 Split horizontally





Splits the graphics window horizontally into two parts. Clicking into one of them makes that window active. The display settings of each window can be set independently. Different load case or combination can be selected for each window.
You can maximize or minimize or restore the graphics windows by using the buttons at the top-right of the windows.

3.6.5 Split vertically



Splits the graphics window vertically into two parts. Clicking into one of them makes that window active. The display settings of each window can be set independently. Different load case or combination can be selected for each window.
You can maximize or minimize or restore the graphics windows by using the buttons at the top-right of the windows. Different load cases can be set in each window but only when displaying results.

3.6.6 Special window layouts

This submenu offers predefined multi-window layouts.

Four windows for principal views
The four equal-sized panes display the front view (X-Z), side view (Y-Z), top view (X-Y), and perspective view (from left to right and from top to bottom) so that the entire model is visible in each pane.

3D view and smaller orthographic views
The larger pane on the left displays the perspective view, with the top view (X-Y), front view (X-Z), side view (Y-Z) (from top to bottom) on the right so that the entire model is visible in each pane.

Top view with a 3D view
The larger pane on the left displays the top view (X-Y) with the perspective view next to it, so that the entire model is visible in each pane.

Side views with a 3D view for vertical structures
The three panes of equal size obtained with vertical divisions display the front view (X-Z), the side view (Y-Z), and the perspective view, so that the entire model is visible in each pane.

Views for beam structures in X-Z plane
The three panes of equal size obtained with horizontal divisions display the front view (X-Z), the side view (Y-Z), and the perspective view, so that the entire model is visible in each pane.

Views for beam structures in Y-Z plane
The three panes of equal size obtained with horizontal divisions display the side view (Y-Z), top view (X-Y), and the perspective view, so that the entire model is visible in each pane.

3.6.7 Close Window

Closes the current graphics window.

3.6.8 Changing label font size

Decrease / Increase label font size These two menu items are to change the font size for all labels on diagrams.

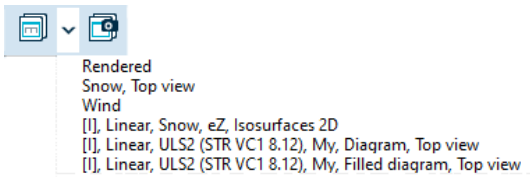
3.6.9 Preventing labels from overlapping

If this option is checked the program prevents labels from overlapping by moving and hiding less important labels. In the latter case zooming in may increase the number of visible labels. Priorities are determined by label types and values. If large number of labels are to be displayed this optimization may be a time-consuming process.
You can also set this option in Display options/Labels. See 2.16.21 Display options

3.6.10 Drawings Library

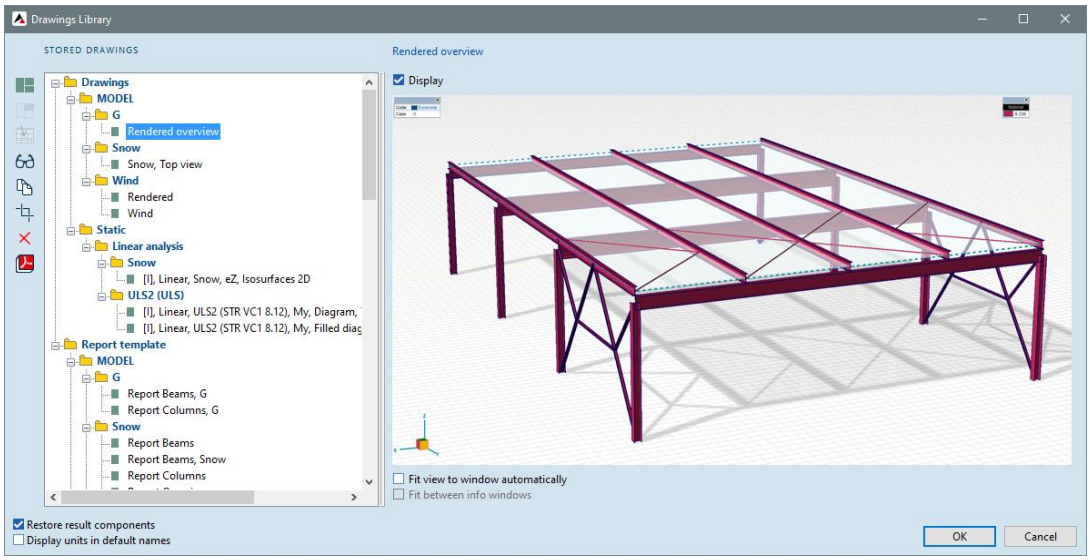
The Drawings Library contains drawings saved in the program. Drawings are not saved pictures but instructions how to draw a view of the model or parts of it including multi-window settings. Drawings can be reloaded to restore saved view and display settings. Including drawings into a report makes it easier to update the report when the model has changed and recalculated as drawings will be updated automatically like tables.

Drawings Library can store displacement, force, stress diagrams of line elements, diagrams of steel and bolted joint design, punching analysis, reinforced concrete column check and beam design in an associative way.



Clicking the arrow beside the tool button an existing drawing can be selected from a pop-up list, restoring its view and display settings.

After clicking the Drawings Library tool button a dialog appears.



This dialog is to overview, maintain and reload saved drawings.

Load the chosen drawing to the main window.
If the button is down and dialog is closed with OK, the drawing and all its settings are loaded to the main window.

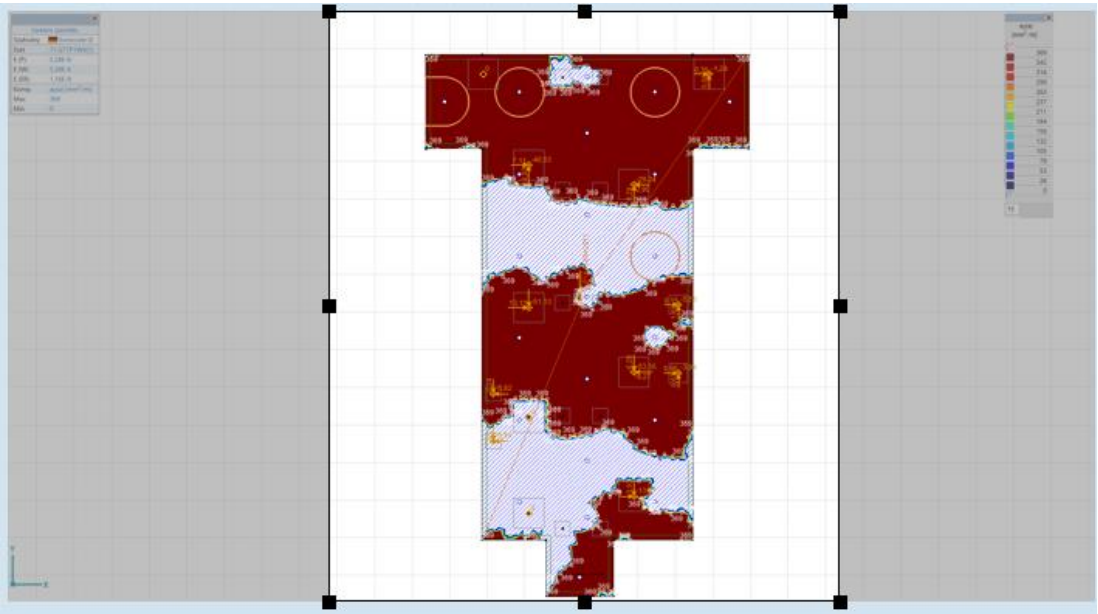
Load the chosen drawing to the active window.
If the button is down and dialog is closed with OK, the drawing and all its settings are loaded to the active window pane of the main window.

Displays a diagram dialog
This function becomes available when a beam result diagram window or a design result diagram window is selected. Such diagrams are displayed when clicking on a beam on the Static tab or on dimensioning tabs of the main window. See e.g. **6.1.10 Truss/beam internal forces** or **6.6.1.3 Diagrams and design calculations.** Clicking on this button opens the dialog or dimensioning module that generated the diagram. After modifying its content, you can choose whether to overwrite the original diagram or save it with a new name.

Graphic symbols
Display of graphic symbols can be modified in library items. Select one or more items and click the button on the toolbar. The status of symbols in the selected drawings is displayed and can be changed. Mixed status is represented by greyed checkmarks.

Add a copy of the selected item
The new drawing will be named appending (1), (2), etc., after the name. You can rename the drawing library item by double-clicking the name – or by right-clicking and selecting *Rename* from the context menu.

Crop the selected item
You can crop the drawing using the selection rectangle that appears over the drawing panel. The corners, mid-sides and the rectangle itself can be dragged using the mouse. Cropping takes place when you double-click on the drawing, or click on the icon again, or close the form by clicking OK.
The cropped drawing behaves exactly the same way as if the main window had been resized to these proportions before saving; therefore, the information windows will also appear on it, and this must be taken into account when adjusting the new size.



Delete
Deletes an item from the Drawings Library

Exports Drawing Library items as a 3D PDF file.
See **3.6.10.1 Export drawings to a 3D PDF file - PDF module**

Restore result components
If this option is checked loading a drawing displaying results restores the result component as well and sets the appropriate tab (Static, Vibration, etc.).
If this option is unchecked loading a drawing does not restore the result component and the tab.

Fit view to window automatically
Check this option if you want the drawing to accommodate to model changes (drawing is zoomed to show all visible parts).

Fit between info windows
Fits the model image into the area between the info windows to avoid overlapping.

Units of measurement in default names
This checkbox controls whether generated names of diagrams contain the units (like eZ[mm]). Changing this setting also updates the names of existing drawings generated with their default name.

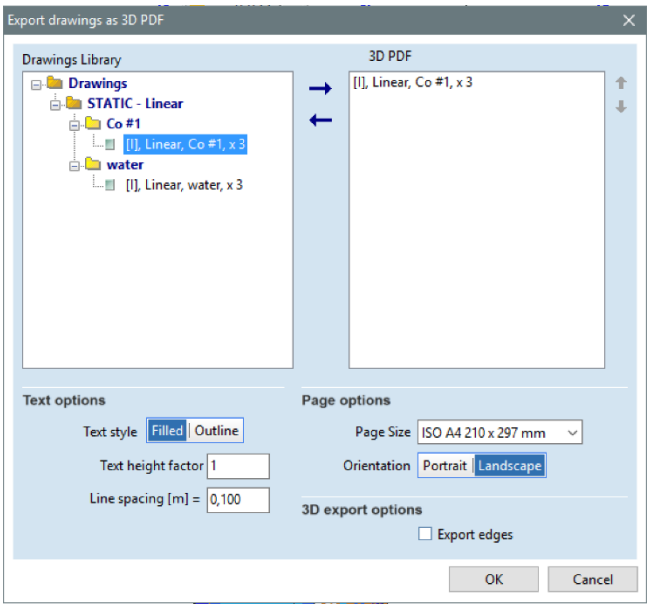
OK Saves the changes and loads the selected drawing.

Cancel Does not save changes.

3.6.10.1 Export drawings to a 3D PDF file - PDF module



Drawing Library items can be exported as a multipage 3D PDF file. To view the interactive 3D images, use Adobe Acrobat Reader (the updated 8.1 Version or later).
Library items selected from the tree view on the left can be moved into the PDF list by clicking on the right arrow. Left arrow removes selected items from the PDF list. Each library item will be rendered on a separate page in the PDF following the order of the PDF list. Items in the PDF list can be rearranged using the up and down arrow. Both 2D and 3D views can be exported. 2D drawings will appear as regular images, 3D views can be rotated, zoomed in and out using Acrobat Reader.



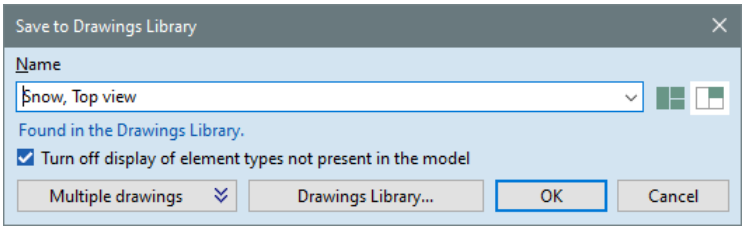
Text options Text size and appearance can be controlled in the *Text options* group.

Page options Size and orientation of the PDF document pages can be set in the *Page options* group.

3D export options Sometimes drawing all edges makes the view a bit blurred. So export of edges can be turned on / off.

3.6.11 Save to Drawings Library





By clicking this tool button one or more drawings can be saved into the Drawings Library.
If the current drawing already exists, a *Found in the Drawings Library* label is displayed in the dialog. It can be overwritten or the drawing can be renamed. *Multiple drawings* button opens additional options. Load cases, load combinations (and result components if results are displayed) can be chosen. AXISVM creates all combinations (i.e., all selected result components in all selected load cases) and saves them into the library with the current view and display settings.
If multiple model views have been created by dividing the main window two options are available:



Saves the whole view with all window panes.

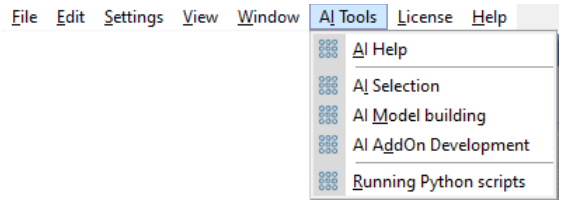


Saves the drawing from the active window pane only.

Turn off display of element types not present in the model It is possible that a certain element type is not present in the model when the drawing is saved to the Drawings Library, but its display is still enabled. If such an element is added to the model, it will appear in the updated library drawing, which can be confusing. This option is to avoid such cases: when enabled, all element types not present in the model are disabled in the drawing.

Clicking the *Drawings Library* button displays the *Drawings Library* dialog.

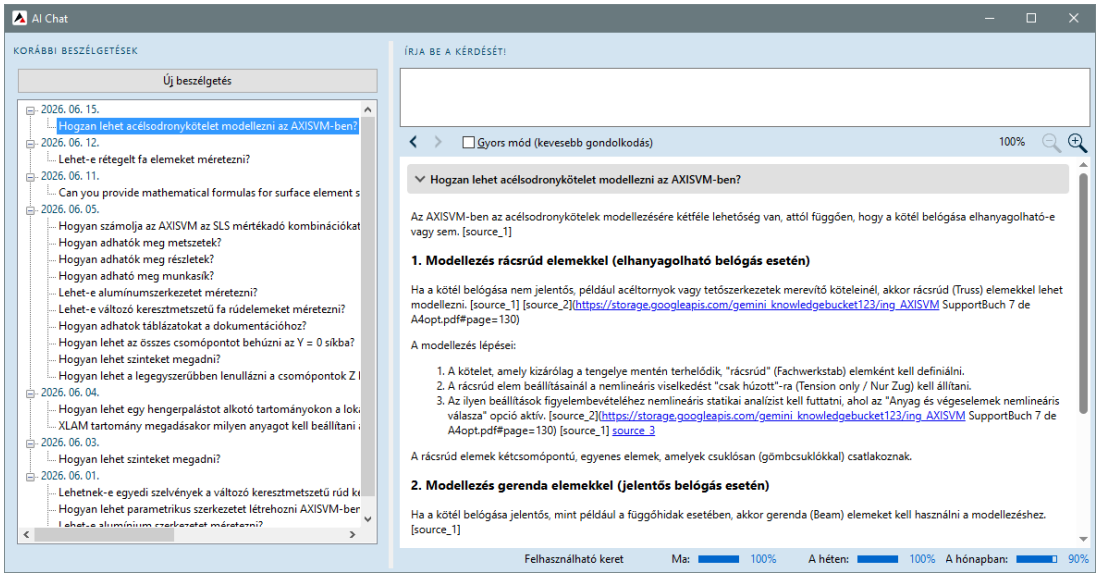
3.7 AI Tools



This menu offers AXISVM functions that use artificial intelligence. These tools are only available with an active internet connection. Running Python scripts is a utility to import, edit, save, store and run Python scripts performing function on the model.

3.7.1 AI Help

Here you can ask the artificial intelligence questions about using AXISVM. Due to limited server capacity, there is a daily, weekly and monthly quota for the number of questions you can ask. You can monitor your remaining quota at the bottom of the window.



The questions are organised into conversations. The significance of this is that the previous exchanges provide the context for subsequent questions. If you wish to change the topic, it is advisable to start a new conversation by clicking the *New conversation* button.
The communication takes place within a web browser window, so you can click on links to jump to the source of the reference. Using the left and right arrow buttons above the chat window, you can jump to the previous or next page viewed. The *Quick mode* option speeds up replying, but this comes at the cost of reduced depth of thinking. On the right-hand side, you can adjust the zoom level of the browser window in 10% increments.
AXISVM logs conversations by date and makes them available for review. In the left-hand tree, you can delete stored previous conversations by pressing the *Delete* key.
The system does not answer questions unrelated to engineering practice or AXISVM.

3.7.2 AI Selection

For details see AI Guide. 3.9.2 Guides

3.7.3 AI Model building

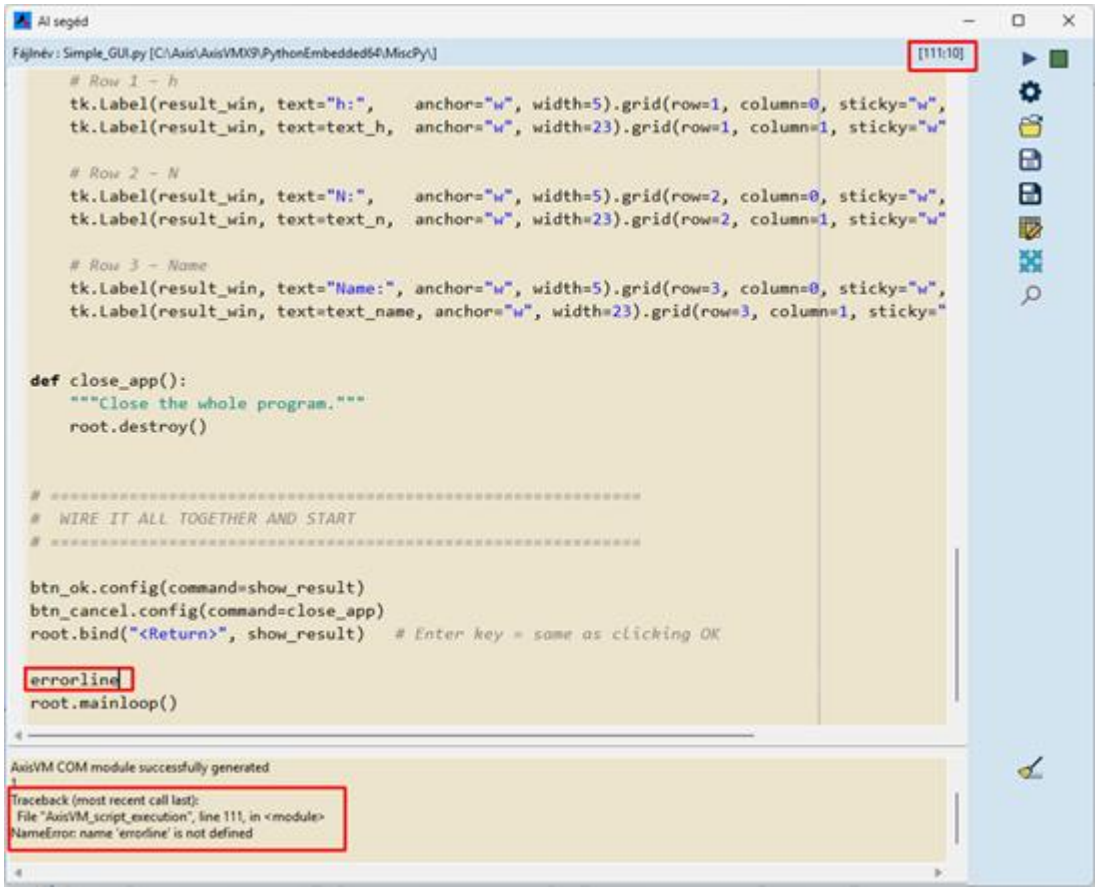
For details see AI Guide. 3.9.2 Guides

3.7.4 AI AddOn Development

For details see AI Guide. 3.9.2 Guides

3.7.5 Running Python scripts

This dialog allows you to edit and run Python source code. By sending calls to the AXISVM COM server, the running AXISVM can be controlled, making the Python programme suitable for selecting elements, as well as building and modifying models.
The Python code is displayed at the top part, its text output is shown at the bottom. The ratio between the two sections can be adjusted by dragging the dividing line. The code can be saved and loaded, imported or copied to elsewhere via the clipboard. The current row and column are displayed in the top right-hand corner. Double-clicking on this field allows you to jump to a specific row, which is useful for debugging.



Another function of the window is to compile a customisable toolbar from Python scripts, which can be moved anywhere above the main window as a floating palette to run the functions assigned to the buttons.



Run the code
The green indicator shows that nothing is currently running. Whilst a script is running, it turns to pink.



Settings
Preamble, postamble, font. You can also reset them to their default values.



Load a Python script



Save

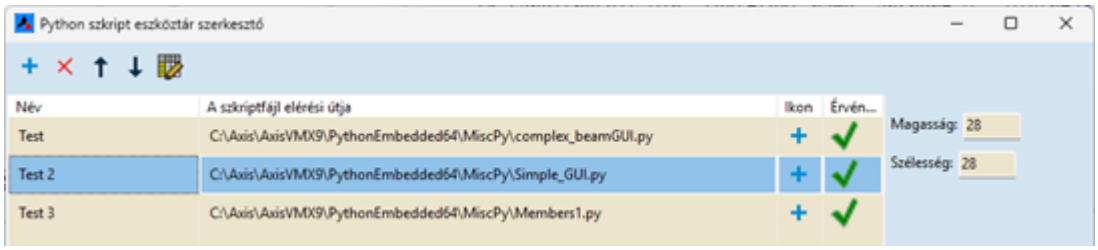


Save as



Python script toolbar editor

The buttons displayed on the floating toolbar can be configured. The Height and Width values specified apply to all buttons. The icons must be images in PNG format. For a 28 x 28 button it is recommended to use a 22 x 22 PNG.
The *Name* appears as a tooltip when the mouse hovers over a button.
The compiled and displayed toolbar palette can be freely resized and arranged in one row, one column or multi-row layouts. Their order is determined by the list.




Add a new button


Delete a button


Move up/down the button in the list.


Edit the button
Set the *Name* of the button, the Python script assigned and the icon.


Python script toolbar
The toolbar can be moved anywhere as a floating palette.


Text search in the code


Clear the text output window

Embedded Python runtime environment

AXISVM also installs an embedded 3.13 Python interpreter in the PythonEmbedded64 subfolder, which already includes the pip package manager that allows installation of additional Python packages and the comtypes package. The *axis_com_for_python.py* script, located in the PythonEmbedded64\MiscPy folder demonstrates how to connect to the AXISVM COM server. A sample icon file (DefaultButton.png) is also located there.

Installing further Python packages

To install additional Python packages, an internet connection and write permissions in the AXISVM installation directory are required. If the programme has been installed in the protected 'Program Files' folder, administrator rights are required to write files.

In the *PythonEmbedded64* directory, you can install a package using the following command:

```
python -m pip install six
```

This installs the package called 'six'. You can check the installation in the PythonEmbedded64\Lib\site-packages directory, where larger packages are usually placed in separate subdirectories.

To remove the installed package:

```
python -m pip uninstall six
```

To answer the command-line prompt, press the Y key (yes).

AXISVM makes its installation directory available via the global variable AXISVM_DIR, which we can also reference in our own programme.

Preamble, postamble

When the source code is run, AXISVM automatically inserts a preamble at the start of the code and a postamble at the end; both can be modified, but it is advisable to retain the section in the preamble that establishes the connection with the COM server:

Preamble

```
import sys
sys.path.insert(0, AXISVM_DIR + r"\PythonEmbedded64\MiscPy")  # Add your projects directory
#sys.path.extend([r"C:\Test\MiscPy", r"C:\Test\OtherDir"]) #You can add multiple paths at once too

#Importing the AxisVM COM server into python
import axis_com_for_python
try:
    import comtypes.gen.AxisVM as ax
    #the following line is an example of raw import, but that assumes you know the Major and Minor
versions (27, 0 in this example)
    #import comtypes.gen._0AA46C32_04EF_46E3_B0E4_D2DA28D0AB08_0_27_0 as ax
except:
    print("AxiVM module import error ! Trying a re-register")
    axis_com_for_python.RegisterCom()
    try:
        import comtypes.gen.AxisVM as ax
    except:
        print("AxiVM module import error, even after re-registering !")
```

The line `sys.path.insert(0, AXISVM_DIR + r"\PythonEmbedded64\MiscPy")` adds the MiscPy directory to Python's search paths, from which it will then import the axis_com_for_python.py file. This section can also be expanded to include other directories. The script then attempts to import the AXISVM COM server. This will not work on first run or after an AXISVM update. In such cases, the script attempts to re-register the COM server of the running programme:

```
    axis_com_for_python.RegisterCom()
    try:
        import comtypes.gen.AxisVM as ax
    except:
        print("AxiVM module import error, even after re-registering !")
```

Then it connects to the COM server

```
axApp = axis_com_for_python.ConnectoAxisVM()
```

Here, any other global variable name may be used instead of axApp, but in that case the post-processing step (see below) must be modified accordingly. As the code will perform operations on the model, it is advisable to store the loaded model in a separate global variable.

```
axDB = axApp.Models[1]
```

Postamble

The postamble frees what the preamble has created. By default, this is the COM server connection and the model's global variable:

```
axApp = None
axDB = None
```

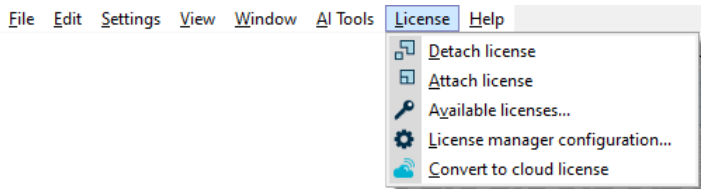
Managing COM-references

Once the COM object references you have received are no longer needed, they must be released at least at the end of the script (as shown in the postamble); failure to do so causes a memory leak, which is generally not a serious problem, but it is worth keeping an eye on.

Entering values from the command line

For testing purposes, it may be useful to enter certain input data via the command line. The recommended way to do this is:
print("Enter a name: ")
s = input()
When the print command is executed, the text appears in the text output, a dialog box pops up, and the text entered is stored in the variable *s*.

3.8 License



This menu item is only displayed for cloud-based (**CL**), software-licensed (**SL**) or Sentinel HL hardware key protection. The following services are not available for SuperPro keys. If you have a SuperPro key and you need any of these services listed here, please ask your distributor to replace the key.

Detach license



If you have a cloud-based (**CL**) or a network software license (**SL**), you can download the license to your computer temporarily. When you detach the license, you can specify the number of days you want to use the license locally.

Please note that after detaching the license, it will only be available on this computer and will not be visible to other users, so the number of available licenses will be reduced by one in case of a network key. The license will automatically expire at the given time limit and you will not be able to use it locally, but it will be automatically available again in the cloud or on the network from which you detached it.
You can also return the license to the network/cloud before the expiry date by selecting *Attach license*. It will be available to others again.
The program will detach the license you are using. If you are using AXISVM in multiple configurations with different license numbers, please check that you are running the correct configuration before detaching the license in *Help / About*. To avoid data loss, the program will automatically save the model before detaching the license. After completing the process, it connects to the local license. You can then disconnect the network connection that was necessary to access the network/cloud license.
In the case of SL network licenses, you can allow and control detachable licenses on the *Detachable Licenses* panel in the *License Manager / Configuration* menu.
Please note that after detachment, the license will act as a local SL software license, so any damage or change in the hardware may cause the license to be lost. If you are planning a hardware replacement while using the detached license, please see **2.2.4 Software key (Sentinel SL)**, refer to the *Sentinel SL license transfer* section and proceed as described.

☞ **In case of a network hardware key, licenses cannot be detached from the server.**

Attach license



Here you can send the license that you have downloaded to your computer with the *Detach license* function back to the server. Attaching the license requires network connection to the server. Before sending back the license, the program saves the model to avoid data loss. After completing the process, it reconnects to the license via the network.

Available licenses



Opens *Sentinel Admin Control Center* (ACC) at the list of licenses available from your computer. This list includes both local and network licenses. Network licenses are only displayed if the network connection is active and the server with the license is available.
In the license row, click the *Session* button in the last column to check which user is using the license. Here you can log out the current user if necessary to free up the license for another user.
If you cannot find a license, check the license manager settings in *License manager configuration*.

License manager configuration



This menu item starts the *Configuration* function of the Sentinel Admin Control Center (ACC) service. Here you can review or modify your license manager settings.
Of all the configuration options available, only the most commonly used are shown here:

Access To Remote License Managers

Set the parameters required to access the remote server here. If you have a network HL hardware license or SL software license, enter the name or IP address of the server that holds the license in the *Remote License Search Parameters* table and press *Submit*.
Turn on *Allow Access to Remote Licenses* and *Broadcast Search for Remote Licenses*. For CL cloud licenses, this field also displays your unique identifier.

Detachable Licenses

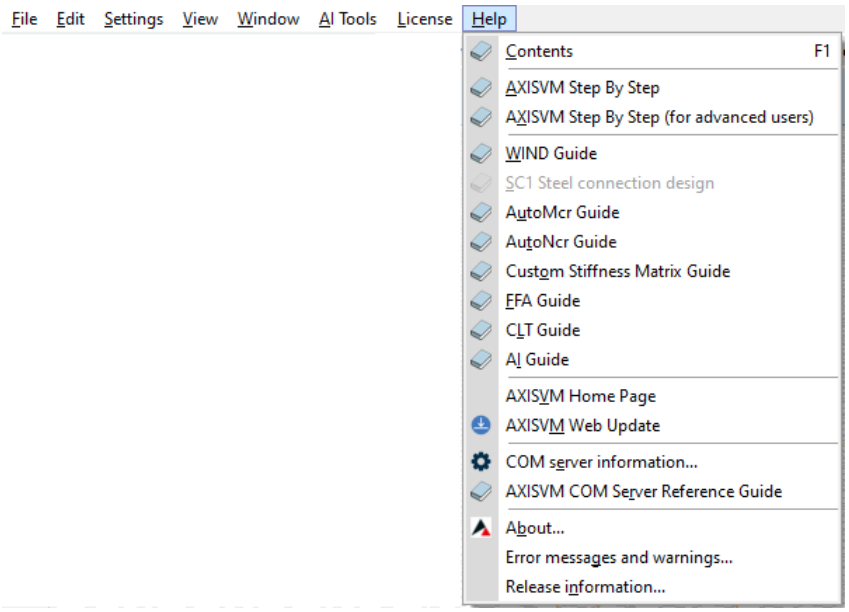
You can set the rules for disconnecting licenses. *On-Demand Detaching of Licenses* enables or disables detaching of licenses. If enabled, you can also specify a maximum detachment time.
Under *Automatic Detaching of Licenses,* you can configure the system to automatically detach the license from the network for the duration of use, so that the license is not affected by a network disconnection.
In the *Common Detach Settings* section you can control how many licenses can be detached at a time, i. e. how many licenses must always remain available on the server.

Convert to cloud license



This menu item is displayed only for SuperPro hardware keys. The licence can be transferred as follows. Using this menu item, you can remove the licence from the key and then send it to the distributor, who will replace it with a cloud licence with the same serial number. This allows the transition to be carried out without having to return the dongle or change the serial number.
When you launch this function, a warning will appear advising you to consult the distributor first, as the process cannot be undone and the licence replacement is not free of charge.

3.9 Help



The Help menu provides you with information on how to use the program and the theoretical background of certain functions. You can access the AXISVM website, download manuals and guides, and upgrade to the latest release.
You can also log into your MYAXISVM account to request and manage trial or educational licenses.

3.9.1 Contents



[F1]

Opens the present User's Manual in PDF format. AXISVM X4 Release 2 reintroduced context-sensitive help: pressing F1 in dialog windows opens the manual at the related topic.
This feature requires an application that can open PDF files at named destinations.
As Acrobat Reader supports this functionality AXISVM tries to open the manual with Acrobat Reader. If it is not installed it tries to open it with the default web browser on the computer. Most browsers can open PDF files, but the support of named destinations is not general. This feature is supported in Firefox, Chrome requires installation of the PDF Viewer extension to handle it, Edge does not support it.

3.9.2 Guides



Guides offer insight into some of the design methods used in AXISVM.
The **WIND Guide** provides a deeper insight into the operation of the WIND module, generation of wind loads based on virtual wind tunnel results, outlines the theoretical background of flow modelling and analysis issues.
SC1 Steel connection design explains how the **SC1** module should be used.
The **AutoMcr Guide** summarizes the calculation method of the lateral-torsional buckling resistance in the **SD1** steel design module. The **AutoNcr Guide** explains the method of determining the buckling length in the same module, shows the area of applications and the proper usage.
The **Custom Stiffness Matrix Guide** explains the theoretical background of calculating elements of a custom stiffness matrix that can be assigned to a domain. See 4.9.6.6 Domain with a custom stiffness matrix.
The **FFA Guide** is about the theoretical background behind the footfall analysis implemented in the **FFA** module.
The **CLT Guide** summarises the theoretical principles underlying the verification of CLT (cross-laminated timber) panels.
The **AI Guide** explains how to use the artificial intelligence-powered tools of AXISVM.

3.9.3 AXISVM Home Page

Visits AXISVM Home Page using the default Internet browser (www.axisvm.eu).

Technical support can be requested primarily through our new, efficient support system.

3.9.4 AXISVM Update

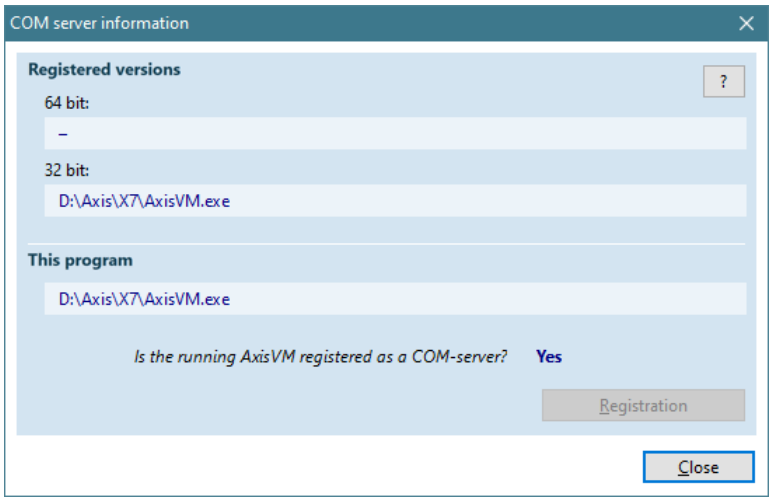
Launches the AXISVM Web Update Wizard. See **3.3.11 Preferences**

3.9.5 MYAXISVM Log in

If you are using the trial or student version of the software, you can go to **my.axisvm.eu** in your browser and log in to your MYAXISVM account. See **9 MYAXISVM – AXISVM Trial, Student, Viewer and Light version.**

3.9.6 COM server information

Provides information about AXISVM COM servers currently registered in the operating system.



You can check which executable program file is registered as the 64-bit or 32-bit COM server. If the running AXISVM is not configured to provide the COM server service, you can register it by clicking on the *Registration* button.

By default, the 64-bit version of the program is installed, and it only registers the 64-bit COM server. Run the installer a second time and select to install the 32-bit version. This will also install the 32-bit COM server.
If the program is installed with administrator rights and no AXISVM COM server (or a previous version) has been registered, the COM server registration will be done automatically. If the registered AXISVM COM server is a higher version, a warning message will be displayed and you can decide whether to overwrite it with the older version.
If you experience problems using the COM server, the first step is to check the COM server registration. There may be several registered AXISVM COM servers in the Windows Registry. To remove them, run *!UnregAxisVMComServer.bat* from the AXISVM folder as administrator. After a more time-consuming two-phase scan, you will need to confirm the deletion. All existing AXISVM COM server registrations will then be deleted and the running AXISVM can be registered as a COM server as described above.

3.9.7 About



This window displays version and release number, configuration, license number and time limit of your AXISVM version.
Available modules are in black, others are in gray.

3.9.8 Error messages and warnings...

Displays a log window that collects every warning and error message from the current AXISVM session. It helps to find error messages shown earlier from the start of the software.
The log is not saved automatically, so if you close the software, the messages will be lost.

3.9.9 Release information...

Latest release information and history of fixes and new developments.

3.10 Main toolbar





See 3.1.1 New model



[Ctrl+O]

See 3.1.3 Open



[Ctrl+S]

See 3.1.4 Save



[Ctrl+P]

See 3.1.12 Print



Making 3D PDF
Checking *Keep the U3D* the intermediary U3D file can be retained for later use. Export of edges can be controlled through the *Export edges* checkbox.
For more information see chapter 3.6.10.1 Export drawings to a 3D PDF file - PDF module



[Alt]+[Bksp]

See 3.2.1 Undo



[Shift]+[Alt]+[Bksp]

See 3.2.2 Redo



[F11]

See 3.3.3 Layer Manager



[F7]

See 3.3.4 Storeys



[F12]

See 2.9 Table Browser



[F10]

See 2.10 Report Maker



See 3.6.10 Drawings Library



See 3.6.11 Save to Drawings Library

4. The Preprocessor

The preprocessor lets you create or modify the geometry of the model, in a completely visual way. The advanced Visual Modeling feature allows quick and reliable modeling and design.

This chapter introduces the AXISVM modeling commands (geometry generation, element / mesh generation, and load case/combination definition).

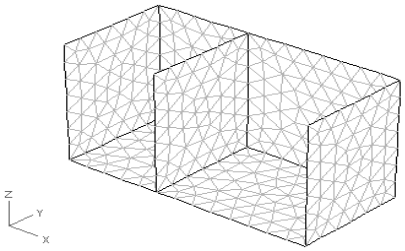
4.1 Geometry

Geometry commands let you interactively and graphically create the model geometry in 3D.
The model geometry is defined by nodes (points), mesh lines (lines) between nodes, and surfaces (triangular or quadrilateral) created from three or four appropriate lines. Later you can define finite elements based on the geometry constructed here.

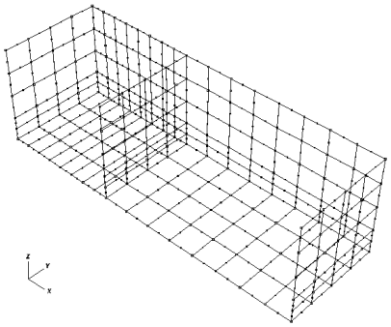
☞ **Direct drawing of objects does not require drawing geometry in advance.**
See 4.9.4 Direct drawing of objects

In the case of surface structures (plates, membranes, or shells) the mesh consists of quadrilaterals that represent the median plane of the elements.

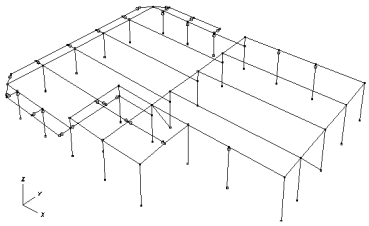
Automatic meshing on domains



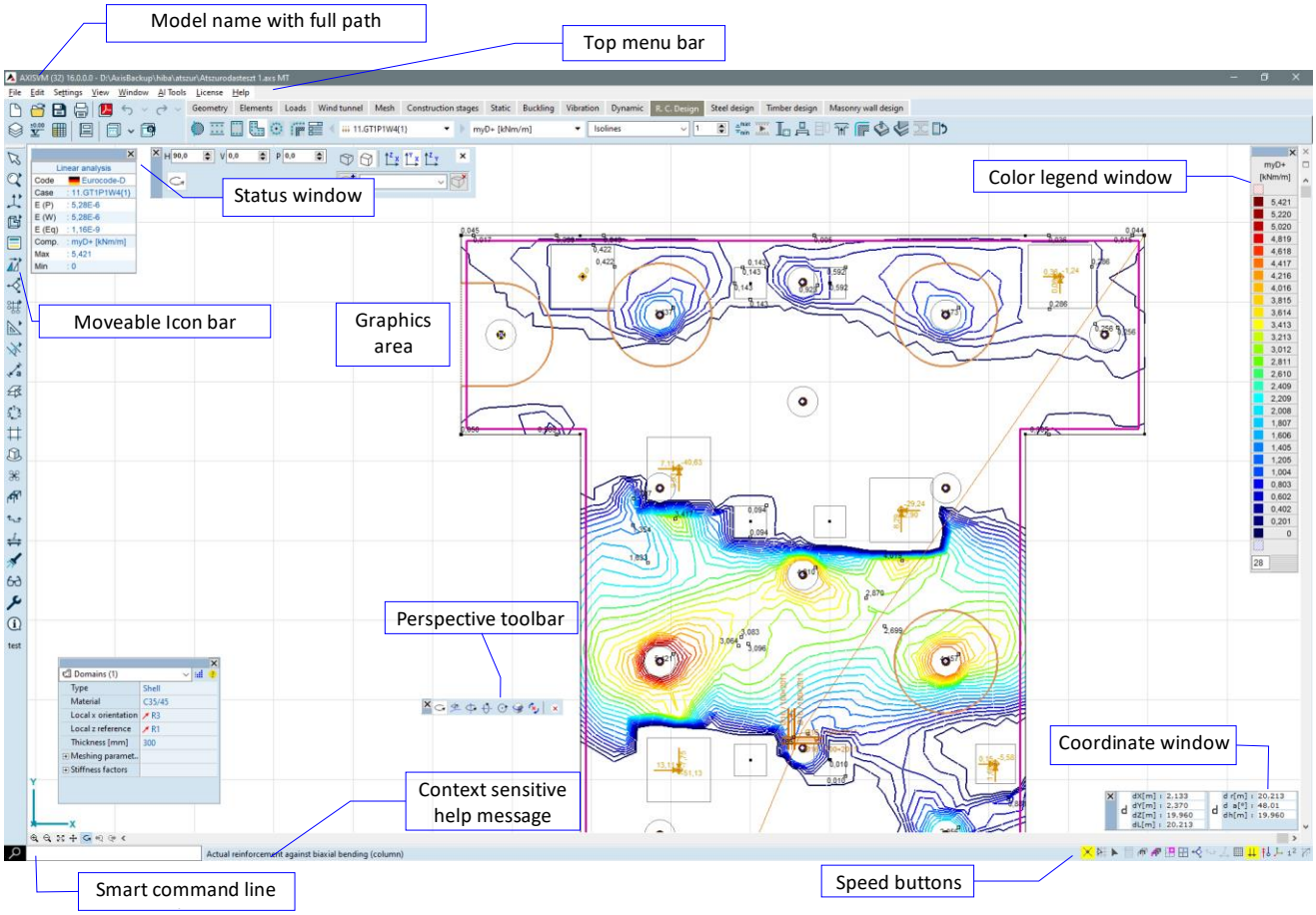
Automatic meshing on macro quads and triangles



In the case of frame structures (beams or trusses) the mesh consists of the axes of the elements.



4.2 The Model Editor



When AXISVM starts, the graphical user interface is ready for geometry editing. In case of a new model X-Y, X-Z or perspective view can be set as the default view. In case of an existing model the latest view settings will be loaded.
Using the horizontal icon toolbar at the top of the graphics area you can apply various commands to construct geometry meshes describing the geometry of your finite element model.
See 4.8 Geometry Toolbar

Using the vertical icon bar on the left you can apply commands that change the display of the model, and can configure the working environment of the editor.
See 2.16 The Icon Bar

4.2.1 Multi-window mode

When the model is complex, it is useful to display different views of the model simultaneously on the screen. AXISVM allows you to split the graphics area horizontally or vertically. Each newly created graphics window has its own settings, and allows the independent display of the model views. This feature is also useful when interpreting results. Find split commands in the *Window* menu.

Split horizontally

Splits the active graphics window horizontally into two equal parts. The top window will become the active window.

See 3.6.4 Split horizontally

Split vertically

Splits the active graphics window vertically into two equal parts. The left window will become the active window.

See 3.6.5 Split vertically

Close Window

Closes the active window if there are more than one graphics windows in use.
The new default window will be that in which you previously worked.

You can change views during any editing command.

☞ **In the perspective view some editing commands cannot be used, or are limited in use.**

4.3 Coordinate systems

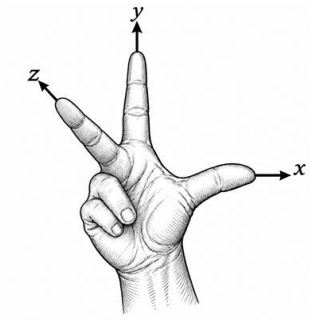
AXISVM uses different coordinate systems, to describe the model. The global coordinate system is used to describe the model geometry. Local coordinate systems are mainly used in the element definitions. The local systems are usually defined by the element geometry and additional references. AXISVM denotes the axes of the global system with capital letters, and the local axes with small letters.
The geometry can be created using Cartesian, Cylindrical or Spherical coordinate systems.

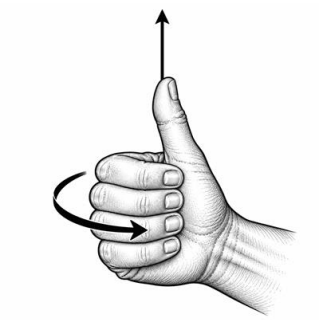
See 4.3.2 Polar coordinates

4.3.1 Cartesian coordinate system

Base coordinate system

AXISVM uses Cartesian coordinates to store geometry data. AXISVM uses the right-hand rule exclusively to define the positive directions of axes and rotation. The illustration shows the positive directions of the axes and of rotation according to the right-hand rule.





Global and relative origin

A new model uses the view selected in the New Model dialog (see 3.1.1 New model). The origin of the coordinate system is shown by a blue X initially located at the left bottom corner of the editor window.
A fixed (X, Y, Z) and a relative (dX, dY, dZ) global system are used to locate points (nodes) in your model. The origin of the relative system can be moved anywhere (using **[Alt]+[Shift]** or **[Insert]**), at any time during modeling.
The Coordinate Window displays either the fixed or the relative global coordinates according to its current settings. If the relative mode is selected, the denotation of axes becomes dX, dY, dZ.
With the help of the Coordinate Window, and according to the movement of the relative origin you can make measurements on the model (distances, angles).
The nodal displacements and mode shapes refer to the fixed global system.

☞ **In the X-Y and Y-Z views the third axis (normal to the view's plane) is oriented towards you. As a result, when a copy is made by translation with a positive increment about the respective third axis, the copies will be placed nearer you. In X-Z view the opposite occurs as the third axis in this case points to the opposite direction.**

See 4.9.25 References

4.3.2 Polar coordinates

In addition to the Cartesian global coordinate system, you can use either a cylindrical or a spherical coordinate system. One of the polar coordinate systems can be selected through its corresponding radio button in *Settings / Options / Editing / Polar coordinates*.

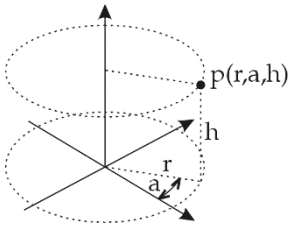
In the Coordinate Window three variables will be displayed depending on selection:

Cylindrical

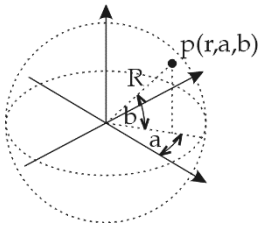
- *h:* the value measured from the view plane to a point on the cylinder's main axis (that is perpendicular to the view plane) oriented outward from the screen
- *r:* radius that is the distance on the view plane from the projection of the point to the cylinder's main axis
- *a:* the angle between the line that joins the point with the origin and the horizontal

Spherical

- *r:* the radius, that is the distance from the point to the sphere's center (origin)
- *a:* the angle on the view plane between the line that joins the projection of the point with the origin and the horizontal
- *b:* the angle between the line that joins the point with the origin and the view plane, which is positive if the point is in front of the view plane (between the user and the view plane).

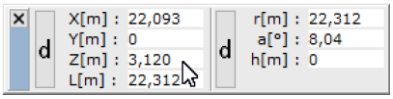


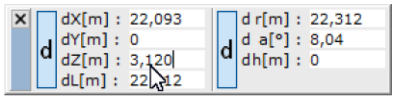
Cylindrical coordinate system



Spherical coordinate system

4.4 Coordinate window





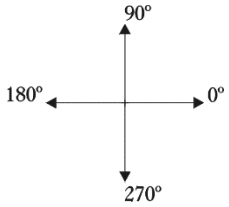
Displays the current absolute and relative values of the cursor position in the global coordinate system (Cartesian and cylindrical or spherical).

You can switch between absolute and relative coordinate displays, by clicking on the letters *d* in the Coordinate Window (delta switch). The display of the *d*'s also show whether the relative coordinates are enabled or not.

During editing it is possible to jump into the coordinate window by shortcuts. The default assignment can be changed, see **3.3.12 Keyboard shortcuts**

Coordinate command	*Shortcut*
X	**X**
Y	**Y**
Z	**Z**
L	**Shift+Ctrl+L**
r	**Shift+Ctrl+R**
a	**Shift+Ctrl+A**
h	**Shift+Ctrl+H**
b	**Shift+Ctrl+B**
Temporary workplane	**Shift+Ctrl+W**
Lock X	**Ctrl+Alt+X**
Lock Y	**Ctrl+Alt+Y**
Lock Z	**Ctrl+Alt+Z**
Lock L	**Ctrl+Alt+L**
Lock r	**Ctrl+Alt+R**
Lock a	**Ctrl+Alt+A**
Lock h	**Ctrl+Alt+H**
Lock b	**Ctrl+Alt+B**
Relative / global coordinates (delta switch)	**Shift+D**
Relative / global polar coordinates	**Shift+E**

The positive angles, α:



☞ **The relative switch (delta) can be used together with the constrained cursor movements.** See 4.7.4 Constrained cursor movements.

☞ **You can enter expressions in the edit fields (e.g.: 12.927+23.439, cos(45), sin(60))**

4.5 Grid

See in detail 2.16.22.1 Grid and cursor

4.6 Cursor step

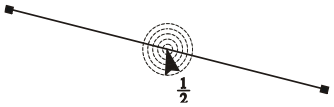
See in detail 2.16.22.1 Grid and cursor

4.7 Editing tools

Editing tools help the work by several features. See 2.16.22.2 Editing

4.7.1 Cursor identification (object snap)

Sets the size of the cursor identification area (in pixels).



When you position the cursor over the graphics area, AXISVM finds the entity of the model that is closest to the center of the cursor from among the entities that are located in or intersect the identification area. The size of the identification area can be set at *Settings / Options / Editing / Cursor identification*.

The current shape of the cursor shows what kind of entity was identified. Depending on entity type, the cursor will have the following shapes:

Node

Mid-side node

Support

Edge hinge

Mesh independent load

Load polygon vertex

Center of an arc

Arc

Tangent

Bézier curve

References

Line

Surface

Domain

Rigid element

ARBO element

CRET element

Text box, label

Reinforcement domain, AirDeck solid area

Edge, corner of a pad footing

Edge, corner of a strip footing

SC1 bolted joint

Virtual beam or virtual strip

Guideline

Structural gridline

Intersection

Perpendicular (normal)

Dimension line

In case of Pick up function

If there are several entities at the same location, the program identifies the first entity according to the ordering of the list above. If there are multiple entities of the same type, the cursor will show a double symbol.

☞ **Use the Coordinate Window to find out which one of the elements was actually identified.**

Background detection

The cursor can be set to snap to the lines on architecture background layers.

For further object snap control options see 2.18.1 Object snap

4.7.2 Entering coordinates numerically

During the model editing, coordinates of the cursor can be specified directly entering the numerical values into the Coordinate Window. There are two ways to enter the numerical values:

1. by pressing the corresponding character button on the keyboard
2. by clicking with the left button on the desired coordinate value display field, and then typing in the value.

If the relative mode is enabled (the letter *d* is pressed), the coordinates you enter will define a point from the relative origin.
If contradictory values are entered (in case of a constraint), the last entered value will update the others.

☞ **You can enter expressions in the edit fields (e.g.: 12.927+23.439, cos(45), sin(60))**

The relative origin can be moved at any time to the current mouse cursor position by pressing the Insert key. So, when drawing a polyline, you can specify the end point coordinates relative to the previous point.

☞ **To draw a line with a given length and direction move to relative origin to the starting point (using [Alt]+[Shift] or [Insert]), enter the angle at d a[°] and enter the length at d r[m] then press the Enter button.**

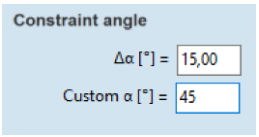
4.7.3 Measuring distance

The distance between two points or the length of a line can be measured by moving the relative origin onto the first point and then identifying the second point by positioning the cursor over it. In this case the value of *dL* in the Coordinate Window is the distance between the points.

The cursor can be moved to a location relative to a reference point by moving the relative origin onto the reference point, then entering the angle in the input field *da* and the distance in the *dr* input field.

4.7.4 Constrained cursor movements

The cursor movement constraints can be customized in the *Settings / Options / Editing* dialog. The constrained cursor movements use the following values:



Δα Holding the **[Shift]** key pressed, the cursor is moving along a line that connects its current positionwith the origin, and that has an *n*Δα* angle, where the value of *n* depends on the current cursor position.

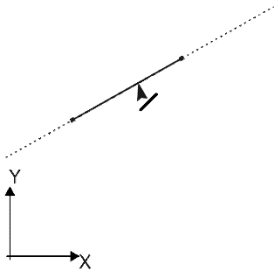
Custom α Holding the **[Shift]** key pressed, the cursor is moved a line that connects its current position with the origin, and that has an α or $\alpha + n*90°$ angle, where the value of n depends on the current cursor position. $\Delta\alpha$ and α can be set in *Settings/Options/Editing/Constraint Angle*.

The meaning of origin depends on the *d* switches of the coordinate palette. Turning off both the origin will be the global origin. Turning on any of the *d* switches the origin will be the local origin.

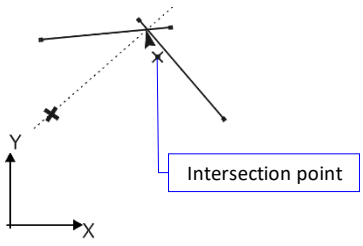
☞ **You cannot use *Δα* and *Custom α* constraints in perspective view.**

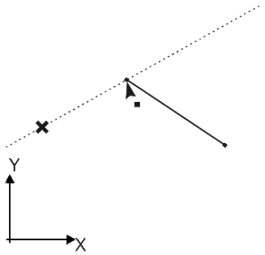
If the cursor is over a line, holding the key **[Shift]** depressed, will constrain the cursor movement to the line and its extension .

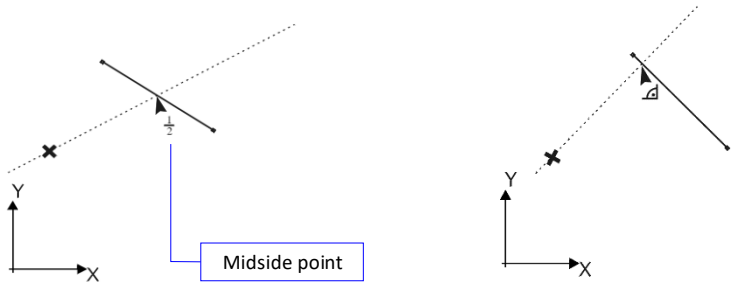
If the cursor identifies a point, holding the key **[Shift]** depressed, makes the cursor move along the line defined by the point and the relative origin..



When the cursor identifies a **domain or surface element** pressing **[Shift]** makes the cursor move in the plane of the element**.**







Geometry tools



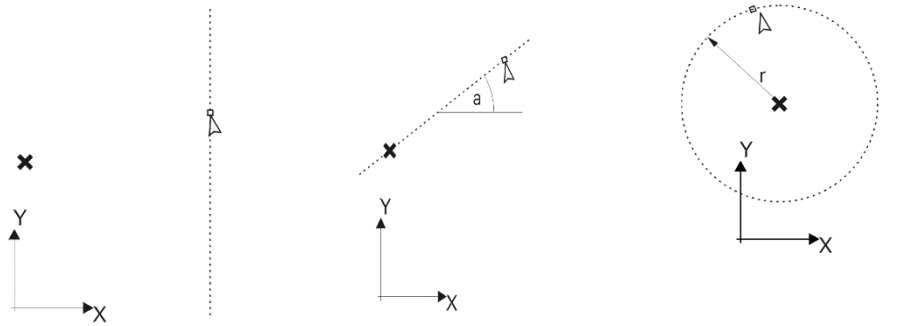
The icons of *Geometry tools* allow you to lock the direction of drawing a line.
See **2.16.10 Geometry tools**

4.7.5 Locking coordinates

You can lock the value of a coordinate, allowing for better positioning. A locked coordinate will not change on cursor motion. Locking can be achieved by using **[Ctrl+Alt] + [X], [Y], [Z], [L], [R], [A], [B], [H]** respectively.

A black rectangle over the coordinate input field shows that the coordinate is locked. To cancel coordinate locking, press the same button combination, that was used to lock it.

Frozen X coordinate *Frozen angle* *Frozen radius*



4.7.6 Auto intersect

At the intersection point of the lines, a node will be generated and the lines will be bisected. If surfaces are intersected by lines, they will be split, and the resulting elements will have the same material and cross-sectional properties as the original. Set the line intersection options in *Settings / Options / Editing / Auto Intersect*. See **2.16.22.2 Editing**
If Auto Intersection is on, surfaces will be divided into smaller surfaces if necessary. Surface finite elements are also divided and the new elements inherit the properties and loads of the original element.

4.8 Geometry Toolbar

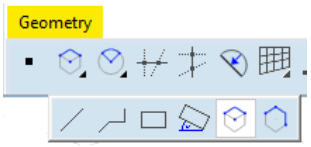


These tool buttons create new geometry or change the existing one..

☞ **If you are working on parts and** *Settings / Options / Editing / Auto / Part Management* **option is checked then all new geometric entities will be added to the active parts.**

The geometric entities can be selected prior to applying the geometry construction commands, as well.

You can make this toolbar shorter via *Settings / Preferences / Toolbar* by grouping similar commands and placing them on a flyout toolbar that appears when clicking near the black triangle at the bottom right corner of the icon or holding down the mouse button longer. The selected command will replace the current icon.



4.8.1 Node (Point)



Lets you place new nodes or modify existing ones.
To place a node:

1. Move the graphics cursor to the desired location and press the **[Space]** key or the left mouse button (in perspective view you can place nodes only to special locations).
2. Enter the node coordinates numerically in the Coordinate Window, and then press **[Space]** or **[Enter]** (it works in all views).

You can place a node on a line or surface. If the *Settings / Options / Editing / Auto Intersect* check-box is enabled, the line or surface will be divided by the new node, otherwise it remains independent of the line.

☞ **If nodes are generated closer to each other than the tolerance specified in** *Settings / Options / Editing / Editing Tolerance* **value, nodes will be merged.**
When working on parts with *Settings / Options / Editing / Auto / Part Management* **turned on all geometric entities created will be automatically added to the active parts.**

4.8.2 Line



The *Line* tool is to construct lines or other simple shapes. If flyout toolbars are activated (see above) the line type can be chosen by clicking on the triangle at the bottom right corner of the currently used *Line* tool icon or holding down the mouse button longer, and then clicking on the desired icon.

The *Line* tool offers the following options to draw simple shapes:



Line



Constructs straight lines by defining their end points (nodes). You must graphically or numerically (by the Coordinate Window) specify the end points (nodes). The command lets you generate one or more independent lines. You can cancel the process by pressing the **[Esc]** key or the right mouse button.
In perspective view lines are drawn on the Z = 0 plane by default. To draw lines in perspective in a different plane workplanes can be used.
See 2.16.7 Workplanes.

Polyline



Constructs a series of connected straight lines (a polyline). You must specify the vertices.

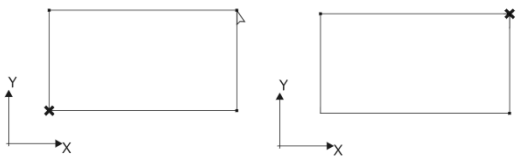
Exit current polyline by any of the following

1. **[Esc]** key
2. **[Esc]** key a second time will exit polyline drawing mode.
3. right button & *popup menu/Cancel*
4. left button while pointing to the last point (node) of the current polyline.

Rectangle



Constructs a rectangle (its corner points (nodes) and edge lines). You must specify two opposite corner points.

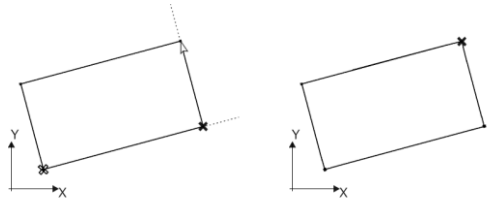


After you specified the first corner you can cancel the command by pressing the **[Esc]** key. This command is not available in perspective view.

Skewed rectangle



Constructs a skewed rectangle (its corner points (nodes) and edge lines). You must specify one of its sides (by its end points), and then the other side.



After you specify the first corner you can cancel the command pressing the **[Esc]** key. In perspective view, you can draw skewed rectangles using only the existing points.

Polygon 

Number of sides has to be defined in a dialog. Polygon has to be defined by entering a centerpoint and 2 polygon points.

Polygon 

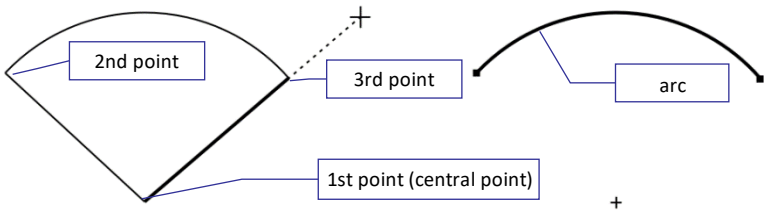
Number of sides has to be defined in a dialog. Polygon has to be defined by entering three points of the arc.

4.8.3 Arc

Draws an arc or a circle. Arcs and circles will be displayed as polygons according to the *Arc resolution* set in *Settings / Preferences / Display*.
[Esc] cancels the command.

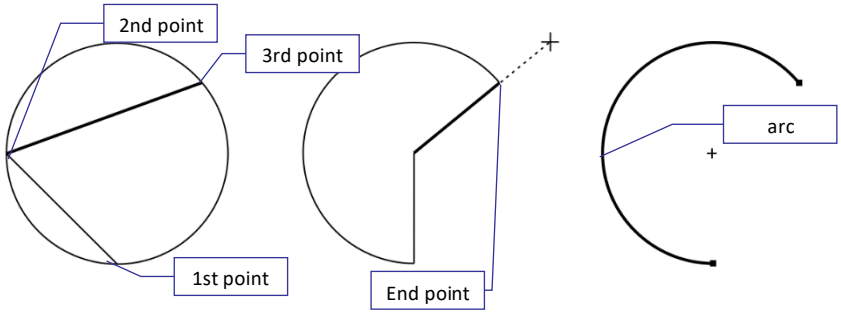


Defining an arc by its radius, and starting and ending points.





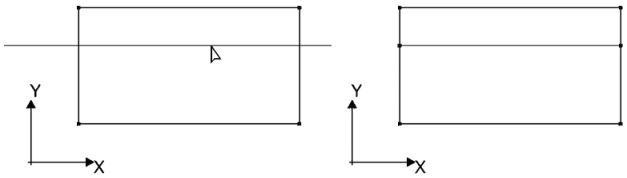
Defining an arc by three points. The command can be applied in perspective setting as well.



4.8.4 Horizontal division



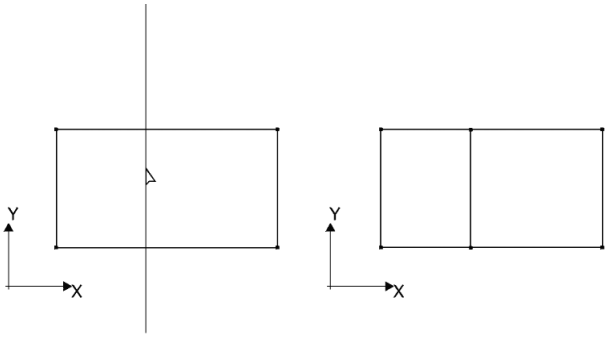
This function creates a horizontal divider line passing through the cursor position. This line is in a plane parallel to the X-Y, X-Z or Y-Z plane depending on the actual view (or parallel to the workplane if a workplane is used). Creates new nodes at the intersections. If finite elements are intersected new elements inherit properties and loads of the original element.



4.8.5 Vertical division



This function creates a vertical divider line passing through the cursor position. This line is in a plane parallel to the X-Y, X-Z or Y-Z plane depending on the actual view (or parallel to the workplane if a workplane is used). Creates new nodes at the intersections. If finite elements are intersected new elements inherit properties and loads of the original element.



4.8.6 Replace selected arcs with straight lines



This function converts the selected arcs to straight lines. It can be useful when arcs are already created and divided in the model but to determine proper intersections with other elements straight line segments are needed.

4.8.7 Quad/triangle division

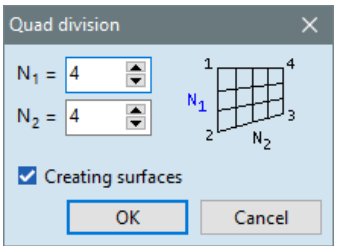


Visibility of these icons is controlled by the option *Settings/Preferences/Toolbar/Show obsolete surface element tools on toolbars*.
Constructs a mesh of quads/triangles over a quad or triangle. This command is to generate a macro mesh before applying a finite element mesh generation command. If the mesh is fine enough, it can be used directly as a finite element mesh. Checking *Creating surfaces* makes surface elements from the triangles or quadrilaterals. Introduction of domains and automatic meshing rendered this method obsolete.

Quad-to-quads



Generates an $n \times m$ mesh between the corners of a 3D quad (not necessarily flat, or with any side lines). You must successively graphically select the corners (four points), and specify the number of segments ($N_1 \geq 1$) between corners 1 and 2, and the number of segments ($N_2 \geq 1$) between corners 2 and 3.



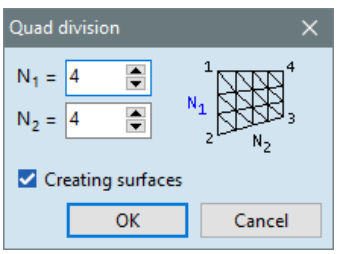
The quad and the mesh are displayed with solid grey lines.

If the mesh leads to quad subdivisions that are distorted (have an angle smaller than 30° or greater than 150°), the quad is displayed with grey dotted lines.
If a quad shape is entered that is not allowed (e.g., concave), the quad is displayed with red dotted lines.

Quad-to-triangles



The command is similar to the quad-to-quads command, but each generated quad is divided into two triangles by its shorter diagonal.
The quad and the mesh is displayed with solid grey lines.
If the mesh leads to triangle subdivisions that are distorted (have an angle smaller than 15° or greater than 165°), the quad is displayed with grey dotted lines.

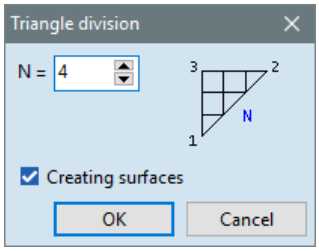


If a quad shape is entered that is not allowed (e.g., concave), the quad is displayed with red dotted lines.

Triangle-to-quads



Constructs a mesh between the corners of a triangle (not necessarily with any side lines). The mesh will also contain triangles along the side that corresponds to the first two corners entered.
You must graphically select the corners successively (three points), and specify the number of segments *N* between corners.



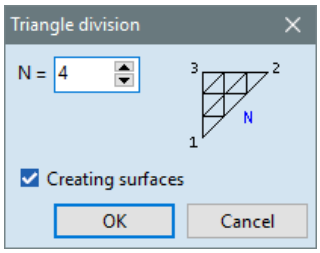
The triangle and the mesh are displayed with solid grey lines.

If the mesh leads to quad subdivisions that are distorted (have an angle smaller than 30° or greater than 150°), or to triangle subdivisions that are too distorted (has an angle smaller than 15° or greater than 165°), the triangle is displayed with grey dotted lines.
If a quad shape is entered that is not allowed (e.g., three collinear corners), the triangle is displayed with red dotted lines.

Triangle-to-triangle



The command is similar to the triangle-to-quads command, except that each generated quad is divided into two triangles by its diagonals which are parallel to the side first entered.



Same as for triangle-to-quads.

4.8.8 Intersect



If the *Settings / Options / Editing / Auto / Intersect* checkbox was not enabled in the dialog window at the time of creating the geometric entity, using this command you can intersect the selected lines. Intersection can be filtered by element types in a dialog. Selected lines will be divided by creating nodes (points) at their intersections.
If finite elements are assigned to the lines, finite elements are also divided and inherit the properties and loads of the original element..

You can select elements for intersection beforehand.

4.8.9 Remove node



Removes the selected nodes at the intersections of lines. It makes it easier to construct trusses crossing but not intersecting each other or to remove unnecessary division points along a line.

Intersection nodes can be removed only if the number of connecting lines are even and lines can be joined.

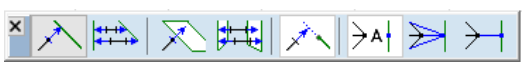
4.8.10 Remove intermediate nodes



Removes unnecessary intermediate nodes on lines. Nodes with two connecting lines are removed provided they can be joined.

4.8.11 Extend lines to meet another line or plane





Extends existing straight lines or arcs to meet another straight line, arc or plane.

Extension modes



Extend lines to meet another line
The first step is to specify the boundary: click on an existing line or arc or enter two points of an imaginary boundary line. Click on lines near their end point to extend.



Extend lines to meet more than one line
The first step is to click on multiple lines defining the boundary. Click OK on the selection toolbar or press Enter. Select the lines to extend. Click OK on the selection toolbar or press Enter. Both ends of the lines will be extended to meet the boundary.



Extend lines to meet a plane
The first step is the specify the boundary plane: click on an existing domain or enter three non-collinear points of an imaginary boundary plane. Click on lines near their end point to extend.



Extend lines to meet more than one plane
The first step is to click on multiple domains defining the boundary. Click OK on the selection toolbar or press Enter. Select the lines to extend. Click OK on the selection toolbar or press Enter. Both ends of the lines will be extended to meet the boundary.

Settings



Extend lines even if there is no actual intersection
This is a two-state button. If it is down (its background turns white) lines will be extended even if the point of intersection does not fall between the start and end point of the boundary line. If it is up (its background is blue) lines will be extended only if the point of intersection is on the boundary line.

The last three buttons control the behaviour of lines connected to the extended point.



Move free ends, add splice if other elements are connected
The program automatically chooses from the two options listed below. If no lines are connected to the extended point, it chooses the first option, otherwise it chooses the second one.



Always move line ends
The node of the extended point is moved to the point of intersection. The direction of lines connected to this node will change.



Always splice line ends
The node of the extended point is not moved. A new node will be created at the point of intersection and a new line (a splice) is inserted between the two nodes. Connected lines remain unchanged.

4.8.12 Trim lines to meet another line or plane





Trims existing straight lines or arcs to meet other straight lines, arcs or planes.

Trimming modes



Trim lines to meet another line
The first step is to specify the cutting line: click on an existing line or arc or enter two points of an imaginary cutting line. Click on the line segments to be removed.



Trim lines to meet other lines
The first step is to click on multiple lines defining the cutting lines. Click OK on the selection toolbar or press Enter. Select the lines to trim. Click OK on the selection toolbar or press Enter. Click on that side of the cutting lines where you want to remove the line segments.



Trim lines to meet a plane
The first step is the specify the cutting plane: click on an existing domain or enter three non-collinear points of an imaginary cutting plane. If the cutting plane intersects the lines click on the line segments to be removed.



Trim lines to meet more than one plane
The first step is to click on multiple domains defining the cutting planes. Click OK on the selection toolbar or press Enter. Select the lines to trim. Click OK on the selection toolbar or press Enter. Click on that side of the cutting planes where you want to remove the line segments.

Settings



Trim lines even if there is no actual intersection
This is a two-state button. If it is down (its background turns white) lines will be trimmed even if the point of intersection does not fall between the start and end point of the boundary line. If it is up (its background is blue) lines will be trimmed only if the point of intersection is on the boundary line.

The last three buttons control the behaviour of lines connected to end point of trimmed lines.

The trimmed line always gets a new end point at the point of intersection.



Move free ends, remove segment if other elements are connected
The program automatically chooses from the two options listed below. If lines are connected to the extended point, it chooses the second option.



Always move line ends
The end point of connected lines will move to the new end point. The old end point is removed.



Always remove line segment
The original end point and connected lines remains unchanged.

4.8.13 Extend/cut lines to their point of intersection





Modes



Extend/trim two selected lines to their point of intersection
Extends/trims pairs of consecutively selected lines to their point of intersection.



Extend/trim selected lines to the point of intersection with the first one
Extends/trims consecutively selected lines to their point of intersection with the first line.

The next two buttons control the behaviour of arcs where there may be two points of intersection.

Settings



Extend the arc end where the extension is shorter
The program always chooses the end where the point of intersection is closer.



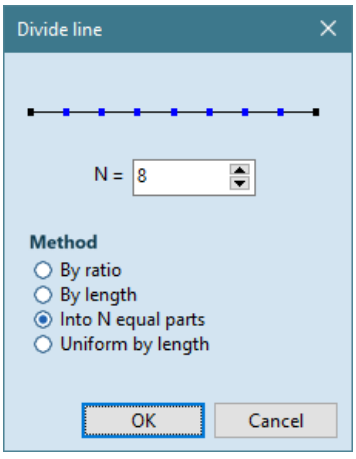
Click to choose the arc end to extend
The end to extend can be selected by clicking near the chosen end.

The last three buttons control the behaviour of connected lines according to the above description.

4.8.14 Line division



Lets you create new point (nodes) on the selected lines.
The following input options are available:



By Ratio: Lets you divide the selected lines into two segments. You must specify the parameter *a* of the location of the inserted node relative to the first node (*i*). The parameter *a* must be between 0 and 1. *a* = 0.5 represents a division of the selected lines into two equal segments.

By Length: Lets you divide the selected lines into two segments. You must specify the length *(d)* of the segment corresponding to the first node (*i* end). The parameter *d* must be between 0 and the total length.

Into N equal parts: Lets you divide the selected lines into several equal-length segments. Specify the number of segments *(N)*.

Uniform by length: Lets you divide the selected lines into several equal-length segments. Specify the length of segments *(d)*.



before division



after division

If finite elements are divided the new elements inherit properties and loads of the original elements.

☞ **If you divide surface edge lines surface elements will be deleted.**

4.8.15 Normal transversal



Creates a connection between two lines along their normal transversal.

4.8.16 Intersect plane with the model



After defining the intersecting plane intersection lines and nodes will be added to the model. Domains, beams and ribs will be divided.

4.8.17 Intersect plane with the model and remove half space

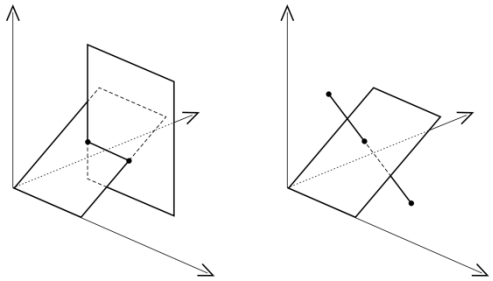


This operation is similar to Intersect plane with the model, but after defining the plane a half space can be selected. Elements within that half space will be deleted.

4.8.18 Domain intersection



Creates intersection lines of domains and line elements. After clicking the tool button select domains to create their intersection or select a domain and a line to create the intersection.



4.8.19 Purge unnecessary lines and nodes



This function removes nodes not connected to any element (line element, domain, support, etc.) and lines without element properties assigned. Possible options are

From the selection: Only selected nodes and lines will be checked.

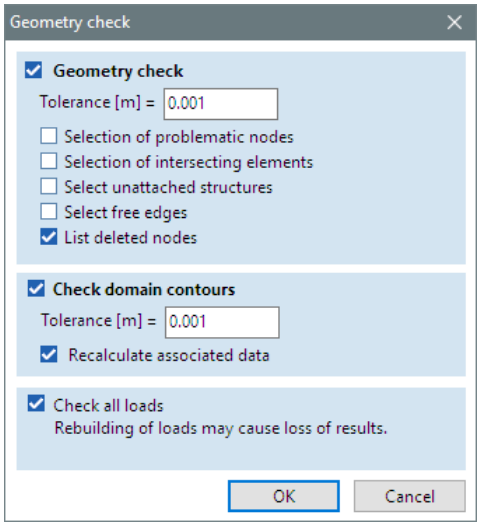
Also check nodes and lines within domains: When mesh-independent loads are placed on the domain additional nodes and lines may be created within the domain to guide the meshing algorithm. Deleting the mesh does not remove these nodes and lines. As these nodes and lines have no physical role in the model they are detected during purging. Leaving this option unchecked keeps them.

Select but don't remove: Unnecessary lines and nodes will be selected and not removed.

4.8.20 Geometry check

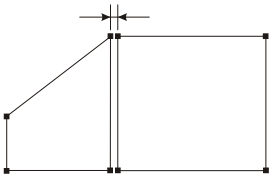


Finds unattached or duplicated nodes and lines within the given *Tolerance*. Nodes closer than this distance are considered to be duplicates.
If *Selection of problematic nodes* is checked nodes closer than *Tolerance* will be selected. If it is unchecked these nodes will be deleted and a new node will be created with averaged coordinates. The command reports the number of merged nodes/lines. If *List deleted nodes* is checked a list of deleted nodes is displayed using the node numbers *before* the deletion. Lines connected to the nodes will be replaced with a single line to the new node.
If *Selection of intersecting elements* is checked it selects intersecting elements where there is no node at the point of intersection.

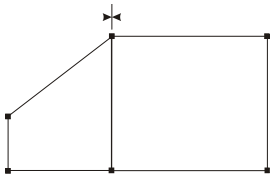


If the *Select unattached structures* is checked a warning will be displayed if there are independent lines or nodes not connected to the rest of the structure.

The command *Select free edges* selects all edges connected to one surface element only. This selection helps to find edges where the meshes of two domains are not connected due some editing problem.



before checking



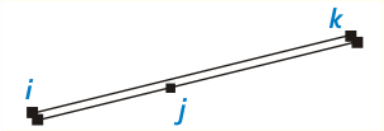
after checking

The function fixes domain contours forcing contour segments into the same plane checks the openings and adjusts arcs if radius is not the same at the start point and the end point.

Checking domain contours If checked the program removes zigzags from nearly straight segments of domain contours if the deviation from the line is below the *Tolerance* set here.

Check all loads This function recreates all loads that depend on the geometry (like surface loads distributed over beams or load panel loads). Rebuilding of loads may cause loss if results.

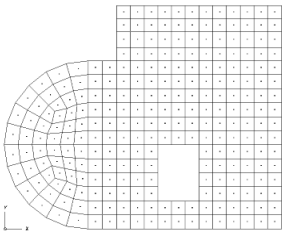
☞ **The case on the right is not identified by the** *Check* **command. To avoid having hiding lines check** *Settings / Options / Editing / Auto / Intersect* **or click** *Intersect* **on the** *Geometry* **toolbar.**

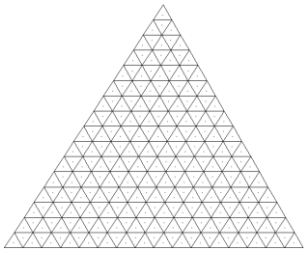


4.8.21 Surface



In any cases when you wish to model surfaces (plates, membranes, or shells) you have to create a mesh that consists of triangles and convex flat quadrilaterals. The mesh then can be refined. The command searches all triangles and quads in the selected mesh of lines. You must select all surface edges when applying the command.
Introduction of domains and automatic meshing rendered this method obsolete.





The number of surfaces detected is displayed in an info dialog.
The reported surfaces are geometry surfaces but not surface elements. You can make them surface elements by assigning material and cross-section properties to them.

☞ **Quads have to be flat. AXISVM takes into account only those surfaces that have an out-of-plane measurement smaller that the tolerance entered in the** *Settings / Options / Editing / Editing Tolerance.*

4.8.22 Modify, transform

Lets you modify existing geometric entities.
To modify nodes or lines:

1. Position the cursor over the node/line/centre of surface.
2. Holding the left mouse button pressed, drag the node/line/surface.
3. Drag the node/line/surface to its new position, or enter its new coordinates in the Coordinate Window, and then press enter or press the left mouse button again.

☞ **If multiple nodes and/or lines are selected, the position of all nodes and lines will be modified.**

Fast modify: Clicking a node you get to the Table Browser where you can enter new coordinate values. If multiple nodes are selected and you click one of them, all the selected nodes will appear in the table.

Moving selected nodes into the same plane: if the plane is a global one you can move selected nodes into this plane easily.

1. Click on any of the selected nodes.
2. Select the entire column of the respective coordinate.
3. Use *Edit / Set common value* to set a common coordinate value.

Using pet palettes Depending on the type of the dragged element different pet palettes appear on the screen. Their position can be set in *Settings / Preferences / Toolbar*. See **3.3.11 Preferences**

Dragging nodes



 Dragging node with connecting lines

 Dragging node disconnecting the selected lines

 Dragging all connecting lines

 Chamfer

 Fillet

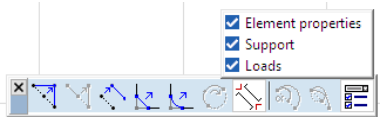
 Lengthening or shortening connecting arcs

 Detaching a copy of the node

 Keeping the central angle of the connecting arc constant.

 The new arc is defined by the dragged node, the start point and midpoint of the original arc.

 Enabled only in detaching mode. It pops up a list of properties to be copied.

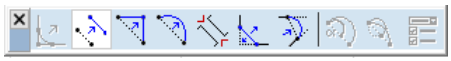


Entering node coordinates: Clicking a node the table of nodes appears where coordinates can be changed. After selecting one or more nodes their coordinates can be edited in the property editor as well.

Examples of aligning nodes to a plane if this plane is parallel to one of the global coordinate plane:

1. Select nodes to align.
2. Enter the required coordinate value in the property editor.

Dragging lines



 Fillet

 Dragging the line parallel to its original position

 Breaking the line at a given point by adding a node

 Converting to arc

 Detaching a copy of the line

 Dragging a cut-off parallel to its original position

 Replacing a straight line with an arc based on two end point tangents.

 See *Dragging nodes*

Modifying arcs



 Fillet

 Dragging the arc parallel to its original position

 Converting to line

 Changing arc radius



Inflating / deflating arc



Detaching a copy of the arc



See *Dragging nodes*

Transforming objects

See **2.16.6 Geometric transformations on objects**

4.8.23 Delete

[Del] See in detail **3.2.9 Delete**

4.9 Finite Elements

The commands related to the definition of the finite elements are described below.



The commands associated with the icons let you define the finite elements used for modeling. In the definition process you must define and assign different property sets.

Properties of finite elements

Depending on the type of finite element, you have to define the following properties:

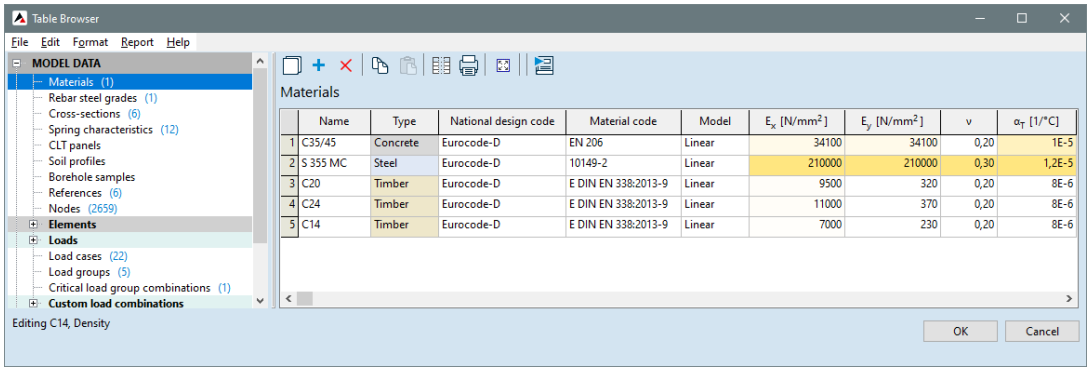
Finite element	*Material*	*Cross-section*	*Reference*	*Stiffness*	*Surface*
Truss	•	•	o		
Beam	•	•	•	o	
Rib	•	•	o		
Membrane	•		•		•
Plate	•		•		•
Shell	•		•		•
Support			•	•	
Rigid					
Spring			o	•	
Gap				•	
Link				•	
Edge hinge				•	

o: optional

Note that some elements like springs and gaps can have nonlinear elastic stiffness properties that are taken into account only in a nonlinear analysis. In a linear analysis the initial stiffness is taken into account for the spring element, and the active or inactive stiffness depending on its initial opening for the gap element.

4.9.1 Materials

Define Materials



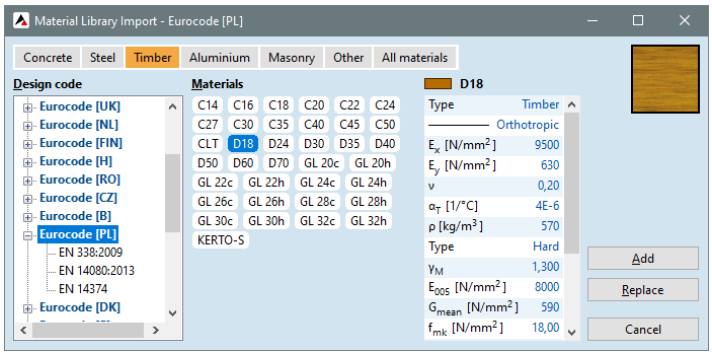
For color coding of table cells and sorting by columns see **2.9 Table Browser.**

Lets you define and save material property sets or load them from a material library.
If you delete a material the definition of the elements with the respective material will be deleted.

Browse Material Library

[Shift+Ctrl+M]

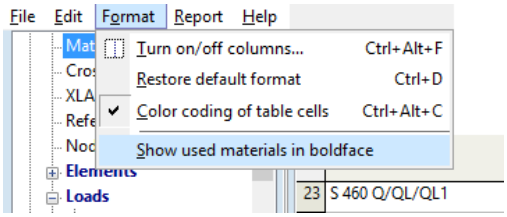
The material library contains material properties of civil engineering materials based on Eurocode, SIA and some other specifications.



Select a material type from the horizontal list at the top to filter the list of available materials.

Click *Add* to add a new material to the table of materials. Click Replace to replace the material of the current table row.

Activating *Format / Show used materials in boldface* helps to avoid deleting a material when it is in use. Listed but unused materials can be easily removed by the command *Edit / Delete unused materials.*



☞ **If a material type is deleted all elements made of this material will be deleted.**

Material Properties

Depending on the type of the finite element you must define the following material properties:

Finite Element	*E*	*ν*	*α*	*ρ*
Truss	•		•	•
Beam	•		•	•
Rib	•		•	•
Membrane	•	•	•	•
Plate	•	•	•	•
Shell	•	•	•	•
Support				
Rigid				
Diaphragm				
Spring				
Gap				
Link				

Displaying and changing material properties is described in **3.1.15 Material Library**

☞ **In AXISVM all the materials are considered to be linear elastic (Hooke's Law) or plastic, and uniform isotropic or orthotropic (for beam, rib, membrane, plate, and shell elements).**
Some elements can have nonlinear elastic material (truss), or stiffness (support, gap, link, spring elements).
Nonlinear material models are taken into account only in a nonlinear analysis.
In a linear analysis the initial stiffness is taken into account for the nonlinear elements.

Only the used materials

When this option is enabled, only materials actually used in the model are listed.

Only materials in the active parts or selection

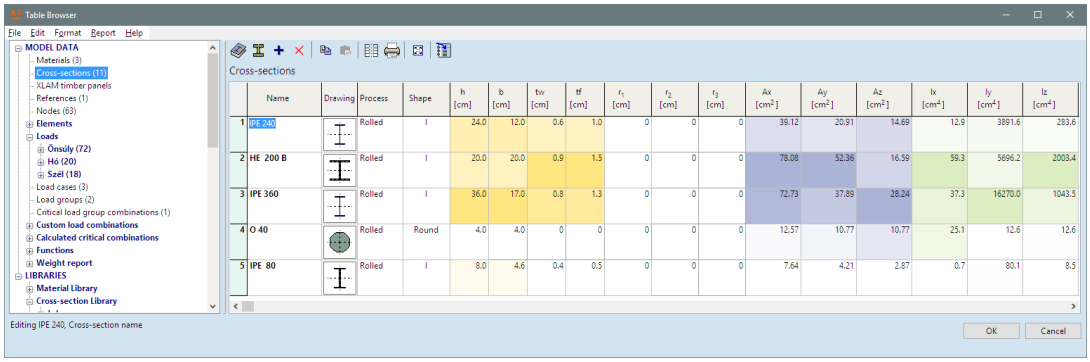
When this option is enabled, only materials for the active details or the selected items are listed.

Show material numbers / Show material names

The switch affects the content of other tables (e.g. Beams): the columns showing the material display either the material number or its name accordingly.

4.9.2 Cross-sections

Define Cross-sections

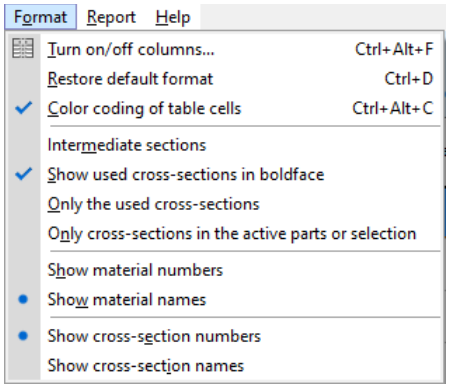


For color coding of table cells and sorting by columns see **2.9 Table Browser.**

Lets you define and save cross-sections or load them from a cross-section library (see **4.9.2.1 Browse cross-section libraries**). The beam, truss, and rib elements require a cross-section. The properties are interpreted in the element's local coordinate system.
For cross-section properties see **3.1.16 Cross-Section Library**

☞ **If you delete a cross-section, the definition of the elements to which it was assigned will also be deleted. The lines will not be deleted.**
You must enter values for all properties.

Cross section properties are defined in the coordinate system of a truss / beam / rib element.



Intermediate sections

If you have beam or rib elements with a variable cross-section, you can use this menu option to show or hide the data for the generated intermediate cross-sections at the end of the cross-section table.

Show used cross-sections in boldface

After invoking the *Edit / Delete unused sections* command these are the sections that will remain in the table.

Only the used cross-sections

When this option is enabled, only cross-sections actually used in the model are listed.

Only cross-sections in the active parts or selection

When this option is enabled, only cross-sections for the active details or the selected items are listed.

Show cross-sections numbers / Show material names

The switch affects the content of other tables (e.g. Beams): the columns showing the cross-section display either the cross-sections number or its name accordingly.

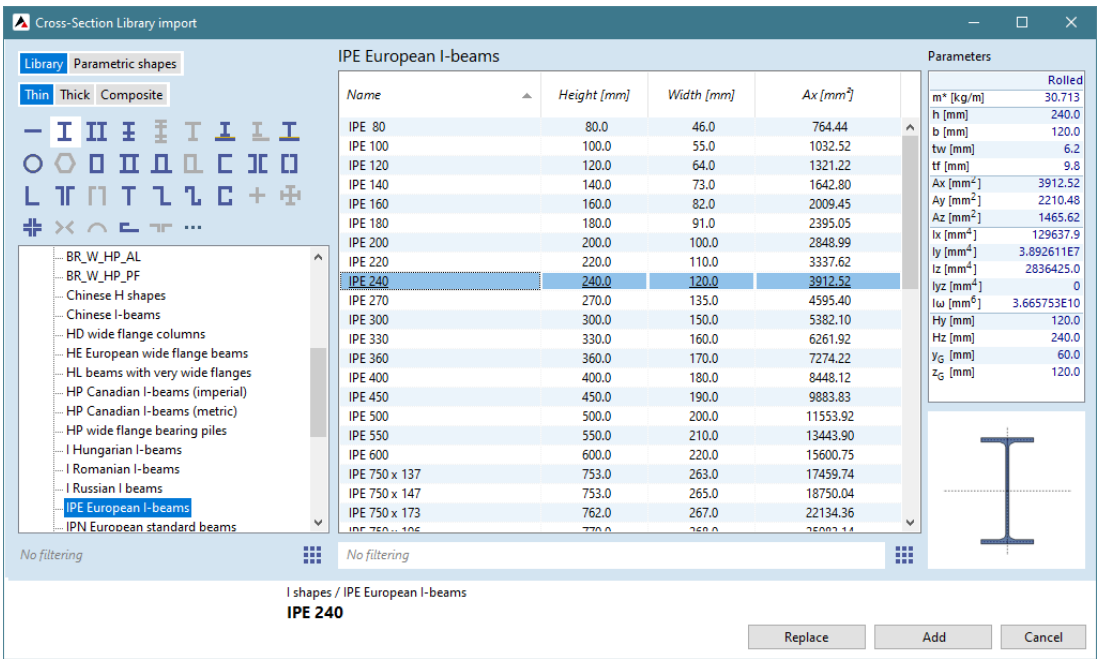
4.9.2.1 Browse cross-section libraries

A cross-section can be imported from the database of the *Library,* or can be defined by entering geometric parameters describing *Parametric shapes*.

Library

Choose thin-walled, thick-walled or composite cross-sections and click on the desired shape on the toolbar. The tree on the left will list the available tables for the selected shape.
The list in the middle shows the cross-sections of the selected table. Clicking on column header sorts the list into ascending/descending order by the respective parameter: *Name, Height, Width, Ax* (cross-section area). One or more cross-sections can be selected. Cross-section parameters and drawing appears on the right.
For cross-section properties see **3.1.16 Cross-Section Library**

☞ **If a button is disabled the library does not contain cross-section tables for the respective shape. If you define a custom cross-section table with that section type, and open this dialog the button will be enabled.**

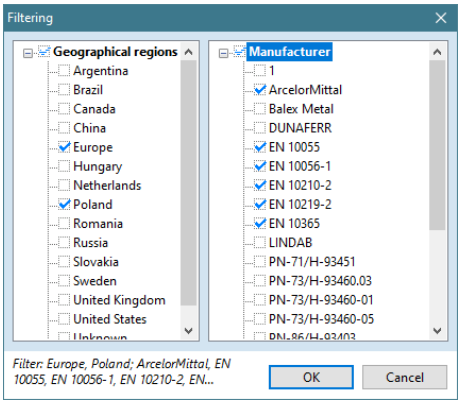


Click on the *Replace* button to replace the the selected row in the table of cross-sections with the item selected here. Click the *Add* button to append selected items to the table of cross-sections. If multiple cross-sections are selected *Replace* will be disabled.

Filters

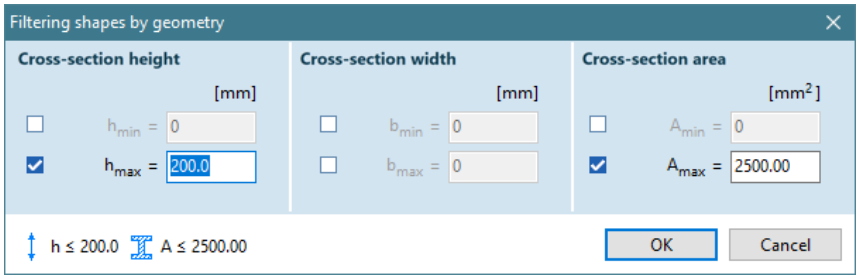


The list of cross-sections with the selected shape can be filtered by geographical regions and manufacturers. Click the *Filter* button under the tree on the left to configure the filter.



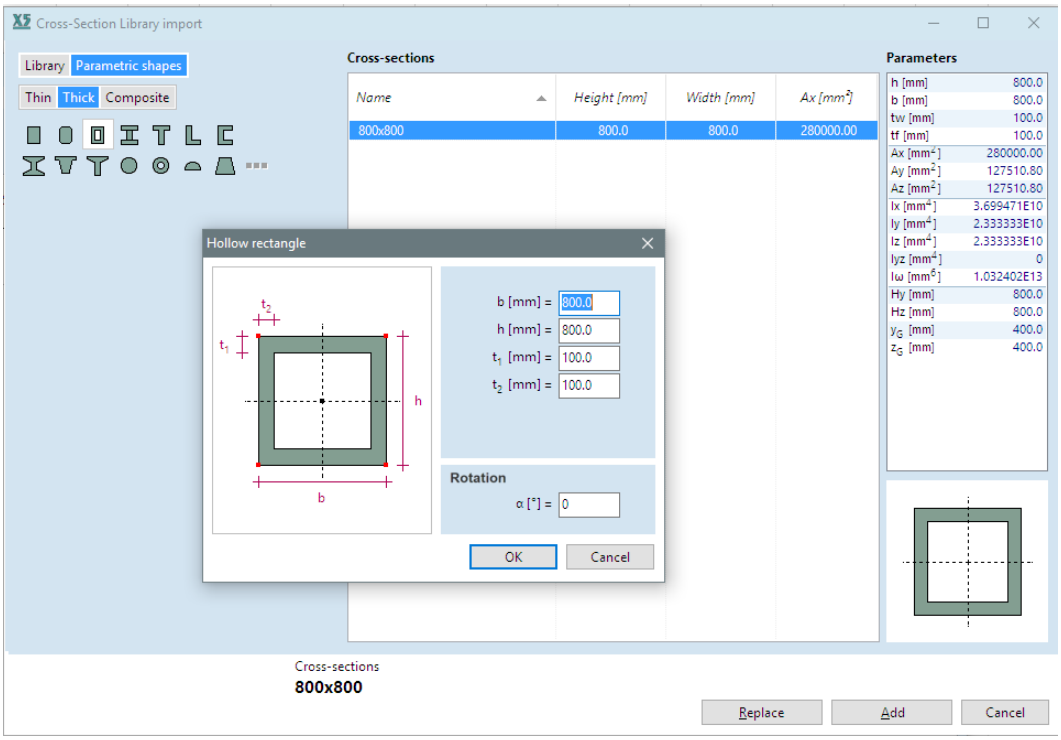
Another *Filter* button is situated under the cross-section list. Click on it to set the desired filtering criteria for minimum / maximum height, width or cross-section area.





Parametric shapes

Choose thin-walled, thick-walled or composite cross-sections and click on the desired shape on the toolbar to open a dialog. Set the parameters describing the geometry and click OK to add the cross-section to the list of parametric cross-sections with a generated name. You can add different parametric shapes to the list.
Double-click on the name to edit it. Press the *Delete* key to remove a selected cross-section(s) from the list.



If there is only one cross-section in the list click on the *Replace* button to replace the the selected row in the table of cross-sections. Click the *Add* button to append all cross-sections in the list to the table of cross-sections. If the list contains more than one cross-section *Replace* will be disabled.

☞ **In case of parametric shapes there is no need to select cross-sections in the list to import them. The entire list is imported.**

4.9.3 Add borehole samples – SOIL module

AXISVM allows you to build a stratigraphic model from data of borehole samples, and a solid element soil model by generating a 3D mesh.

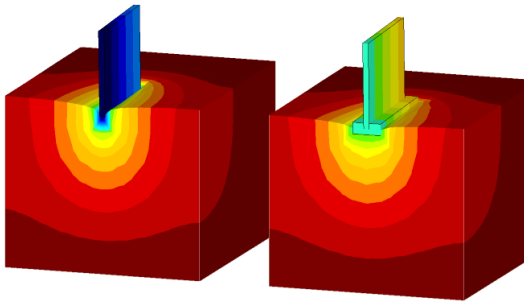
Stratigraphic model

The *stratigraphic model* can be used to interpolate the soil profile under nodal, line or surface supports, to estimate their stiffness (see **4.9.14.1.2, 4.9.15.1.2, 4.9.16.1.1 Estimating support stiffness from soil profile and footing parameters**). Accordingly, for line and surface supports, stiffness may vary from element to element in the finite element mesh.

Soil model

A more accurate model of the soil-structure interaction can be created by constructing a *soil model* from solid elements. This model can also be used to follow the stress distribution and deformation of the soil. In this case, the superstructure is not supported by nodal, line or surface supports, but by the soil model.

Pad and strip footings must be modelled as individual domains considering their real physical dimensions.
To ensure realistic behaviour, structural elements cannot be attached directly to the soil model, as this would result in unrealistically high settlement and singular stress peaks.
At the bottom and on the sides of the soil model, nodes are rigidly supported in the normal direction.
For details on building a soil model, see **4.9.9 Soil modeling domain – SOIL module**.



Borehole samples

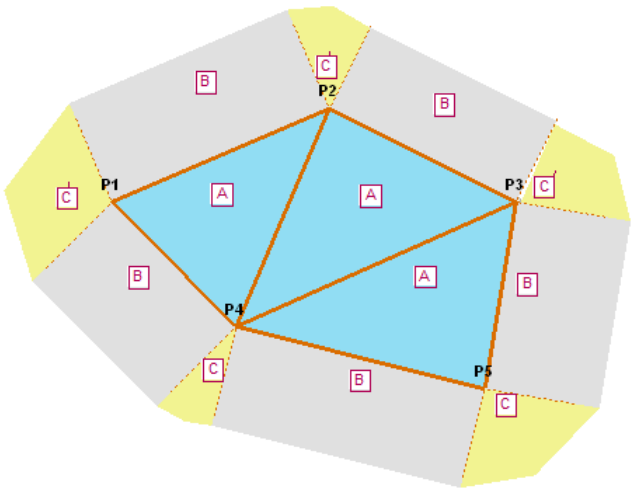
A *borehole sample* is a soil profile determined at a given position in the global model space. The position represents the uppermost (zero) point of the soil profile. Only one borehole sample can be defined at a given position.

Terrain model

Borehole samples defined at different positions are connected by triangulation, called *terrain model*. The program builds a stratigraphic model that allows finding the soil profile at a given position P. If the Z coordinate of point P is above the top of the soil profile, the top layer is extended upwards, if it is lower than the bottom of the soil profile, the bottom layer is extended downwards.
To find the soil profile the terrain model and the point P are projected onto the base plane of the global model space (Z = 0) to determine in which triangle of the terrain model P lies.

Stratigraphic model

Within the triangles (blue A regions) the soil profile under point P is determined by interpolating between the three soil profiles. Outside the convex hull of the projection of the terrain model onto the base plane (grey B regions and yellow C regions) the soil profile is determined by extrapolation. In grey B regions, a linear interpolation is used between the two borehole samples (e. g. P1 and P4). In the yellow C regions, the soil profile of the single borehole sample in the region is used. If borehole samples fall on a single straight line, a linear interpolation is used between the sample points, and an extrapolation is used beyond the start and end points.



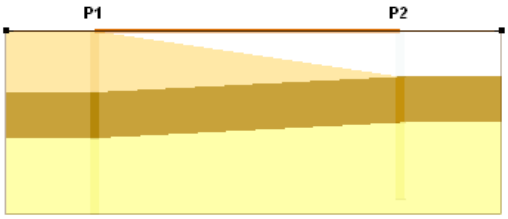
If there is only one borehole sample in the model, the soil profile is determined from that.
The terrain model is also displayed in the model to show regions of interpolation and extrapolation.

Interpolation between soil layers

If the soil layers of two borehole samples have the same properties and are in the same order (i. e. only their thickness is different), the interpolation procedure is unambiguous.
When interpolating between layers with different properties (the top layer of borehole samples P1 and P2 in the figure is different), it is not possible to determine if the top layer of sample P1 or P2 is on the top in intermediate points. In this case, the program simply choses one of these options.



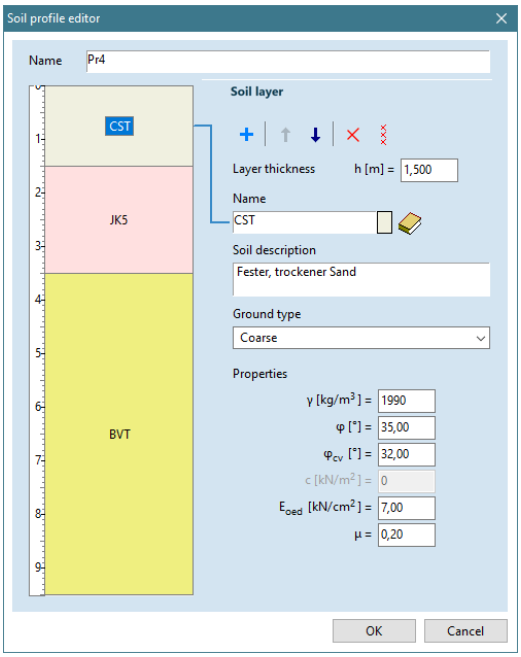
If it is known from other considerations that the top layer of P2 is on top in intermediate points, you can add a 1 mm layer from the top layer of P2 on top of P1, which does not affect the soil behaviour significantly, but ensures that the top layer of P2 will be on top in intermediate points.



Add borehole samples



To add a borehole sample, enter a name for the sample and its location in the global model space (you can also select a point by clicking on the *Pick up (X,Y,Z)* button), and set the soil profile using the toolbar on the right.



Soil profile

There are several ways to set or change a soil profile.



Create a new soil profile
Add layers and set their properties, see **6.5.16.3 Soil profile editor**



Load soil profile
Choose from the list of existing soil profiles in the model.



Soil profile library
Choose a soil profile from the model-independent soil profile library.
See **6.5.16.4 Soil profile library**.



Pick up the interpolated soil profile at the given position
After clicking on this button, an interpolated soil profile is calculated at the specified sample location based on the terrain model of existing borehole samples. It can be used primarily for verification purposes.



Save soil profile to the library
The current soil profile can be saved in the soil profile library, which is accessible from any model. See **6.5.16.4 Soil profile library**.

Delete borehole samples

Borehole samples can be deleted in the Table Browser (*MODEL DATA / Borehole samples*).

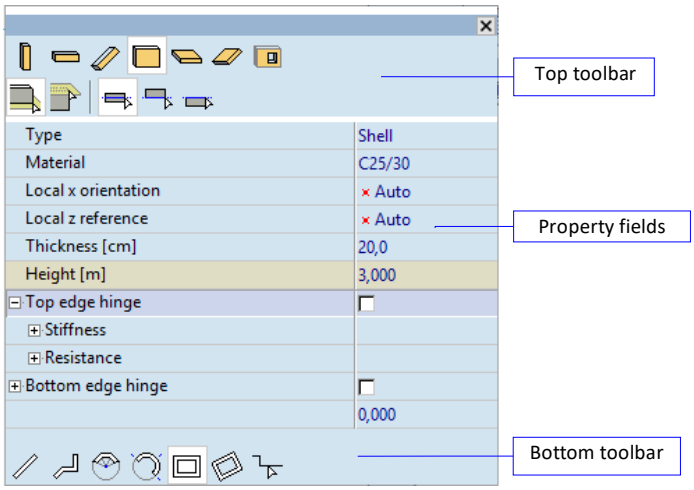
Using the stratigraphic model

The stratigraphic model determined from the terrain model based on borehole samples can be used on its own without creating a soil modelling domain. It can be used to interpolate the soil profile under nodal, line or surface supports, to estimate their stiffness (see **4.9.14.1.2, 4.9.15.1.2, 4.9.16.1.1 Estimating support stiffness from soil profile and footing parameters**). Accordingly, for line and surface supports, stiffness may vary from element to element in the finite element mesh.
When designing a pad footing, it is not necessary to enter the different soil profiles one by one if you enable the automatic interpolation of soil profiles (see **6.5.16.1 Pad footing design**).

For creating a 3D soil model, see sections **4.9.9 Soil modeling domain – SOIL module** and **4.12.1.3 Meshing of soil modeling domains**.

4.9.4 Direct drawing of objects





After clicking the icon, a direct drawing toolbar and property editor appears. With the help of this window columns, beams, walls, slabs and holes can be drawn. Their properties can be set previously and changed any time during the drawing.
The top toolbar shows the type of the object to draw and the orientation of the object (for columns and walls). Property fields can be edited like in the Property Editor.
The bottom toolbar shows the drawing methods available for the object (one segment, polyline, polygon, rectangle, etc.).
Clicking a domain contour before drawing holes forces the drawing into the plane of the domain.

Object types

 Column (in global Z direction)

 Beam (in global X-Y plane)

 Beam (spatial)

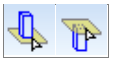
 Wall (always vertical with a constant height, i.e., its normal and upper/lower edges are parallel to the global X-Y plane)

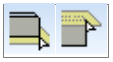
 Slab domain (parallel to the global X-Y plane)

 Surface domain (spatial)

 Hole

Object drag points

 Column upwards / downwards

 Wall upwards / downwards



Wall justification: during drawing walls it can be set whether the line drawn is the centerline or left / right side of the wall (or interior / exterior side in case of arcs and rectangles).

Object geometry



Column



Single segment beam or wall



Beam or wall polyline



Arced beam with center point, start and end point



Arced beam with three points



Polygonal beam or wall



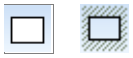
Walls on a rectangle



Walls on a slanted rectangle



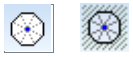
Walls along existing AXISVM lines or background layer lines



Rectangle shaped slab / opening



Slanted rectangle shaped slab / opening



Polygon shaped slab / opening



Round shaped slab / opening

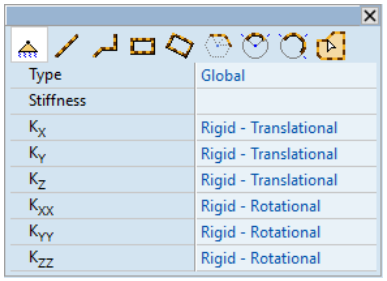


Complex slab / opening

4.9.5 Direct drawing of supports



This function allows direct drawing of nodal and line supports with predefined properties.





Nodal support



Line support, single line



Line support, polyline



Line support, rectangle



Line support, skewed rectangle



Line support, polygon by center point and two points



Line support, arc by center point and two points



Line support, arc by three points



Line support on domain edges

4.9.6 Domain



A domain is a planar structural element with a complex geometric shape described by a closed polygon made of lines and arcs. A domain can contain holes, internal lines and points.
Polygon vertices, holes and internal lines must be in same plane.

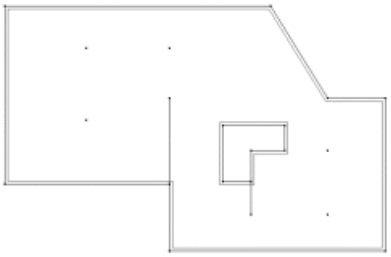
A domain has the following parameters:

Element type (membrane, plate, shell)
Material
Thickness
Eccentricity
Local coordinate system
Custom color for rendered view

The following parameters can be assigned to the polygon, hole edges, internal lines and points of a domain:

point, line, and surface support
rib element
distributed load
dead load
thermal load
nodal degrees of freedom (DOF)

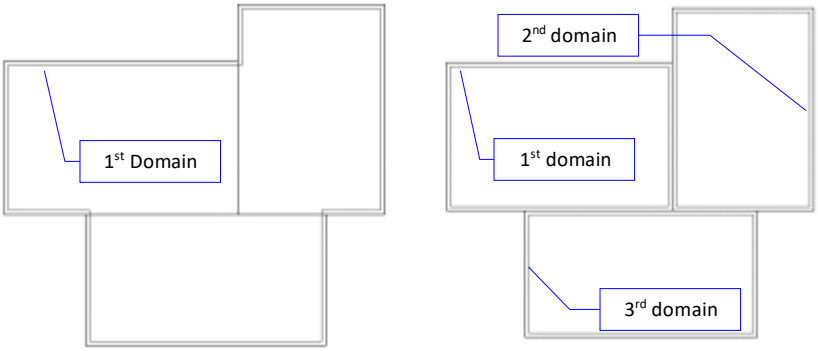
A domain is displayed by a contour line inside of the domain's polygon, with a color corresponding to the domain's element type (blue for membrane, red for plate, and green for shell).



Domains can be defined for floors, walls, and any other complex structural surface element.

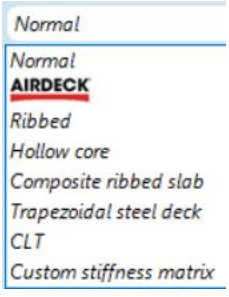
The domain can be meshed automatically. See 4.12.1.2 Meshing of domain

More than one domain can be used to model a structural element.



A domain can contain other (sub-) domains.

Different domain types are available from the *Type* dropdown list at the top



- *Normal domain* (for simple slabs, walls, etc.)
- *AirDeck domain* (for voided slabs, requires ADK module)
- *Ribbed domain* (for parametric ribbed slabs)
- *Hollow core* (for hollow core slabs)
- *Composite ribbed slab* (for composite ribbed slabs)
- *Trapezoidal steel deck* (for steel plate or composite slabs)
- *CLT* (for cross-laminated timber panels)
- *Custom stiffness matrix*

4.9.6.1 Defining a normal domain

Finite element

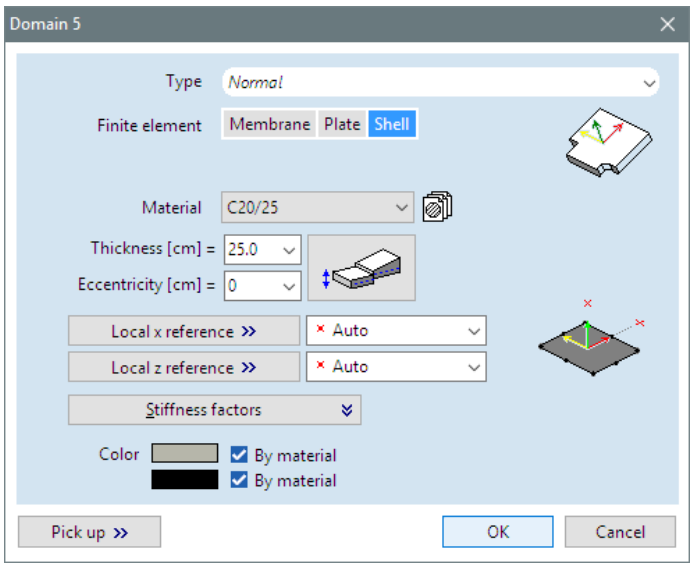
Domains can be modelled with different type of finite elements (membrane, plate, shell). To understand the meaning of these types, see **4.9.13 Surface elements** and **5.5 Finite elements**

.

Select lines on the contour of the domains you want to define. If you select more lines or lines from different planes, AXISVM will find the planes and the contour polygons of the set. The program applies the parameters you entered in a dialog window.

Material

Select a material from the list of materials used in the model or pick one from the material library.



Thickness, Eccentricity

To define a domain with a constant thickness and eccentricity only the two values must be entered.

Tapered domains and/or domains with variable eccentricity can be defined by clicking the button beside the edit fields.



k,torsion

If the material is concrete, it is possible to set reduced torsional strength of RC plate or shell.
k,torsion factor must be between 0.1 and 1, i.e., the torsional strength of the RC plate or shell can be 10%-100% compared to the elastic, isotropic material.

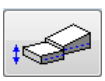
k,shear

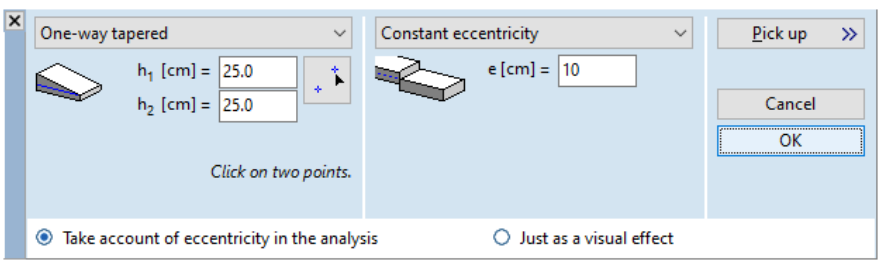
If the material is masonry, it is possible to set reduced shear strength.
k,shear factor must be between 10^{-6} and 1,0, i.e., the shear strength of the masonry membrane or shell can be 10^{-4}%-100% compared to the elastic, isotropic material.
If the material is concrete, it is possible to set reduced shear strength of RC membrane or shell.
k,shear factor must be between 10^{-6} and 1,0, i.e., the shear strength of the RC membrane or shell can be 10^{-4}%-100% compared to the elastic, isotropic material.

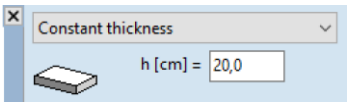
k,bending

If the material is concrete, it is possible to set reduced bending strength of RC plate or shell.
k,bending factor must be between 0.1 and 1, i.e., the bending strength of the RC plate or shell can be 10%-100% compared to the elastic, isotropic material.

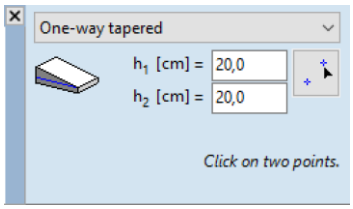
☞ In case of concrete surfaces with reinforcement, nonlinear analysis does not take into account *k,shear* and *k,torsion* factors if Use reinforcement in calculation is checked. (See **6.5.6 Nonlinear analysis of RC surfaces**)



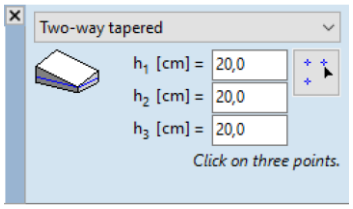




In case of *Constant thickness*, the domain thickness *h* must be entered.



For *One-way tapered* domains enter h_1, h_2 and click on the button then click on two thickness reference points. Domain thickness will change linearly between the reference points, being h_1 at the first point and h_2 at the second one.



For *Two-way tapered* domains enter h_1, h_2, h_3 and click on the button then click on three thickness reference points. Domain thickness will change linearly between reference points, being h_1 at the first point h_2, at the second one, h_3 at the third one. The three reference points cannot be on the same line

☞ **If domain thickness would be reduced to zero on certain points a warning will appear.**

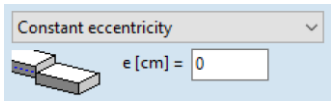
☞ **If thickness gradient is over 5% surface reinforcement calculation becomes unavailable.**

☞ **The reference plane of the static model is the middle plane of the domain. Eccentricity is relative to this plane.**

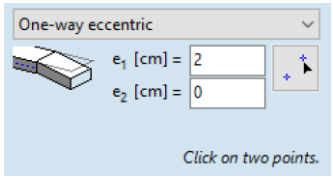
Eccentricity

Setting eccentricity is optional. Choosing *Constant eccentricity* from the list or setting $e = 0$, the midplane of the domain will be the same as the plane of the statical framework. If you select the option *Take account of eccentricity in the analysis*, the program calculates with eccentrically connected domains. If you select the option *Just as a visual effect,* the program ignores the specified eccentricity in the analysis.

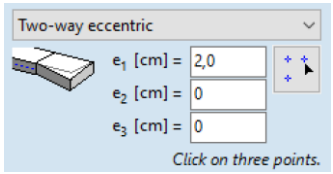
Other options are



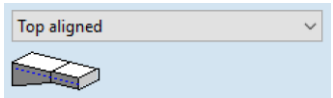
Constant eccentricity: Midplane of the domain gets an offset of e in the local z direction.

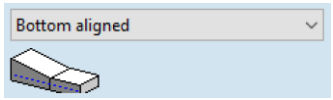


One-way eccentric: Enter e_1, e_2, click on the button then click on two eccentricity reference points on the model. The eccentricity of the midplane will change linearly between reference points, being e_1 at the first point and e_2 at the second one.



Two-way eccentric: Enter e_1, e_2, e_3, click on the button then click on three reference points on the model. The eccentricity of the midplane will change linearly between reference points, being e_1 at the first point, e_2 at the second one and e_3 at the third one.





Top / Bottom aligned eccentricity is useful when working with tapered domains or many domains connecting with different thickness. In these modes, eccentricity is calculated automatically. Tapered domains will have zero eccentricity at their thinnest point - unless they are within an eccentricity group (see below).
Top aligned domains have their upper plane parallel to the local *x-y* plane. *Bottom aligned* domains have their lower plane parallel to the local *x-y* plane

If multiple domains are selected and the *Eccentricity group* option is activated domain eccentricities will be set to make the upper or lower plane of the domains align. Changing the thickness of any domain in the group will update the eccentricity of other domains within the group to keep the planes aligned.
The *Pick up* button is to pick up thickness and eccentricity from another domain. As reference points are also picked up the thickness change will follow them plane defined by the domain.
Eccentricities and eccentricity groups can serve as a base for color coding (see **2.19.3 Color coding**) and eccentricity groups can be displayed as parts (see **2.16.17 Parts**).

☞ **In the case of eccentricity adjustment, during the calculation of the stiffness matrix, the eccentric elements are connected to the statical frame by the means of rigid elements. Additional forces from the eccentricity influences the analysis results, which effect should be taken into account in the course of the evaluation of the results.**

Local x and z reference

The local *x* and *z* directions of the domain can be set by choosing reference elements, see part **4.9.25 References** or can be set to *Auto*. The local *y* direction comes from the right hand rule.

Color

Domains can have their own fill and outline color used in rendered display mode. The default values are taken from the material colors. If a color coding is applied the domain color is determined by the color coding both in wireframe and rendered modes.
See **2.16.5 Color coding**

Modify a domain

Select the domain (click on the contour line of the domain) you want to modify and make the changes in the dialog displayed.

Delete a domain

Press the **[Del]** button, select the domains (click on the contour line of the domain) you want to delete and click OK in the dialog.

4.9.6.2 Composite ribbed domain

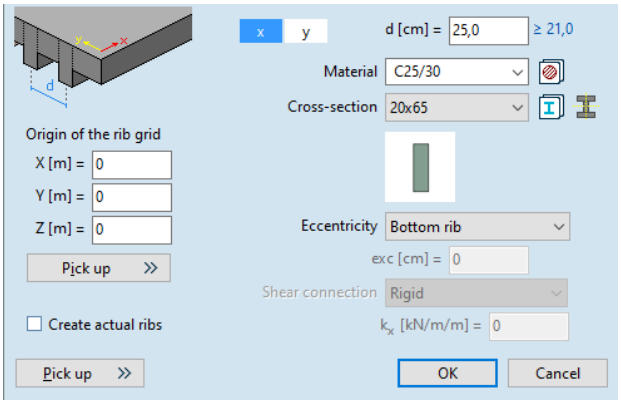
Defining a composite ribbed domain is an easier way to place a system of ribs over a domain. Unlike the parametric ribbed slabs (1) the ribs can run in local *x* or *y* direction but not both, (2) their material can be different from that of the domain and (3) they can have any cross-section taken from the cross-section library or created in the cross-section editor.
Beyond basic domain parameters (see **4.9.6.1 Defining a normal domain**) the following parameters can be specified:

☞ **Composite ribbed domains require plate or shell element type.**

Origin of the rib grid

Ribs can run in the local *x* or *y* direction. The origin of the rib grid can be entered numerically or picked up from the model.

The spacing of ribs (*d*) cannot be smaller than the width of the cross-section.
Eccentricity of the ribs can be set, shear connection can be set to *Elastic* or *Rigid* with a given k_x stiffness.



Create actual ribs

In case of steel ribs cast partially or entirely in concrete *Create actual ribs* must be left unchecked. If upper or lower ribs are attached to the surface of the slab or the spacing is high compared to the dimensions of the domain it is recommended to activate this option.

☞ **The reference plane of the static model is the middle plane of the concrete plate.**

Application of logical ribs

If *Create actual ribs* is left unchecked, composite ribs will be represented as logical entities, no finite elements will be created. Ribs will be taken into account when creating the material stiffness matrix of the plate. From the 3D geometry we calculate the effective orthotropic elastic constants of the material, suitable to represent the stiffness of a periodic body with a 2D model. This is carried out according the theory of asymptotic homogenization. The effectiveness of such a method is mainly influenced by the ratio of the length of the period and the dominant size of the domain being modelled (in this case the ratio of parameter *d* and the span of the domain along the rib-perpendicular direction).

☞ **As a general guideline we can say that the spacing of the ribs be way smaller than the dominant in-plane size of the problem domain.**

The material stiffness matrix of a general shell element:

$$\begin{Bmatrix}\{N\}\\ \{M\}\end{Bmatrix} = \begin{bmatrix}\mathcal{A} & \mathcal{B}\\ \mathcal{B}^{\mathrm{T}} & \mathcal{D}\end{bmatrix}\begin{Bmatrix}\{\epsilon_0\}\\ \{\kappa\}\end{Bmatrix},$$

$$\begin{Bmatrix}Q_x\\ Q_y\end{Bmatrix} = K_s\begin{bmatrix}A_{44} & A_{45}\\ A_{45} & A_{55}\end{bmatrix}\begin{Bmatrix}\gamma_x\\ \gamma_y\end{Bmatrix}$$

where matrices

$$\mathcal{A} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{12} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix};\ \mathcal{B} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{12} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix};\ \mathcal{D} = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{12} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix}$$

can be originated from the 6x6 stiffness matrix of general Hooke's law. Matrix $\mathcal{B}$ represents the coupling effect between bending and membrane or in other words the eccentric effects. This matrix can be zero if and only if the ribs both have zero eccentricity. In every other case the structure must be modelled using shell elements to avoid a serious loss of information. The entries of the matrices are determined by applying independent strain states on a 3D model, and retrofitting the obtained stress field.
Since a shell has 8 generalized strain components, we need to perform 8 finite element calculations in the background.

A shell equipped with the so calculated stiffness properties is equivalent to the original problem in a way that the work needed to deform the body to a certain level remains the same, but it doesn't entail the equivalence of all displacement components at all points.

☞ **Logical ribs are most effective when the spacing of the ribs is dense or the plate is mostly loaded by distributed loads. This is due to the local effects of concentrated forces.**

4.9.6.3 Hollow core domain

Beyond basic domain parameters the following parameters can be specified:

☞ **Hollow core domains require concrete material and plate or shell element type.**

Void type



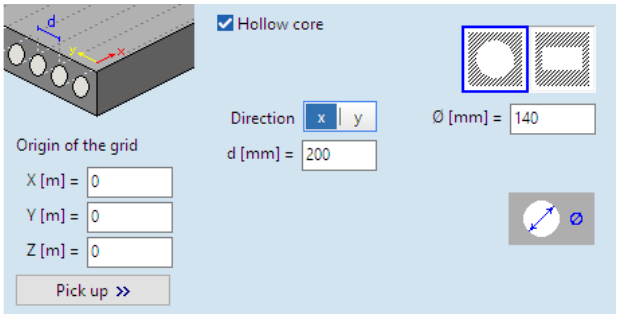
Tubular



Rectangular

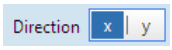
Origin of the grid

Voids are running in local x or y direction along a grid.
The origin of this grid can be specified numerically or picked up from the model.



Voids in x / y direction

The direction of voids can be chosen here.



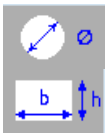
☞ **The reference plane of the static model is the middle plane of the concrete plate.**

Tubular voids

Parameters: *Ø* is the diameter of the void, *d* is the distance between the centers of the voids.

Rectangular voids

Parameters: *b* is the width of the void, *h* is the height of the void, *d* is the distance between the centers of the voids.



Calculation

The calculation method is the same what can be seen at **Composite ribbed domain**, since present configurations can be considered as special cases of that. Consequently, all the properties of the method written there applies for this domain type too.

☞ **As a general guideline we can say that the spacing of the voids be way smaller than the dominant in-plane sizes of the problem domain.**

In the context of elastic anisotropy, the material stiffness (ABD matrix) of a general flat shell element can be described with the following system of equations

$$\begin{Bmatrix} \{N\} \\ \{M\} \end{Bmatrix} = \begin{bmatrix} \mathcal{A} & \mathcal{B} \\ \mathcal{B}^{\mathrm{T}} & \mathcal{D} \end{bmatrix} \begin{Bmatrix} \{\epsilon_0\} \\ \{\kappa\} \end{Bmatrix};\ \begin{Bmatrix} Q_y \\ Q_x \end{Bmatrix} = K_s \begin{bmatrix} A_{44} & A_{45} \\ A_{45} & A_{55} \end{bmatrix} \begin{Bmatrix} \gamma_y \\ \gamma_x \end{Bmatrix}$$

where

$$\mathcal{A} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{12} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix};\ \mathcal{B} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{12} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix};\ \mathcal{D} = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{12} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix}$$

matrices can be derived from the 6x6 stiffness matrix of the generalized Hooke's law. $\mathcal{B}$ represents the material relation between normal forces and bending, which has only zero values int the present case. Nonzero stiffness values are determined by the calculation of average stresses, due to pure strain states applied on the representative volume element of the material.

4.9.6.4 Parametric ribbed plates

Beyond basic domain parameters the following parameters can be specified:

Rib type



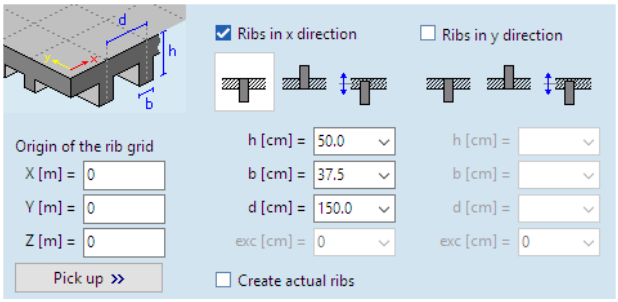
Bottom rib



Top rib



Custom eccentricity



Origin of the rib grid

Ribs follow a grid of lines in the local *x* and *y* directions. The origin of this grid can be entered numerically or picked up from the model. The ribs may be modelled with logical or actual ribs.

Ribs in x / y direction

Ribs running in x or y direction has the following geometric parameters *h* is rib height, *b* is rib width, *d* is distance between ribs, *exc* is the eccentricity (if custom eccentricity is selected).

☞ **The maximum value of the rib eccentricity = (plate thickness + h) / 2**

☞ **The reference plane of the static model is the middle plane of the concrete plate.**

Calculation

The calculation method is the same what can be seen at **Composite ribbed domain,** since present configurations can be considered as special cases of that. Consequently, all the properties of the method written there applies for this domain type too.

☞ **As a general guideline we can say that the spacing of the ribs be way smaller than the dominant in-plane size of the problem domain.**

Another consequence of the homogenization is that stiffness peaks at the rib intersections will be smoothed, which may lead to insufficient design. The limits of application regarding a ribbed plate as a homogeneous one is collected in Eurocode 2, chapter 5.3.1. The program automatically detects is these suggestions are not met.

☞ **In case of violating geometric restriction, the user is informed with a message window.**

Create actual ribs

The homogenization method is not applicable in nonlinear analysis. To get proper nonlinear results actual ribs must be created. This option allows automatic generation of rib elements according to the parameters.

4.9.6.5 Trapezoidal steel deck

For the definition of basic parameters see **4.9.6.1 Defining a normal domain.**

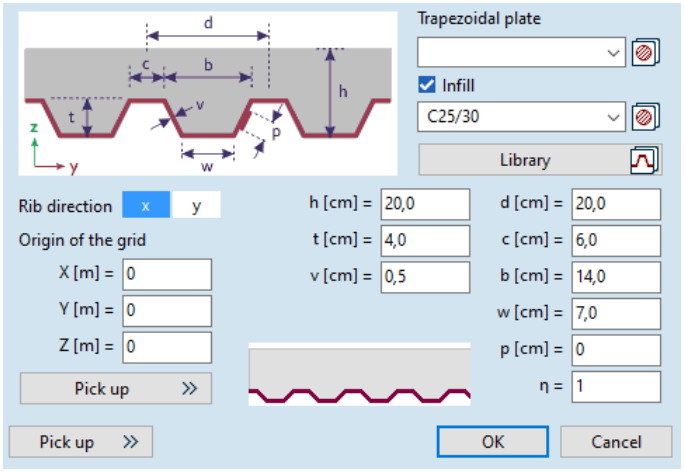
☞ **The trapezoidal plate must be steel, the infill must be concrete.**

Infill

If this field is unchecked the domain is just a trapezoidal steel plate. If checked it is a composite plate where a steel deck is filled with concrete.

Origin of the grid

Ribs of the trapezoidal plate can run in the local *x* or *y* direction of the domain depending on the status of the *Rib direction* switch. Enter X, Y, Z global coordinates to define the origin of the rib system or click the *Pick up* button to define it by clicking.
The direction of the ribs is marked on the domain, if the display of the domain local systems is switched on (the gridlines of parametric domains must be checked among the display options).

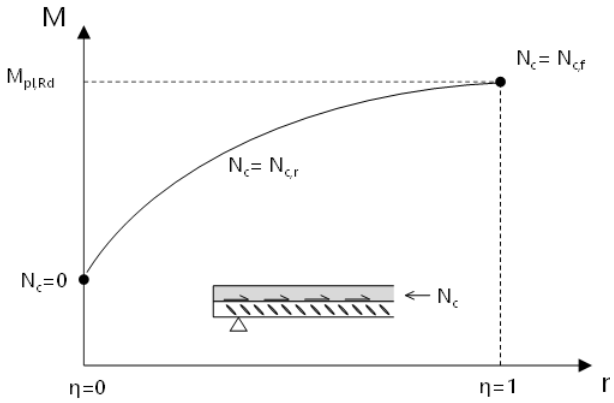


Parameters The figure above shows the main geometrical parameters of the composite slab. Rolled mechanical embossments and end anchorages typically ensure the shear connection between the trapezoidal sheet and concrete. The parameters η and p are the degree/effectiveness of shear connection and the width of rolled mechanical embossments, respectively. The software ignores the rolled mechanical embossments in the course of calculation of the bending resistance for trapezoidal sheet.

η expresses the proportion of the full compression resistance ($N_{c,f}$) of the upper concrete layer transferred to the trapezoidal sheet. If η is lower than 1.0, the bending resistance of the composite slab is lower than in the case of full shear connection.

$$N_{c,f} = 0.85 f_{cd}(h - t),$$

where f_{cd} is the compressive strength of the concrete.



If only rolled embossments are applied for shear connection, the degree of shear connection is 0 at the supports and it increases with the distance from the supports. In such case, it is recommended to split the domain parallel to the supports and to define different discrete η values for subdomains. The exact values can be calculated from the strength of shear connection ($\tau_{u,Rd}$) that depends significantly on the geometry of embossments thus, it may vary for different trapezoidal sheets. If $\tau_{u,Rd}$ is known, the degree of shear connection can be calculated in the following way:

$$\eta = \frac{\tau_{u,Rd} L}{N_{c,f}} \leq 1,$$

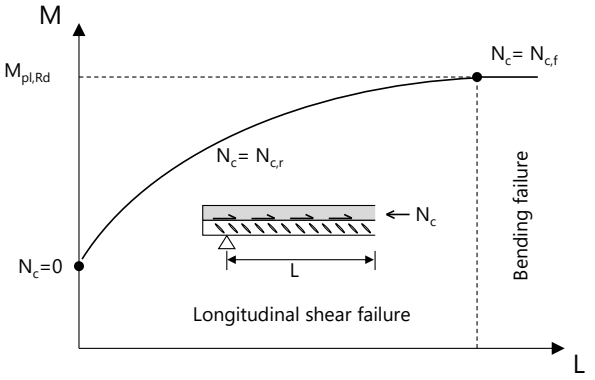
where L is the distance from the supports.

If only end anchorages are applied, the degree of shear connection can be calculated in the following way for the entire domain:

$$\eta = \frac{V_{ld}}{N_{c,f}} \leq 1,$$

where V_{ld} is the resistance of end anchorages.

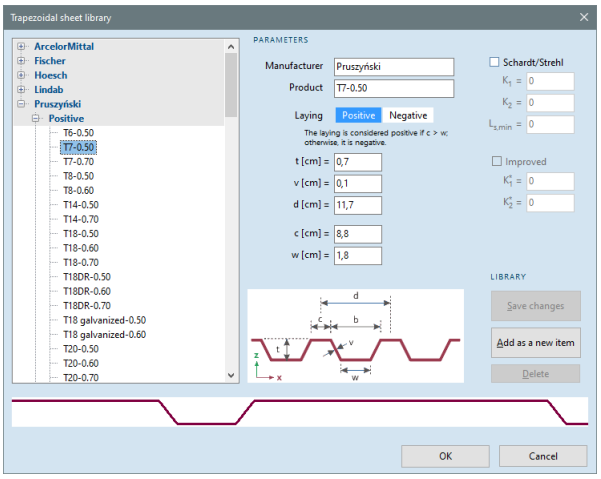
If both end anchorages and rolled embossments are applied, the degree of shear connection can be calculated as the sum of η values shown above.



☞ **If no infill is applied the reference plane of the static model is the middle plane of the steel plate (t/2) otherwise the middle plane of the total composite slab (h/2).**

Library

The *Trapezoidal sheet library* offers products from various manufacturers and it also allows saving custom geometries and building custom libraries.
The orientation can also be specified. The orientation is positive if c > w, i.e. the ridges are wider than the valleys; otherwise, it is negative.
See also **4.9.18.1.1 Trapezoidal sheet library**



Analysis

The calculation method is the same what can be seen at **Composite ribbed domain**, since present configurations can be considered as special cases of that. Consequently, all the properties of the method written there applies for this domain type too.

4.9.6.6 Domain with a custom stiffness matrix

Entering a custom stiffness matrix provides full control over the relations between internal force components and strains. This solution is suggested in every case when other modelling solutions are not possible and the user is capable of generating the necessary values, or is in posession of suggested values from any source. The common term for this matrix is ABD matrix. ABD matrices must be positive definite, this criterion is checked by the program.
Indices of matrix elements follow the Voigt notation with the slight modification of interchanging the incices of τ_{xz} and τ_{yz}. Therefore

$$\sigma_1 = \sigma_{11},\ \sigma_2 = \sigma_{22},\ \sigma_3 = \sigma_{33},\ \sigma_4 = \sigma_{23},\ \sigma_5 = \sigma_{13},\ \sigma_6 = \sigma_{12}$$

and

$$\varepsilon_1 = \varepsilon_{11},\ \varepsilon_2 = \varepsilon_{22},\ \varepsilon_3 = \varepsilon_{33},\ \varepsilon_4 = \varepsilon_{23},\ \varepsilon_5 = \varepsilon_{13},\ \varepsilon_6 = \varepsilon_{12}.$$

Hence the mapping from strains to stresses is given with the general form of Hooke's law for anysotropic materials:

$$\begin{Bmatrix}\sigma_1\\ \sigma_2\\ \sigma_3\\ \sigma_4\\ \sigma_5\\ \sigma_6\end{Bmatrix} = \begin{bmatrix}C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16}\\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26}\\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36}\\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46}\\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56}\\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66}\end{bmatrix} \begin{Bmatrix}\varepsilon_1\\ \varepsilon_2\\ \varepsilon_3\\ \varepsilon_4\\ \varepsilon_5\\ \varepsilon_6\end{Bmatrix}$$

where a pair of indices refer to the element of the material stiffness matrix which is related to the given stiffness component.

The meaning of blocks can be described as follows

A_{ij} – membrane stiffness (normal forces – relative strain)

D_{ij} – plate stiffness (moments – bending)

S_{ij} – adjusted shear stiffness (shear forces – relative shear strain)

B_{ij} – connecting stiffness (normal forces – bending or moments – relative strain)

$$\begin{bmatrix} n_x \\ n_y \\ n_{xy} \\ m_x \\ m_y \\ m_{xy} \\ v_{xz} \\ v_{yz} \end{bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} & B_{11} & B_{12} & B_{16} & & \\ A_{12} & A_{22} & A_{26} & B_{12} & B_{22} & B_{26} & 0 & \\ A_{16} & A_{26} & A_{66} & B_{16} & B_{26} & B_{66} & & \\ B_{11} & B_{12} & B_{16} & D_{11} & D_{12} & D_{16} & & \\ B_{12} & B_{22} & B_{26} & D_{12} & D_{22} & D_{26} & 0 & \\ B_{16} & B_{26} & B_{66} & D_{16} & D_{26} & D_{66} & & \\ & 0 & & & 0 & & S_{44} \; S_{45} & \\ & & & & & & S_{45} \; S_{55} & \end{bmatrix} \cdot \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{bmatrix}$$

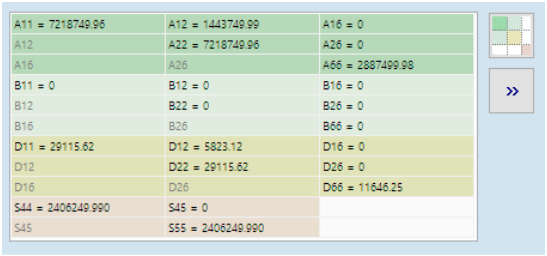
The basic properties of the domain can be entered as for normal domains. Those parameters will affect only the visual representation as the physical properties are controlled by the stiffness matrix.

☞ **Corrected shear stiffness components must include the shear correction factors as described in the chapter about CLT domains. In a general case these may be other than 5/6, used for homogeneous sections and may be different in local directions. Determining shear corretion factors is the responsibility of the user.**

☞ **The meaning of the different stiffness parameters is explained in a dedicated theory guide.**

Stiffness matrix

At the bottom of the dialog stiffness values assigned to the A, B, D, S submatrix elements are displayed.





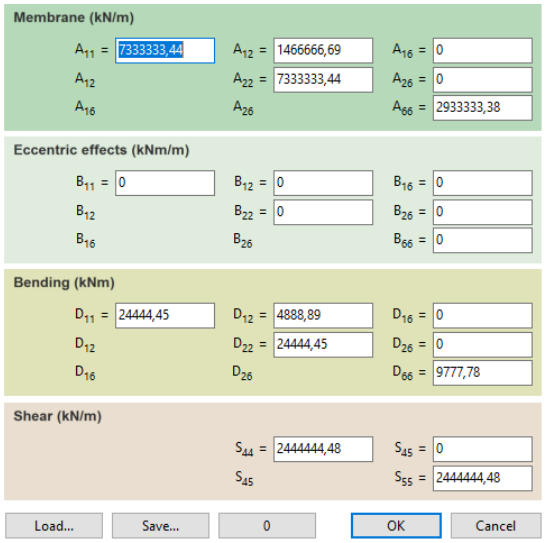
Edit stiffness matrix

Click on this button to edit matrix elements.
The matrix can be saved to a *smx* file by clicking on *Save...*, and can be loaded from a *smx* file by clicking on *Load...*.
This file is a simple text file with the stiffness values. So, it is also possible to create *smx* files manually or by other programs.
Click on *0* to set all stiffness components to zero.



Pick up stiffness matrix

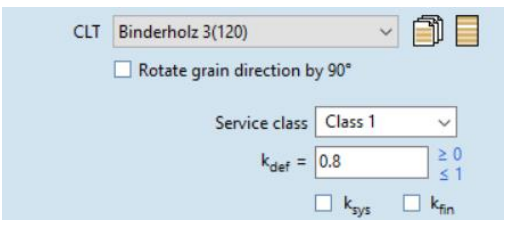
Click on this button to pick up stiffness matrix of other domains (even if they do not have custom stiffness).



For further information see *Help / Custom Stiffness Matrix Guide*.

4.9.6.7 CLT domain

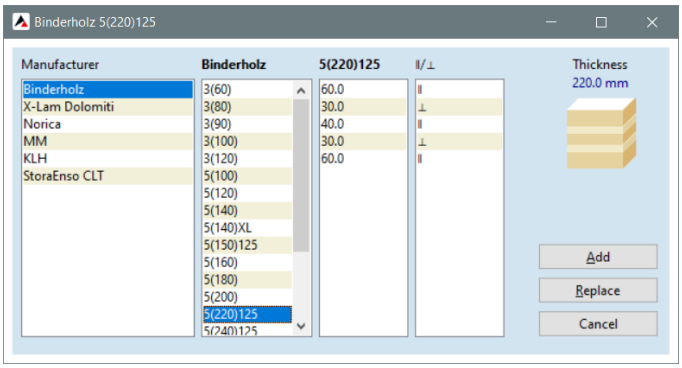
This domain type requires the CLT module handling CLT (cross-laminated timber) panels. AXISVM offers a library of many common products but custom layer structure can also be entered.



Analysis provides displacements, forces, stresses and utilizations in CLT domains.

Definition

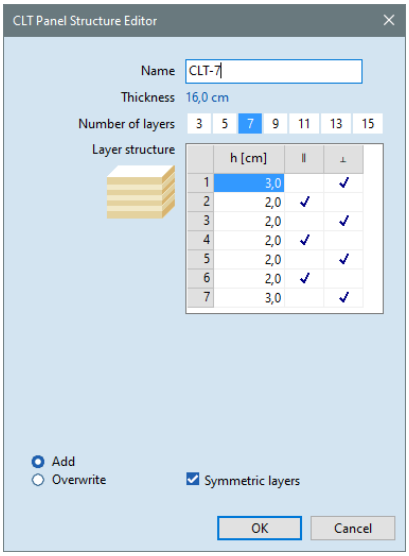
Browse CLT libraries



AXISVM has a wide variety, built-in databases from the CLT manufacturers, which is constantly expanding.

CLT layer structure can be loaded from libraries.

Panel Structure Editor



This editor allows to define custom layer structure.

The *Name* field contains the name of the panel structure.

Thickness is the calculated total thickness of the panel.

The *Number of layers* is always odd, the layer structure can be symmetrical or asymmetrical. Besides the thicknesses of the individual layers, the user can change the orientation as well.

Add: adds the CLT layer structure as a new item to the model.

Overwrite: overwrites the current item with the new layer structure (available only if all selected domains use the same item).

Service Class

This is a classification based on the moisture content of the material and the relative humidity. For details see *Service class* in **6.7.1 Timber beam design – TD1 module**

Rotate grain direction by 90°

If checked, the grain direction is rotated by 90°. By default, the parallel grain of the CLT panel is oriented in the local x direction. The direction of the top layer is marked on the domain if the display of the domain local systems is switched on (the gridlines of parametric domains must be checked among the display options).

Top layer grain direction

Grain direction of the topmost layer must be specified it can be the local *x* or *y* direction.

k_{sys}

When checked, the system strength factor is taken into consideration when calculating the strength properties of the member. For further information see the "CLT Theory and Design Guide".

k_{fin}

When checked, the flexural strength is taken into account with reduced value. Check it on if the lamellas are finger-jointed. For further information see the "CLT Theory and Design Guide".

Calculation

Layered structures built from homogenous layers an equivalent orthotropic material stiffness matrix can be calculated. This method converts the geometric inhomogeneity into material orthotropy
Material stiffness of a general shell element can be described with the following system of equations

$$\begin{Bmatrix}\{N\}\\ \{M\}\end{Bmatrix} = \begin{bmatrix}\mathcal{A} & \mathcal{B}\\ \mathcal{B}^{\mathrm{T}} & \mathcal{D}\end{bmatrix}\begin{Bmatrix}\{\epsilon_0\}\\ \{\kappa\}\end{Bmatrix};\quad \begin{Bmatrix}Q_y\\ Q_x\end{Bmatrix} = K_s\begin{bmatrix}A_{44} & A_{45}\\ A_{45} & A_{55}\end{bmatrix}\begin{Bmatrix}\gamma_y\\ \gamma_x\end{Bmatrix}$$

where

$$\mathcal{A} = \begin{bmatrix}A_{11} & A_{12} & A_{16}\\ A_{12} & A_{22} & A_{26}\\ A_{16} & A_{26} & A_{66}\end{bmatrix};\ \mathcal{B} = \begin{bmatrix}B_{11} & B_{12} & B_{16}\\ B_{12} & B_{22} & B_{26}\\ B_{16} & B_{26} & B_{66}\end{bmatrix};\ \mathcal{D} = \begin{bmatrix}D_{11} & D_{12} & D_{16}\\ D_{12} & D_{22} & D_{26}\\ D_{16} & D_{26} & D_{66}\end{bmatrix}$$

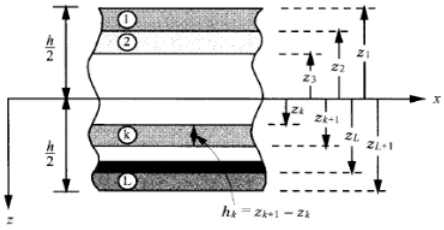
matrices can be derived from the 6x6 stiffness matrix of the Hooke model for orthotropic materials. $\mathcal{B}$ represents the material relation between normal forces and bending. K_s denotes the shear correction factor, which is uniquely determined for each lamination scheme. AXISVM handles symmetric and asymmetric cases as well, the layer orientations must be aligned with one of the local coordinate directions of the domain.

The above matrices are calculated as

$$A_{ij} = \sum_{k=1}^{L} (\bar{Q}_{ij})_{(k)} (z_{k+1} - z_k),$$

$$B_{ij} = \frac{1}{2} \sum_{k=1}^{L} (\bar{Q}_{ij})_{(k)} ({z_{k+1}}^2 - {z_k}^2) = 0,$$

$$D_{ij} = \frac{1}{3} \sum_{k=1}^{L} (\bar{Q}_{ij})_{(k)} ({z_{k+1}}^3 - {z_k}^3).$$



Lamination pattern. (Reddy, J. *Mechanics of Laminated Composite Plates and Shells.* CRC Press, 2004.)

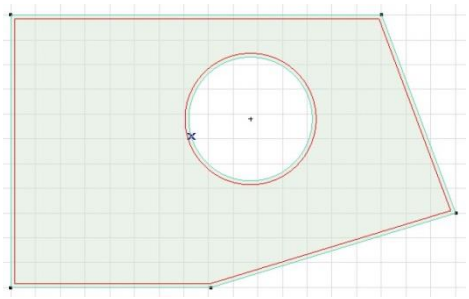
where $\bar{Q}_{ij}$ is the Hooke-modell value transformed into the panel coordinate-system.
For further details on the stiffness calculation procedure, check **Help / Custom Stiffness Matrix Guide.**

4.9.7 Hole



Holes can be defined in domains. Holes have to be inside the domain and in the domain's plane.

Select the (closed) polygons that are the edges of the holes you want to define. More than one outline can be selected. If an outline is not in the plane of the domain no hole will be created. You can move holes from one domain to another, or change their shape. If the hole outline intersects the domain outline, the hole will be removed.
Holes are displayed by a contour line with the color of the domain in which they are located.



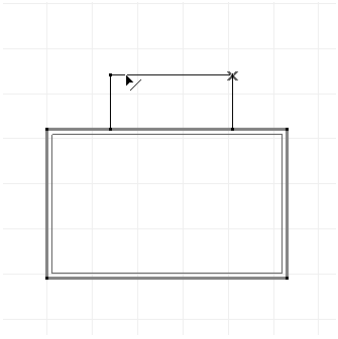
4.9.8 Domain operations

Domain contours can be changed, cut and a union of domains can be calculated.

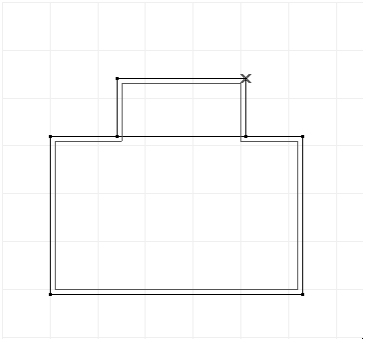
Change domain contour



1. Click the *Change domain contour* icon on the toolbar.
2. Select a domain to change. Domain countour will be selected.
3. Change selection to modify domain contour and click OK on the selection toolbar.



Before



After

☞ **Domain properties (material, thickness, local system) will be retained but the existing mesh will be removed.**

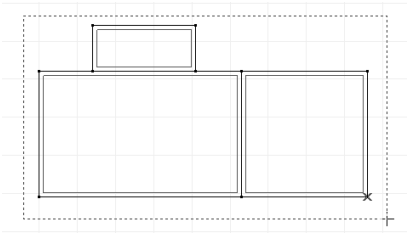
If loaded areas are removed from the domain, loads will automatically be removed.

Union of domains

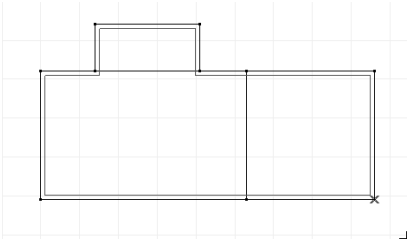


Union can be created from adjacent domains.

1. Click the *Union of domains* icon on the toolbar.
2. Select the domains and click OK on the selection toolbar.
3. If domains have different properties (thickness, material or local system) one of the domains has to be clicked. The union will inherit properties from the clicked domain.



Before



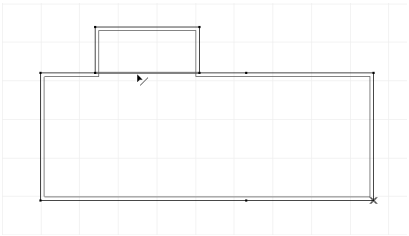
After

Cut domains

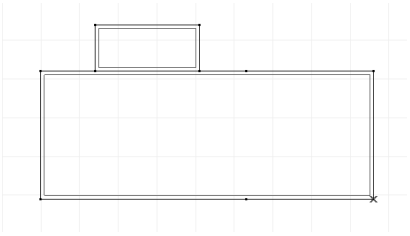


To cut domains along en existing line:

1. Click the *Cut domains* icon on the toolbar.
2. Select the domains.
3. Select the cutting line and click OK on the selection toolbar.



Before



After

4.9.9 Soil modeling domain – SOIL module



After entering borehole samples (see **4.9.3 Add borehole samples – SOIL module**), a 3D soil model can be built up with solid elements.
Using a 3D soil model makes it possible to get results for the stress distribution and deformation of the soil and build a more accurate model of the soil-structure interaction.

Soil model

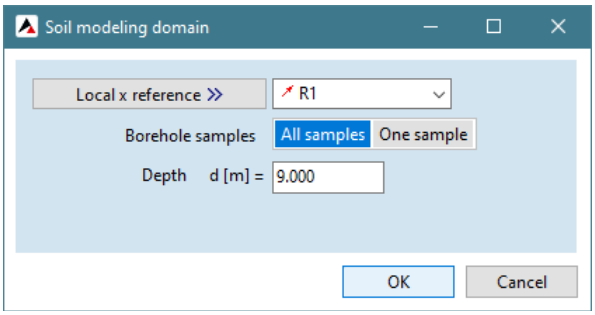
Building a soil model means that the superstructure is not supported by nodal, line or surface supports, but by a soil model. Pad and strip footings should be modelled as individual domains with their real physical dimensions. At the bottom and on the sides of the soil modeling block, the nodes are rigidly supported in the normal direction.

Soil modeling domain

Building a soil model requires definition of at least one soil modelling domain. It is a planar polygon just like a domain used to model surface elements (**4.9.6 Domain**). It is defined by selecting its boundary lines and assigning a few parameters to it.
The soil model is a region of space defined by a polygon and a constant depth. It is meshed with solid elements, where solid elements properties follow the stratigraphic model obtained from the borehole samples (or a selected borehole sample).
The size of the soil domain should always be significantly larger than the superstructure. The required dimensions (horizontal size and depth) depend on the stiffness of the soil and the structure, the load intensity, the stress distribution, the deformation of the soil. It is advisable to choose the dimensions of the soil model in such a way that further increases in dimensions do not lead to significant changes in the results.
Multiple soil model domains can be specified for a model, but the program will use the same soil modeling and mesh generation parameters for the domains that are in contact (see below).

Initial state of soil

For the calculation of initial state of soil please see part **5.1 Static analysis**.



Local x reference The local x direction of the soil modeling domain can be determined by reference elements, see part **4.9.25 References** or can be set automatically. The local z direction is always perpendicular to the plane of the domain. The local *y* direction comes from the right hand rule.

Borehole samples Soil modelling domains can take into account either all borehole samples or a selected sample. In the first case soil properties are determined from the stratigraphic model, in the second case the same soil profile is assumed at every position. This option is applied to all connecting soil modeling domains.

Depth The value of *d* is the minimum depth of the soil model. The actual depth can be greater than this as the soild element mesh will be generated according to the maximum depth of connected domains.
This is also true for mesh generation parameters: the program uses the same mesh size for the contiguous domains to ensure mesh continuity. See **4.12.1.3 Meshing of soil modeling domains**.

4.9.10 Line elements

Line elements are defined and modified in a common dialog. After choosing the element type specific truss / beam / rib element parameters can be set.
Line elements are handled as structural members and not as finite elements. Meshing a line element divides a beam or a rib into finite elements. Existing line elements can be joined to form a single element if the geometry and their properties allow it *(Edit / Find structural members)*. Numbering, labeling, listing functions will consider it to be a single structural member. Structural members can be broken apart by *Edit / Break apart structural members)* See **3.2.14 Assemble structural members, 3.2.15 Break apart structural members**

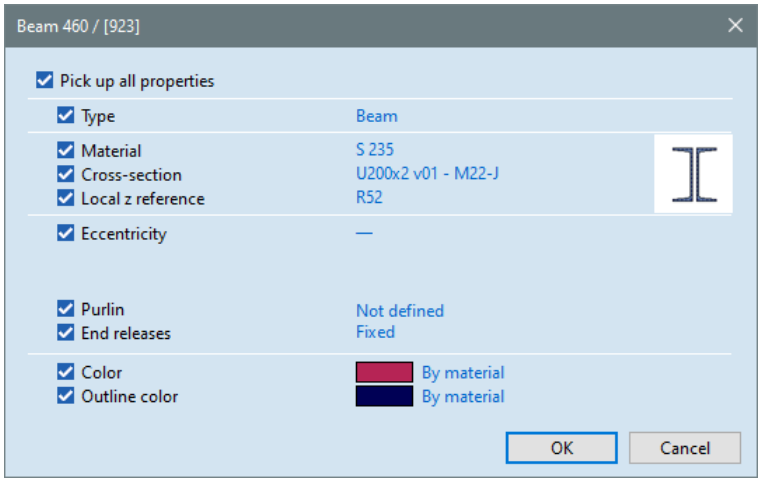
Definition and modification Select the lines to which you wish to assign the same material, cross-section and reference properties. The existing properties of the selected truss, beam and rib elements are displayed in the dialog. If a property does not have the same value on the selected elements, the field remains blank; if no value is entered, the property value will not be affected.
If simple geometric lines have also been selected and wish to define a line element, you must specify the value for every property. As it may be the case that we have selected geometric lines only by accident, the programme will in such cases offer the option to modify only the properties of the line elements, without adding any new elements. If we have selected only geometric lines, the dialog will appear with default values.

Color Elements can have their own fill and outline color used in rendered display mode. Default colors are taken from the material colors. If a color coding is applied the color of a line element is determined by the color coding both in wireframe and rendered modes.
See **2.16.5 Color coding**

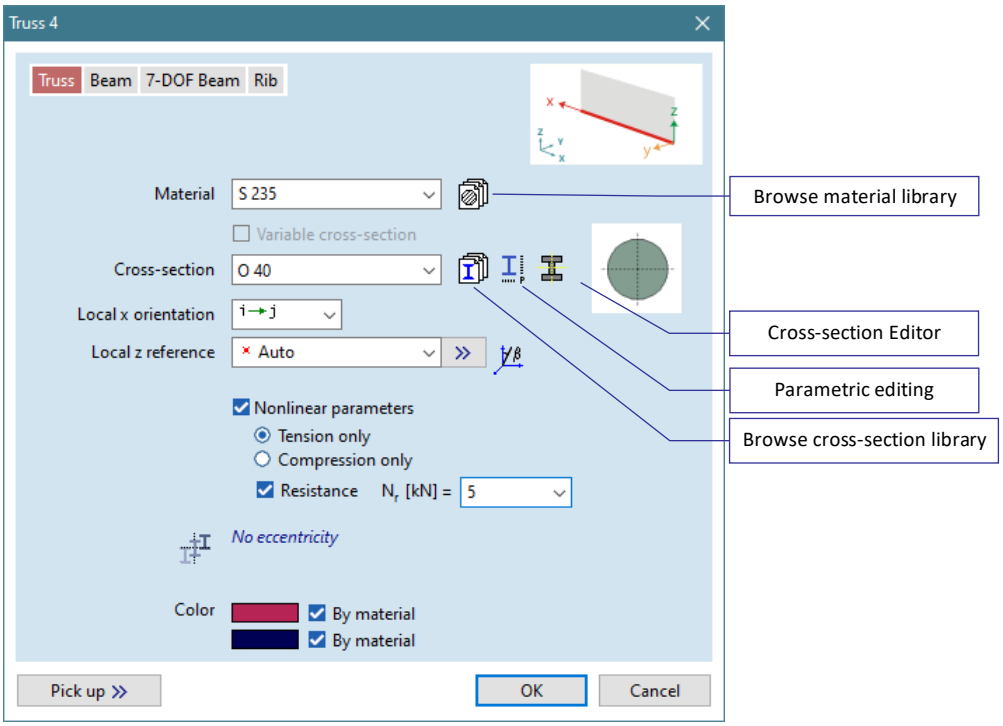


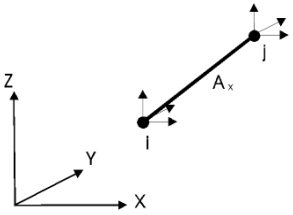
By clicking on the icon, you can assign purlin properties to the line element. (See **4.9.11 Modeling of cold-formed steel purlins)**
This icon is visible only when defining or modifying beam, 7 DOF beam or rib elements made of steel and only if the cross-section is accepted for purlins and wall studs.

Pickup The *Pickup* button can be used when defining any line element. It hides the dialog to choose an element of the model whose properties you wish to copy. A window will then appear showing the properties of the selected element, and you can check the properties you wish to copy.



4.9.10.1 Truss





Truss elements can be used to model truss structures. Trusses are two node, straight elements with a constant cross-section along the truss length. A maximum of three translational degrees of freedom are defined for each node of the elements. The elements are pin-ended (spherical hinges). Only the *Nx* axial internal force is calculated. It is constant along the element.

i denotes the truss end with the lower node index (first node). By default the element *x* axis goes from the node (*i*), to the node (*j*). It can be changed by selecting the other orientation from **Local x orientation.**

Define

You must select the lines to which you want to assign the same material and cross-sectional properties in order to define truss elements.
If elements of different type are selected element definition will be activated.

Defining materials and cross-sections

Materials and cross-sections can be selected from built-in libraries or from a list of the materials/cross-sections already defined.

Allows browsing of the material library to assign a material to the element. The material selected will be added to the material table of the model.



Clicking on the icon allows you to modify sections parametrically, i.e. if the current section was selected from the library or entered parametrically, you can change the geometric parameters according to the predefined type, creating a new section in the model.



Allows browsing of the cross-section library to assign a cross-section to the element. For details see **4.9.2.1 Browse cross-section libraries.** Only one cross-section can be selected from the library or only one parametric shape can be created.



Launches the *Cross-section Editor*. The cross-section created in the Editor will be registered in the list of model cross-sections.



The truss elements are displayed on the screen as red lines.

Service class

If the current design code is Eurocode or SIA and a timber material is selected from the respective database, the service class can be set here. For details see **6.7.1 Timber beam design – TD1 module**

Local x Orientation

Local x direction of a beam can be set to point from Node *i* to Node *j* or vice-versa.

i → *j :* local x axis is directed from the end node with a lower number to the node with the higher one
j → *i :* local x axis is directed from the end node with a higher number to the node with the lower one

Setting this parameter to automatic means that the program determines this orientation based on the end point coordinates.
The orientation can be reversed any time using the shortcut **[Ctrl+E]** or in the dialog or in the property editor window.

Cross-section

In the calculation of the element stiffness, only the cross-sectional area A_x is considered from the cross-sectional properties.

Local z Reference

A reference point can be assigned to define the element orientation.
This allows a correct display of the cross-section on the screen. In case of selecting Auto the reference(s) will be set by the program. Affects only the display of references.
See **4.9.25 References**

Reference angle



Rotation of cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel to the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

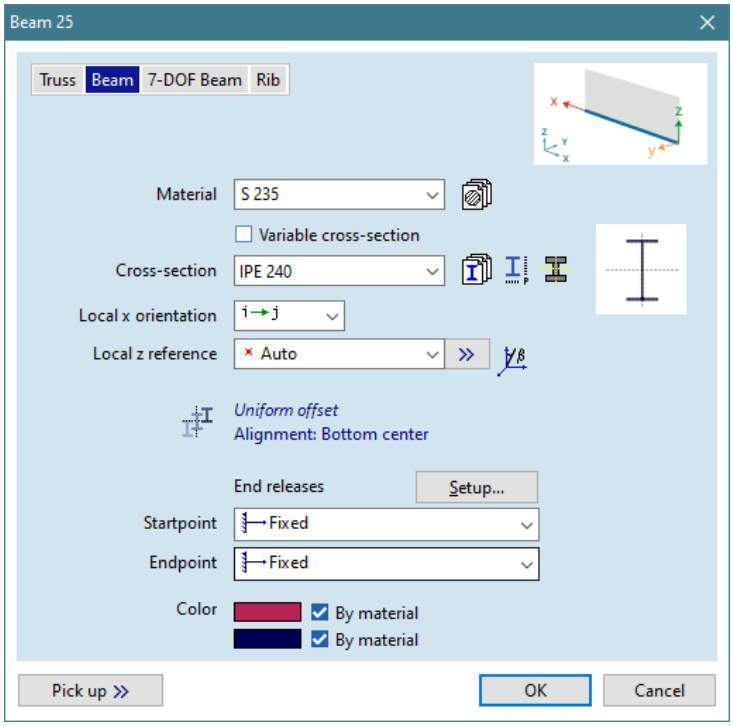
Nonlinear parameters

In a nonlinear analysis you can specify that a truss has stiffness only if it is in tension or compression. You can optionally enter a resistance value as well. A nonlinear elastic behavior is assumed for the nonlinear truss elements.

The nonlinear parameters are taken into account only in a nonlinear analysis.

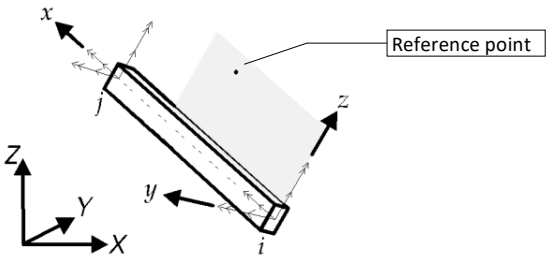
The initial elastic stiffness of a truss element is taken into account if a linear static, vibration, or buckling analysis is performed, disregarding any nonlinear parameter entered.

4.9.10.2 Beam



Beam elements may be used to model frame structures.
Beams are two-node, straight elements with constant or variable (linearly changing) cross-section properties along the beam length. A reference point is used to arbitrarily orient the element in 3-dimensional space (to define the local *x-z* plane). A maximum of three translational and three rotational degrees of freedom are defined for each node of the elements. The ends of the elements can have arbitrary releases.
Three orthogonal internal forces, one axial and two shear (N_x, V_y, V_z), and three internal moments, one torsional and two flexural (T_x, M_y, M_z) are calculated at each cross-section of each element.

i denotes the beam end with the lower node index (first node). By default the element *x* axis goes from the node (*i*), to the node (*j*). It can be changed by selecting the other orientation from *Local x orientation*.



Material, cross-section, local x orientation

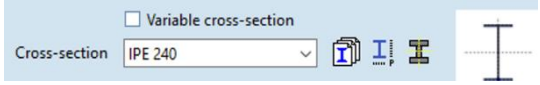


Defining material, cross-section and local direction X are similar to truss elements.
If *Variable cross-section* is checked the start and end cross-sections can differ. To calculate proper intermediate cross-sections certain criteria must be met.
1. Both cross-sections must be thick-walled or thin-walled.
2. Thick-walled sections must have the same number of openings and the respective polygons must have the same number of points.
3. Thin-walled sections must be of the same type.

Haunched I profiles



Choosing *Parametric editing of the cross-section* for I profiles, haunched or double haunched I profiles displays a small palette to choose a parametric type.





If you choose a different type than the current one, the program will import the appropriate parameters from the current cross-section. If a simple I shape is selected instead of a haunched one, the parametric dialog displays the core cross-section. The core cross-section is the upper I of the haunched profile and the middle I of the double haunched one.

When specifying a haunched I profile of variable cross-section, it is common that the haunch at one end-point of the element is reduced to almost zero. Since the start and end I-sections must be of the same type, the end without haunch can only be approximated by setting the minimum haunch height. The parametric dialog for haunched profiles provides a separate *Close haunch* button to set such an end section **(3.1.16 Cross-Section Library).**

Automatic reference

The reference vector will be generated by the program according to the section **4.9.25 References.**

The orientation of the local x axis of the element can be reversed or can be set to Auto which means that local x directions will be set automatically based on the beam end coordinates.

Reference angle

Rotation of cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel to the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

The beam elements are displayed on the screen as blue lines.

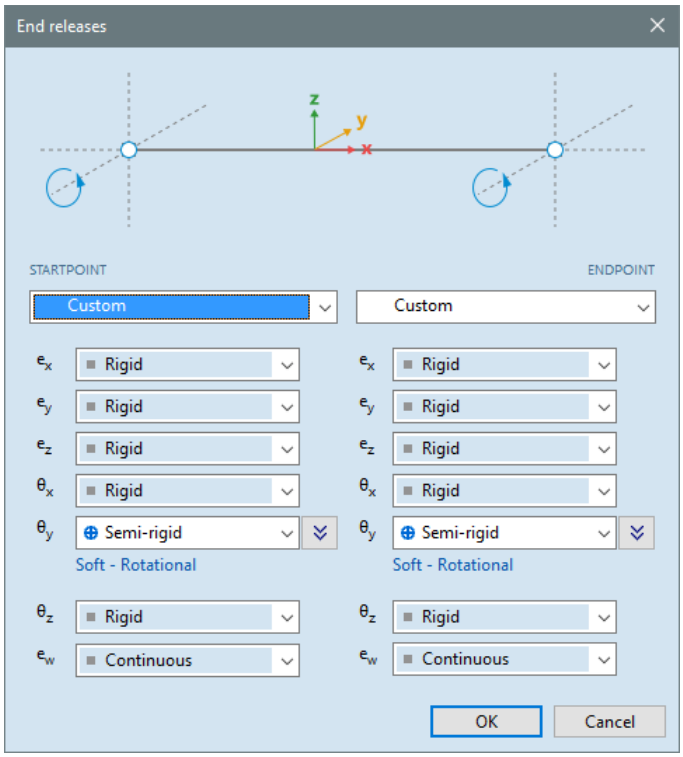
Eccentricity

Beam elements can have eccentricity in the local *y* and *z* directions relative to the logical axis connecting the end nodes. See **4.9.10.2.1 Eccentricity of beams and ribs**

In the case of eccentricity adjustment, during the calculation of the stiffness matrix, the eccentric elements are connected to the statical frame by the means of rigid elements. Additional forces from the eccentricity influences the analysis results, which effect should be taken into account in the course of the evaluation of the results.

End releases

Click on the *Setup...* button to set end releases of the selected elements. End releases remove the connection between the selected elements' degrees of freedom (in the local coordinate system) and the nodes. The end-releases are defined by six or seven (for 7DOF beams, see **4.9.10.3 7 DOF beam element – 7 DOF module** es for each end: *ex, ey, ez, θx, θy, θz, ew*. Each code corresponds to one local internal force component, *ew* represents warping. By default the beam ends are considered rigidly connected (all codes are of rigid connection) to the nodes. Setting a code as hinged connection will result in the corresponding internal force component of the respective end to be released. A semi-rigid connection code can be assigned to the in-plane rotation components of the beam ends.
End release types are displayed in the table of *Beams* or *Ribs*, but separate tables are also created in the Table Browser for *Beam end releases* and *Rib end releases* with all details.



At the top a list of predefined typical configurations is displayed in two dropdown lists, one for the start point and one for the endpoint but end releases can be customized further.

Graphical symbol of a *rigid* connection code (the corresponding local displacement or warping component of the beam end is transferred to the node)

Graphical symbol of a *hinged* connection code (the corresponding local displacement or warping component of the beam end is not transferred to the node)

Graphical symbol of a *semi-rigid* connection code (the corresponding local displacement or warping component of the beam end is partially transferred to the node). The semi-rigid connection requires spring characteristic, as in case of springs and nodal support elements.

The drop-down menu contains just the applicable spring characteristics. New characteristic can be defined by clicking on the spring icon, see **3.1.17 Spring characteristics library** . Characteristics can be any monotonic increasing function.

Graphical symbol of a *steel plastic* connection: the maximum value of the moment at the end points is calculated from the material and cross-section properties. Available only for steel elements and only for the *ex, θy, θz components*.

Graphical symbol of a customizable *pushover* hinge: the corresponding force-displacement or moment-rotation relationship is defined by the user. Click the double arrow to select, load or edit the moment-rotation curve. Available only for the *ex, ey, ez, θy, θz components*. The *ex* component can be used to model masonry infill panels using the compression strut analogy [46; Chapter 7.5.2.1]. The *ey, ez* and *θy, θz* components can be used to model plastic shear and bending hinges.

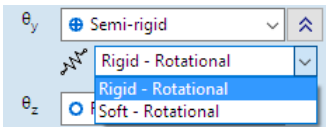
The table below demonstrates the use of end releases for some typical configurations:

End Release	*Symbol*
*Hinge in **x**-**y** plane.* Can't transmit *Mz* moment.	
*Hinge in **x**-**z** plane.* Can't transmit *My* moment.	
*Hinge in **x**-**y** and **x**-**z** plane.* Can't transmit *Mz* and *My* moments.	
*Hinge in **x**-**y** and **x**-**z** plane and free rotation about local **x** axis (spherical hinge).* Can't transmit *Mx, My,* and *Mz* moments.	
*Free translation along local **y** axis.* Can't transmit *Vy* shear force.	
*Free translation along local **z** axis.* Can't transmit *Vz* shear force.	

☞ **If a connection that does not transmit torque is created at both ends of a beam (e.g. using ball joints), then it will be capable of rigid-body motion, which is not permitted (rotation about the x-axis). In such cases, rotation about the x-axis must be prevented at least at one end of the beam.**

Semi-rigid connection

To define semi-rigid hinges choose *Semi-rigid* from the dropdown list and enter the corresponding translational or rotational spring characteristics. To modify it click the double arrow to show the button and the second dropdown list. Semi-rigid hinges can be assigned to all components (3 translational and 3 rotational components).



Click on the spring icon to edit a spring characteristic or select one of the predefined ones.
The moment - relative rotation diagram of a connection is modeled by a linear or nonlinear elastic rotational spring. The nonlinear characteristic can only be used in a nonlinear static analysis. In a linear static, vibration, or buckling analysis, the initial stiffness of the connection is taken into account.
If a new characteristic must be defined the default stiffness values (K_u translational and K_r rotational stiffnesses) are computed on the basis of the cross-sectional characteristics, *but are multiplied by 1000* as follows.

K_u translational stiffness:

$$x: K_u = 1000 \cdot \frac{A_x \cdot E_x}{\ell}$$

where *Ax* is the cross-sectional area, *Ex* is the Young's modulus and ℓ is the length of the element,

K_r rotational stiffnesses:

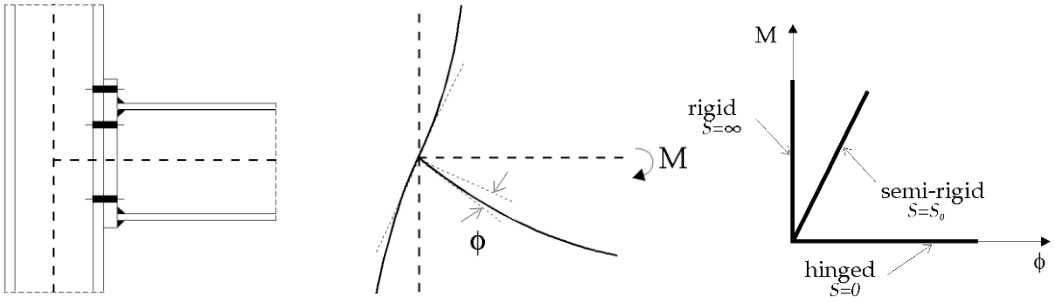
$$yy: K_r = 1000 \cdot \frac{4 \cdot E_x \cdot I_y}{\ell}, \qquad zz: K_r = 1000 \cdot \frac{4 \cdot E_x \cdot I_z}{\ell}$$

where *Iy* and *Iz* are the flexural inertias of the cross-section.
The program does not calculate a default value for y and z shear deformation components and for the torsional component.

The stiffness of the joints has to be determined according to its local coordinate system. Generally, the initial stiffness (or its certain quotient) of the nonlinear characteristics is given.

The following example demonstrates how to interpret the rotational stiffness of a steel joint:

Connection: *Model:* *Moment - Relative Rotation Diagram*



☞ **For example, in the case of steel frame structures, Eurocode EN 1993-1-8 gives the details of application.**

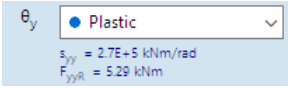
Resistance

To semi-rigid connections a resistance can be assigned, that is the maximum force/moment that can develop in the connection. The value can be defined in the window of spring characteristics.

☞ **In case of steel material, the default values for Fu, My and Mz resistance values are calculated based on the cross-sectional characteristics and the material yield strength. In case of any other material, no default resistance value is calculated by the program.**

☞ **The resistance parameter is only used in case of a nonlinear static/dynamic analysis.**

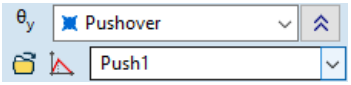
Steel plastic hinge



To define steel plastic hinges choose *Plastic* from the dropdown list.
Plastic hinge can only be set for *x* directional displacement and *y*, *z* rotational components. The calculated axial or moment resistance will be displayed but cannot be edited.
If elements with different materials or cross-sections are selected no value will appear in the edit field but hinges will be defined with the appropriate resistance.
After completing the nonlinear analysis and displaying beam internal force diagrams hinges that got into plastic state at the current load step become red. The number beside the hinge shows the order of getting into a plastic state. Hinge with number 1 is the hinge getting plastic first. Where hinges are not red, plastic limit moment is not reached yet.

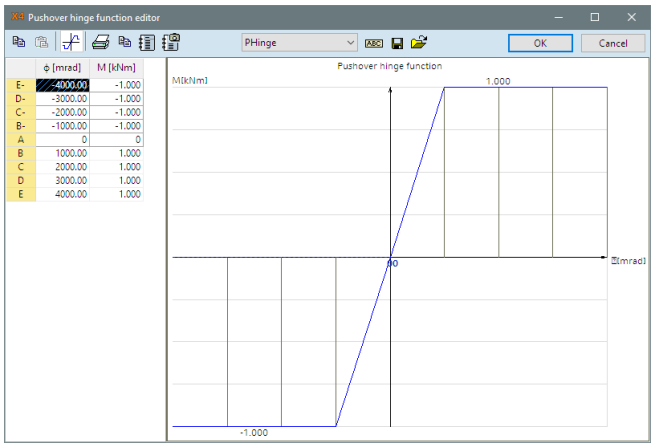
☞ **Steel plastic hinges can only be used with steel beams but are considered only in nonlinear static/dynamic analysis.**

Pushover hinge



To define a pushover hinge choose *Pushover* from the dropdown list then select a function from the dropdown list, load a previously saved function from the library or edit force-displacement or the moment-rotation diagram by clicking on the *Function editor* button. To modify a pushover hinge click the double arrow to show the buttons and the second dropdown list.

A total of five points can be defined for both directions of the force-displacement or moment-rotation diagram. This allows modeling of complex connection behaviour including the possibility of hardening, softening and strength degradation. Behaviour after the last point is extrapolated based on points D and E. The diagram is defined by specifying the corresponding moment and rotation coordinates in the table on the left side of the window. The created digaram is symmetric by default, but this can be overriden by clicking on the *Symmetrical function* button. The created diagrams can be saved and used for other elements in the model.
To facilitate numerical analysis and prevent convergence difficulties, it is recommended to avoid sudden drops of capacity and perfectly plastic sections in the diagram. Instead, relax the diagram, by making sure that there is at least a small difference in both coordinates of consecutive points. This does not influence the results, but improves numerical stability significantly.



After completing the nonlinear analysis and displaying beam internal force diagrams hinges that have got into plastic state by the current load step become red. The number beside the hinge shows the order of getting into a plastic state. Hinge with number 1 is the hinge getting plastic first. Where hinges are not red, plastic limit moment has not been reached yet.

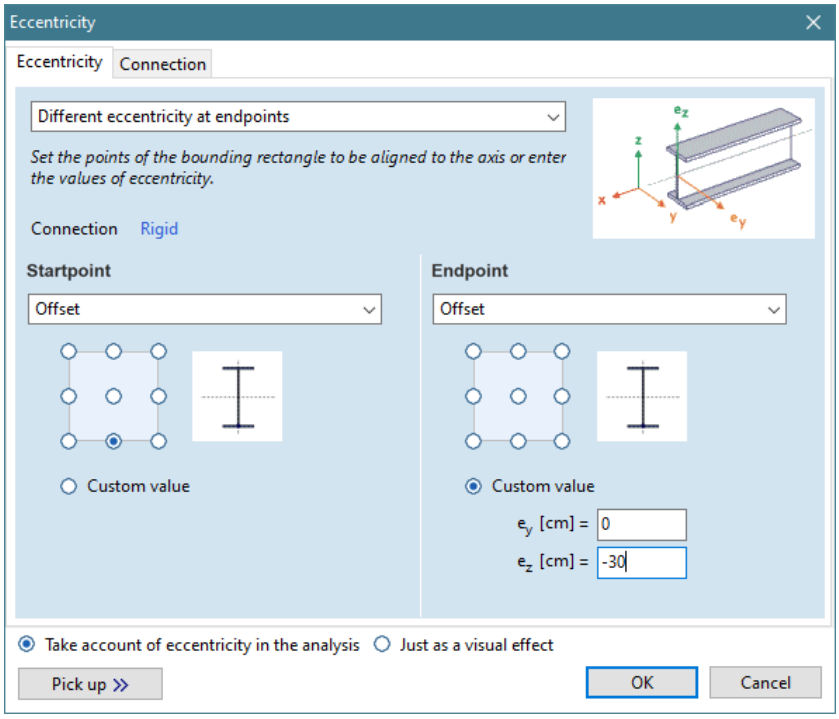
If any beam end release code is of a hinged connection, the beam end is displayed on the screen as a blue circle. If it has a stiffness value a blue cross is inscribed. If the end release corresponds to a spherical hinge, it is displayed as a red circle.
The plastic hinges are displayed as solid circles.
The defined beams appear as dark blue lines.

4.9.10.2.1 Eccentricity of beams and ribs



Beam or independent rib elements can have eccentricity in the local *y* and *z* directions relative to the logical axis connecting the end nodes. Eccentricities assigned to the endpoints can differ, in this case the eccentricity of the intermediate cross-sections changes linearly between the two values.
Eccentricity can be edited in two ways: either from the beam dialog or directly from the toolbar on the *Elements* tab.
You can set whether the eccentricity should be set physically on the model *(Take account of the eccentricity in the analysis)*, or the offset should appear *Just as a visual effect*. This can be controlled using the radio buttons at the bottom

The dialog has two tabs. *Eccentricity* is to define an *ey* and *ez* eccentricity, while *Connection* is to fine-tune the properties of the eccentric connection.



Eccentricity

Methods of definition

The appropriate method can be selected from the dropdown list.

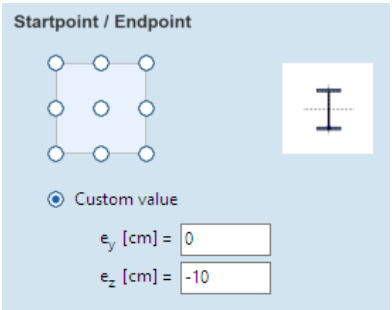
- *No eccentricity*
- *Uniform offset*
- *Aligned to eccentricity group*
- *Aligned to another structural member*
- *Different eccentricity at endpoints*

No eccentricity

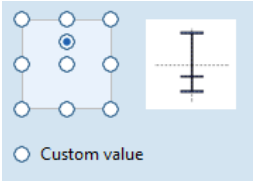
Clears the eccentricity of the structural member.

Uniform offset

The e_y and e_z eccentricities are constant along the structural member. The offset can be set by clicking on the radio buttons placed on selected points of the bounding rectangle of the cross-section, thus making the selected point fit on the logical axis of the element or choosing *Custom value* and entering the values.
The bounding box is displayed in the usual local y-z system, i.e., the *y* axis points horizontally to the right while the *z* axis points vertically upwards.

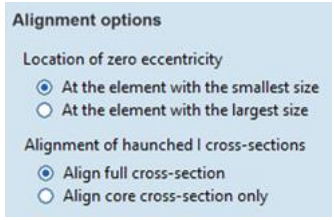


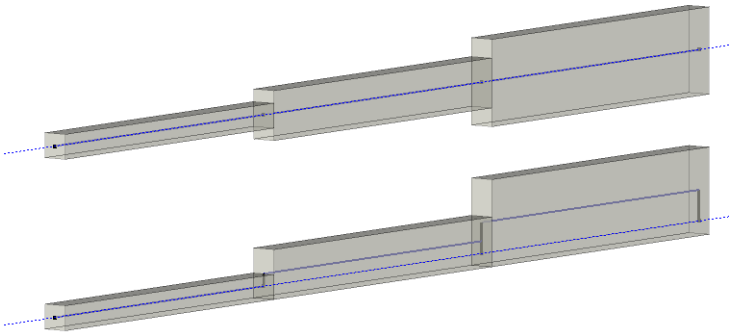
If the element is a haunched or double haunched I beam, an additional position appears among the options, the center of gravity of the core cross-section. The core cross-section is the upper I of the haunched profile and the middle I of the double haunched one.



Aligned to eccentricity group

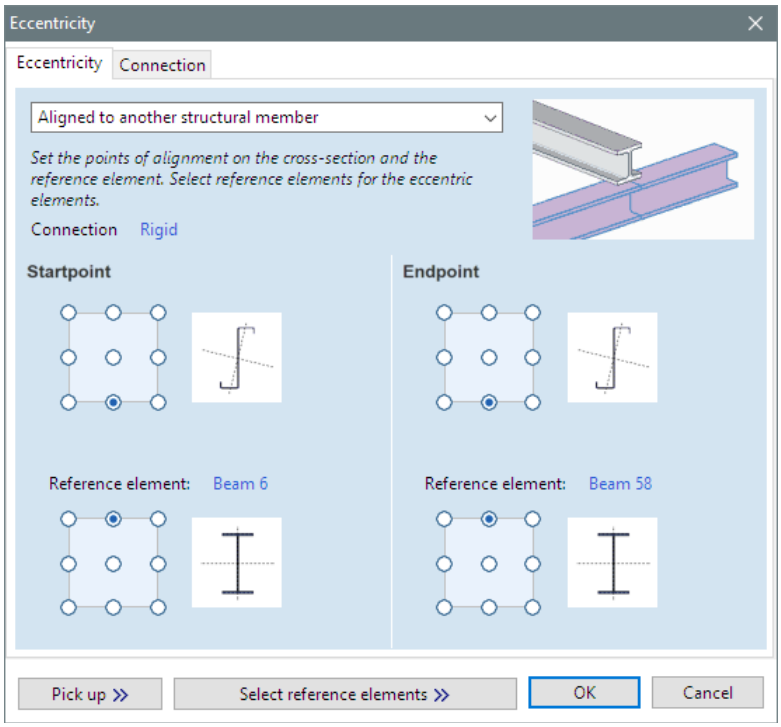
The selected structural members form an eccentricity group just like domains. The e_y and e_z eccentricity of the element smallest/largest in the respective direction (see *Alignment options*) will be set to zero, while other elements will shift so that their selected point on the bounding rectangle remains aligned. Special alignment options are available for haunched I profiles. When the *Align full cross-section* is selected, alignment will be calculated the same way as for other cross-sections. If *Align core cross-section only* is selected haunches will be ignored in the alignment calculation.





Aligned to another structural member

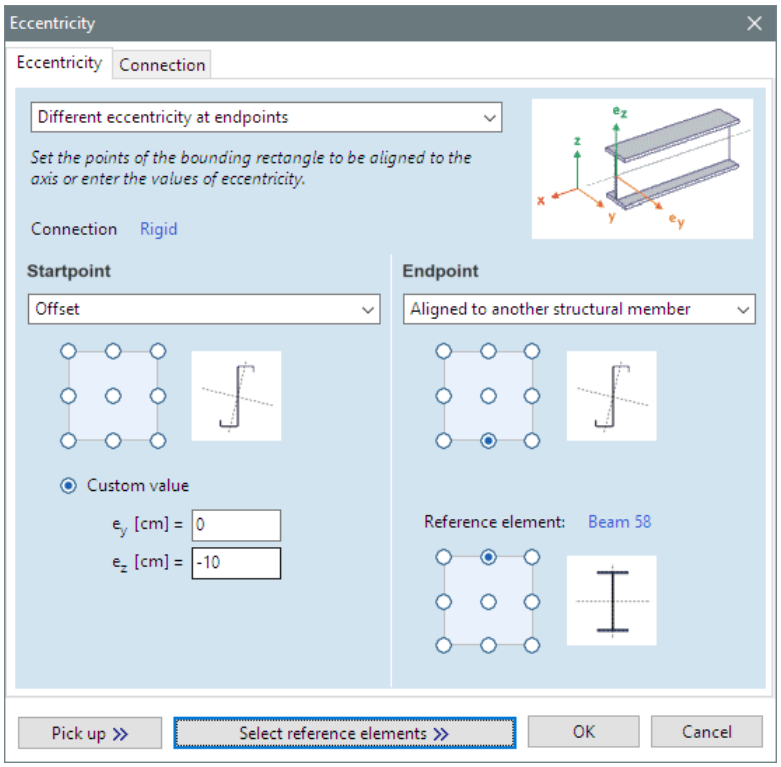
It is a common situation when a beam rest on another one. Choose *Aligned to another structural member* to set the appropriate eccentricity.



Set the point of alignment at the start point and endpoint of the structural member at the top then set the point of alignment for the reference elements connected to the respective points.
By clicking on the *Select reference elements* button the selection is retained but trusses, beams or ribs that can be used as reference elements can also be selected. A reference element must have a common node with the elements to modify. The reference element may also be eccentric, but it must not be aligned to another structural member.
If the selected reference elements meet the above criteria the eccentricity of the element is adjusted matching the specified points of the two bounding rectangles.
Changing the eccentricity or cross-section of a reference element immediately updates the eccentricity of the elements aligned to it.

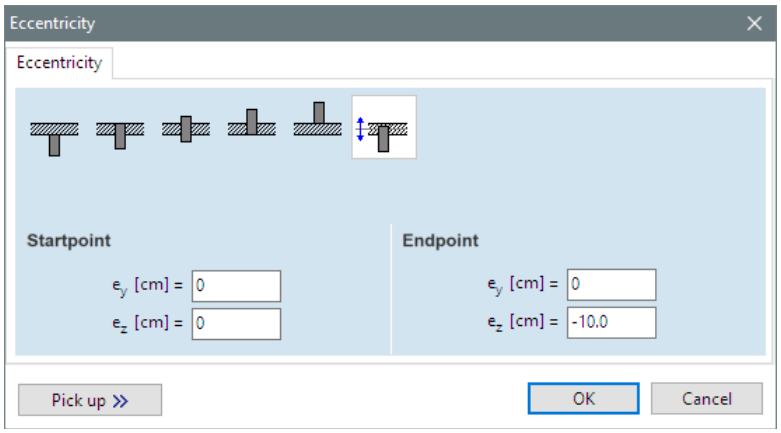
Different eccentricity at endpoints

Allows to define a different eccentricity at the endpoints either by setting an *Offset* or making the endpoint *Aligned to another structural member*.
For details see the description of these methods above.



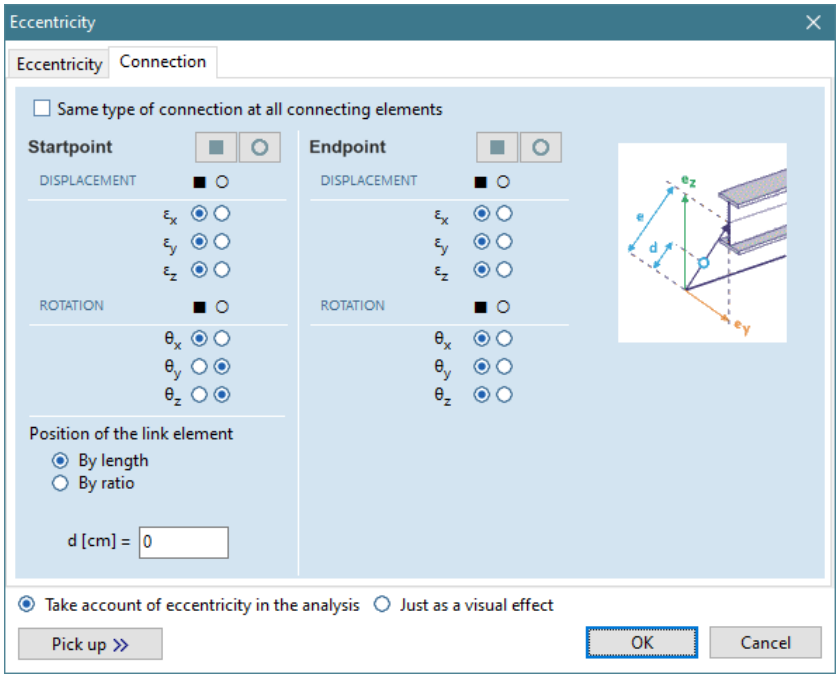
Eccentricity of ribs connected to surface elements

In case of ribs connected to surface elements the last option on the toolbar is *Custom value*, where local *y* and *z* eccentricities can be set independently at the endpoints. In other cases, e_y = 0. See **4.9.10.4. Rib.**



Connection

An eccentric connection can be rigid or hinged according to each displacement or rotation component.
Signs: ■ = rigid connection, ○ = free displacement (ε) or rotation (θ).
If the connection is not rigid, a link element is automatically generated between the end node and the eccentric endpoint.
The position of the link element can be defined *By length* or *By ratio*. If the element is aligned to a reference element a third option is also available: *Calculate the position*. Knowing the cross-section of the two structural members the program calculates the position of the link element.



The two buttons set a predefined connection type

Rigid connection
Sets all components to rigid.

Hinged connection
Sets all components to rigid except the rotation about the *y* and *z* axes.

Same type of connection at all connecting elements
If the eccentric element is connected to more than one element, this switch sets the same connection at each connection point.

Pick up

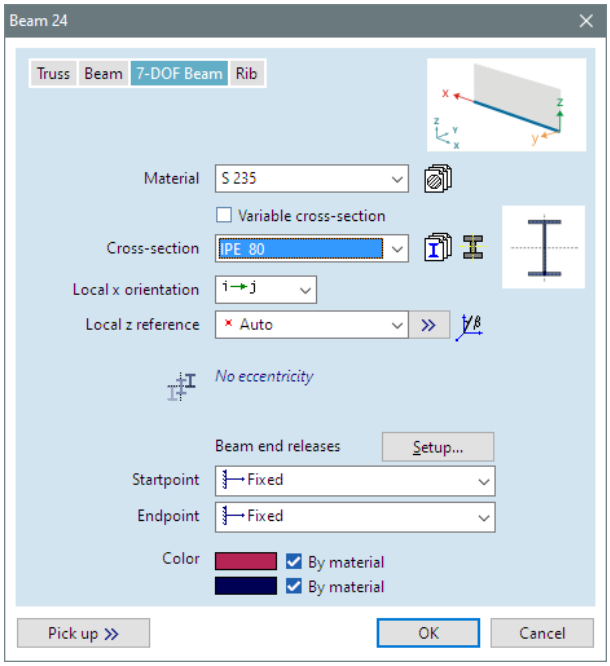
Beam and rib eccentricites can be copied from another beam or rib if that eccentricity is not determined from alignment to a reference element.

4.9.10.3 7 DOF beam element – 7 DOF module

7 DOF BEAM



7 DOFBeams are two-node, straight elements with constant or variable (linearly changing) cross-section properties along the beam length. A reference point is used to arbitrarily orient the element in 3-dimensional space (to define the local *x-z* plane). Besides its three translational, three rotational degrees of freedom a seventh degree of freedom related to warping is defined for each node of the elements. The ends of the elements can have arbitrary releases.
Three orthogonal internal forces, one axial and two shear (N_x, V_y, V_z), and three internal moments, one torsional and two flexural (T_x, M_y, M_z) and a bimoment (B) are calculated at each cross-section of each element. [16, 24, 39, 40, 41].
Other parameters are the same as those of beam elements.



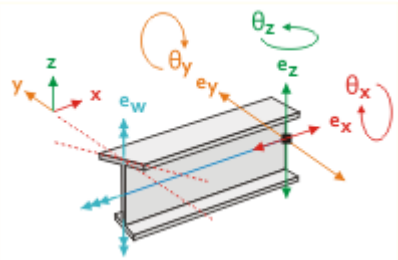
For single and double symmetric, haunched and double haunched variable I shapes, the effect of the variability is taken into account by the element, see [45].

End releases

End releases can be defined for the warping degree of freedom *(e_w)*.

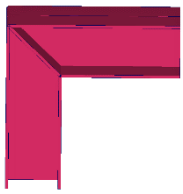


- *Continuous:* Warping degree of freedom at the beam end is the same as that of the node.
- *Free:* Beam can warp freely at the end.
- *Semi-rigid:* This type of connection is to control the transfer of warping based on works of *Basaglia et al.* [39] This special kind of spring characteristics is defined by a coefficient describing the transmission of warping between the node and the beam end, so connecting beams can have different warping. The constraint applied to warping transmission is the following

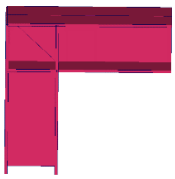


$$w_{beam\ end} = WF \cdot w_{node}$$

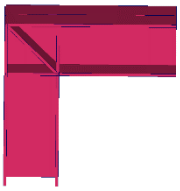
The value of *WF* depends on the joint type. The most common types are:



Diagonal



Box



Box-diagonal

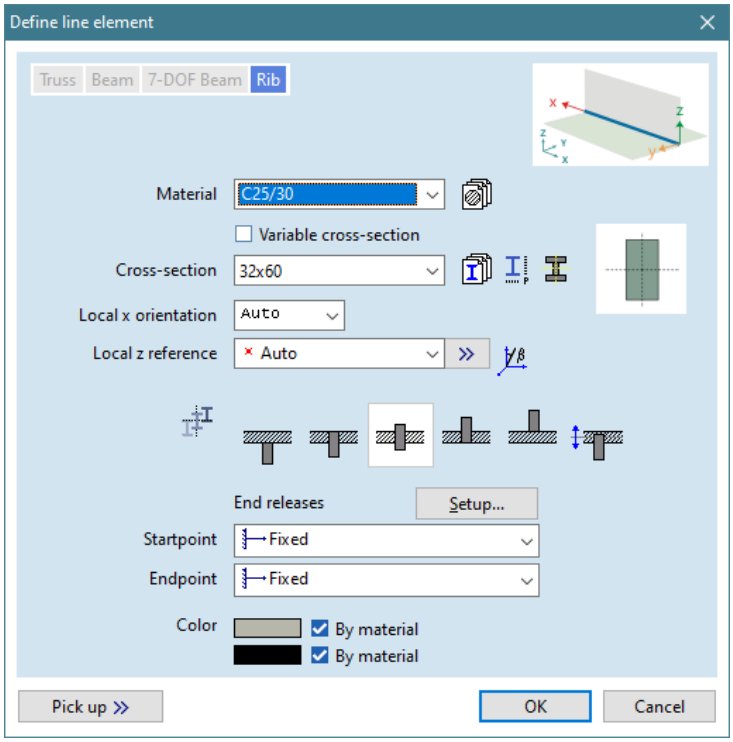
A diagonal joint provides complete and direct warping transmission, a box joint provides a complete and inverse warping transmission, while a box-diagonal joint does not provide transmission.
The respective *WF* coefficients are

Complete, direct (diagonal):	*WF* = 1
Complete, inverse (box):	*WF* = –1
Rigid (box-diagonal):	*WF* = 0

For other joint types WF can be determined by building a shell model of the joint.

☞ **Restrained warping at the end of a beam not connected to another beam can be defined by WF = 0 rigid warping transmission characteristic.**

4.9.10.4 Rib



Rib elements may be used, independently or in conjunction with surface elements (plates, membranes, and shells) to model ribbed surface structures. When used attached to surface elements, the ribs can be connected centrically or eccentrically to the surface elements. The properties of the corresponding surface elements are used to orient the element in the 3-dimensional space (to define the local *x-z* plane).
When used independently, the ribs can model frame structures in a similar way as the beam element, but it can take into account the shear deformations. A reference point or vector is required to arbitrarily orient the element in the 3D space.
Rib elements are isoparametric three node, straight elements with constant or variable (linearly changing) cross-section properties along the rib length, and with quadratic interpolation functions. Three translational and three rotational degrees of freedom are defined for the nodes of the element. Three orthogonal internal forces, one axial and two shear (N_x, V_y, V_z), and three internal moments, one torsional and two flexural (T_x, M_y, M_z) are calculated at each node of each element.

The variation of the internal forces within an element can be regarded as linear.

Define

You must assign the following properties:

Material, Cross-section, Local x orientation

Defining material, cross-section and local direction *x* are similar to truss elements.
If *Variable cross-section* is checked the start and end cross-sections can differ. To calculate proper intermediate cross-sections certain criteria must be met.
1. Both cross-sections must be thick-walled or thin-walled.
2. Thick-walled sections must have the same number of openings and the respective polygons must have the same number of points.
3. Thin-walled sections must be of the same type.

Material

The material of the rib can be different from the surface material (if it is connected to a surface).

Cross-section

The rib element's cross-section is taken into account as is shown in the figure below:

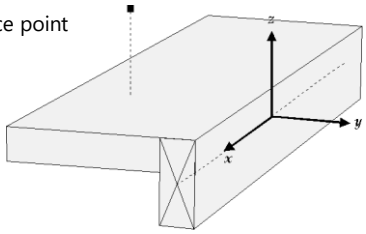
Automatic reference

The reference vector will be generated by the program according to the section 4.9.25 References

Reference

Independent rib:

The local coordinate system is defined as follows: the element axis defines the *x* local axis; the local z axis is defined by the reference point or vector; the *y* local axis is according to the right-hand rule.



Rib connected to a domain:

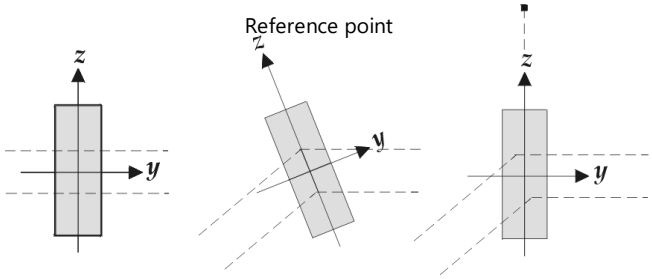
The local coordinate system is defined as follows: the element axis defines the *x* local axis; the local z axis is parallel to the *z* axis of the surface element; the *y* local axis is parallel to the plane of the surface element, oriented according to the right-hand rule.

The figure below shows that when the rib is located on the edge of two surface elements that makes an angle, the local z axis is oriented by the average of normal axes of the surfaces. If more than two surfaces are connected to the edge and you select one or two of them then an automatic reference will be created when defining the rib.

if multiple domains are connecting to the rib select one domain (or two domains) before defining a rib to register the rib to that domain. The domains registered to the rib can be checked in the element tooltip popping up if the *Elements* tab is active and the mouse hovers over the element.

Domain registration also takes into account the visible parts. If a rib is on the edge of multiple domains, and only one of these domains is visible (the parts the other domains belong to are turned off), the rib will be registered to the only visible domain.

The cross-sectional properties must be defined in this coordinate system.



Reference angle



The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel to the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

End releases

End releases can be defined for ribs the same way as for beams. By default both ends are fixed.

End release types are displayed in the table of *Ribs*, but a separate table is also created in the Table Browser for *Rib end releases* with all details.

Eccentricity of ribs

Ribs are used mostly connected to surface elements (domains) but they can be defined as independent line elements as well. In the latter case their eccentricity can be defined the same way as for beams (see **4.9.10.2.1 Eccentricity of beams and ribs**). If ribs are connected to surface elements (domains) their eccentricity can be set as follows.

If more than two surfaces are connected to the edge also select one or two of them to define eccentricity for the rib. The eccentricity of a rib is given by the distance of the center of gravity of its cross-section to the plane of the model of the surface (neutral plane). It is positive if the center of gravity is on the positive direction of its local z axis.

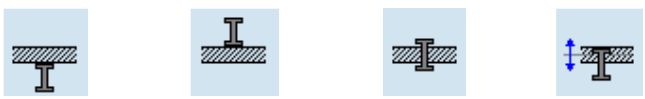
There are standard options to set the rib eccentricity.

Rib types in case of concrete slab and concrete rib



Bottom rib | *Bottom rib (embedded)* | *Rib in midplane* | *Top rib (embedded)* | *Top rib* | *Custom eccentricity*

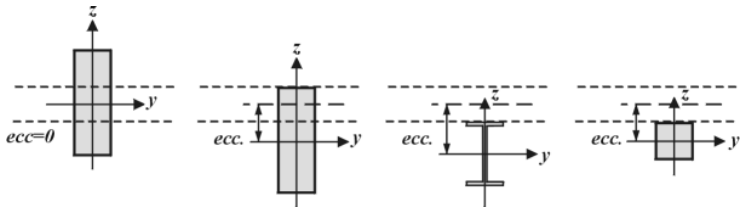
Rib types in case of concrete slab and a rib of a different material



Bottom rib | *Top rib* | *Rib in midplane* | *Custom eccentricity*

In the first three cases the actual eccentricity is calculated from the rib cross-section and the plate thickness. If the rib is made of concrete the definition of top and bottom ribs is different, so button pictures change according to the rib material. If rib cross-section or plate thickness changes the eccentricity is automatically recalculated.
If the rib is made of steel or timber, connected to a shell and is defined as a top or bottom rib, an additional axial connection stiffness can be defined.

☞ **In case of reinforced concrete plate-rib connections, rib cross-section must include the plate thickness. In other cases (steel or timber structures), the cross-section is attached to the top or bottom plane of the plate.**



☞ **For plates, the eccentricity of the rib will modify the flexural inertia of the rib as follows:**

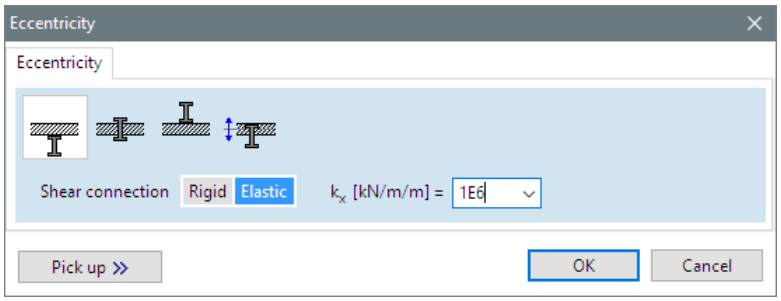
$$I_y^* = I_y + A \cdot exc^2$$

For shells, due to the eccentric connection of the rib to the shell, axial forces will appear in the rib and shell.

☞ **In the case of eccentricity adjustment, during the calculation of the stiffness matrix, the eccentric elements are connected to the statical frame by the means of rigid elements. Additional forces from the eccentricity influences the analysis results, which effect should be taken into account in the course of the evaluation of the results.**

Shear connection

In case of top and bottom ribs (except for concrete structure), the shear connection between the slab and rib can be set, which can be *Rigid* or *Elastic*. The elastic stiffness – *kx* component - has to be set by the user, it is not calculated by the program (the default value is 1E+6 kN/m/m).
Using *Elastic* setting, the specific longitudinal shear connection force (*Vxz* component) is also calculated (this is not calculated with rigid settings).



Stifffness reduction

Using Eurocode standard, stiffness reduction can be set for certain architectural elements (such as columns, beams, walls, slabs and other type of elements) for modal response spectrum analyis. For details see chapter **3.3.10 Stiffness reduction.** In case of concrete ribs two kind of reduction factor can be set: k_A and k_I.

Factor kA lowers the cross-sectional area of the concrete. It has an effect only on the elongation stiffness and has no effect on the shear stiffness. Factor kI lowers only the flexural stiffness of the cross-section.

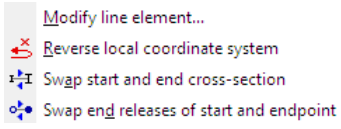
Ribs appear as blue lines.

Pick up>>

Properties of another element can be picked up and assigned to the selected elements. Clicking the *Pick up* button closes the dialog. Clicking an element picks up the value and shows the dialog again. Only those properties will be copied where the checkbox is checked.

Popup menu

If the *Element* tab is selected click the right mouse button on line elements to find further useful tools operating on selected line elements. If no element has been selected these functions will change the element you clicked on.



Reverse local coordinate system

Sets the local *x* direction of the selected line element in the reverse direction. Start and end cross-sections of tapered beams/ribs and their end releases will remain in their original position only their interpretation will be different.

Swap start and end cross-section

Swaps the start and end cross-sections of the selected variable cross-section beams/ribs.
The local *x* direction remains unchanged.

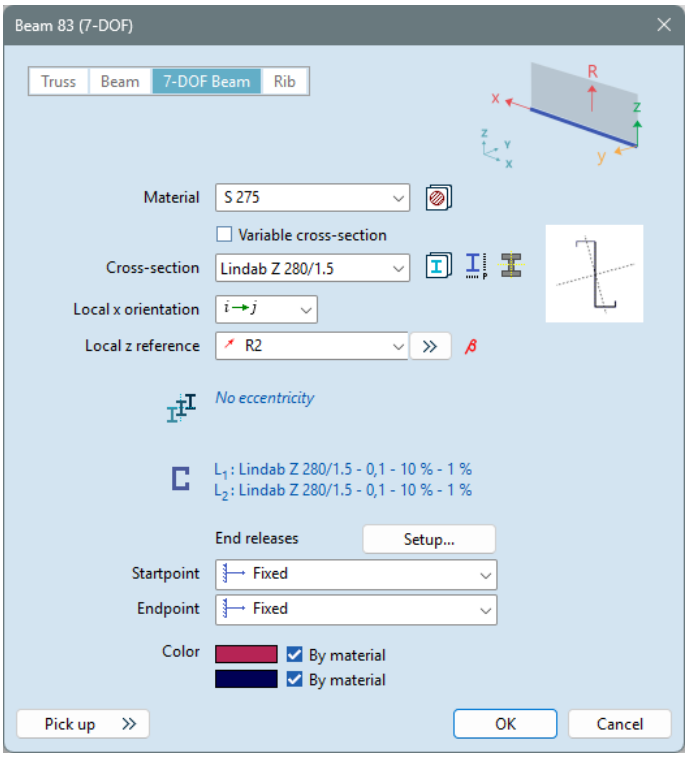
Swap end releases of start and end point

Swaps the end releases of the selected beams/ribs.
The local *x* direction remains unchanged.

4.9.11 Modeling of cold-formed steel purlins

Cold-formed thin-walled steel purlins and wall studs are frequently used to support the external cladding, particularly in hall structures. These claddings are most commonly cold-formed trapezoidal sheets or sandwich panels. Since cold-formed thin-walled profiles have open cross-sections and low stiffness, they are highly sensitive to various forms of loss of stability and torsional loads. Consequently, it is highly recommended to model the purlins as 7DOF beam elements. Alongside the standard 6 displacements and rotations per node, the 7DOF beam element also features a seventh degree of freedom for warping. Only this element type is capable of accurately calculating and taking into account warping and the stresses arising from restrained torsion. For open Z-, C-, or Sigma-sections, this is critical in terms of stability loss (e.g., lateral-torsional buckling).
To ensure the flexibility of the software, in addition to the 7DOF type, the use of conventional 6DOF beam elements and rib elements is also permitted for purlins. In these cases, however, warping effects are only taken into account approximately or not at all in the model. Truss elements cannot have purlin properties.

☞ **In the software and the manual, we generally use the term purlin as a collective term for the modeling and design of cold-formed thin-walled steel beams (purlins and girts). It is important to note that purlins also include girts.**



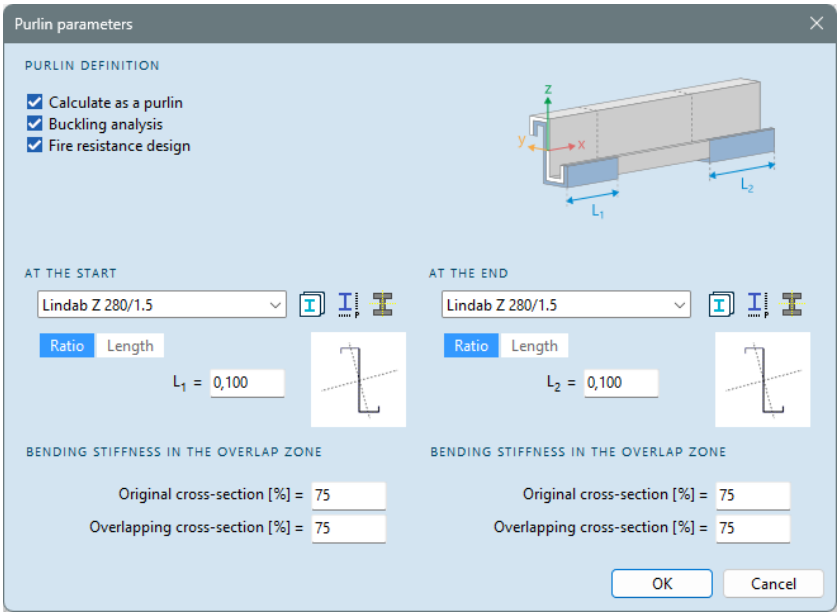
By clicking the icon shown in the figure, purlin properties can be assigned to the examined line element. The icon is only visible when defining or modifying Beam, 7DOF beam, and Rib type line elements. It is also necessary that the material of the element in question is steel, and its cross-section is an accepted cross-section for purlins and girts.

Applicable materials: A material according to the part of Eurocode 3 for cold-formed steel structures (EC3-1-3) must be assigned to the specified cross-section. The material database contains the relevant standard steel grades (e.g., S280GD, S320GD, S350GD), which take into account the yield strength increase resulting from cold forming and the manufacturing tolerances,

Accepted cross-section types: The feature is dedicated to supporting the design of thin-walled, cold-formed steel cross-sections. Ready-made profiles from major manufacturers in the international and domestic markets (e.g., Lindab, Kingspan) can be directly imported from the software's built-in cross-section library. Supported and manually creatable types include stiffened and plain C-sections, U-sections, Z-sections, as well as Sigma and Zeta sections, which are typically equipped with lips and stiffening ribs.

Although the purlin group wizard (see **4.9.12 Purlin group**) is available in the software for mass, surface-based generation, the manual definition of purlins and girts is essential for hall roofs and wall framing structures with unique geometry, asymmetric, or special designs.

When clicking the icon for an element with an appropriate cross-section and material in the beam element dialog box, the following special design parameters become available:



Calculate as a purlin

By toggling the switch, it can be set whether the line element has purlin properties and parameters.

Overlap at the start and end of the element

In the case of multispan hall roofs and wall framing systems, purlins are often spliced over the rafters (frames) in order to increase bending moment capacity and optimize deflection. The following can be specified in the dialog box:

- Overlapping cross-sections at the start and at the end (defined by the local *x* orientation)
- *Overlap length (L_1, L_2):* The actual length of the overlapping zone defined separately at the start node and end node of the beam, or a ratio proportional to the length of the beam.
- *Bending stiffness in the overlap zone:* In practice, the connections in the overlapped sections are not infinitely rigid due to the flexibility of the bolted connections, the deformation of the elements and possibly local buckling of the plates. Therefore, the bending stiffness cannot simply be taken as the sum of the bending stiffnesses of the two cross-sections. The extent to which the increased stiffness should be taken into account in the overlapping zone can be specified as a percentage based on the second moments of area of the constituent cross-sections. In the case of a perfectly rigid connection, 100-100% should be set. The 75 % used as the default value is consistent with the results of other specialist software.

☞ **During the construction of the stiffness matrix, the software can take into account the increased bending stiffness in the overlapping zones. For this, the element must be meshed and its meshing must align with the boundaries of the overlapping zones. If the purlins are not meshed properly, the program will issue a warning.**

Buckling analysis

The switch controls whether the software should account for the buckling of purlins when running a Buckling analysis, or neglect it so they only have a supporting effect. The software achieves the latter by ignoring second-order effects (geometric stiffnesses). This may be necessary due to the large number and low stiffness of purlin and girt elements. It can easily occur that during a Buckling analysis, dozens or hundreds of mode shapes will show the lateral-torsional buckling or torsional buckling of the purlins, making it difficult to determine the mode shapes and critical load multipliers characteristic of the main girders. The feature works similarly to a Buckling analysis run on a substructure (For details, see... **6.2 Buckling).**

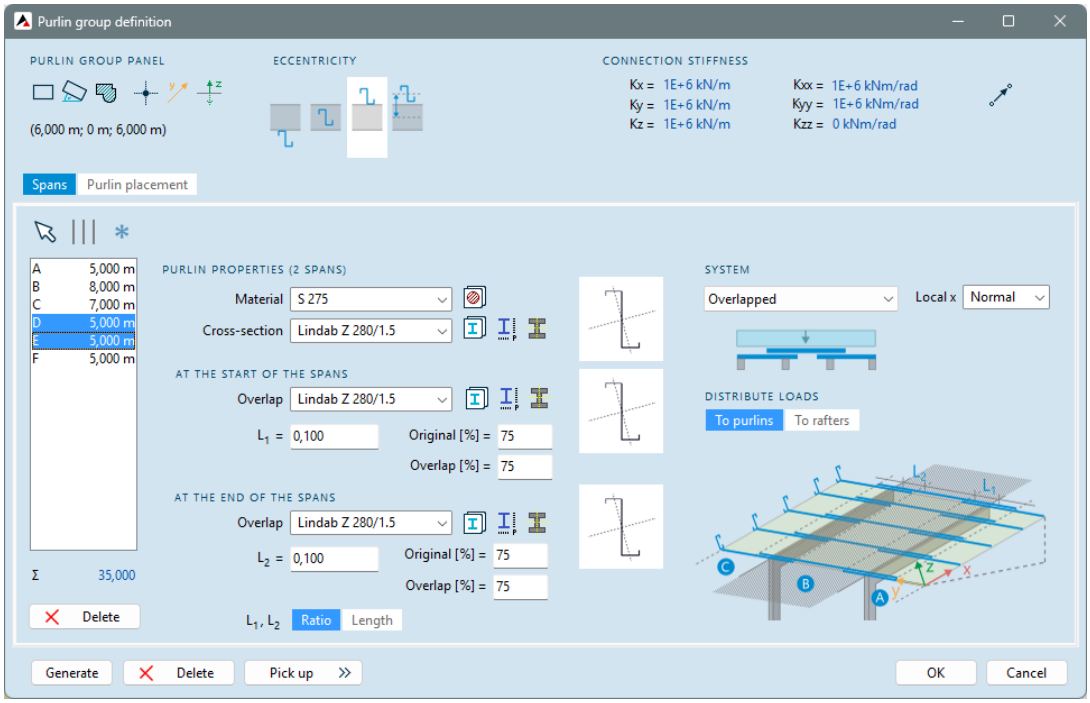
Fire resistance design

Cold-formed, thin-walled steel elements lose their stiffness extremely quickly in the event of fire. This is particularly problematic for the design of the rafters, because purlins and girts cannot be considered as supports during the fire design of the frames. Using the switch, we can control whether the software should consider the purlin as a support in the AutoMcr method during the fire design of the rafters connected to the line element.

4.9.12 Purlin group definition



This is essentially a wizard tool, a powerful automation feature in AXISVM specifically designed for the quick and precise modeling of cladding support systems (roof purlins and wall studs) of hall buildings, warehouses, and industrial facilities.
The primary purpose of this feature is the mass generation of purlins. It eliminates the need for the user to manually define and draw hundreds of line elements one by one, along with their corresponding nodal connections (e.g., to the main girder). By using this wizard tool, modeling time is drastically reduced, and the chance of errors from manual data entry is minimized.



Defining a purlin group is based on an intelligent, polygon-based surface definition. To generate the purlin system, the user must first define a purlin group panel on the roof plane or side wall, which in rectangular cases is possible by specifying just two or three corner points.
After defining the polygon, the built-in algorithm automatically maps the space under the panel. It recognizes the existing load-bearing rafters (frames, beams, columns) and, based on these, independently determines the lengths of the spans. The program spans the purlin system onto the recognized structural framework based on the parameters specified in the dialog box.
The parameters are linked to the purlin group panel as an object. It stores the parameters and the index numbers of the generated purlins. Potential modifications (cross-section, distribution) can be easily performed, and the purlin distribution can be easily regenerated. When deleting a purlin group panel, the purlins generated are also deleted.
The purlins and girts generated by the wizard are automatically created in the model as 7DOF beam elements with active purlin properties corresponding to the settings. A load panel can also be generated simultaneously with the creation of the purlins in the plane of the elements for the purpose of defining self-weight, live, and meteorological loads. The automatically created load panels linked to the purlin group panels are fitted by the software to the load panels of adjacent purlin groups.

☞ **The polygon-based recognition algorithm works most efficiently for regular or moderately asymmetric (e.g., slightly tapering) frames and roof planes. For highly articulated roofs with many openings (e.g., large skylights) or broken planes, it is recommended to divide the surface into several separate polygons and run the purlin group generation plane by plane in multiple phases.**



Origin of the panel and the purlin placement scheme: By clicking on the icon, you can select the position of the origin for the element group panel into the model space. When elements are generated, their placement starts from the origin.
The origin must lie within the plane of the panel.



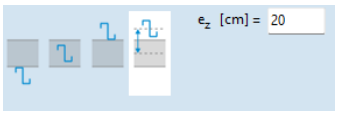
Set purlin direction: The local y-axis of the purlin group panel is parallel to the direction of the purlins. By clicking the icon, we can modify the direction of the panel's local y-axis.



Flip local z-direction: By clicking the icon shown in the figure, we can reverse the panel's normal vector. The local z-vector plays a role when assessing eccentricities.

Eccentricity

We can set the eccentricity of the generated purlins by choosing from four options. These include alignment to the bottom plane of the main girder, centric alignment, alignment to the top plane, and an arbitrarily specified eccentricity relative to the axis of the main girder.



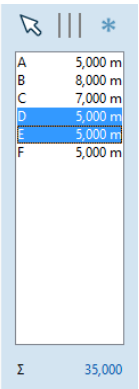
Connection stiffness



The generated purlins are connected to the main girders by node-to-node link elements (see... **4.9.22 Gap**). By clicking the icon shown in the figure, we can specify the stiffness of the link elements. We can choose from rigid, semi-rigid, and hinged options. The stiffnesses corresponding to individual degrees of freedom can also be set manually.

Spans

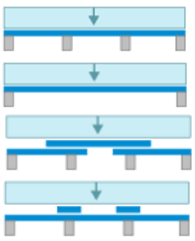
Further parameters can be specified on two tabs. The first one is the *Spans* tab.



Spans have letter codes. Rafters can be identified automatically by clicking on the centre icon, or by selecting them in the model space and clicking on the icon on the left. Different cross-sections and overlap parameters can be assigned to the spans. By selecting multiple spans, the cross-section and overlap parameters can be set for several spans at once. The star icon selects all spans.
The overlap parameters are the same as those for individual purlins (see... **4.9.11 Modeling of cold-formed steel purlins**).

System

The structural system of the purlins can be specified from four options:
This determines how the generated beam elements are connected at the rafters.



Continuous: no overlap, rigid connection

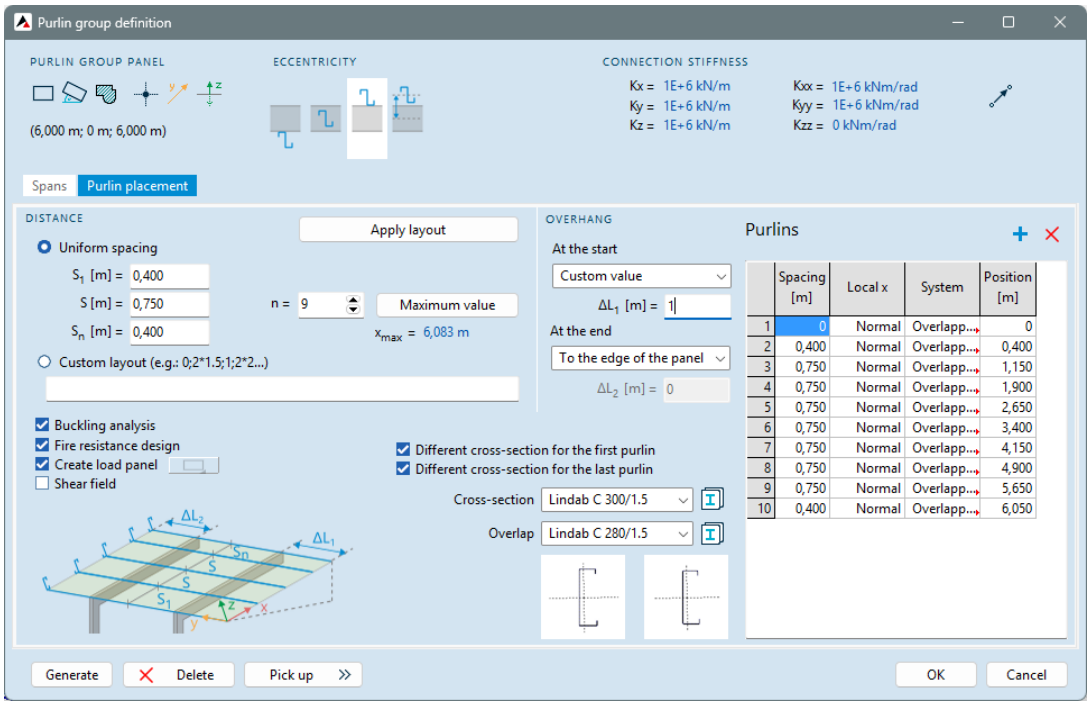
Single span: no overlap, hinged connection

Overlapped: with overlaps

Sleeved: sleeved connections

Distribute loads

It can be set whether the automatically generated load panels assigned to the purlin group panel distribute the loads to the purlins or to the rafters. This latter option can be useful in seismic design. By distributing the loads to the rafters, it is possible to avoid a multitude of modal shapes with small modal masses when running a vibration analysis.



Purlin placement

The second tab is to enter parameters related to the purlin distribution, as well as some additional parameters related to modeling and design.

Distance

The transverse distribution of the purlins can be specified with uniform or custom spacing. A figure at the bottom of the window helps interpret the S distances. By clicking the *Apply layout* button, the generated distribution appears in the table on the right. Here, the properties of the individual purlins can still be modified before the actual generation (structural system, or the direction of the local x-vector if we want to reverse the purlin).

Overhang

An overhang can also be specified at the ends of the purlins, either up to the edge of the purlin group panel or by specifying an actual overhang value.

First or last purlin cross-section of is different

The software supports the common industry practice where the first (or last) purlin of the roof plane at the eaves requires a different cross-section due to the fastening of the wall cladding or the nodal design of the gutter. The dialog allows a configuration where purlins have a Z-section, but the bottom and top purlins have a C-section with the appropriate orientation in the edge positions.

Buckling analysis

See **4.9.11 Modeling of cold-formed steel purlins**

Fire resistance design

See **4.9.11 Modeling of cold-formed steel purlins**

Create load panel

If this feature is enabled, the software also generates a load panel so that the loads acting on the cladding can be applied as surface loads. Depending on the settings, the load panel distributes this load either to the purlins or to the rafters. If the feature was not active when the purlins were generated, the load panel can also be generated later without regenerating the purlins. By clicking the button next to the label, load panels covering gates/openings can be specified, which strictly distribute the load to the rafters below/behind them and not to the purlins. It is necessary to create such panels so that the self-weight, and possibly meteorological loads acting on the gates, doors, windows, and skylights, are also taken into account.

Shear field

Shear field parameters, i.e., the stabilizing shear stiffness of the cladding, can be specified for the generated purlins.

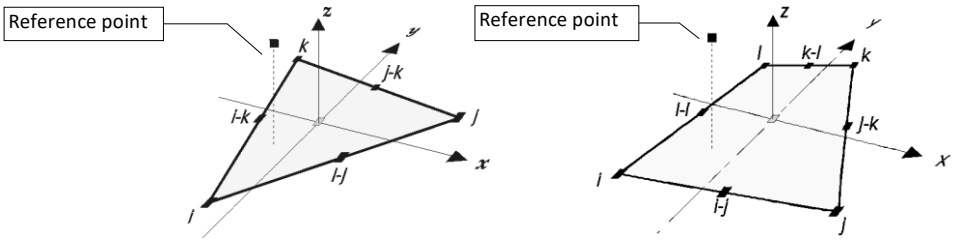
4.9.13 Surface elements

Surface elements can be used to model membranes (membrane element), thin and thick plates (plate element) and shells (shell element) assuming that the displacements are small.
As surface elements you can use a six node triangular or eight/nine node quadrilateral finite elements, formulated in an isoparametric approach. The surface elements are flat and have constant thickness within the elements.

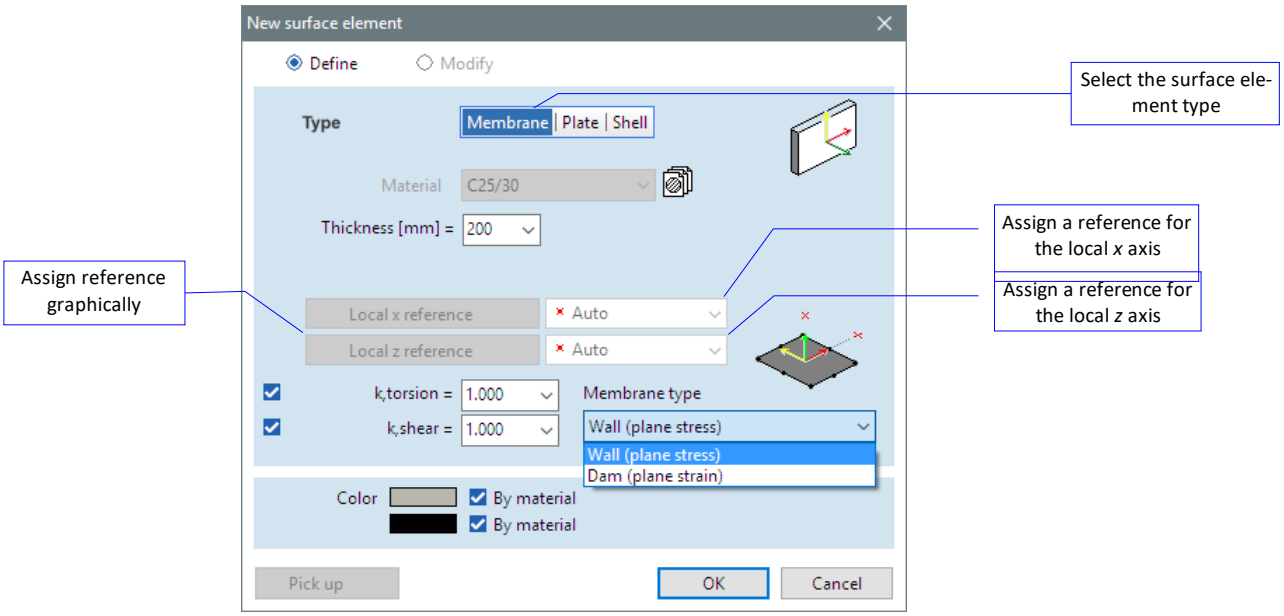
Introduction of domains and automatic meshing rendered the definition of surface elements obsolete.

☞ **It is preferable for the element thickness not to exceed one tenth of the smallest characteristic size of the modeled structural element, and the deflection (w) of a plate or shell structural element is less than 20% of its thickness (displacements are small compared to the plate thickness).**

Use of elements with the ratio of the longest to shortest element side lengths larger than 5, or with the ratio of the longest structural element side length to the thickness larger than 100 are not recommended.
In some cases when the elements are used (that are flat with straight edges) to approximate curved surfaces or boundaries, poor results may be obtained.
For the factors *k, torsions* and *k, share* see **Domain**.



4.9.13.1 Membrane



Membrane elements may be used to model flat structures whose behavior is dominated by in-plane membrane effects. Membrane elements incorporate in-plane (membrane) behavior only (they include no bending behavior).

☞ **The membrane element can be loaded only in its plane.**

AXISVM uses an eight node Serendipity,
plane stress ($\sigma_{zz} = \sigma_{xz} = \sigma_{yz} = 0, \epsilon_{xz} = \epsilon_{yz} = 0, \epsilon_{zz} \neq 0$) or
plane strain ($\epsilon_{zz} = \epsilon_{xz} = \epsilon_{yz} = 0, \sigma_{xz} = \sigma_{yz} = 0, \sigma_{zz} \neq 0$)
finite element as membrane element.
The membrane internal forces are: n_x, n_y, n_{xy}. In addition the principal internal forces n_1, n_2 and the angle α_n are calculated.

The variation of internal forces within an element can be regarded as linear.

The following parameters should be specified:

1. Plane strain or plane stress
2. Material
3. Thickness
4. Reference (point/vector/axis/plane) for local x axis
5. Reference (point/vector) for local z axis

Allows browsing of the material library to assign a material to the element. The material selected will be added to the material table of the model.

Automatic reference:
The axis of element local directions *x* and *z* can be determined by reference elements, see part **4.9.25 References** or can be set automatically.

The center of the membrane elements is displayed on the screen in blue.

4.9.13.2 Plate

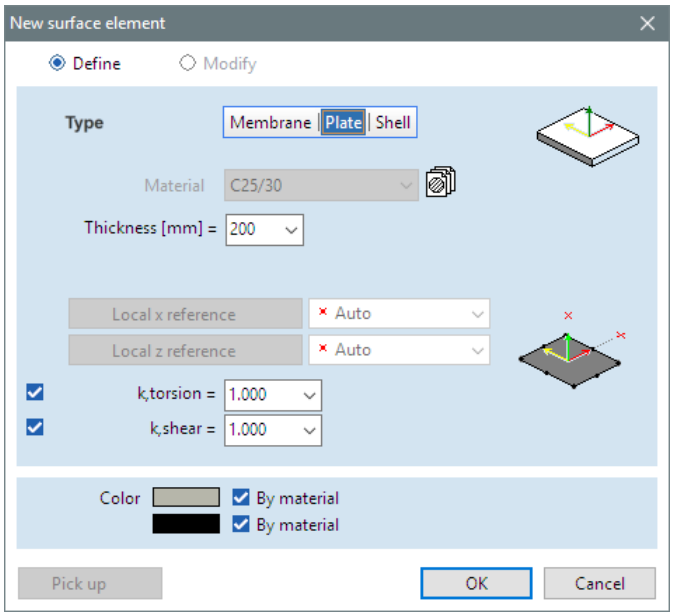


Plate elements may be used to model flat structures whose behavior is dominated by flexural effects.

AXISVM uses an eight/nine node Heterosis finite element as plate element, that is based on Mindlin-Reissner plate theory that allows for transverse shear deformation effects). This element is suitable for modeling thin and thick plates as well.
Plate elements incorporate flexural (plate) behavior only (they include no in-plane behavior).

The element can only be loaded perpendicular to its plane.

The plate internal forces are: m_x, m_y, m_{xy} moments, and v_x, v_y shear forces (normal to the plane of the element). In addition, the principal internal forces: m_1, m_2, the angle α_m and the resultant shear force q_R are calculated.
The variation of internal forces within an element can be regarded as linear.

The following parameters should be specified:

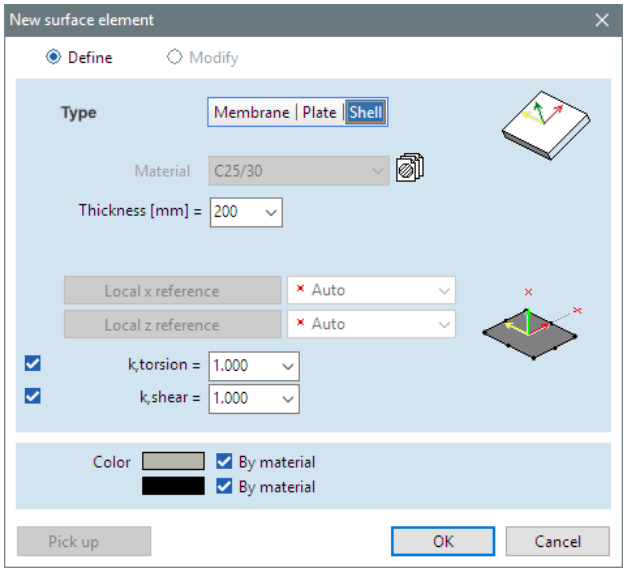
1. Material
2. Thickness
3. Reference (point/vector/axis/plane) for local x axis
4. Reference (point/vector) for local z axis

Allows browsing of the material library to assign a material to the element. The material selected will be added to the material table of the model.

Automatic reference:
The axis of element local directions *x* and *z* can be determined by reference elements, see part **4.9.25 References** or can be set automatically.

The center of the plate elements is displayed on the screen in red.

4.9.13.3 Shell



Shell elements may be used to model structures with behavior that is dependent upon both in-plane (membrane) and flexural (plate) effects.
The shell element consists of a superimposed membrane and plate element. The element is flat, so the membrane and plate effects are independent (first order analysis).

The element can be loaded in its plane and perpendicular to its plane.

The shell internal forces are: n_x, n_y, n_{xy} forces (membrane components), m_x, m_y, m_{xy} moments, and v_x, v_y shear forces (plate components). In addition, the principal internal forces and moments n_1, n_2, m_1, m_2, the angles α_n, α_m and the resultant shear force vSz are calculated.
The variation of internal forces within an element can be regarded as linear.

The following parameters should be specified:

1. Material
2. Thickness
3. Reference (point/vector/axis/plane) for local x axis
4. Reference (point/vector) for local z axis

Allows browsing of the material library to assign a material to the element. The material selected will be added to the material table of the model.

Automatic reference:
The axis of element local directions *x* and *z* can be determined by reference elements, see part **4.9.25 References** or can be set automatically.

The center of the shell elements is displayed on the screen in green.

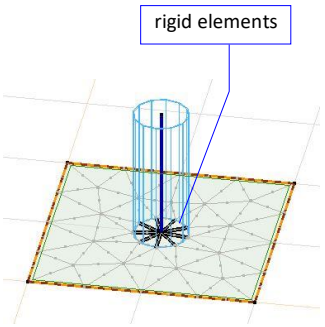
Modifying

Selecting elements of the same type *Modify* will be activated. Checked properties can be changed or picked up from another element. Selecting elements of different types *Define* will be activated.

Pick up>>

See *Pick up* at **4.9.10 Line elements**

The rotational DOF about the axis normal to the surface is a special DOF. It is the 3rd DOF of membrane elements and the 6th DOF of shells. It is coupled with the planar translations through a very small stiffness only. This is necessary to avoid singularities and provide a rotation field compatible with the in-plane translations without increasing the in-plane stiffness of the element by the added DOF. Since the drilling DOF has only a small stiffness a directly loaded node of a membrane or shell element works as if it was released about the axis normal to the surface. Using rigid elements the moment resistance of this node can be increased.



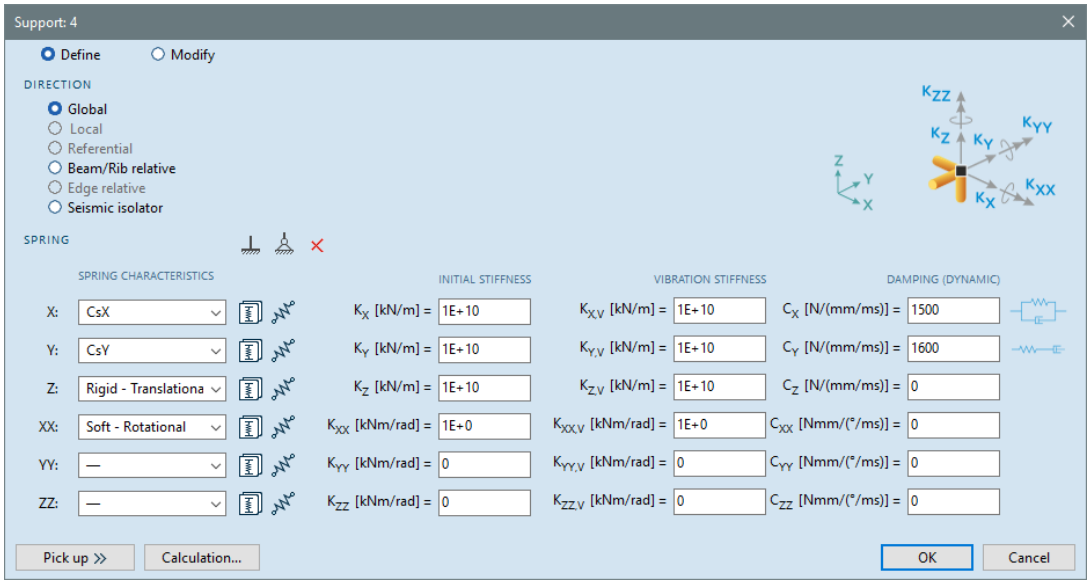
4.9.14 Nodal support

Nodal support elements may be used to model the point support conditions of a structure. Nodal support elements elastically support nodes, while the internal forces are the support reactions. Midside nodes of surface edges cannot be supported. References are used to arbitrarily orient the *x* and z axes of the element.The x axis is directed from a reference point to the attachment node (the node to which it is attached).

You can specify the translational and/or rotational (torsional) properties about the element axes by spring characteristic (see **4.9.21 Spring**) . The values of initial and vibration stiffness and the damping coefficient are also displayed.

Components with a linear spring characteristics have editable fields to enter stiffness and damping directly. Closing the dialog will create a new spring characteristics with the new values automatically.

Nonlinear parameters can be assigned to each direction. To change the characteristics click the spring characteristic icon (see **2.9.1 Spring characteristics**). The values of damping coefficient are visible if at least one of them is not zero.



Spring presets

You can apply the following spring configuration presets

All components are rigid

Rigid for translation, free for rotation

All components are free

The default stiffness values are 1E+10 [kN/m], [kNm/rad].

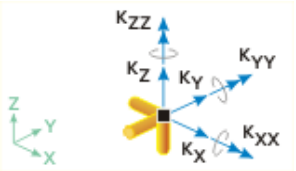
The support elements are displayed on the screen in yellow (translational spring) or orange (rotational spring).

The support can be defined in the following systems:

- *Global*
- *Local*
- *Beam/rib relative*
- *Reference*
- *Edge relative*
- *Seismic isolator*

Global

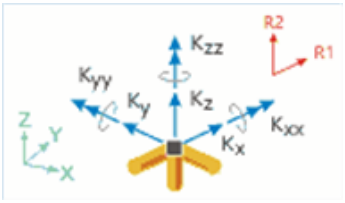
Defines nodal support elements parallel to global coordinate axes. You must select the nodes that are identically supported, and specify the corresponding translational (K_X, K_Y, K_Z) and rotational (K_{XX}, K_{YY}, K_{ZZ}) stiffnesses.



☞ **You can define only one global support for a node. You cannot define nodal support for a midside node of a surface element.**

Local

Minimum two applicable reference is necessary to define local nodal support element. This kind of nodal support is actually a rotated version of the global one. The firs reference is the direction of local x-axis and the second reference together with the first one deterimines the plane of the local z-axis. Beside the references the corresponding stiffness (spring characteristic) must be specified.

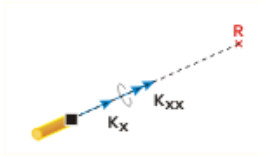


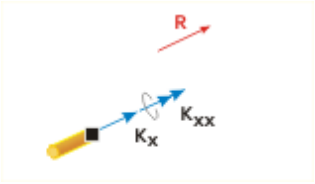
☞ **You can define only one local support for a node.**

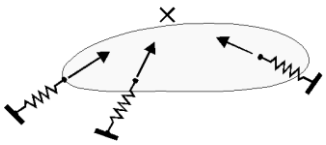
Reference

Defines nodal support elements in the direction of a reference (point or vector). You must select the nodes that are identically supported, and specify the corresponding stiffness (translational K_x, and rotational K_{xx}).

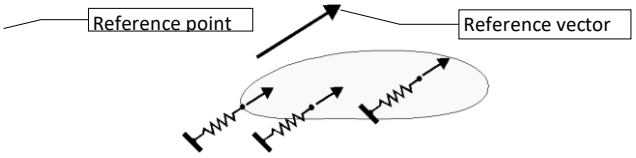
The direction of the reference vector is defined by the element node and its reference point or reference vector in the following way:







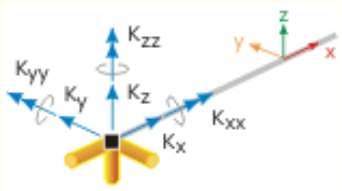
Support elements oriented toward a reference point



Support elements parallel to a reference vector

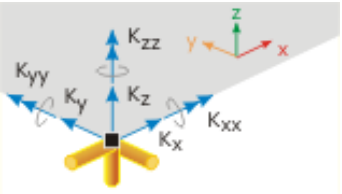
Beam/rib relative

Defines nodal support elements about local coordinate axes of beam / rib elements. You must select the beam / rib elements and the nodes that are identically supported, and specify the corresponding translational (K_X, K_Y, K_Z) and rotational (K_{XX}, K_{YY}, K_{ZZ}) stifnesses.



Edge relative

Defines nodal support elements about local coordinate axes of surface element edges. You must select the surface element edges and the nodes that are identically supported, and specify the corresponding translational (K_X, K_Y, K_Z) and rotational (K_{XX}, K_{YY}, K_{ZZ}) stifnesses.



If one surface is connected to the edge the local coordinate axes of the edge are:

x = the axis of the edge
y = the axis is oriented toward inside of the surface element in its plane
z = parallel to the *z* local axis of the surface element

If two surfaces are connected to the edge the local *z*-axis direction is bisecting the angle of surfaces. The *y*-axis is determined according to the right hand-rule.
If more than two surfaces are connected to the edge and you select one or two of them then support local system will be determined based on the selected surfaces.

Positive displacement elongates the support spring that causes tensile force. Negative displacement shorten the spring that causes compressive force.

Nonlinear characteristics is taken into account only in static or dynamic analysis with material nonlinearity. Linear static, linear dynamic and buckling analyses take into account the *Initial stiffness* only, while vibration analysis takes into account the *Vibration stiffness* only.

The damping coefficients are considered only in dynamic analysis.

Nodal supports appear as brown (K_X, K_Y, K_Z) and orange (K_{XX}, K_{YY}, K_{ZZ}) pegs in 3 orthogonal direction.

Calculation... See **4.9.14.1 Support stiffness calculation**

Modifying Selecting elements of the same type Modifying will be activated. Checked properties can be changed or picked up from another element. Selecting elements of different types Definiton will be activated.

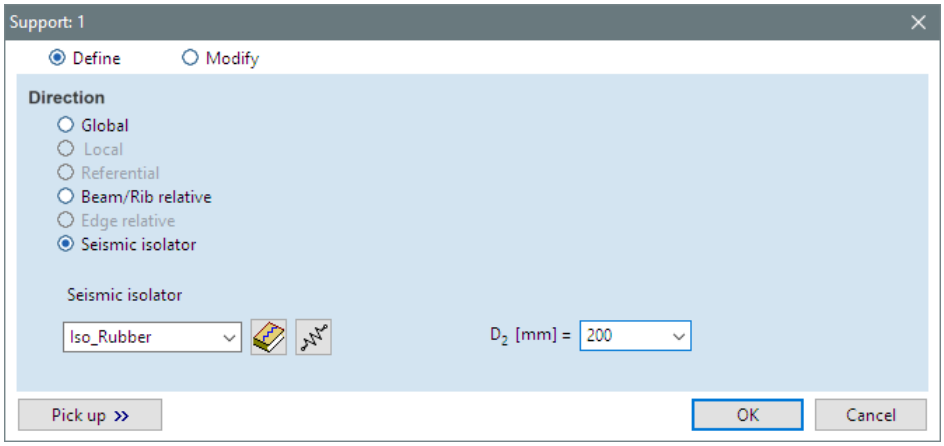
Pick up>> See *Pick up* at **4.9.10 Line elements**

Import support stiffness values from an Excel sheet

The values of the initial stiffness can be copied from an Excel spreadsheet to the node support table via clipboard. In this case supports with linear characteristic are created with initial stiffness components according to the Excel sheet.
The columns of the copied table correspond to the values K_x, K_y, K_z, K_{xx}, K_{yy}, K_{zz} respectively.

	A	B	C	D	E	F
1	2,00E+00	3,42E+02	2,34E+03	2,00E+00	2,22E+02	2,22E+02
2	3,00E+00	3,75E+04	1,21E+05	1,00E+10	4,22E+06	1,00E+10
3	3,52E+04	7,89E+04	1,00E+10	8,85E+04	8,68E+03	1,00E+10
4	2,35E+04	5,35E+05	1,00E+10	7,81E+06	7,57E+06	1,00E+10
5	1,00E+10	8,24E+04	7,68E+05	8,78E+02	2,45E+06	1,00E+10
6	1,00E+10	1,23E+07	1,00E+10	5,00E+00	4,58E+04	1,00E+10
7	1,00E+10	5,45E+04	1,00E+10	5,50E+01	8,75E+06	1,00E+10
8	1,00E+10	1,23E+02	1,00E+10	5,00E+00	4,45E+04	1,00E+10

Seismic isolator



D_2 Maximum design displacement in ULS

The axes of seismic isolators are parallel with the global coordinate axes.
See **3.1.17.2 Seismic isolators**

A node can have either a seismic isolator or a nodal support assigned to it. Defining a seismic isolator to a node deletes all nodal supports from that node. Defining a nodal support to a node deletes all seismic isolators from that node.

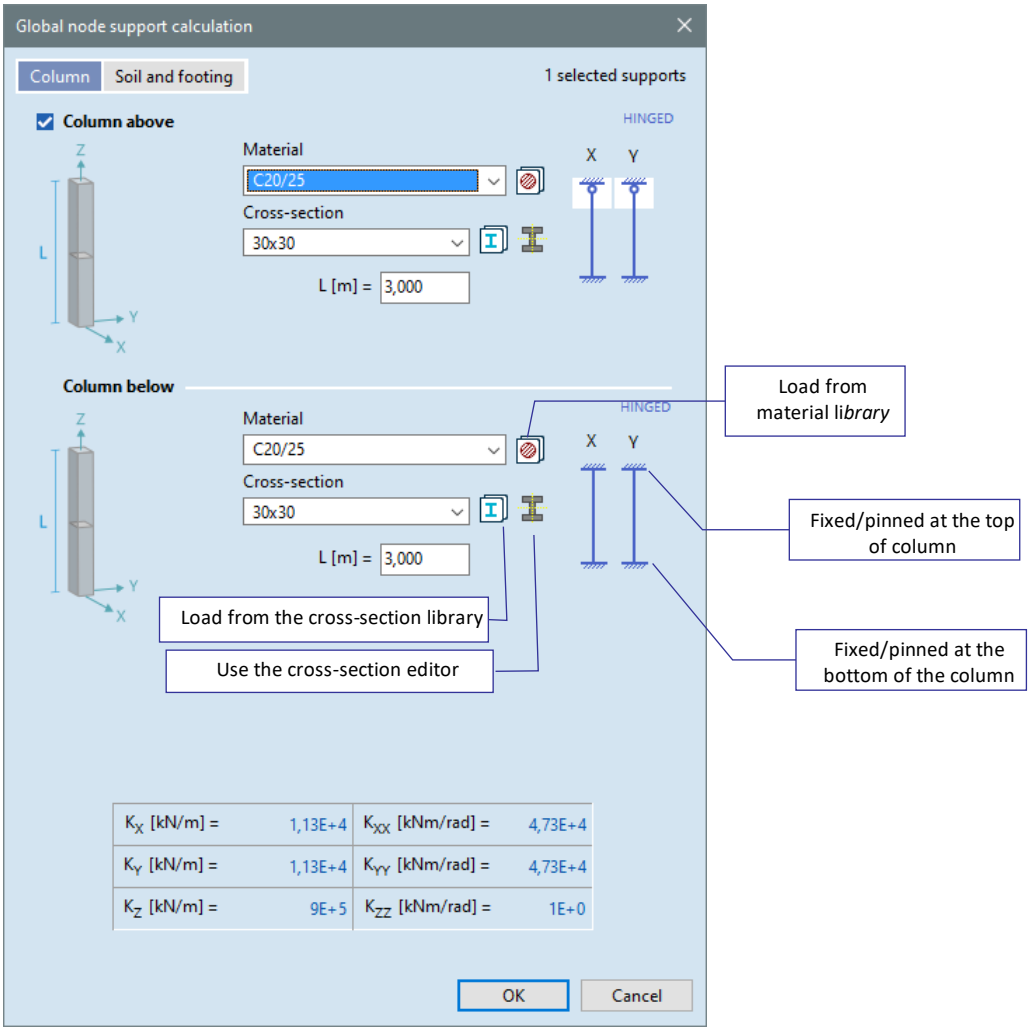
4.9.14.1 Support stiffness calculation

Use the *Calculation...* button on the support dialog to calculate or estimate the support stiffness.
Two methods are implemented *Column* or *Soil and footing*. In the first case the nodal support is modelled by a column of some material and cross-section, in the second by a pad footing on a soil with a specified soil profile.

4.9.14.1.1 Calculation from column parameters

The support stiffnesses are determined based on the end releases, material, and geometry of the column. Using the parameters, spring characteristics are generated by the program.

Calculating nodal support stiffness, a column below and a column above the node can be specified separately. These column parameters can also be used in punching analysis (especially in the case of intermediate slabs). The columns and walls modeling the supports can be displayed and the cursor can identify them.



4.9.14.1.2 Estimating support stiffness from soil profile and footing parameters

When selecting *Soil and footing* the program estimates the support stiffness in global Z direction (K) based on footing parameters and a soil profile.

☞ **If the local directions of the support are significantly different from the global directions (none of the local directions is closer than 10° to the global Z direction), the support stiffness cannot be calculated using this method.**

Two calculation methods are available: *From estimated load,* or *From settlement at the ultimate bearing capacity*.

From estimated load

The program determines the settlement under the footing due to an estimated central compressive force F on the ground, then calculates an estimated support stiffness as $K = F/s$.
The settlement is calculated in the same way as described in **6.5.16.1 Pad footing design.**
The calculation assumes that $F \leq R_{v,d}$, i.e. the load does not exceed the ultimate bearing capacity of the soil under the footing, so the dialog does not allow entering F above the ultimate capacity.

From settlement at the ultimate bearing capacity

If we know the ultimate bearing capacity of the soil under the footing and the associated settlement s_{lim}, the rigidity of the support can be estimated as $K = R_{v,d}/s_{lim}$.

In both cases, it is necessary to determine the ultimate bearing capacity. In the first method it is needed to find the upper limit of F, in the second method it is needed to calculate the stiffness.
The general formula is
For drained conditions: $R_{v,d} = (c'N_c b_c s_c i_c g_c d_c + qN_q b_q s_q i_q g_q d_q + \frac{1}{2}\gamma B' N_\gamma b_\gamma s_\gamma i_\gamma g_\gamma d_\gamma) \cdot A'$
For undrained conditions: $R_{v,d} = \left((\pi + 2)c_u b_c s_c i_c \gamma_c d_c + q\right) \cdot A'$
Where *A'* is the effective area of the footing, *B'* is the effective size of the foundation, γ is the design value of the weight density of the soil under the footing, *c'* is the characteristic value of the cohesion of the soil, *q* is the pressure under the footing. The *N* factors are the bearing capacity factors depending on the soil's angle of shearing resistance *φ'*, the *b* factors are the base inclination factors, *s* factors are the shape factors, the *i* factors are the load inclination factors, the *g* factors are the ground inclination factors, the *d* factors are the depth factors.
Multiple methods are available, they calculate the various *N, b, s, i, γ, d* factors differently: *EN 1997-1 Annex D, Terzaghi (Bowles), Terzaghi (Sprangler-Handy), Meyerhof, Hansen, Vesic.*
D is the depth of the foundation, *θ* is the angle of the load inclination angle to the vertical, *β* is the ground inclination angle.

EN 1997-1 Annex D

The method that takes most factors into account.

$$N_q = e^{\pi \operatorname{tg}\varphi'} \operatorname{tg}^2\left(45° + \frac{\varphi'}{2}\right), \quad N_c = \left(N_q - 1\right) \operatorname{tg}\varphi', \quad N_\gamma = 2\left(N_q - 1\right) \operatorname{ctg}\varphi'$$
$$b_q = b_\gamma = (1 - \alpha \operatorname{tg}\varphi')^2, \quad b_c = b_q - \frac{1 - b_q}{N_c \operatorname{tg}\varphi'}$$
$$s_q = 1 + \frac{B'}{L'} sin\,\varphi', \quad s_\gamma = 1 - 0{,}3\frac{B'}{L'}, \quad s_c = \frac{s_q N_q - 1}{N_q - 1}$$
$$i_c = i_q - \frac{1 - i_q}{N_c \operatorname{tg}\varphi'}, \quad i_q = \left(1 - \frac{H}{V + A'c' \operatorname{ctg}\varphi'}\right)^m, \quad i_\gamma = \left(1 - \frac{H}{V + A'c' \operatorname{ctg}\varphi'}\right)^{m+1},$$

where $m_B = \frac{2 + B'/L'}{1 + B'/L'}$ $m_L = \frac{2 + L'/B'}{1 + L'/B'}$, $m = m_\theta = m_L \cos^2\theta + m_B \sin^2\theta$

Terzaghi (Bowles)

The Terzaghi method is used mostly for centrally loaded shallow foundations, where the base plane depth of the footing, *D* does not exceed the footing size. Inclination of footing or the load is ignored, therefore it uses the following factors only:

$$N_q = \frac{e^{2\left(0{,}75\pi - \frac{\varphi'}{2}\right)\operatorname{tg}\varphi'}}{2\cos^2\left(45° + \frac{\varphi'}{2}\right)}, \quad N_c = \left(N_q - 1\right) \operatorname{ctg}\varphi', \quad N_\gamma = \frac{\operatorname{tg}\varphi'}{2}\left(\frac{K_{pg}}{\cos^2\varphi'} - 1\right),$$

where K_{pg} is the passive earth pressure coefficient, and

$$s_q = 1, \quad s_\gamma = 0{,}8, \quad s_c = 1{,}3$$

Terzaghi (Sprangler-Handy)

The *Terzaghi (Sprangler-Handy)* method uses N factors of *EN 1997-1 Annex D*, and s factors from the *Terzaghi (Bowles)* method, *but*

$$N_\gamma = 1.1\left(N_q - 1\right) \operatorname{tg}(1{,}3\varphi')$$

Meyerhof

Only the factors *N, s, i, d* are taken into account:

$$N_q = e^{\pi \operatorname{tg}\varphi'} \operatorname{tg}^2\left(45° + \frac{\varphi'}{2}\right), \quad N_c = \left(N_q - 1\right) cot\,\varphi', \quad N_\gamma = \left(N_q - 1\right) tan(1{,}4\varphi')$$
$$s_c = 1 + 0{,}2K_p\frac{B'}{L'}, \quad s_\gamma = 1 + 0{,}1K_p\frac{B'}{L'}, \quad s_q = s_\gamma$$
$$i_c = 1 - \theta/90°, \quad i_\gamma = (1 - \theta\ /\ \varphi')^2, \quad i_q = i_c,$$
$$d_c = 1 - 0{,}2\sqrt{K_p}\frac{D}{B'}, \quad d_\gamma = 1 - 0{,}1\sqrt{K_p}\frac{D}{B'}, \quad d_q = d_\gamma,$$

where $K_p = \operatorname{tg}^2(45° + \varphi')$

Hansen

Describes cohesive soils better, using *Mayerhof* N factors but

$$N_\gamma = 1{,}5\left(N_q - 1\right) \operatorname{tg}\varphi'$$
$$b_c = 1 - \frac{\alpha}{147°} \qquad b_q = (1 - \alpha \operatorname{tg}\varphi')^2 \qquad b_\gamma = b_q$$
$$s_c = 1 + \frac{N_q}{N_c}\frac{B'}{L'} \qquad s_q = 1 + \frac{B'}{L'}\operatorname{tg}\varphi' \qquad s_\gamma = 1 - 0{,}4\frac{B'}{L'},$$

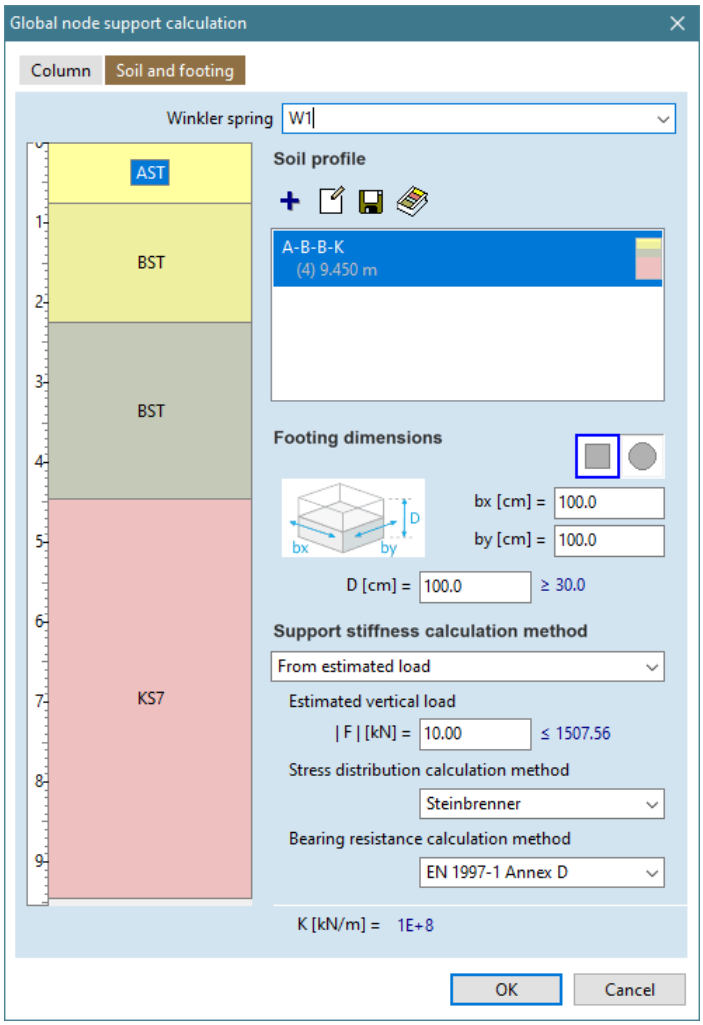
for purely cohesive soils: $s_c = 0{,}2B'/L'$

$$i_c = 1 - \theta/90°, \quad i_\gamma = (1 - \theta\ /\ \varphi')^2, \quad i_q = i_c,$$
$$g_c = 1 - \frac{\beta}{147°}, \quad g_q = (1 - \operatorname{tg}\beta)^2 \; g_\gamma = g_q$$
$$d_c = 1 + 0{,}4k, \quad d_q = 1 + 2\operatorname{tg}\varphi'(1 - \sin\varphi')^2 k,$$

where $k = D/B'$, if $k > 1$ then $k = \operatorname{arc\,tg}\,(D\ /\ B')$

Vesic Factors of the Hansen method are used, but

$$N_\gamma = 2(N_q - 1)\ tg\ \varphi'$$



Winkler spring A combination of a soil profile, the parameters of the footing and the calculation is called a Winkler spring representing an estimated support. Winkler springs are saved into the model under a name. The same Winkler spring can be assigned to multiple supports. Existing Winkler springs can be selected from the topmost combo box.

Soil profile The toolbar above the list of available soil profiles offers the same functions as described in **6.5.16.1 Pad footing design.**

Create new soil profile
A new soil profile can be built using the soil profile editor **(6.5.16.3 Soil profile editor)**

Edit soil profile
Edits the current soil profile in the soil profile editor

Saves the selected soil profile
Saves the selected soil profile to the soil profile library

Import soil profile(s) from the library
The soil profile library **(6.5.16.4 Soil profile library)** is available from any model.

Automatic interpolation If borehole samples have been entered, the soil profile can be determined in any position from the stratigraphic model (see **4.9.3 Add borehole samples – SOIL module**). If *Automatic interpolation* is checked the soil profile interpolated at the support position is used in the calculation.
In case of changes in position of the support or borehole samples, the stiffness of the global Z-direction is updated automatically.

Footing dimensions

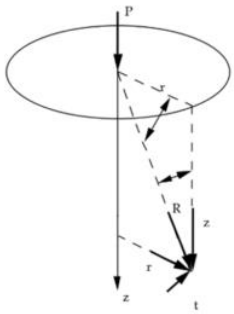
The shape of the footing can be rectangular or circular.
Rectangular footings are defined by the *bx* and *by* sizes, circular footings by the d_p diameter. In both cases the depth of the foundation relative to the zero level of the soil profile *(D)* must be set.

Stiffness calculation

The calculation can be proformed *From estimated load,* or *From settlement at the ultimate bearing capacity*.

From estimated load

In this case an estimated absolute value of the vertical load on the ground must be entered (F). The program calculates the ultimate bearing capacity of the soil profile, to check the upper limit of this load. The method for calculating the stress distribution (Boussinesq, Westergaard, Steinbrenner, 2:1 ratio) and the method for calculating the ultimate bearing capacity (see above) must be selected.

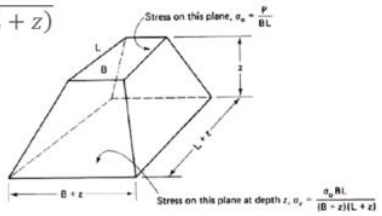


Boussinesq → $\sigma_z = \frac{3F}{2\pi}\frac{z^3}{R^5}$

Westergaard → $\sigma_Z = \frac{F}{z^2 2\pi}\frac{\alpha_\mu}{\left(\alpha_\mu{}^2+\left(\frac{r}{z}\right)^2\right)^{3/2}};\ \alpha_\mu = \sqrt{\frac{1-2\mu}{2-2\mu}}$

Steinbrenner → $\sigma_Z = \frac{\sigma_0}{2\pi} arc\tan\left(\frac{b}{z}\frac{a(a^2+b^2)-2az(r-z)}{(a^2+b^2)(r-z)-z(r-z)^2}\right)+\left(\frac{bz}{b^2+z^2}a\frac{r^2+z^2}{(a^2+z^2)r}\right)$ $\quad r=\sqrt{a^2+b^2+z^2}$

Ratio 2:1 → $\sigma_Z = \sigma_0\frac{BL}{(B+z)(L+z)}$



From settlement at the ultimate bearing capacity

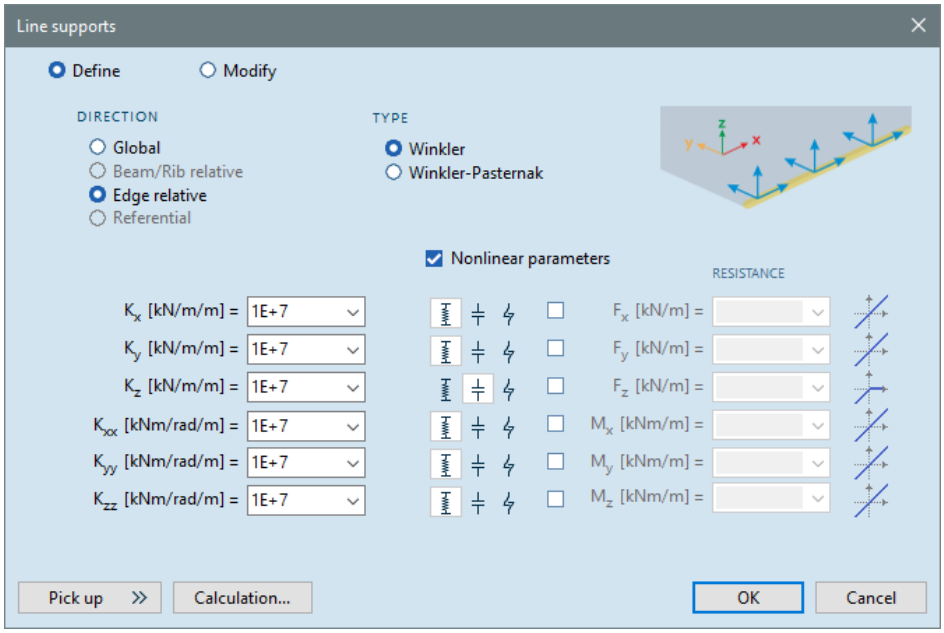
The settlement associated with the ultimate bearing capacity s_{lim} and the method for calculating the ultimate bearing capacity (see above) must be specified.

Once all the parameters for the chosen method have been specified, the estimated support stiffness K is displayed at the bottom of the dialog box. After clicking on the *OK* button, the program assigns a support characteristic to the support component corresponding to the global Z-direction based on the Winkler spring. The name of the generated spring is built from the name of the Winkler spring and the characters _x, _y, _z indicating the component. The program sets the stiffness of this component but leaves the other components unchanged.

Update support parameters based on the actual footings

If results of a static analysis are available, the design of footings can be carried out **(6.5.16.1 Pad footing design).** The parameters (dimensions of the footing and the soil profile) used in footing design can be used to update support stiffness values through the main menu **(Edit / Update support parameters based on the actual footings).** This command recalculates stiffness of the selected nodal and line supports.
If the stiffness calculation was carried out using an estimated load, the programme will also import the force value used in the settlement calculation during foundation design. This operation will of course invalidate the existing results.
See **3.2.22 Update support parameters based on the actual footings**

4.9.15 Line support



Line support elements may be used to model the line support conditions of a structure. Line support elements are elastically supporting beams, ribs, or surface (domain) edges. The internal forces are the support reactions.

Select domain edges, beams or ribs then enter the translational and rotational stiffness values. Nonlinear parameters can be assigned to each direction. To change the characteristics click one the three buttons (bidirectional, compression only, tension only) and set the resistance checkbox and specify a value if necessary.

The support can be defined in the following systems: *Global, Beam/rib relative, Edge relative, Referential* (taking the direction from a reference point or reference vector).
Two types of supports are available: *Winkler* or *Winkler-Pasternak*.

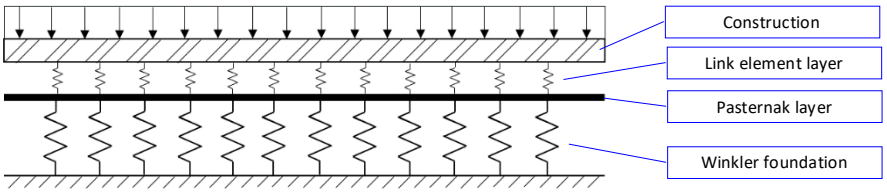
Winkler type supports

These support elements represent a Winkler type elastic foundation. Select domain edges, beams or ribs then enter the translational and rotational stiffness values.

☞ **The default stiffness values are 1E+7 [kN/m/m], or [kNm/rad/m].**

Winkler-Pasternak type supports

These support elements represent a Winkler type elastic foundation against rotation and displacements in local *x* and *y* directions but modeling of the support against displacements in local *z* direction is more sophisticated. A new Pasternak layer modeling the vertical shear stiffness of the soil is placed above the Winkler foundation. It has a constant R_G shear resistance. Above the Pasternak layer a line-to-line link layer is generated with a stiffness of $1000 \cdot K_z$. In linear analysis the only function of this layer is to transfer the *Rz* support reaction between the soil and the structure.



☞ **Both line element or domain edge supports are considered to be structural elements, i.e., when a mesh is created these elements will be automatically divided internally but tables and labels always refer to the structural elements.**

Global

Defines line support elements parallel to global coordinate axes. You must specify the corresponding translational K_X, K_Y, K_Z and rotational K_{XX}, K_{YY}, K_{ZZ} stiffnesses.

Beam/Rib relative

Defines line support elements for beam/rib elements in their local coordinate system acting as an elastic foundation. You must specify the corresponding translational K_x, K_y, K_z and rotational K_{xx}, K_{yy}, K_{zz} stifnesses.

☞ **The beams/ribs with line supports must be divided into at least four elements.**

In addition, the following condition must be satisfied:

$$L \le l_k = \frac{1}{2}\min\left(\sqrt[4]{\frac{4E_xI_z}{k_y}}, \sqrt[4]{\frac{4E_xI_y}{k_z}}\right)$$

where *L* is the beam / rib length.

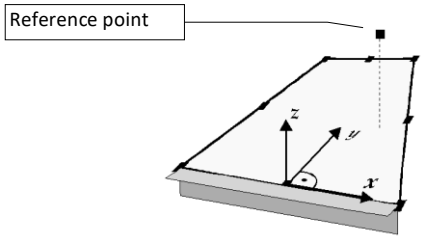
☞ **AXISVM warns you if the condition is not satisfied (by one or more elements). In this case the Winkler's modulus of the defined elements are set to zero, therefore you can divide the elements and repeat the definition/modification process.**
If you specify line supports the internal forces are linearly interpolated between the ends of the element, therefore the division of the elements is required.

Edge relative

Defines edge support elements relative to local coordinate axes of the edges. You must specify the corresponding stiffness (translational K_x, K_y, K_z and rotational K_{xx}, K_{yy}, K_{zz}).
If one surface is connected to the edge the local coordinate axes of the edge are:

- ***x*** = the axis of the edge
- ***y*** = the axis is oriented toward inside of the surface element in its plane
- ***z*** = parallel to the *z* local axis of the surface element

If two surfaces are connected to the edge the local *z*-axis direction is bisecting the angle of surfaces. The *y*-axis is determined according to the right hand-rule.
If more than two surfaces are connected to the edge and you select one or two of them then the local system will be determined from the selected surfaces.



Referential

Defines edge support in a right-handed ***x***, ***y***, ***z*** coordinate system, where
- ***x*** = edge direction
- ***y*** = perpendicular both to the edge and the *z* reference direction
- ***z*** = direction defined by the reference point or reference vector

Positive displacement stretches the support spring creating tensile force. Negative displacement compresses the spring creating compressive force.

Nonlinear behavior

Nonlinear force-displacement characteristics can be specified for this element as follows: compression only (very small stiffness in tension), tension only (very small stiffness in compression). A resistance value can also be entered.

☞ **The nonlinear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account.**

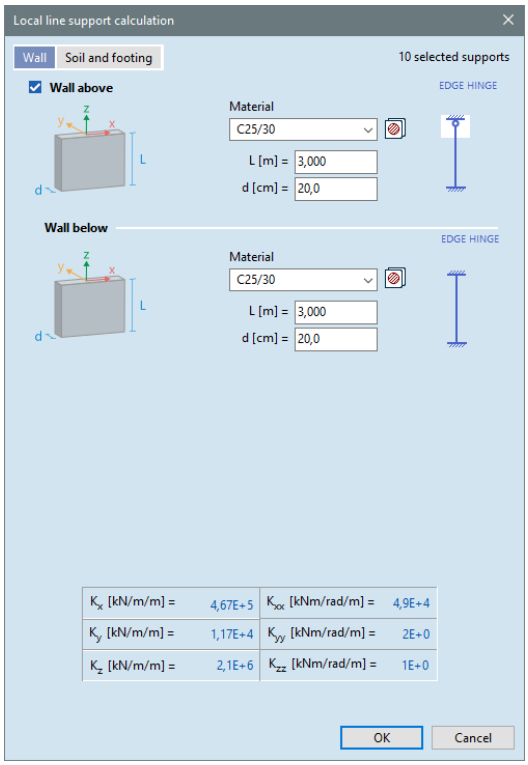
The nonlinear behaviour of Winkler-Pasternak supports in local *z* direction derives from the nonlinear characteristic of the line-to-line link layer. In case of compression-only supports the link layer stretches freely letting the construction rise from the ground. The *Rz* reaction force will be zero but due to the Pasternak layer the Winkler foundation will remain loaded.

Line supports appear as brown (K_X, K_Y, K_Z) and orange (K_{XX}, K_{YY}, K_{ZZ}) lines.

4.9.15.1 Support stiffness calculation

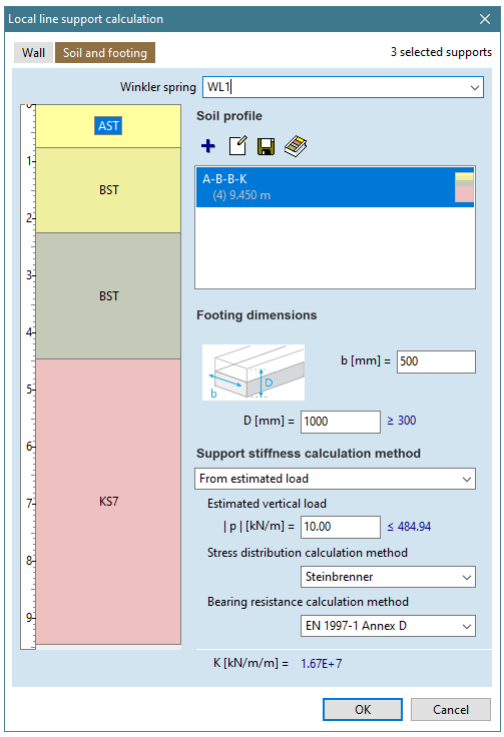
Use the *Calculation...* button on the support dialog to calculate or estimate the support stiffness.
Two methods are implemented *Wall* or *Soil and footing*. In the first case the nodal support is modelled by a wall of some material and geometry, in the second case by a strip footing on a soil with a specified soil profile.

4.9.15.1.1 Calculation from wall parameters



The support stiffness components are determined based on the edge hinges, material, and geometry of the wall. Using the parameters, spring characteristics are generated by the program.

4.9.15.1.2 Estimating support stiffness from soil profile and footing parameters



The support stiffness is estimated in the same way as for nodal supports **(4.9.14.1.2 Estimating support stiffness from soil profile and footing parameters),** but of course the load *p* and the resulting stiffness *K* are also specific values.
The parameters (dimensions of the footing and the soil profile) used in footing design **(6.5.16.2 Strip footing design)** can be used to update support stiffness values through the main menu **(Edit / Update support parameters based on the actual footings).** This command recalculates stiffness of the selected nodal and line supports.
This operation will of course invalidate the existing results.
If the stiffness calculation was carried out using an estimated load, the force value used in the settlement calculation during foundation design will also be imported.

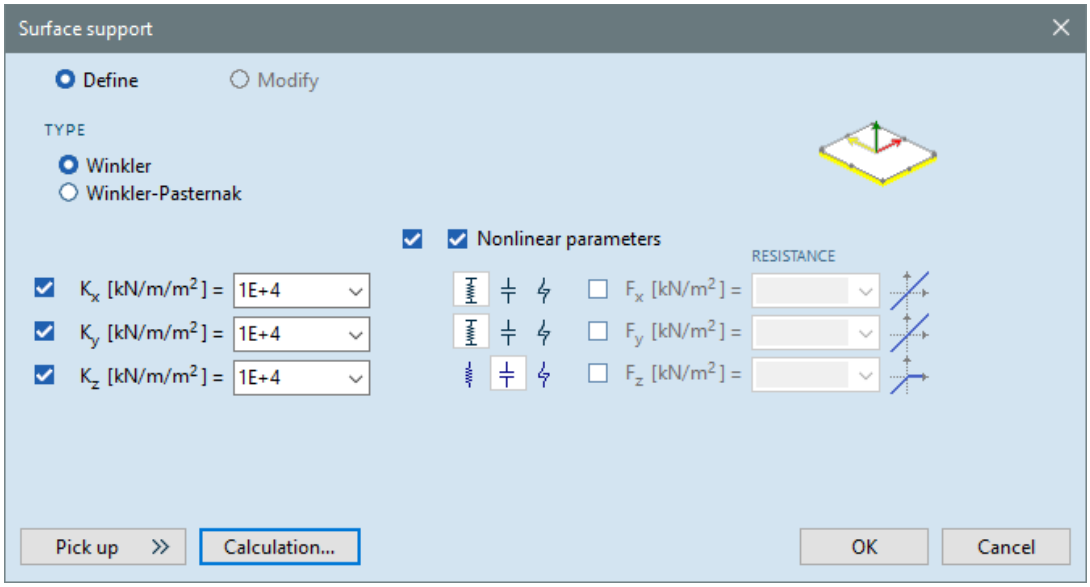
☞ **If the local directions of the support are significantly different from the global directions (none of the local directions is closer than 10° to the global Z direction), the stiffness of the support cannot be calculated using this method.**

Automatic interpolation

If borehole samples have been entered, the soil profile can be determined in any position from the stratigraphic model (see **4.9.3 Add borehole samples – SOIL module**). If *Automatic interpolation* is checked the soil profile interpolated at the support position is used in the calculation.
Since line supports are usually divided into finite elements, support stiffness is not a single value, but can vary along the support.
In case of changes in position of the support or borehole samples, the stiffness of the finite elements in global Z-direction is updated automatically.

4.9.16 Surface support





Surface supports provide support for surface elements and domains, with stiffness component interpreted in the local coordinate system of the surface or domain.

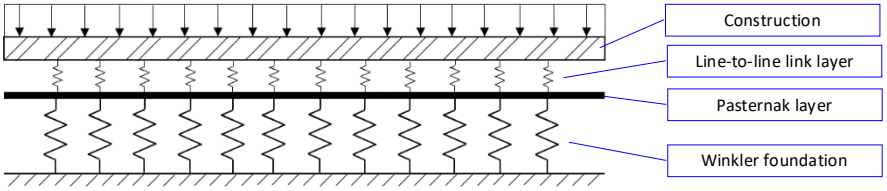
Winkler

Defines a surface support element (Winkler type elastic foundation) to surface elements. You must specify a translational stiffness in the surface element local coordinate system. The surface support behaves identically in tension and compression and is considered constant within the element.
You must specify the support stiffness K_x, K_y, K_z (Winkler's modulus) about the surface element local *x*, *y*, and *z* axes.

☞ **The default stiffness values are 1E+4 [kN/m/m], or [kNm/rad/m].**

Winkler-Pasternak type supports

These support elements represent a Winkler type elastic foundation against rotation and displacements in local *x* and *y* directions but modeling of the support against displacements in local *z* direction is more sophisticated. A new Pasternak layer modeling the vertical shear stiffness of the soil is placed above the Winkler foundation. It has a constant R_G shear resistance. Above the Pasternak layer a line-to-line link layer is generated with a stiffness of 1000 · K_z. In linear analysis the only function of this layer is to transfer the *Rz* support reaction between the soil and the structure.



Positive displacement stretches the support spring creating tensile force. Negative displacement compresses the spring creating compressive force.

Nonlinear behavior

Nonlinear force-displacement characteristics can be specified for this element as follows: compression only (very small stiffness in tension), tension only (very small stiffness in compression), or with resistance (the same stiffness for compression and tension).

☞ **The nonlinear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account.**

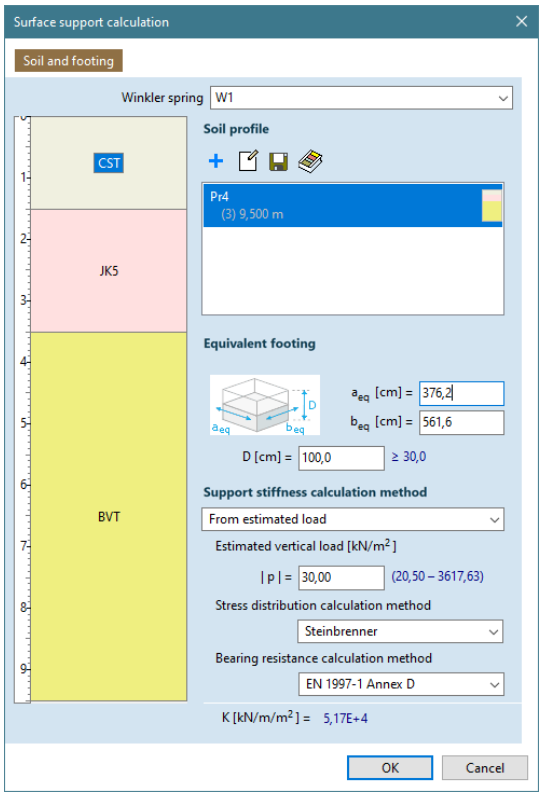
The nonlinear behaviour of Winkler-Pasternak supports in local *z* direction derives from the nonlinear characteristic of the line-to-line link layer. In case of compression-only supports the link layer stretches freely letting the construction rise from the ground. The *Rz* reaction force will be zero but due to the Pasternak layer the Winkler foundation will remain loaded.

Surface supports appear as an orange square-hatched fill.

4.9.16.1 Support stiffness calculation

Use the *Calculation*... button to estimate support stiffness. The surface support is modelled by a soil with a specified soil profile.

4.9.16.1.1 Estimating support stiffness from soil profile and footing parameters



The support stiffness is estimated in the same way as for nodal supports (**4.9.14.1.2 Estimating support stiffness from soil profile and footing parameters**), but of course the load *p* and the resulting stiffness *K* are also specific values per m^2.

The size of the footing does not need to be specified, as the load will be distributed over the entire area of the surface support. The only parameter is the footing base depth *D*.

In case of the *Steinbrenner* stress distribution calculation method, it is required to enter the a_{eq}, b_{eq} sides of an equivalent rectangle.

If only one domain was selected when defining or modifying the surface support, the equivalent rectangle is determined automatically. The equivalent rectangle has the same area as the domain and has the aspect ratio of the minimum bounding rectangle of the domain.

If more than one domain was selected, the minimum and maximum values of the sides can be displayed as a tooltip.

If the selected domain was an internal domain of another one the equivalent rectangle is determined from the containing domain.

Automatic interpolation

If borehole samples have been entered, the soil profile can be determined in any position from the stratigraphic model (see **4.9.3 Add borehole samples – SOIL module**). If *Automatic interpolation* is checked the soil profile interpolated at the support position is used in the calculation.

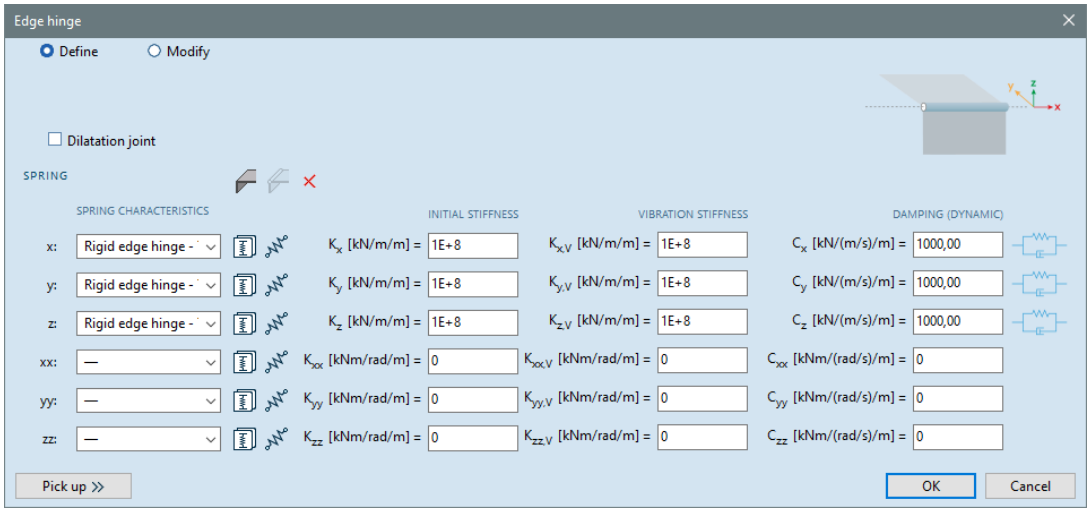
The program computes an interpolated soil profile at the centre of the mesh elements, therefore support stiffness is not a single value, but can vary across the elements of the finite element mesh.

In case of changes in position of the support or borehole samples, the stiffness of the finite elements in global Z-direction is updated automatically.

4.9.17 Edge hinge



Edge hinge can be defined between domain edges or between a rib and a domain edge. Select edge and a domain. Hinge stifness can be defined in the local system of the edge of the selected domain.
The properties can be set in the same way as for a spring element. Choose a characteristic for the components interpreted in the local coordinate system of the edge (see **4.9.21 Spring**). The values of the initial stiffness, the vibration stiffness and the damping factor are displayed and can be edited. If values are modified and the dialog is closed with OK, new spring characteristics are created with the modified values. The damping factor is only displayed if it is non-zero for at least one component. The quick setup of characteristics is also available.



Spring presets

You can apply the following spring configuration presets



All components are rigid

Rigid for translation, free for rotation

All components are free

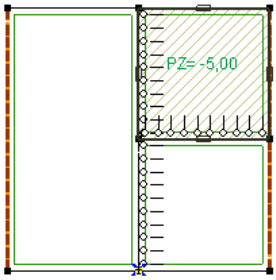
The option *Dilatation joints* is described in the next subsection.

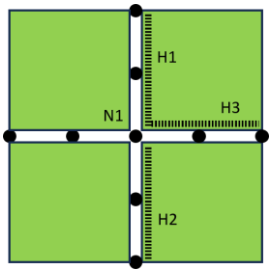
4.9.17.1 Dilatation joints

Dilatation joint

For the purpose of numerical analysis AXISVM converts the structural model to a pure finite element model. During this process the structural elements are replaced by one or more finite elements. For example, edge hinges are replaced by line-to-line spring elements. Hinged surfaces are disconnected from the hinged edge by doubling nodes and lines and inserting line-to-line spring elements between the original and the new edge.
It is also possible to model dilatation joints with edge hinges. The distinction between dilatation joints and other edge hinges is only necessary if there is at least one node in the model where both types of edge hinges are connected. In this case, the edge hinges should be processed in two steps. First, the model should be dissected along the dilatation joints to create structural units separated by them. Then, the other edge hinges should be inserted dissecting their respective structural units only.
The effect of this method is illustrated in the following model. The slab is divided by edge hinges. The walls are modelled by line supports. The upper right plate is subjected to uniformly distributed load. Edge hinges H1 and H2 connect the plates on the left to the plates on the right and have stiffness in the local x direction only. Edge hinge H3 placed between the plates on the right have stiffness in all three directions.
If the *Dilatation joint* option is off, edge hinges H1 and H3 are connected to each other to form a continuous separation line. Thus, edge hinge H3 is also connected to the centre node. Therefore, the left plate is also loaded at the centre point by the upper right plate through edge hinge H3 having stiffness in all three directions. In contrast, when the *Dilatation joint* option is on, edge hinges H1 and H2 are connected to each other to model a dilatation joint.

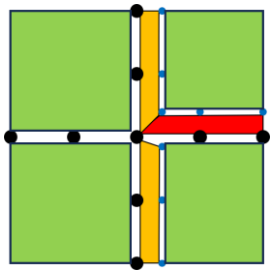
This way edge hinge H3 connects only the right plates to each other. Thus, there is no connection in vertical direction between the left and right sides of the model.



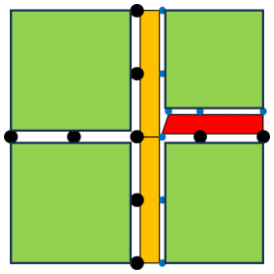


Centre of the structural model

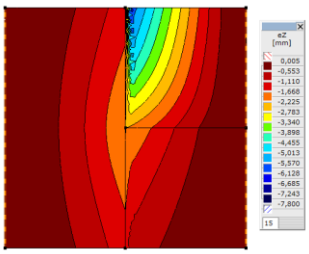
Original node
New node
Surface element
Edge hinge – Kx
Edge hinge – Kx, Ky, Kz



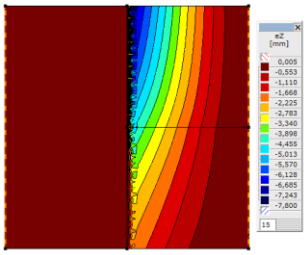
FE model without dilatation



FE model with dilatation



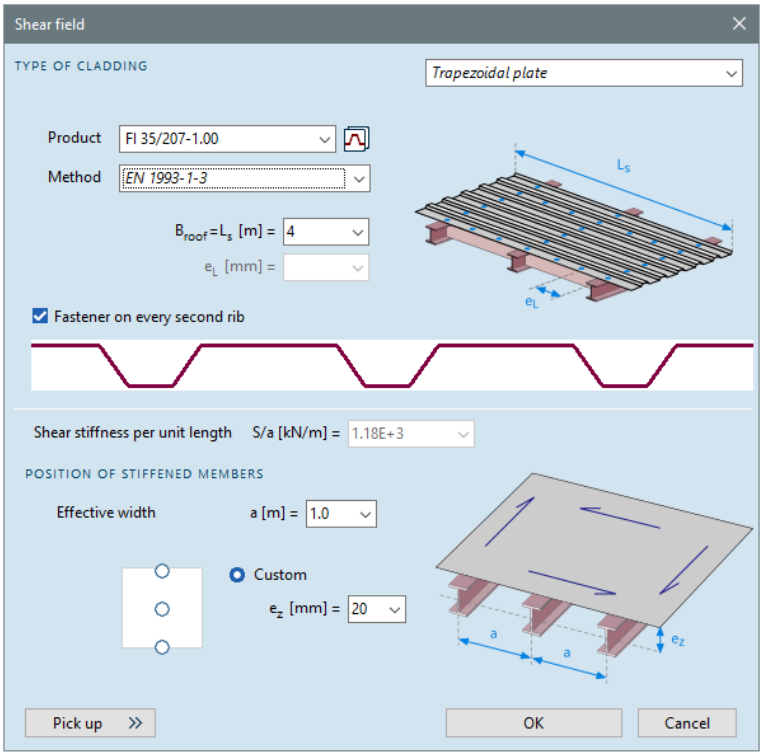
eZ translation result without dilatation



eZ translation result with dilatation

4.9.18 Shear field

Using a shear field, we can model the supporting effect of cladding on beam elements without having to include the cladding itself in the finite element model.



Type of cladding — The following types of cladding are available *Trapezoidal sheet*, *Sandwich panel*, *Liner tray*, *Custom*.

Shear stiffness per unit length — Provided the correct data is entered, the form calculates the shear stiffness per unit length.

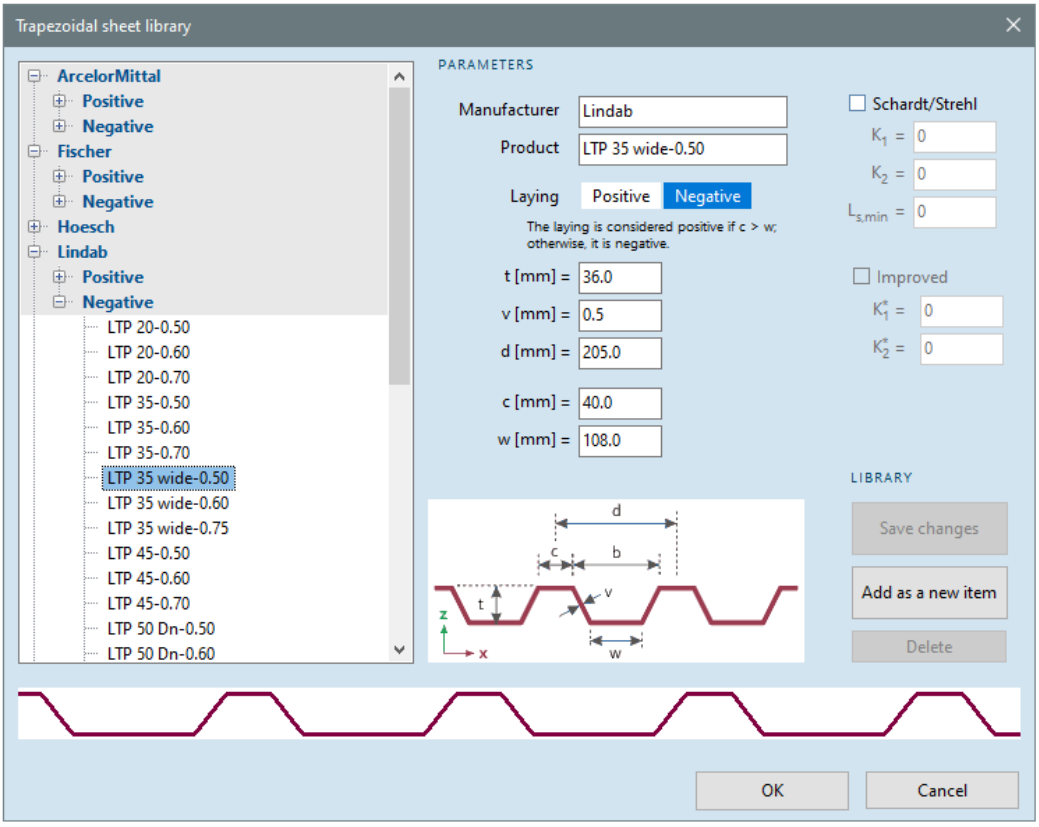
Position of stiffened members — In this panel, the user can specify the effective width of the shear field and the eccentricity, which in all cases is to be interpreted in the local z-direction of the beam element.

4.9.18.1 Trapezoidal sheet

Product — The user can select a trapezoidal sheet product already listed in the database from the drop-down menu. To add a new sheet type to the model, click the *Trapezoidal sheet library* button.

Fastener on every second rib — If this checkbox is selected, the program takes into account 20% of the total shear stiffness, in accordance with EN 1993-1-3.

4.9.18.1.1 Trapezoidal sheet library



In the tree on the left, you can select a trapezoidal sheet from the manufacturers' catalogue; the *Parameters* panel shows the characteristics of the selected element and allows definition of custom elements. For factory elements, the parameters can be modified, but the values in the catalogue cannot be overwritten. *Library* operations:
Add as a new item: Adds a trapezoidal sheet with the specified parameters to the library as a new custom item.
Save changes: The library parameters can only be modified for custom trapezoidal sheets.
Delete: Only custom items can be deleted.

Parameters:

Manufacturer	The name of the manufacturer can be entered or amended.
Product	The name of the product can be entered or amended.
Laying	The orientation can be set to positive or negative. Changing laying swaps of the *c* and *w* parameters.
t	Profile height
v	Sheet thickness
d	Rib spacing
c	Upper flange width
w	Lower flange width
Schardt/Strehl	If the Schardt/Stehl parameter, such as K_1, K_2, $L_{s,min}$, are known, the user can assign them to the trapezoidal sheet by checking the checkbox.
Improved	If the improved Schardt/Stehl parameters, such as K_1^*, K_2^*, are known, the user can assign them to the trapezoidal plate by checking the checkbox.

Method

The user can choose between the methods defined in *EN 1993-1-3, Schardt/Strehl*, and *improved Schardt/Strehl*. The *EN 1993-1-3* method is always active, while the other two methods are active only if the appropriate parameters have been defined for the profile.

EN 1993-1-3 If this method is selected, the shear stiffness is determined based on Equation (10.1b) in EN 1993-1-3.

$$S/a = \frac{\sqrt{v^3}}{t}\left(50 + 10\sqrt[3]{B_{roof}}\right)\left[\frac{kN}{m}\right]$$

v	Sheet thickness (in *mm*)
t	Profile height (in *mm*)
B_{roof}	Shear field length in the direction of ribs (in *mm*)

Schardt/Strehl This method follows the calculation method specified in the DIN 18807-1 standard:

$$S/a = \frac{10^4}{K_1 + \frac{K_2}{L_s}}$$

K_1, K_2	Stiffness parameters, based on experiments
L_s	Shear field length in the direction of ribs

Improved Schardt/Strehl Improved version of the previous method, with two additional parameters:

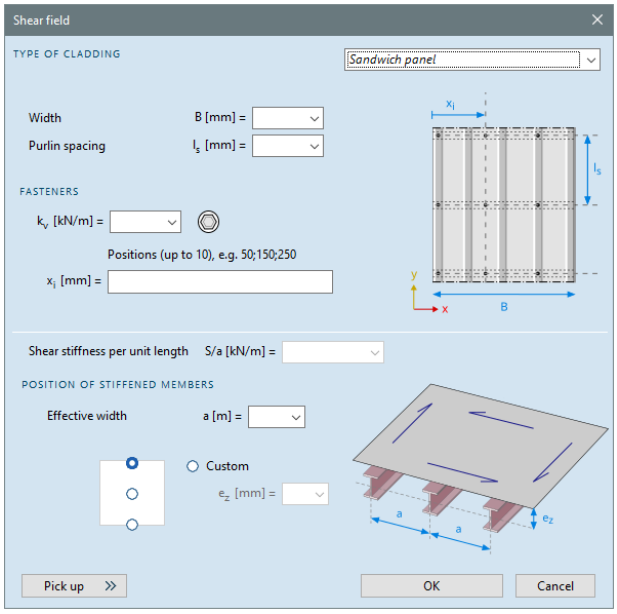
$$S/a = \frac{10^4}{K_1 + K_1^* e_L + \frac{K_2 + K_2^*}{L_s}}$$

K_1, K_2, K_1^*, K_2^*	Stiffness parameters, based on experiments
L_s	Shear field length in the direction of ribs
e_L	Distance between fasteners in the direction of the ribs.

☞ **The basic assumption of the Schardt/Strehl models is that the shear field is supported on all four sides by interconnected edge supports wich can withstand the resulting shear forces. In contrast, the model according to EN 1993-1-3 does not require the shear field to be supported on all four sides; the calculation can also be applied in cases where the trapezoidal sheet is supported on only two sides.**

Fastener on every second rib If this checkbox is selected, the program takes into account 20% of the total shear stiffness, in accordance with EN 1993-1-3.

4.9.18.2 Sandwich panel



The program performs the calculation in accordance with the *European Recommendations on the Stabilization of Steel Structures by Sandwich Panels*. These recommendations are consistent with Chapter 11.4.1.4 of the second-generation Eurocode EN 1993-1-3:2024:

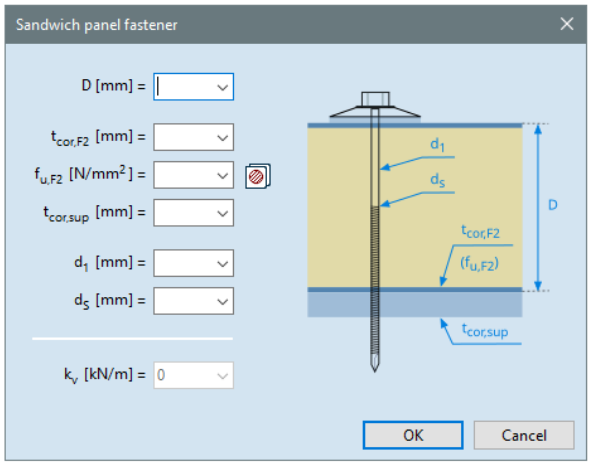
$$S/a = \frac{k_v}{Bl_s} \sum_{k=1}^{n_k} (x_i - c)^2$$

B	Width of the sandwich panel
l_s	Purlin spacing
x_i	Fastener positions, separated by semicolons, a maximum of 10 elements may be specified; 'c' denotes the centre of gravity of the connection.
k_v	Translation stiffness of the fastener

The stiffness in kN/m can be estimated on the basis of the following table:

Core thickness of internal face [mm]	Material of the internal face		
	S220GD	S280GD	S320GD
0.40	1600	1900	2000
0.50	2000	2300	2500
0.63	2400	2900	3100
0.75	2800	3300	3600

The stiffness can also be calculated by clicking the button.



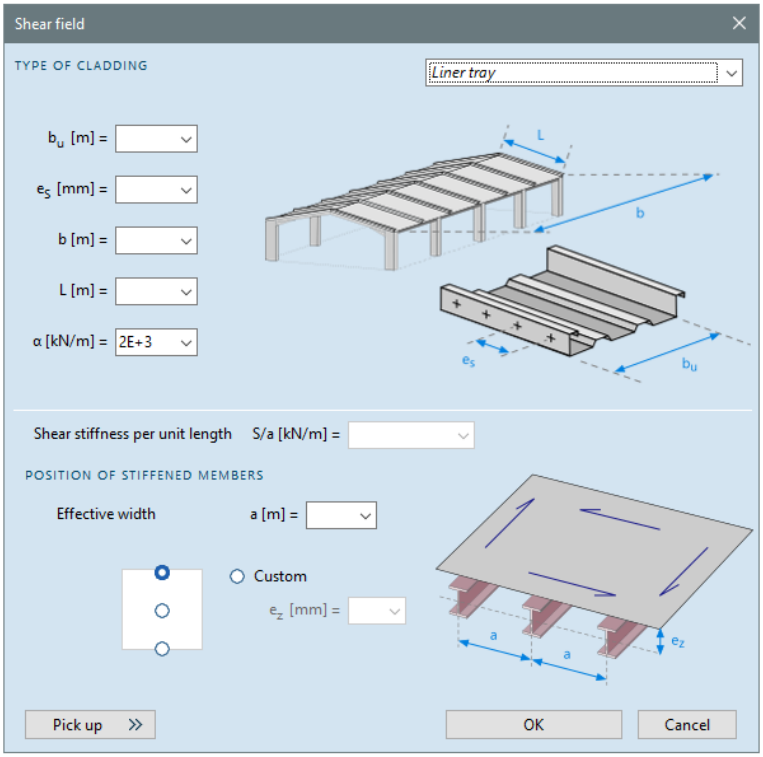
D	Thickness of panel at the point of fastening
$t_{cor,F2}$	Core thickness of internal face
$f_{u,F2}$	Tensile strength of the internal face Click on the icon to pick it up from the *Material library*
$t_{cor,sup}$	Core thickness of the supporting structure
d_1	Minor diameter of threaded part of the fastener
d_S	Diameter of the unthreaded shrank

The calculation applies within the following ranges:

$5.5 \text{ mm} \leq d \leq 8.0 \text{ mm}$
$40 \text{ mm} \leq D$
$0.40 \text{ mm} \leq t_{cor,F2} \leq 1.00 \text{ mm}$
$1.50 \text{ mm} \leq t_{cor,sup} \leq 10.0 \text{ mm}$

☞ **The calculation assumes that the connection stiffness is the same for all elements. If it differs, specify the shear field using the Custom option.**

4.9.18.3 Liner tray



☞ **The program calculates the shear stiffness of the liner tray in accordance with EN 1993-1-3 (10.24):**

$$S/a = \frac{\alpha L b_u}{e_s}$$

b_u	Width of the wide flange
e_s	Fastener spacing
b	Overall width of the shear field
L	Shear field length in the direction of the liner tray
α	Stiffness factor, default value is 2000 kN/m

4.9.18.4 Custom

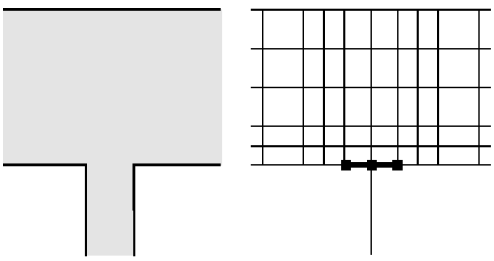
Selecting Custom type, the shear stiffness can be specified directly. This option covers cases where the supporting effect does not correspond to any of the three cases discussed above, for example, in the case of OSB boards.

4.9.19 Rigid elements

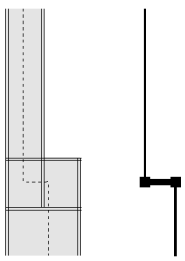
Rigid elements may be used to model parts with a rigid behavior relative to other parts of the structure. The elements can be defined by selecting the lines that connect its nodes. The selected lines that have common nodes define the same rigid element. There is no limit to the number of nodes of any element.

☞ **The degrees of freedom of the nodes of a rigid element cannot be constrained (fixed).**

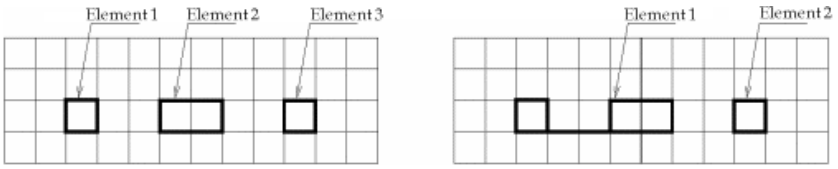
Modeling membrane–beam element connection:



Modeling eccentric beam–beam element connection:



Define Lets you define rigid elements. You must select the lines that connect the nodes attached to rigid elements. Recall that the lines with common nodes define the same rigid element.



You can join or split rigid elements using the modify command.
If you select lines that connect nodes of different rigid elements, the elements will be joined. If you deselect lines of rigid elements interrupting their continuity, the respective elements will be split.

☞ **A finite element cannot have all of its lines assigned to the same rigid body.**

If we want to calculate the mass of the body in a vibration analysis, place a node to the center of gravity, connect it to the body and make this line a part of the rigid body. Assign the mass of the body to this node.

The geometrical stiffness of the element is taken into account in buckling analysis and geometrically nonlinear simulations, see [43].

The rigid elements are displayed on the screen with thick black lines.

4.9.20 Diaphragm

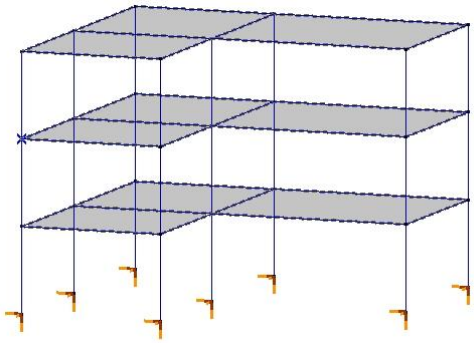


Using diaphragms means simplifying the model. Diaphragms are special rigid bodies where the relative position of the element nodes remain constant in a global plane. Diaphragms considerably reduce the amount of calculation. It can be an advantage running vibration analysis of big models. Diaphragms can represent plates totally rigid in their planes.

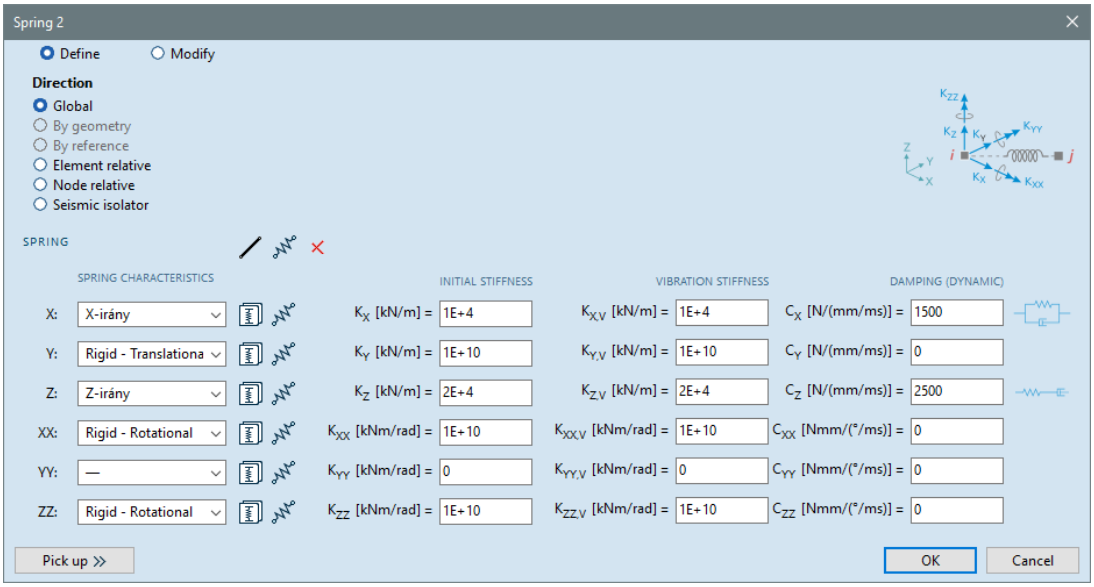
Definition Select lines to define diaphragms. Each set of connecting lines will form a diaphragm.

Diaphragms are displayed as dashed thick purple lines.

After definition you must set the working plane of the diaphragm. The relative position of element nodes remains constant in this plane. For rigid plates in the X-Y plane choose XY.
After changing, deleting or adding diaphragms all diaphragms will be checked again to find connected groups of lines. Unconnected groups will form separate diaphragms.



4.9.21 Spring



Spring presets

You can apply the following spring configuration presets



All components are rigid

Rigid for translation, free for rotation

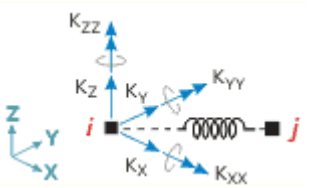
All components are free

The spring element connects the degrees of freedoms of two nodes of the model. The element has its own coordinate system. You can specify the translational (*X, Y, Z*) and/or rotational (*XX, YY, ZZ*) spring characteristics. The values of initial and vibration stiffness and the damping coefficient are also displayed. Nonlinear parameters can be assigned to each direction. To change the characteristics click the spring characteristic icon (see **3.1.17 Spring characteristics library**)

Springs can be defined in the following systems: *Global, By geometry, By reference, Element relative, Node relative,* or as a *Seismic isolator.*

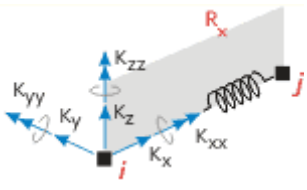
Global

Defines spring elements parallel to global coordinate axes. You must select the line and specify the corresponding translational (K_X, K_Y, K_Z) and rotational (K_{XX}, K_{YY}, K_{ZZ}) characteristics.



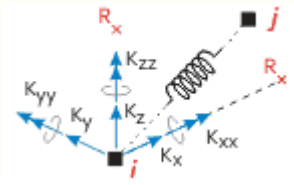
By geometry

Specifying a spring element based on geometry, the direction of the selected line will be the local x-axis of the spring element. The plane containing the local z-axis is determined by reference z and the local x-axis.
The model requires at least one applicable reference to be specified.



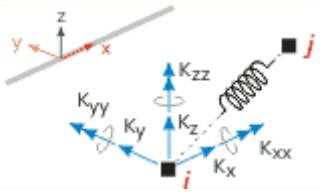
By reference

The spring element defined by reference can be specified by two references. The direction of the local x-axis is assigned by x-reference. The z-reference, together with the local x-axis determines the plane in which the local z-axis is located. After selecting the line, references and spring characteristics must be specified.
The model requires at least two applicable references to be specified.



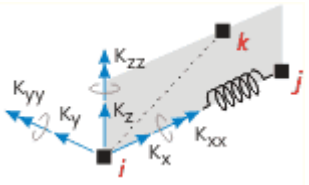
Element relative

The local coordinate system of the spring element is aligned to the local coordinate system of the selected beam/rib. You must select the required line and the beam/rib element and finally the spring characteristics for each direction. The model must include at least one beam or rib element to define this kind of spring element.



Node relative

Spring element with node relative is determined by two applicable nodes in addition to the selected line. The local x-reference node with the lower number node of the spring line specifies the local x-axis of the spring. The local z-reference node with the local x-direction determines the plane that contains the local z-axis.
The model requires at least two applicable nodes to be specified.



Spring characteristics library

Spring characteristic can be imported from *Spring characteristics library* see

. The program automatically filters out the spring properties corresponding to the degree of freedom.

Spring characteristic editor

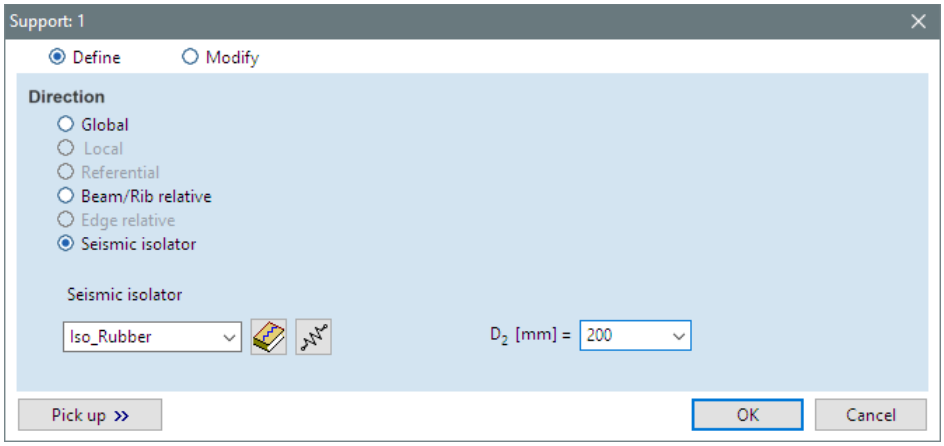
The components can be edited with the spring characteristic icon.
See **3.1.17 Spring characteristics library**.
Components with a linear spring characteristics have editable fields to enter stiffness and damping directly. Closing the dialog will create a new spring characteristics with the new values automatically. The values of damping coefficient are visible if at least one of them is not zero.

☞ **Nonlinear spring characteristics are taken into account only in case of materially nonlinear static and materially nonlinear dynamic analyses.**

Initial stiffnesses are taken into account in materially linear static, materially linear dynamic and buckling analyses even if the spring properties are nonlinear. In case of Vibration I/II the vibration stiffnesses are taken into account, that values remain constant during the analyses.
The damping coefficients are considered only in dynamic analysis.

The spring elements are not real structural elements. They connect the nodal DOFs of the end points one by one. There is no geometric coupling between the different DOFs (translation-rotation, force-moment).

Seismic isolator



D_2 Maximum design displacement in ULS

The axes of seismic isolators are parallel with the global coordinate axes.
See **3.1.17.2 Seismic isolators**

Seismic loads
SE1 module

During different analyses the seismic isolators are taken into account in the following way.
The seismic isolator system is considered as a single degree of freedom model, with the following parameters:

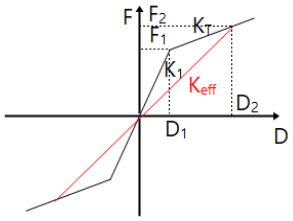
T_{is}	the natural period of the system
ξ_{esi}	damping ratio of the system
K_{esi}	effective horizontal stiffness
K_v	effective vertical stiffness
dc_x, dc_y	the coordinates of the centre of the isolator system on the X-Y plane

The effective horizontal stiffness and damping ratio

In order to determine the effective horizontal stiffness of the nonlinear isolators one must know the ultimate limit displacements of the isolators (D_2, see **4.9.14 Nodal support, 4.9.21 Spring**). As displacements of the isolators depend on the seismic load the solution can only be provided iteratively. At the end of the procedure the relative difference between the design displacement and the actual maximum displacement must be less than 5%.

The effective stiffnesses and damping ratios can be determined in the following way for different type of isolators:

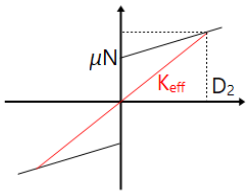
Rubber



Parameters:

$$D_1 = F_1/K_1$$
$$F_2 = F_1 + K_T(D_2 - D_1)$$
$$K_{eff} = F_2/D_2$$
$$\xi_{eff} = \frac{2}{\pi}\left(\frac{F_1}{F_2} - \frac{D_1}{D_2}\right)$$

Curved surface slider



$$K_{eff} = N\left(\frac{1}{R} + \frac{\mu}{D_2}\right)$$
$$\xi_{eff} = \frac{2}{\pi}\left(\frac{1}{1 + \frac{D_2}{\mu R}}\right)$$

where N is the normal force on the isolator element

Other

The user defines the effective stiffness (E_{xy}) and ratio (ξ) (see **3.1.17.2 Seismic isolators**).

The vertical and horizontal stiffness of the isolator system

$$K_v = \sum E_{z,i}, K_{esi} = \sum K_{eff,i}$$

The natural period of the isolator system

$T_{is} = \frac{2\pi}{\sqrt{K_{esi}/M_{is}}}$ where M_{is} is the mass of the isolated structure

Damping ratio of the system

$$\xi_{esi} = \frac{\sum \xi_{eff,i} K_{eff,i}}{K_{esi}}$$

The stiffness centre of the isolator system

$$dc_x = \frac{\sum x_i K_{eff,i}}{K_{esi}},\ dc_y = \frac{\sum y_i K_{eff,i}}{K_{esi}}$$

During response spectrum analysis isolators are considered in the following way (available only if NTC, Italian code is selected)

Horizontal spectrum

Elastic spectrum, where

$$\xi = \begin{cases} 5\%, & T < 0.8T_{is} \\ \xi_{esi}, & T \geq 0.8T_{is} \end{cases}$$

- For SLO, SLD limit states

$$\eta = \max\left(0.55, \sqrt{\frac{10}{5+\xi}}\right)$$

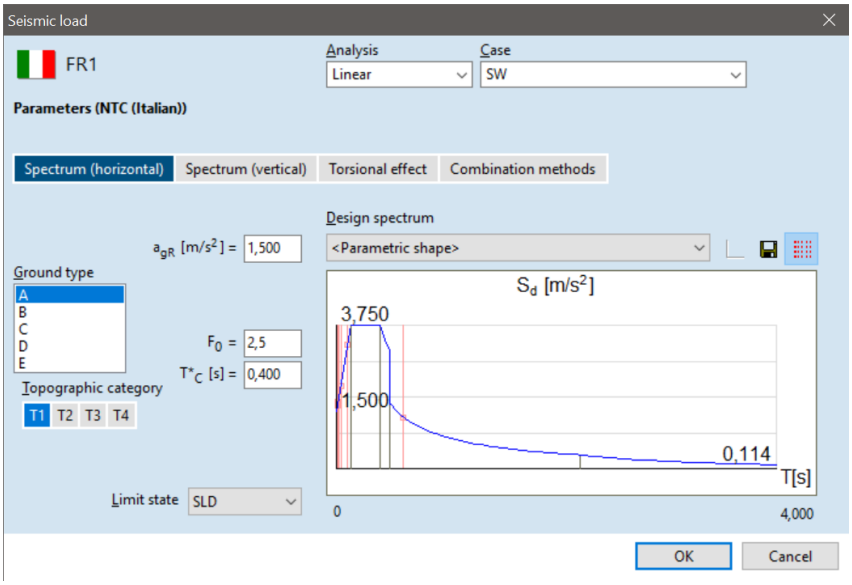
- For SLV, SLC limit states

$\eta = \max\left(\frac{2}{3}, \sqrt{\frac{10}{5+\xi}}\right)$, with this modification the equivalent behaviour factor cannot be larger than 1.5.

Vertical spectrum

Elastic spectrum, where $\xi = 5\%$.

The horizontal spectrum if seismic isolators are applied:



Linear static analysis

In this case the program takes into account the effective stiffness of the rubber isolator ($K_{eff,i}$), which is introduced above. In the case of curved surface slider the effective stiffness is used only if earthquake load case is defined. However, in that case it is applied to all load cases, including non-earthquake load cases. If no earthquake load case is defined the horizontal stiffness is set to 1E10 kN/m.

Nonlinear static analysis

For pushover analysis the rubber isolators and the curved surface sliders are considered with a bilinear constitutive law. For curved surface sliders the initial stiffness is $K_1 = 1E10\,\mathrm{kN/m}$, the limit force is $F_1 = \mu N$ and the tangent stiffness is $K_T = N/R$. For non-pushover analysis the curved surface sliders are considered with linear behavior with a stiffness of 1E10 kN/m.

Vibration

For first and second order analysis the effective stiffness of the isolator ($K_{eff,i}$) is taken into account, which is introduced above. If the model contains isolator elements, first a linear static analysis is carried out to determine the mass of the isolated system.

Buckling

In buckling analysis the elements are taken into account with initial stiffness (K_1). In the case of curved surface slider the horizontal stiffness is set to 1e10 kN/m.

Dynamic

Without following nonlinear behaviour of materials and finite elements

The effective ($K_{eff,i}$) stiffnesses of isolators are considered, which are introduced above. Furthermore, the program takes into account the dissipated energy due to the elastic-plastic constitutive behaviour in the i. isolator element:

$$c_i = \frac{W_{d,i}}{\pi\omega {D_{2,i}}^2}$$

where $\omega_{is} = \frac{2\pi}{T_{is}}$, $W_{d,i}$ is the dissipated energy during one cycle, which is $W_d = 4(F_1 D_2 - F_2 D_1)$ for rubber element and $W_d = 4\mu N D_2$ for slider element.

With following nonlinear behaviour of materials and finite elements

Rubber bearing isolators and curved surface sliders are taken into account with a bilinear constitutive law.

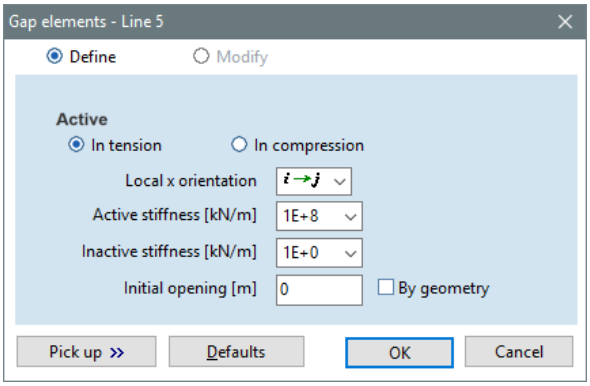
Note: if a curved surface slider is defined in the model, the analysis begins with a linear static analysis in order to determine the normal force on the slider element for the selected load case. The program neglects the oscillation of the pressure force on the isolator element.
The nonlinear behaviour of the isolators are represented by separate springs in *x* and *y* direction. No interaction is considered.

4.9.22 Gap

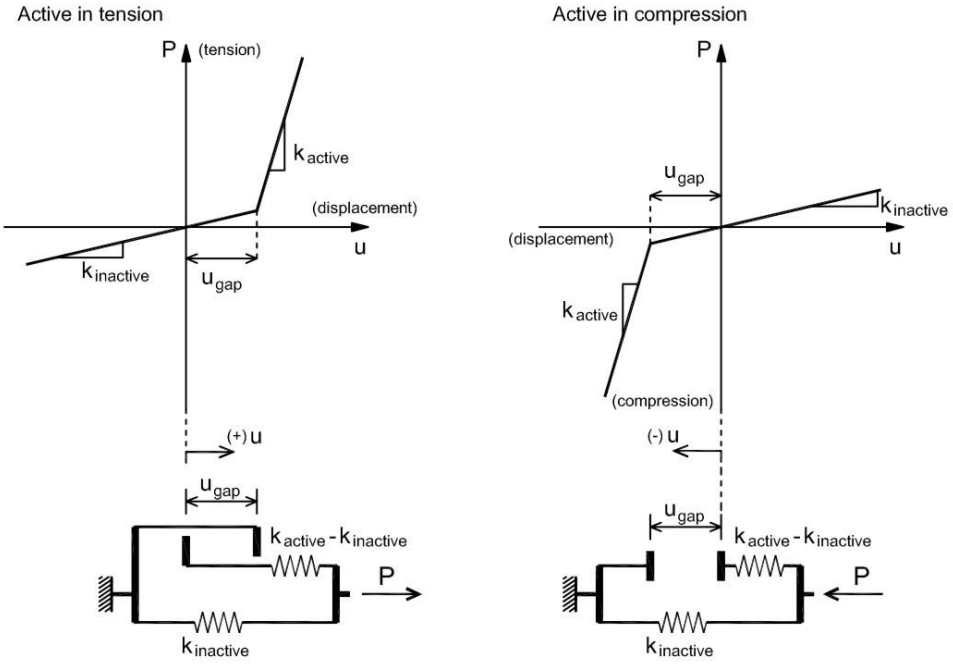
The gap element is used to model point-to-point contact.

The element has two states:

- one ***active***, when its stiffness is *several orders of magnitude higher* than in inactive state (simulates that a contact is achieved)
- one ***inactive***, when it has a small stiffness value (simulates that no contact is achieved).



As the inactive state has a small but nonzero stiffness a gap element is an approximate model of this type of connection. But using this element the stiffness matrix remains a band matrix and any number of gaps can be used in a model.



The gap element is a nonlinear element that can impose difficulties to the solution of the nonlinear problem, due to large changes of element stiffness when it changes status (active/inactive).
If numerical convergence problem occurs, it may be desirable to reduce the ratio of active and inactive stiffness, thereby mitigating the nonlinearity induced by the contact element. This has to be checked by the user, that it is possible or not.

☞ **A gap element is defined by selecting two nodes.**
Defining *Local x orientation* is the same as for beam elements.

Active: The active state that can be tension (a tension bolt connection) or compression (contact of two plates)

Orientation (from one of its node to its other node)

Active stiffness: By default it is 1E+8 kN/m.

Inactive stiffness: By default it is 1E-2 kN/m.

Initial opening\penetration: By default it is 0. The initial opening can be set based on element geometry as well (Check *By geometry*). The initial opening is a positive or zero value. While the initial opening does not close, the gap is considered inactive.

☞ **If the gap element is used in an analysis different from a nonlinear static analysis, the element will be taken into account as a spring with a stiffness corresponding to its initial opening. If the initial opening is zero, the active stiffness will be taken into account.**

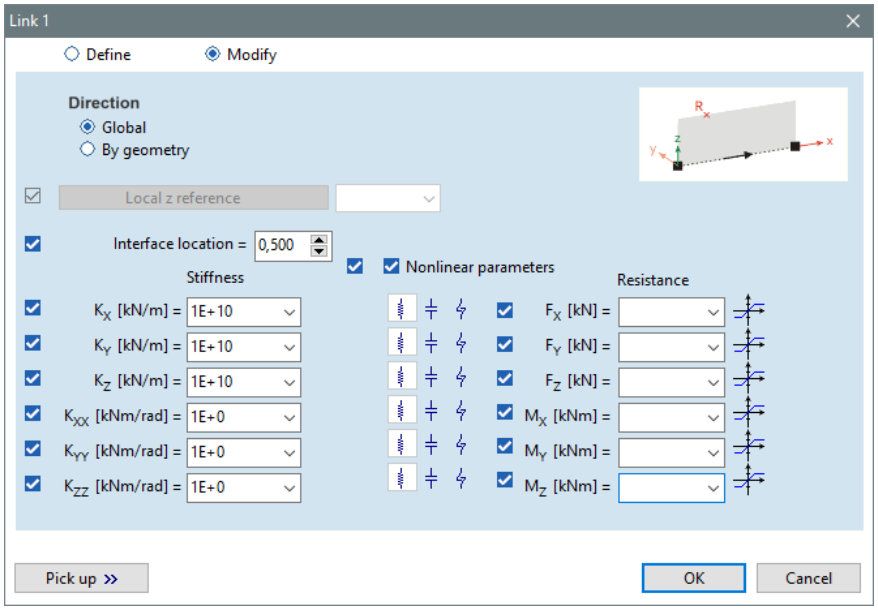
4.9.23 Link

Link elements

Link elements connect two nodes (N-N) or two lines (L-L) and have six stiffness components (defined in their coordinate system) that are concetrated on an interface (located between the connected nodes/lines). Its position can be entered relative to one node/line that is considered as reference.
Link elements can have a nonlinear parameter called limit resistance that limits the force they are able to transfer.

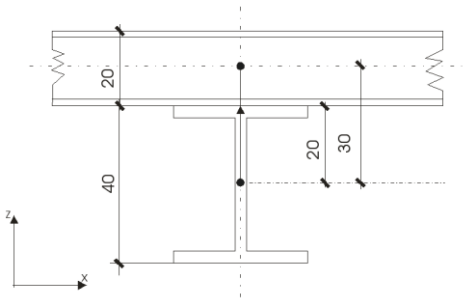
Node-to-Node (N-N) Link

Connects two nodes. The stiffness components are defined in the global coordinate system. The position of the interface can vary from 0 to 1 relative to the master node (selected by the user). If the location of the interface is = 0 the interface is at the master node. If it is = 1 the interface is at the opposite node. For any value greater than 0 or lower than 1 the reference is between the nodes.



Typical applications are: main girder-purlin connection; some types of grillage connections; bracing connections; etc.

***Example*:** A main girder-purlin connection (see SteelFrame.axs in the examples folder)
Let assume that the vertical axis is Z being parallel to the local z axis. The main girder is an IPE-400 in X-Z plane, the purlin is an I-200. You would like to transfer forces from the purlin to the main girder but not the moments.



These elements are represented by their line of gravity. The link has to be placed between these two axes at their point of intersection (if seen from above). Therefore, this link has to be assigned to a vertical line having a length equal to the distance of axes i.e., 30 cm (40/2 + 20/2). Select the node on the main girder to be the master node of the link. The inter-face always has to be placed at the actual point of contact. In this case the interface is located 20 cm far (40/2) from the master node (i.e., the main girder axis). So, the interface position is 20/30 = 0.666. You assume that the connection is fixed against displacements but can rotate. Therefore, you enter 1E10 for translational stiffnesses and 0 for rotational ones. If the purlins are supported only by these links you have to enter KYY=0.001 or a similar small value to eliminate rotation around the main girder axis.

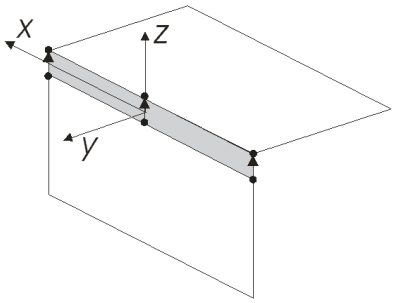
Nonlinear parameters

Nonlinear parameters can be assigned to each nonzero stiffness component. To change the characteristics click one the three buttons (bidirectional, compression only, tension only) and set the resistance checkbox and specify a value if necessary.

☞ **The geometrical stiffness of the element is taken into account in buckling analysis and geometrically nonlinear simulations, see [43].**

Line-to-Line Link

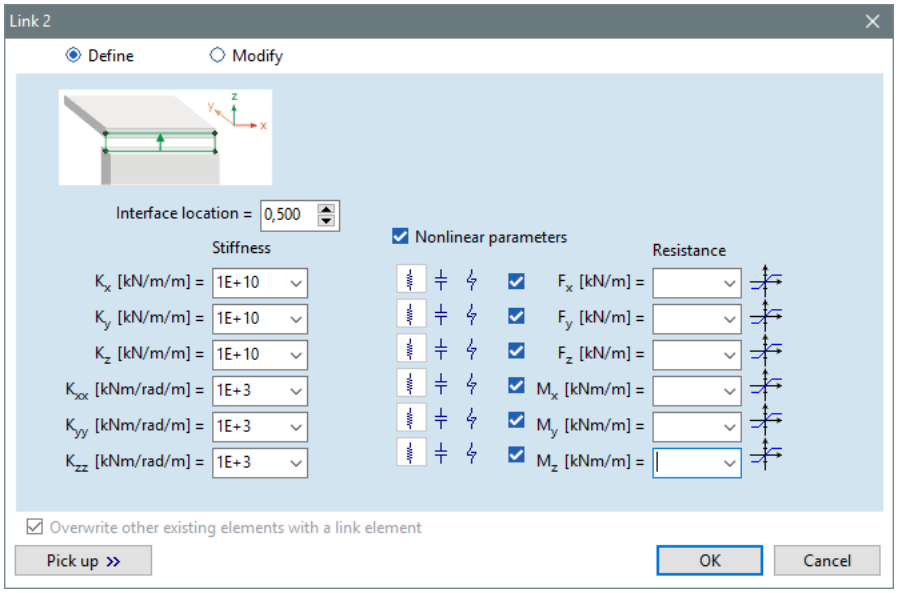
Connects two lines with three nodes each that can be rib elements and/or edges of surface elements. A line-to-line link has 6 nodes. The stiffness components are defined in the local coordinate system of the link that is in the plane of the link element with the x local axis parallel to the master line, and the local z axis oriented toward the other line in the plane of the link and is orthogonal to the local x axis.



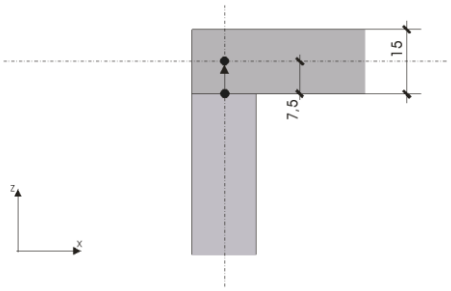
The position of the interface can vary from 0 to 1 relative to the master line (selected by the user).

If the location of the interface is 0, the interface is at the master line (at the start point of the arrow). If it is 1 the interface is at opposite line (at the end point of the arrow). For any value greater than 0 or lower than 1 the interface is between the lines.

Typical applications are: floor-wall hinged connections; semi-composite / full-composite layered beams; Semi-rigid rib-shell connections; etc.



Example: A floor-wall hinged connection.



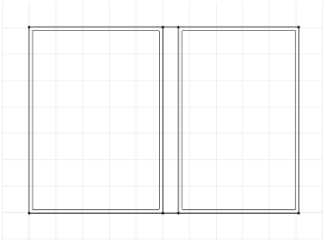
Let's assume that the vertical axis is Z, the wall is in Y-Z plane, the floor is parallel to the X-Y plane and walls are represented by shell elements. Floor thickness is 15 cm.
You would like to transfer forces from the floor to the wall but not the moments.
Elements are represented by their middle plane. The wall has to reach until the bottom plane of the floor. Links have to be placed between the upper wall edge and the floor edge.
In this case the link elements have to be in the plane of the wall. The distance between the edges is 7.5 cm (15/2). Select wall edge nodes to be the master nodes. The interface has to be at the actual point of contact which is in the bottom plane of the floor and is 0 cm far from the master node. Therefore enter 0 for the interface position. You assume that the connection is fixed against displacements but can rotate. Therefore, you enter 1E10 for translational stiffnesses and 0 for rotational ones.

Nonlinear parameters

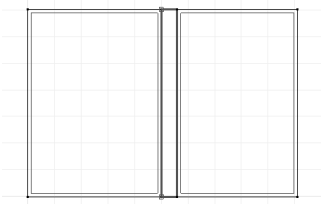
A limit resistance can be specified for each corresponding component with non-zero stiffness.

When used in conjunction with domains the following steps can be followed to define line-to-line link elements:

1. Define the domains (See **4.9.6 Domain**) and connect the cor-responding opposite nodes of the domains with lines (the number of nodes on the edges of the domains should be equal).

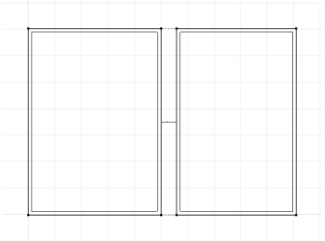


2. Select the quadrilateral between the domains. Click **OK** on the Selection Toolbar.



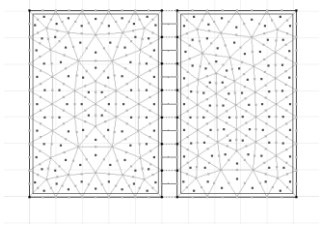
3. Select the master line of the link element. Click **OK** on the Selection Toolbar.

4. Define the link stiffness, and set the interface location. By default the interface is in the midpoint of the link element. The link element(s) are created.



5. Now you can mesh the domains.
See **4.12.1.2 Meshing of domain**

6. Link elements are divided according to the domain mesh.



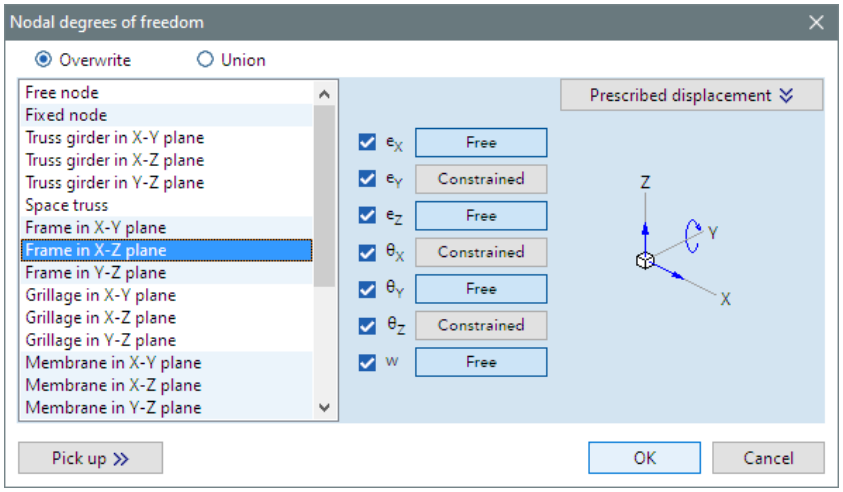
☞ **The geometrical stiffness of the element is taken into account in buckling analysis and geometrically nonlinear simulations, see [43].**

4.9.24 Nodal DOF (degrees of freedom)

The nodes themselves are not finite elements but their degrees of freedom is closely related to finite element analysis. The dialog lets you constrain the six (or seven) nodal degrees of freedom that are: translations (e_X, e_Y, e_Z), rotations (θ_X, θ_Y and θ_Z) and warping (w) (if the model contains one or more 7 DOF beams, see **4.9.10.3 7 DOF beam element – 7 DOF module**). In the default setting no nodes have constrained degrees of freedom.
During the analysis equilibrium equations will only be written in the direction of the free displacements (translations/rotations).
Any combination of the six nodal degrees of freedom (e_X, e_Y, e_Z, θ_X, θ_Y and θ_Z) can be selected. However, in many cases typical combinations of degrees of freedom can be used. In these situations, you can quickly apply a predefined setting by selecting it from the list box.

The following particular structures are listed:

Plane truss girder / Space truss / Plane frame/ Grillage / Membrane / Plate



Define a nodal DOF

Use the buttons to set the degrees of freedom. Button captions will reflect the current value. Changes will be applied only to those nodal DOF which have their corresponding check-box checked. Unchecked components will retain their original values in the selection.
You have two options to change nodal DOF:

Overwrite
The new setting overwrites the existing degrees of freedom settings of the selected nodes.
Union
Sets a degree of freedom constrained if either the existing or the new value is constrained. This option is useful in the definition of symmetry conditions.

Example of union

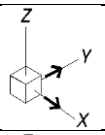
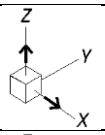
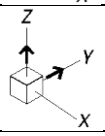
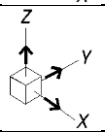
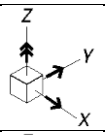
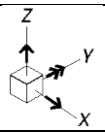
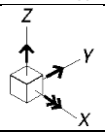
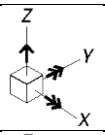
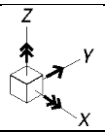
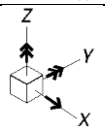
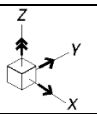
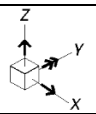
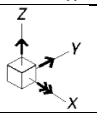
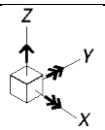
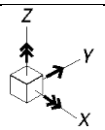
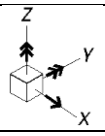
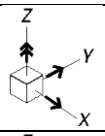
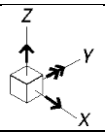
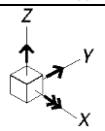
	ex	*eY*	*eZ*	*ΘX*	*ΘY*	*ΘZ*	*w*
initial value	*free*	*constr.*	*free*	*constr.*	*free*	*constr.*	*free.*
new value	*free*	*free*	*free*	*constr.*	*constr.*	*constr.*	*free*
result	*free*	*constr.*	*free*	*constr.*	*constr.*	*constr.*	*free*

The seven nodal degrees of freedom $(e_X, e_Y, e_Z, \theta_X, \theta_Y, \theta_Z, w)$ are set by a six digit code comprised of f (free) and c (constrained) symbols.
Each digit corresponds to one degree of freedom component. By default the nodes are considered free (all digits are f-free symbols). By setting a digit to c (constrained) the corresponding degree of freedom component is constrained. The default DOF code of a node is [f f f f f f f].

☞ **The loads that apply in the direction of a constrained degree of freedom are not taken into account. Loads in the direction of the constrained degrees of freedom will appear in the table of unbalanced loads.**

The nodes with DOF different from [f f f f f f f] are displayed on the screen in cyan.

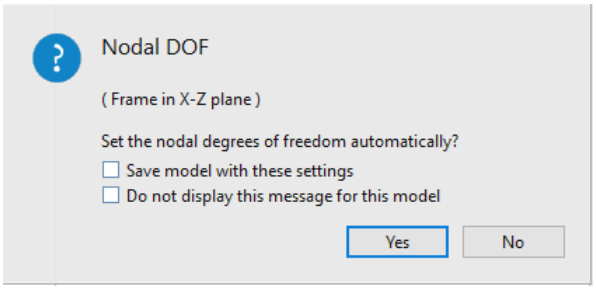
Notations: ↑ free translation, ↟ free rotation about the specified axis.

Degrees of freedom	*Free displacements*	*Degrees of freedom*	*Free displacements*
Truss girders			
Truss girder in **X-Y** plane		Truss girder in **X-Z** plane	
Truss girder in **Y-Z** plane		Space truss	
Frames			
X-Y plane frame		**X-Z** plane frame	
Y-Z plane frame			
Grillages			
Grillage in **X-Y** plane		Grillage in **X-Z** plane	
Grillage in **Y-Z** plane			
Membranes			
Membrane in **X-Y** plane		Membrane in **X-Z** plane	
Membrane in **Y-Z** plane			
Degrees of freedom	*Free displacements*	*Degrees of freedom*	*Free displacements*
Plates			
Plate in **X-Y** plane		Plate in **X-Z** plane	
Plate in **Y-Z** plane			
Symmetry			
X-Y symmetry plane		**X-Z** symmetry plane	
Y-Z symmetry plane			

Pick up>> Degrees of freedom can be picked up from another node and assigned to the selected nodes.

Checking nodal DOF before analysis

Before calculation, the program checks if the degrees of freedom are consistent with the geometry and loads of the structure and, if necessary, sends a message to change the degrees of freedom. The setting can be temporary or can be saved to the model by activating *Save model with these settings*.



Activating the second option *Do not display this message for this model* makes AXISVM remember our choices and this message will never appear in this model even if nodal DOF changes have not been saved.
The display of this message can be reactivated in *Settings/Preferences/Analysis* (checking *Show a message about nodal degrees of freedom before analysis)* See **3.3.11.14 Analysis.**

☞ **If a node has no connecting element, all degrees of freedom will be constrained automatically. If a node has only truss and solid connecting elements, all rotational degrees of freedom will be constrained automatically. These operations are performed without notice. However loads applied in the direction of these constrained degrees of freedom will appear in the table of unbalanced loads.**

4.9.25 References





Lets you define reference points, vectors or axes, and planes. The references determine the orientation of the local coordinate systems of the finite elements in the 3D space. The local coordinate system of the elements defined with the references is used to define cross-sectional properties and to interpret results. The element properties are defined and the internal forces (*Nx, Vy, Vz, Tx, My, Mz* for beams, *mx, my, mxy* for plates, *nx, ny, nxy* for membranes, etc.) are computed in that local system.

Quick modify: Clicking on the symbol of a reference the Table Browser is invoked displaying the table of the references. The reference vector and axis can be defined by two points, the reference plane by three points. When closing the table the reference vectors, and axes are normalized with respect to 1.

Color codes: ***x*** = red, ***y*** = yellow, ***z*** = green.

The following references can be used:

Automatic references

Automatic references for truss and beam elements:
A reference vector is generated and assigned to the truss and beam elements as follows:
If the axis of the element is parallel to the global Z axis the reference vector will be parallel to the global X axis.
In any other case it will be parallel to the global Z axis.
For arcs: if the arc plane is parallel to the global X-Y plane, automatic reference is perpendicular to it and points to the +Z direction. If the arc is in a different plane its reference vector is in the arc plane and points outwards from the arc center point.

Automatic references for rib elements:
If the rib is independent the reference vector will be generated and assigned to the element as for the beam elements.
If the rib is connected to a surface element, the generation of the reference vector is as follows:
The reference vector will be parallel to the bisector of the local z axes (normal to the surfaces) of the surfaces that have the rib element attached.

Automatic references for domains and surface elements:
Reference vectors will be generated and assigned to the surfaces as follows:
Local x-axis reference
If the plane of the surface is parallel to the X-Y plane the reference vector for the x local axis will be generated as a vector parallel to the global X axis.
In any other case, it will be parallel to the intersection line of the surfaces and X-Y plane.
Local z-axis reference
If the plane of the surface element is parallel to the Z axis, the generated reference will be a vector oriented toward the origin of the global XYZ system. If th global origin is in the plane of the surface, the

reference vector points to the positive half space in the X or Ydirection. In any other case it will be parallel to the global Z axis.

The *Edit / Convert automatic references* menu item converts automatic references into reference vectors.

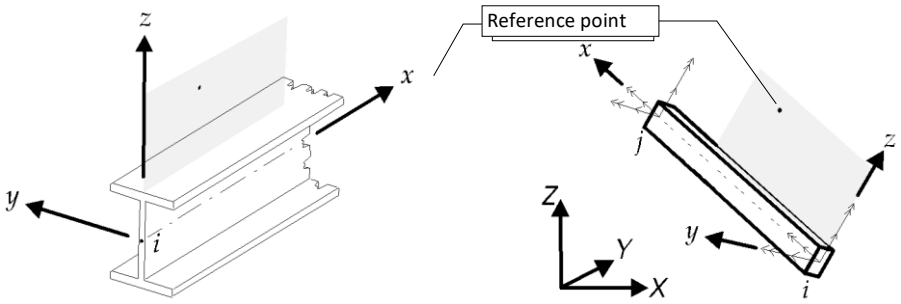
Reference point



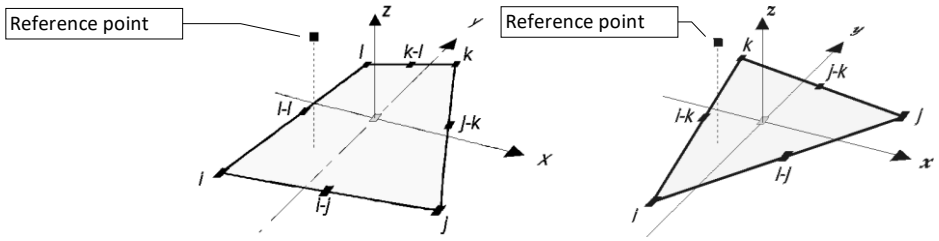
Reference point is used to define the orientation (local coordinate system) of beam, rib, support, and spring elements or to define the positive local x and z axes of surface elements.
The reference points are defined (by its coordinates) in the global coordinate system.

The reference points are displayed on the screen as small red + symbols.

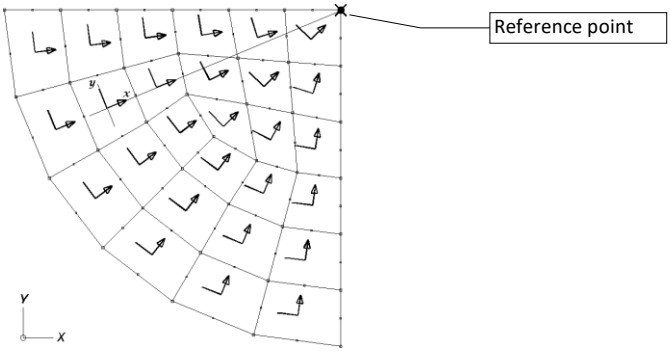
Beams, ribs, and springs:
The reference point and the element's local ***x*** axis defines the local ***x***-***z*** plane. The positive local ***y*** and ***z*** axis direction is determined by the right-hand rule.



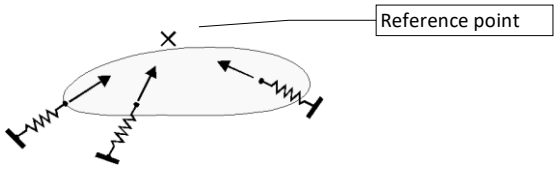
Surface elements:
The positive local *z* axis is oriented toward the half-space in which the reference point is located, and is perpendicular to the element's plane. Once the local *x*-axis is defined local *y*-axis is determined according to the right hand-rule.



The local ***x*** axis will be oriented in the direction of the reference point.
In the case of a surface element the reference point must be located in the plane of the element.



Supports:
In the case of a support element you can use a reference point to define local x axis.



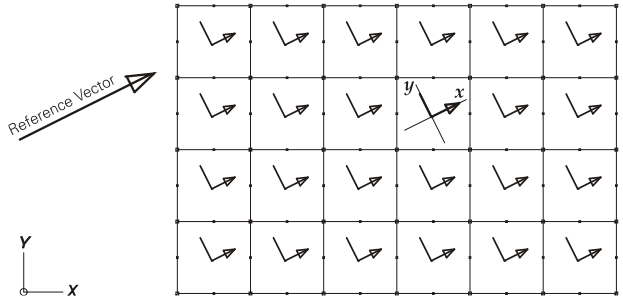
Reference vector



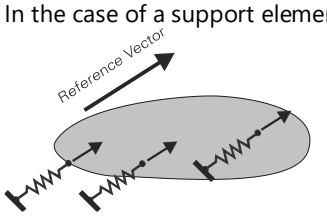
Lets you define the local ***x*** axis for surface, support, and spring elements. Also defines the orientation of local z coordinate axis of beam, rib and spring elements.

The reference vectors are displayed on the screen as red arrows.

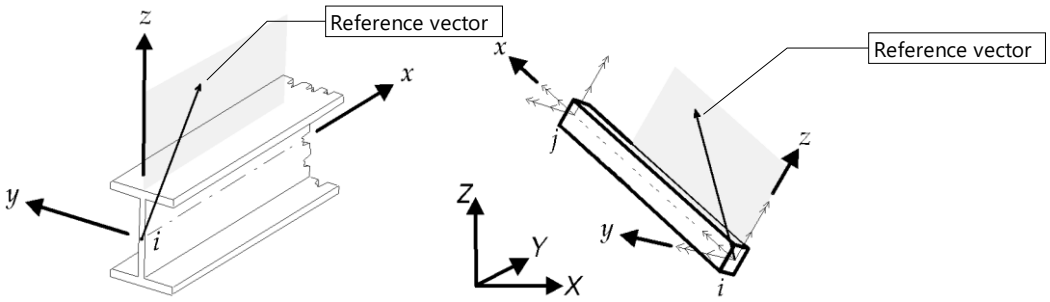
Surfaces:
The local ***x*** axis will be parallel to the reference vector. In the case of a surface element the reference vector must be parallel to the plane of the element.
The orientation of local *z*-axis can also be defined by a reference vector.



Supports:
In the case of a support element you can use a reference vector to define local x axis.



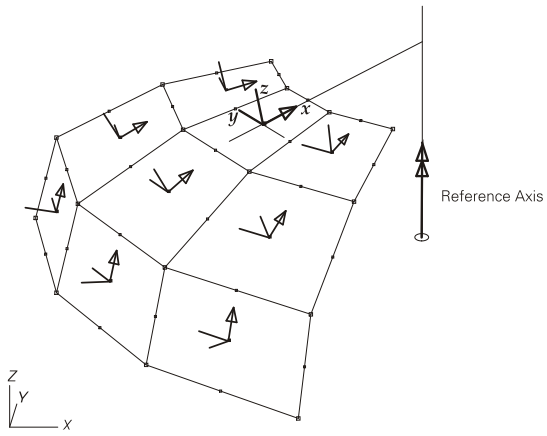
Beams, ribs, and springs:
The reference vector and the element's local ***x*** axis defines the local ***x***-***z*** plane.
The positive local ***y*** and ***z*** axis direction is determined by the right-hand rule.



Reference axis



Reference axis is used to define the local ***x***-axis of surface elements, that will be oriented towards the reference axis. The reference axis must not include element centerpoint.

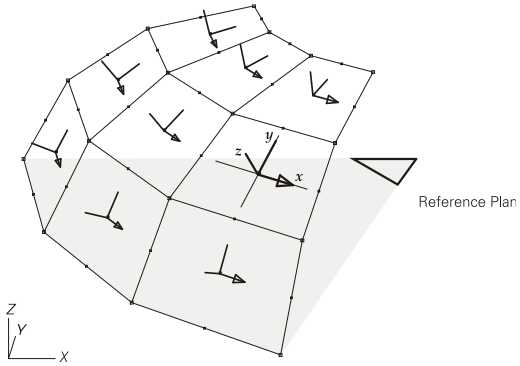


The reference axises are displayed on the screen as red arrows.

Reference plane



Reference plane is used to define the local ***x*** axis of surface elements, that will be parallel to the intersection line of the reference plane and the plane of the element. The reference plane must not be parallel to the plane of the element.



Reference angle



Rotation of truss / beam / rib cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel to the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

The reference plane is displayed on the screen as a red triangle.

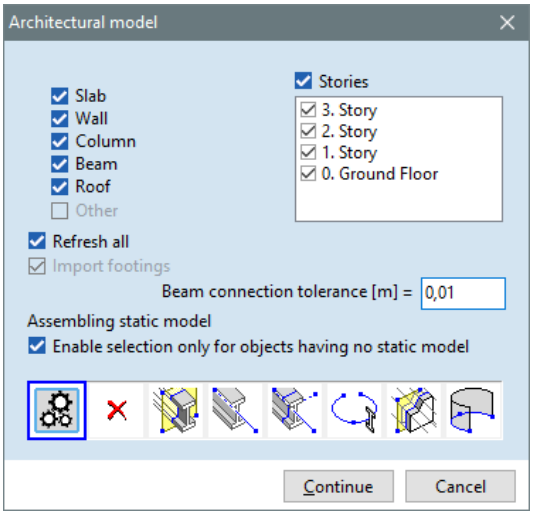
Unused references can be deleted in the Table Browser by selecting *References* and choosing *Edit / Delete unused references* from the menu.

4.9.26 Creating model framework from an architectural model



Creating model framework requires IFC module.

This icon starts the conversion operation of the architectural model if previously an IFC file (*.IFC) was loaded by *File / Import* as a background layer. See 3.1.7.2 Importing IFC files



Display — Select architectural project storeys and element types you want to be displayed.

Use the built-in Filter to enhance selection.

If you create model framework or delete objects and nothing is selected the Selection Toolbar appears. Click the Property Filter icon to select beams and columns within a certain range of section size according to their minimum side length or select walls or slabs within a certain range of thickness.
If you want to restore the whole range click the button at left bottom.
If the *Only objects without static model* is checked only elements not having static model will be selected.

Refresh all — Makes the architectural model visible in all windows.

Import footings — If activated, footings in the IFC file are also processed.

Deleting an architectural object having a static model will not delete its associated static model.

Create Model Framework

Model framework will be created from selected layer elements. Columns will be reduced to their axis, walls, slabs and roofs will be reduced to their center plane. Framework nodes and lines become part of the AXISVM model and are independent of the background layer.
When importing IFC files, it is not always possible to detect the statical frames of objects. Those objects whose statical frame detection failed, are drawn to the screen with a dashed line. User now has many possibilities to create, modify or delete statical model though:



Automated detection of statical frame: in as much as the automated detection method has been successful while importing, the program can use those precalculated data to create statical frame. **More than one item can be selected for this operation.**



Deleting selected IFC objects: with this function user can delete the selected IFC objects from the model. **It is important to note, though, that if we delete objects, whose statical frame has been created before, this frame will not be deleted with the object. More than one item can be selected for this operation.**



Clipping beam-like objects by clipping plane: the selected objects are clipped by the plane the user has supplied by giving 3 points. Then the algorithm defines the sections to all the selected elements, and calculates their normals and centres of gravity. These values give us lines imprinting the selected bodies. The resulting segments shall than be treated as statical frames of the objects, and the calculated polygons as their cross-sections. **More than one item can be selected for this operation.**



Handling beam-like objects by supplying their middle lines: user must support two points that will form a line. This line then is imprinted to the selected bodies, and the resulting segments will serve as the middle lines of the objects. Taking the middle points of these segments and their direction vectors, clipping planes can be obtained. At the final step, the segments are moved to the centres of gravity of the cross-sections. **More than one item can be selected for this operation.**



Handling beam-like objects by supplying their middle lines and position of the cross-section: almost the same like the previous operation, except for that here, the position of the cross-section must be given by the user, too. Then the center line of the selected body is modified in a way, that the start point is left unchanged, and the end point is the mirror image of the start point to the cross section. **Only one item can be selected for this operation.**



Handling circular beam-like elements: user must supply an arc by 3 points. Then the algorithm calculates the intersections of this arc and the selected body. If more than 2 intersections are obtained, then it takes the two extremes. The middle point of the resulting arc and the normal of the arc provide a clipping plane that the algorithm uses to get the cross-section. **More than one item can be selected for this operation.**



Clipping plates by clipping plane: the selected domains are clipped by the plane given by 3 points, and the resulting polygons will be the statical frames of the objects. **More than one item can be selected for this operation.**



Clipping cylindrical swept bodies by a clipping cylinder: the user must supply a cylinder that the algorithm will use for clipping. This cylinder is adjusted in a two-step operation. First, an arc is obtained by 3 points. Secondly, this circular arc is offsetted when moving the cursor. This resulting arc shall serve as the base of the cylinder. Its height is calculated automatically from the geometric parameters of the selected body. Finally, the statical frame is the result of the intersection of the cylinder and the selected body. **Only one item can be selected for this operation.**

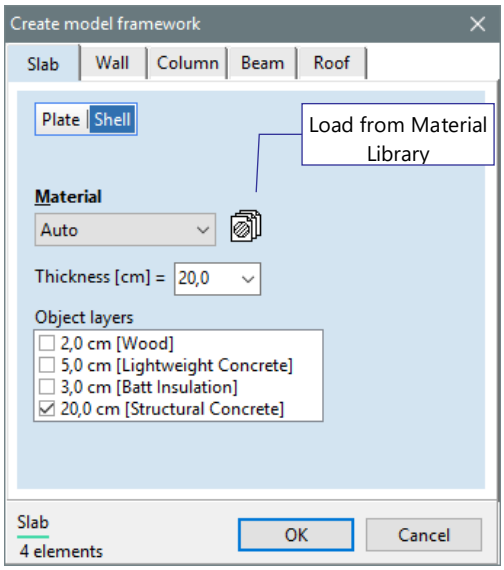
In all editing operations a real-time visual feedback is given to the user about the current statical frame, thus it is possible to edit the correct statical model of the object. Sometimes some pre-editing can be necessary, though. An optional filter is given to filter out IFC objects that already have statical models calculated. However, if we edit an object with an existing statical model, the result of the operation will overwrite the former statical frame.

Parts will automatically be created for levels and object types and the elements created for the static model will be included in the appropriate parts.

Hinged wall connections can be modeled using edge hinges when creating a model framework from the architectural model.

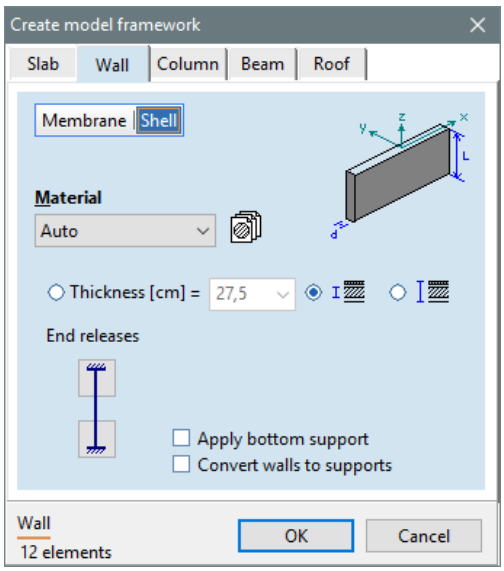
If the *Material* field is set to *Automatic*, and the IFC file includes material data and assigments the model will import them.
You can assign properties to the selected architectural objects as follows:

Slab



Floors can be defined as plates or shells. Assign a material and a thickness. For layered floors, the thickness of the layers will appear in the layer list. You can select the layers that you want to take into account.

Wall

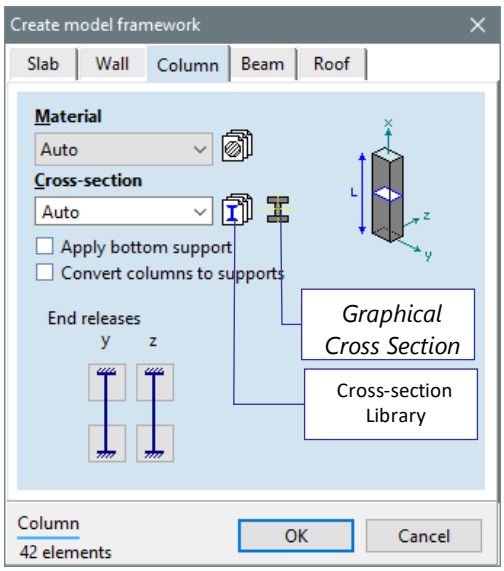


Walls can be defined as membranes or shells. Assign a material and a thickness. For layered walls you can choose to apply the thickness of the load bearing layer, the total thickness or a custom value.

Apply bottom support:
You can automatically assign a support to the bottom edge of the selected walls.

Convert walls to supports: You can convert wall objects to supports by enabling this checkbox. The support will be placed at the top edge of the corresponding wall. The support stiffness will be computed based on the top and bottom end releases.

Column

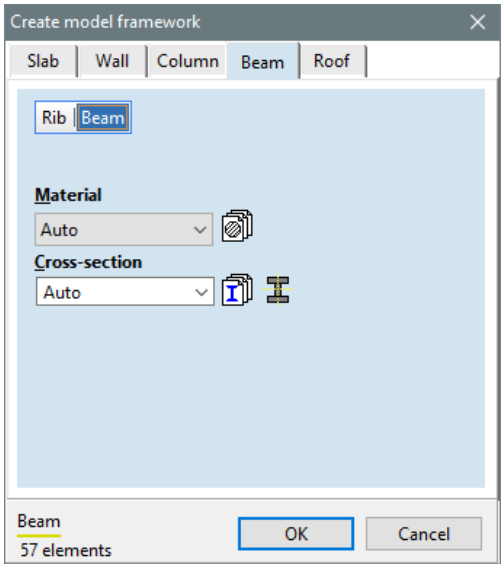


Column objects are always converted to beam elements. Assign a material and a cross-section. If *Auto* is selected the cross-section is created based on the geometrical description of the architectural object.

You can assign a support to the bottom of the column.

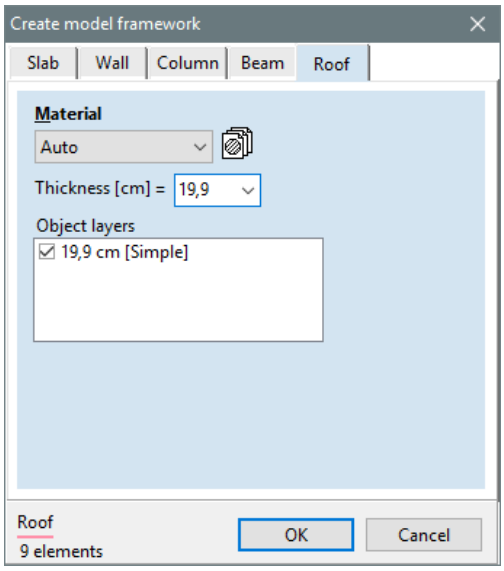
Convert columns to supports: The selected column objects can be converted to supports. Support stiffness is established based on the end releases. Supports will be placed at the top of the column.

Beam



Beam objects are always converted to beam elements. Assign a material and the cross-section.
If *Auto* is selected the cross-section is created based on the geometrical description of the architectural object.

Roof



Roof objects are always converted to shell elements. Assign a material and a cross-section.
For layered roofs, the thickness of layers will appear in the layer list.

You can select the layers that you want to take into account.

4.9.27 Modify

Lets you modify the definition of the selected elements.

1. Holding the **[Shift]** key down, select the elements to modify. You can use the Selection icon as well.
2. Click the element's icon on the Elements Toolbar.
3. In the element's dialog window check the properties you want to modify. Property fields show the common value in selection. If selected elements have different values the field is empty.
4. Modify the respective properties as desired.
5. Click the **OK** button to apply the modifications and exit the dialog window.

☞ **In fact, the modification is similar to the element definition, but does not assign properties to undefined geometrical elements and allows access to a specific property without altering others. You can switch to the element definition radio button to define all properties of all the selected elements, lines or surfaces.**

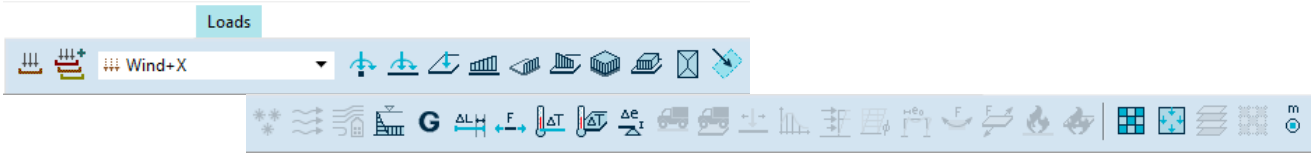
Immediate mode

If the Geometry or Elements tab is active click a finite element to modify its properties. If more finite elements have been selected they can be immediately modified by clicking one of them. If you click an element which is not selected, selection disappears and you can modify the element you clicked. If you click on a node its nodal degrees of freedom can be edited immediately.
You can also modify the properties using of Property Editor.
See 3.6.1 Property Editor

4.9.28 Delete

[Del] See 3.2.9 Delete

4.10 Loads



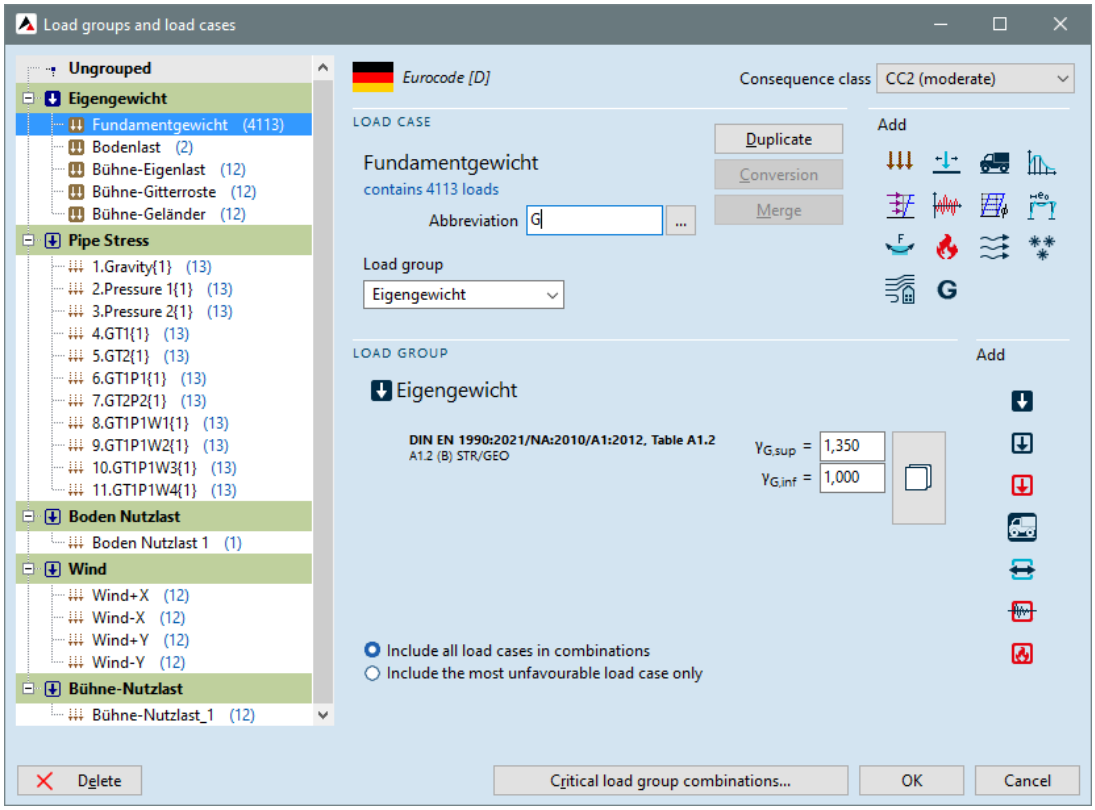
Lets you apply various static loads for static, dynamic and buckling analysis, and define concentrated masses for vibration analysis. The current load case can be selected from the dropdown list.

4.10.1 Load cases, load groups

Load Case



Lets you set the current, create new, and modify or delete existing load cases. Any load you create will be stored in the current load case. In the professional version the number of load cases is not limited. In the standard version a maximum of 99 cases can be created. Load groups can also be created from different load cases.
The name of the current load case is displayed in the Info window and on the *Loads* toolbar. Subsequently defined loads will be added to the current load case. The selected load case on the left will be the current load case if you click OK.



Consequence class

In Eurocode structures are categorized according to the consequences, risks and socio-economic factors of failure or malfunction.
The value of K_{FI} parameter is selected according to the consequence class. This parameter is used as a multiplication factor for partial factors according to EN 1990. The values can be set in *Settings / Design codes / Load combinations*.

New load case

You must assign a different name to each case. These are the possible types of load cases:



1. Static

The static load case can be applied to static, vibration, and buckling analysis. In case of vibration analysis, the loads can also be taken into account as masses.

The load case can be included into a load group. When calculating the critical load combination, the load case will be taken into account according to the parameters of the load group to which it belongs.

Critical combination can be determined only from the results of a linear static analysis.

2. Influence line

Lets you apply a relative displacement load to obtain the influence line of a result component, of a truss or beam element.

When the influence line load case type is selected you can apply only the influence line load.

3. Moving load

In this type of load case only moving (line or surface) loads can be defined. When defining a moving load, a group of new load cases will be created. The number of these load cases is equal to the number of steps specified in the moving load definition dialog. Their name is created automatically like *MOV_xx*. As they get into a load group the most unfavourable effect of the moving load can be checked displaying the result of the critical combination. These auto-created load cases can be moved together only and only into another moving load group.

If more than one moving load is applied in the same load case the number of steps (and auto-created load cases) will be equal to the maximum number of steps specified. If the maximum number of steps is *k,* and another moving load has *i* steps *(i < k)*, then this load will remain at the end of the path in steps *i+1, i+2, ..., k.*

See details **4.10.30 Moving loads**

Moving load cases can be placed only in moving load groups. Static loads can also be defined in moving load cases. They will be present in all steps but only in moving load cases.

4. Seismic (SE1 module is required)

When selecting seismic load case type you can specify the parameters for calculation of earthquake loads. Prior to creating an seismic load case, you must perform a vibration analysis. Based on the mode shapes, and on the structural masses, AXISVM generates seismic loads case, in a *k*+2 number, where *k* is the number of available smallest frequencies. The two additional cases corresponds to the signs **+**, and **-**, that contain the critical combinations.

Multiple seismic actions can be defined for the structure. Seismic actions cannot be combined.

See details **4.10.25 Seismic loads – SE1 module**

When selecting seismic load case the only icon available on the Toolbar will be Seismic parameters.

If the model has construction stages, seismic design can be performed only in the final state.

5. Pushover (SE2 module is required)

When selecting pushover load case type, you can specify parameters for generating load distributions that can be used in pushover analyses. Prior to creating a pushover load case, you must perform vibration analysis. Based on specified mode shapes AXISVM generates nodal forces on each node of the model. A total of four load cases are generated initially. They represent a uniform (U) and a modal (M) distribution in the direction of each of the horizontal axes (X and Y by default). The uniform load distribution option generates nodal forces proportional to the masses assigned to each node in the model. The modal load distribution uses the mode shape weighed by the masses at each node to generate the nodal force distribution. In both cases the sum of forces generated is 1 kN in the same horizontal direction.

See details **4.10.26 Pushover loads – SE2 module**

When selecting pushover load case the only icon available on the Toolbar will be Pushover parameters.

If the model has construction stages, pushover analysis can be performed only in the final state.



6. Global imperfection and imperfection from buckling shapes

If an imperfection load case is created it is automatically placed into an imperfection load group which can contain only imperfection load cases. This load group has no parameters and is automatically deleted if their load cases are deleted. Imperfection load cases can be used in nonlinear analysis with geometric nonlinearity. When generating critical ULS load combinations in the load combination table imperfection load cases can be included. Load combinations including an imperfection load case require nonlinear analysis with geometric nonlinearity.

See details **4.10.27 Global imperfection, 4.10.28 Geometric imperfections based on buckling shapes**

☞ **Imperfection load cases does not contribute to critical combinations of linear analysis results.**



7. Tensioning (PS1 module is required)

If tensioning calculation according to the current design code is supported, tensioning load cases can be created. These load cases always get into a tensioning load group. After defining a load case with the name *name,* two load cases will be created. *name-T0* will contain the equivalent load calculated for the end of tensioning process, *name-TI* will contain long term values of the equivalent load. Any of these load cases can be selected to define tensioning. After definition just loads for *name-T0* will be calculated as static analysis results are required to determine the long-term equivalent loads. See details **4.10.29 Tensioning - PS1 module**

☞ **When selecting tensioning load case the only icon available on the Toolbar will be Tensioning.**



8. Dynamic load case (DYN module is required)

After defining a dynamic load case and selecting it the *Loads* tab will allow definition of dynamic loads and nodal acceleration.

See details **4.10.31 Dynamic loads (for time-history analysis) – DYN module**

☞ **Dynamic load cases cannot be included in load groups and load combinations. Loads within dynamic load cases will be applied only in Dynamic analysis.**



9. Snow load cases (SWG module is required)

AXISVM can calculate and apply snow loads on the structure. The limits of automatic snow load generation in the program are explained in **4.10.13 Snow load – SWG module.**

Snow loads can be placed on flat load panels in arbitrary planes. As a first step a temporary snow load case is created in the snow load group and its name can be set. If the design code requires verification for exceptional snow load, an exceptional snow load case is also created in an exceptional snow load group (except for exceptional snow loads as per Annex B of Eurocode 1-3). After defining load panels and setting the snow load parameters the program replaces the temporary load case with automatically generated snow load cases.

The undrifted load case receives a UD suffix. Drifted load cases receive a D suffix with 2-4 extra characters. These characters indicate the wind direction (X+, X–, Y+, Y–). Besides winds parallel to the coordinate axes, winds in 45°+*n**90° directions (where *n* = 0, 1, 2, 3) are also taken into account and are indicated by the quadrant where the wind speed vector points to. X+Y– refers to a wind in the 315° direction for example (0° being defined by the positive X axis, and angles measured counter clockwise).

Only the necessary load cases are created by the algorithm, therefore the number of load cases and their type depends on the type of structure under consideration and the snow load parameters given by the user. Load parameters shall be defined after closing the Load Cases Dialogue by clicking on the Snow Load icon in the Toolbar. Before this step it is recommended to define load panels of the roof using the Load Panels icon in the Toolbar.

For details of snow load generation see **4.10.13 Snow load – SWG module.**

☞ **If a snow load case is selected only two buttons are enabled on the Loads toolbar. These are the load panel and snow load definition.**

☞ **If the model has construction stages, SWG loads can be applied only to the final state.**



10. Wind load cases (SWG module is required)

During parametric definition of wind loads it is necessary define one or more substructures exposed to wind. If *Separate load cases for each substructure* is checked wind loads of different substructures will be generated into different load cases. Otherwise load cases will contain wind loads for more substructures.

Calculated wind loads are reliable only for certain building types as described in the design code. It is recommended to use these automatic methods only for geometries within the limits explained in detail at **4.10.14 Wind load – SWG module.** In other cases, a wind tunnel simulation available in the WIND module can be used to generate realistic wind loads.

Wind loads can be placed on flat load panels in arbitrary planes. As a first step a temporary wind load case is created in the wind load group and the name of the wind load case can be set. After defining load panels and setting the wind load parameters the program replaces the temporary load case with automatically generated wind load cases.
Wind load cases are generated with a name code corresponding to the loading situation.
The first two characters after the name of the load case describe the wind direction (X+, X–, Y+, Y–). The logic behind this notation is the same as for snow loads. The next one or two characters denote the type of wind action.

P and S denote pressure and suction, respectively. For pitched roofs the design code requires to check all possible combinations of wind actions on the two sides of the roof. Therefore, for pitched roofs Pp, Ps, Sp and Ss load cases are created. Here the first character refers to the windswept side, the second one refers to the sheltered side. For special torsion actions T+ and T– are used referring to the two different torsion directions.
The last character denotes the type of internal action. The internal wind actions are ignored for O; P stands for internal pressure, S stands for internal suction. These load cases are needed only if no further information is available on internal pressure so a positive and a negative critical value must be used as an approximation on the conservative side. If the last character is C, the load case was created based on a user-defined μ value which depends on the layout of openings of the structure. If a C load case is created for a given direction P and S load cases are not required, hence not created.
Only the necessary load cases are created by the algorithm, therefore the number of load cases and their type depends on the type of structure under consideration and the wind load parameters given by the user. Load parameters shall be defined after closing the Load Cases Dialogue by clicking on the Wind Load icon in the Toolbar. Before this step it is recommended to define load panels of the building walls and roof using the Load Panels icon in the Toolbar.
For details of wind load definition see **4.10.14 Wind load – SWG module.**

☞ **If a wind load case is selected only two buttons are enabled on the Loads toolbar: *Load panel* and *Wind load* definition.**

☞ **If the model has construction stages, SWG loads can be applied only to the final state.**



11. Fire load cases (one of the SD8, TD8, RC8-B, RC8-S modules is required)

Fire effects can be assigned to steel, timber and concrete line elements or concrete domains.
Fire load groups contain only fire load cases. This load group has no parameters and is automatically deleted if their load cases are deleted. In the process of finding critical combinations AXISVM takes into account fire load cases only in accidental load combinations. However, it is also possible to consider fire load cases in user specified load combinations.
The definition of fire effects is discussed in

4.10.32 Fire effect on steel line elements – SD8 module,
4.10.33 Fire effect on timber line elements – TD8 module,
4.10.34 Fire effect on concrete line elements – RC8-B module
4.10.35 Fire effect on concrete domains – RC8-S module

☞ **If a fire load case is selected only *Define fire effects on line elements* and/or *Define fire effects on domains* buttons are enabled on the Loads toolbar.**

☞ **If the model has construction stages, fire design is available only for the final state.**



12. Simulated wind load (WIND module is required)

Parameters of the wind simulation for the virtual wind tunnel can be defined on the *Loads* tab. Clicking on this button in the *Load cases, load groups* dialog creates a temporary load case with the default name *w1-0°* and a load group with the default name *WG* (if it does not exist). Load cases in this load group are always mutually exclusive.
When entering a simulated wind load (**4.10.15 Simulated wind used in wind tunnel analysis – WIND** module), multiple wind directions can be added with the same wind profile. Separate load cases will be created in the load group for each wind direction (e.g. w1-0°, w1-90°, w1-180°, w1-270°). This set of load cases is called wind rose.
It is not necessary to delete the directions that are not tested. Any of these load cases can be left empty, moreover the selected directions can be modified, deleted, new directions can be added in the *Simulated wind* dialog. The load cases of the wind rose will be created or deleted accordingly.
By renaming any of the loads in the wind rose, the names of all the loads in the wind rose will be changed accordingly.
Any number of wind roses can be added to the model with different wind profiles. All of them will be included in the same simulated wind load group. New temporary load cases will be created under the

default names *w2-0°*, *w3-0°*, etc. The number of generated load cases will be equal to the number of wind directions added when the wind rose was specified.
Clicking on the *Convert* button converts the simulated wind load group to an incidental load group, and the load cases to ordinary static load cases in that group.
For the details of the wind tunnel analysis see **4.11 Wind tunnel – WIND module** or the *WIND Guide* (see **3.9.2 Guides**).

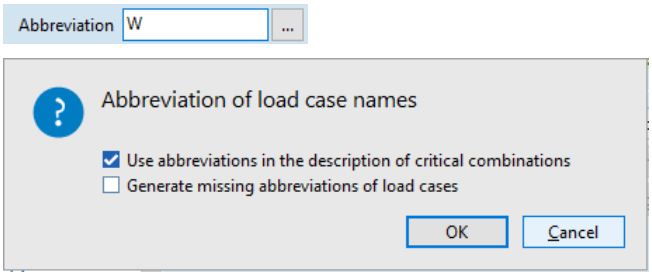
☞ **If the model has construction stages, simulated wind loads can be applied only to the final state.**

13. Self-weight load case

Self-weight loads can be defined in any static load case, but you can also create a dedicated self-weight load case, where the self-weight of each structural element is automatically displayed and taken into account. Only one self-weight load case can be defined in a model. Additional loads cannot be entered in this load case.

Load-duration class Timber design module requires information on the load duration. So, if a timber material has been defined in the model load case duration class can be entered. *(Permanent: > 10 years; Long term: 6 months–10 years; Medium term: 1 week–6 months; Short term: < 1 week; Instantaneous; Undefined)*

Abbreviation Descriptions of critical load combinations can become extremely long if load cases have long names. To improve clarity and prevent display and printing problems, it is advisable to provide abbreviations for the load case names and turn on the use of abbreviations (see below).



Check *Use abbreviations in the description of critical combinations* to shorten names of critical combinations. Check *Generate missing abbreviations of load cases* to create missing abbreviations for long load case names. For names shorter than five characters, and for load cases generated automatically no abbreviation is created.

Duplicate Lets you make a copy of the selected load case under another name. You must specify the new name, and a factor that will multiply the loads while copying. The factor can be a negative number as well.

☞ **Selected loads can be copied or moved to another load case by changing load case during the copy or move process.**

Conversion Generated snow or wind load cases can be converted to regular load cases by clicking the *Conversion* button. All load cases of the selected type will be converted. After conversion converted loads and load cases can be deleted and modified.

Merge Loads of static loads can be merged by selecting the load cases and clicking on the *Merge* button. You can specify whether the merged loads are to be merged into a new load case or into an existing load case. The original load cases will be deleted.

Delete Lets you delete the selected load case or load group. Any related load cases and load groups will also be deleted (e.g., deleting a seismic load group removes all generated seismic loads and their load group as well).

To change the order of load cases click the right mouse button over a load case in the tree. Select *Alphabetical order* or *In the order of definition* from the popup menu to reorder the load cases within the group.
To change the position of ungrouped load cases choose one of the two options (*Ungouped load cases first* or *...at the end*).

Load Groups Load groups are used when generating of critical (design) values of the results.

New Group Lets you define a new load group. You must specify the name and the type (permanent, incidental, exceptional) of the load group, and the corresponding factors according to the current design code.

Later you can specify which load cases belong to a specific load group. Clicking any icon within the New Group group box will create a new group in the tree and you can specify a name for it. Existing load group names will be rejected. After creating a load group you have to specify the value of its paremeters (like the partial factor, dynamic factor, simultaneity factor, etc.). A load case can be assigned to a load group by choosing a group from the dropdown list or dragging the load case under a load group in the tree. See **4.10.2 Load combinations**
The default behaviour of new load cases can be set in *Settings / Preferences / Load group*.
See **3.3.11 Preferences**



Click on these buttons to select from the list of recommended values according to the current design code.

The following load groups are allowed depending on the design code:



Permanent
Includes dead load, permanent features on the structure...

Include all load cases in combinations
All load cases from the group will be taken into account in all load combinations with their upper or lower partial factor.

Include the most unfavourable load case only
Only the most unfavourable load case will be taken into account from the load group with its upper or lower partial factor.



Incidental
Includes live load, wind load, snow load, crane runway load...

Simultaneous load cases
Any number of load cases from the group can act simultaneously in critical combinations.

Mutually exclusive load cases
In a critical load combination only one load case from the group will be taken into account at one time.



Exceptional
Includes earthquake, support settlements, explosion, collision... Only one load case from the group will be taken into account in a load combination at one time. That load case must have the simultaneity factor of zero.



Moving load group
Auto-created load cases for the moving loads in a moving load case get into a moving load group.



Tensioning load group (if tensioning can be calculated according to the current design code)
Tensioning load group is handled as a permanent load group. It can contain only tensioning load cases. Both load cases for the same tensioning (name-T0 and name-TI) cannot be included in any load combination.



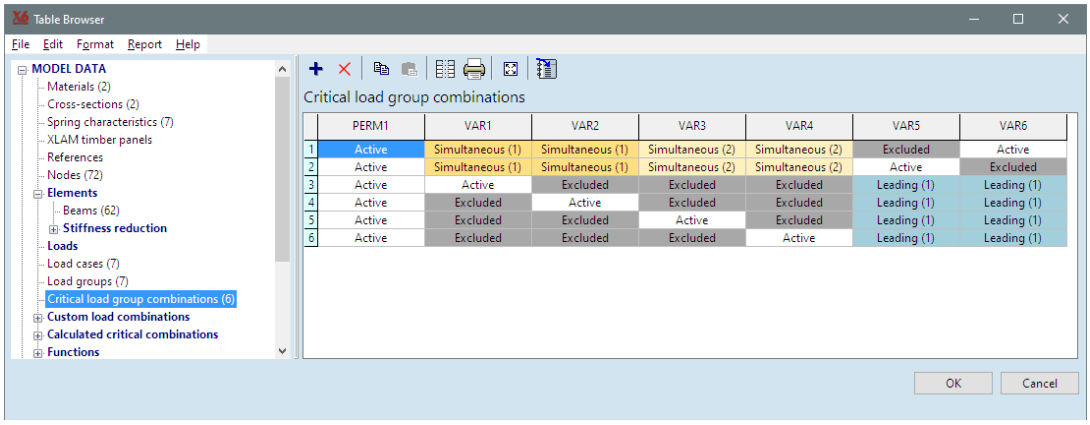
Seismic load group (Eurocode, SIA 26x, and Italian code)
Only one load case from the group will be taken into account in a load combination at one time. That load case must have the simultaneity factor of $\alpha = 0$.



Fire load case (Eurocode)
Fire load groups contain only fire load cases. Load cases can be simultaneous or mutually exclusive. One line element can have only one fire effect in simultaneous load cases.

Critical load group combinations

Critical combinations are determined according to the load groups. Certain exclusive loading situations (like snow and exceptional snow) are detected (so critical combinations do not include both snow and exclusive snow loads). If it is required to ensure exclusivity or linkage between load groups it is possible to control this through critical load group combinations.
Go to the Table Browser and find *Critical load group combinations*.



Columns of the table list the load groups defined in the model. Each row describes a rule for generating critical combinations.

Click on the *New row* icon to add a new rule. The *Active* status of permanent load groups can also be changed to *Excluded*. If a load group has the *Excluded* status in all combinations, it will not appear at all in the relevant combinations; in this case a warning message will be displayed. In case of variable load groups, the way of contributing to critical combinations can be selected from the dropdown list.

Active Load cases of the load group can be included in the automatic combinations built internally.

Excluded Load cases of the load group cannot be included in the automatic combinations built internally. Mutually exclusive load groups can be created by adding as many rules as the number of exclusive load groups, and only one group must be set to *Active* in each row, leaving other groups *Excluded*.

Simultaneous (1), (2), ... Load groups acting simultaneously as simultaneity groups. Load groups in the same group always act together, i.e., they are included in automatic combinations together. More than one simultaneity group can be defined. Having two load groups with a status of *Simultaneous (1)* makes *Simultaneous (2)* also available automatically in the dropdown list, and so on.

Leading (1), (2), ... Load groups acting simultaneously as leading actions. Load groups in the same leading group are like simultaneous load groups but they are not necessarily present in the combination. At least one group will be included in the automatic combinations as leading action, but additional groups of the leading group can be present, depending on which combination causes an extreme. More than one leading group can be defined. Having two load groups with a status of *Leading (1)* makes *Leading (2)* also available automatically in the dropdown list, and so on.

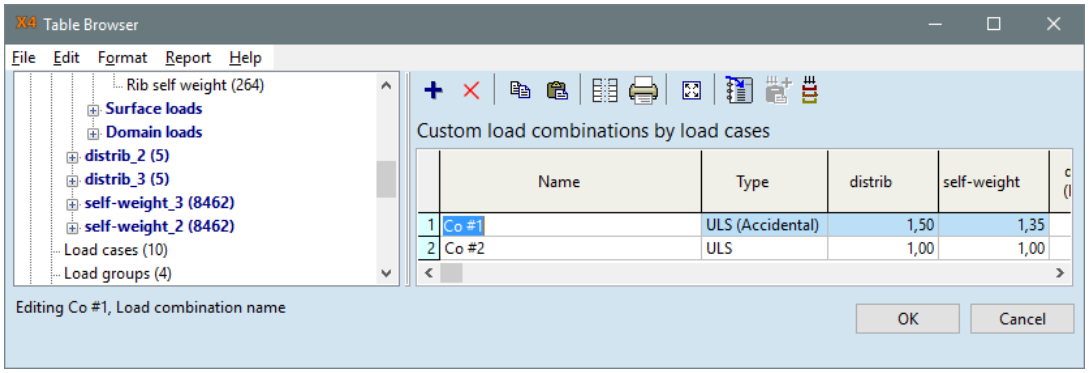
AXISVM builds automatic combinations internally according to the rules defined in this table then finds the most unfavourable combination as critical combination.
The table of *Custom load combinations* allow generation of these automatic combinations to check the combination building process (the total number of combinations can be huge depending on the number of load groups) but generation of the actual combinations are not necessary to obtain critical forces.

Load types The following loads can be applied to the elements:

Load	*Element*
Concentrated	node, beam
Line (distributed)	beam, rib, plate, membrane, shell
Edge (distributed)	plate, membrane, shell
Dead load	truss, beam, rib, plate, membrane, shell
Temperature	truss, beam, rib, plate, membrane, shell
Fault in length	truss, beam
Tension/Compression	truss, beam
Forced support displacement	support
Fluid	plate, shell
Seismic	node
Influence line	truss, beam
Tensioning	beam,rib
Moving	beam, rib, plate, shell

Load	*Element*
Snow	beam, rib, plate, shell
Wind	beam, rib, plate, shell
Fire	beam, rib, truss, membrane, plate, shell

4.10.2 Load combinations



Lets you define load combinations of the defined load cases. You can specify a factor for each load case in a load combination.
The results of a load combination will be computed as a linear combination of the load cases taking into account the specified load case factors. A zero factor means that the respective load case does not participate in the load combination. To find the most unfavourable of the custom combinations defined here create an envelope for the combinations.
Load combinations can be listed, defined and deleted by load cases or by load groups (the second option is easier to overview).
In the former case a factor must be entered for each load case.
In the latter a factor must be entered for each load group. The actual load combinations will be created using these factors and according to the load group properties (e.g., if the load cases within the load groups are exclusive or not. See the previous chapter for details). Combining load groups with many load cases may result in huge number of individual load combinations.

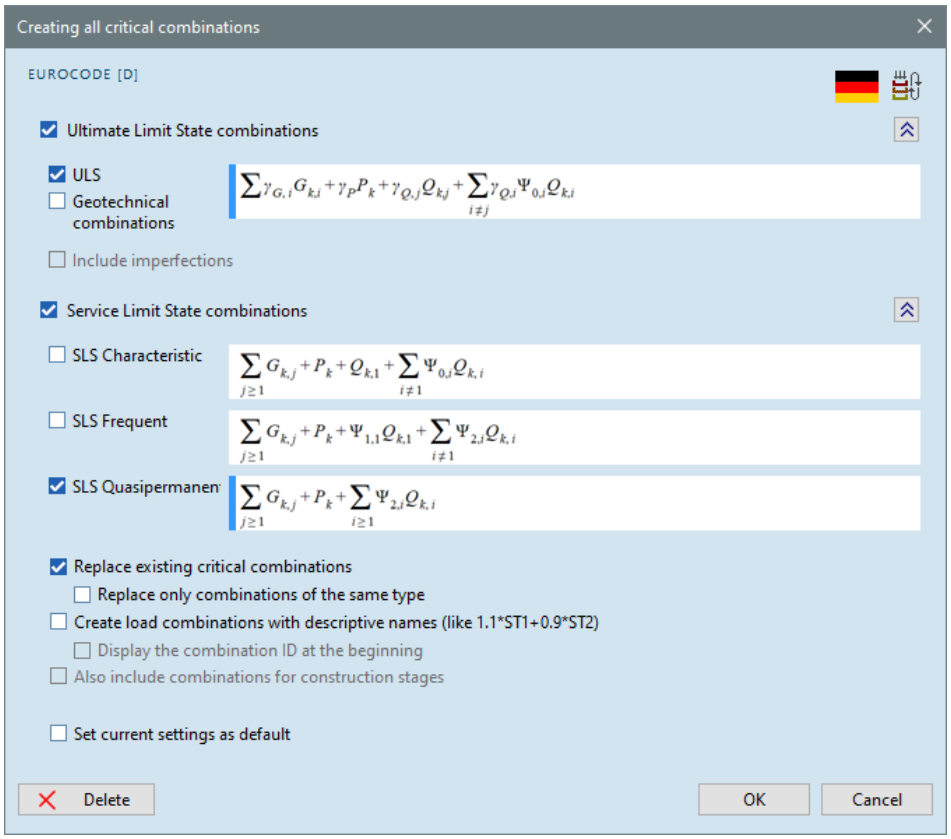
☞ **It is not necessary to create load combinations to determine the critical combination.**

If load groups are defined and load cases are placed within load groups the program automatically finds the critical combination in each node of the structure without actually creating the combinations.

Inserts a load combination table to the current report.

Pastes the load combinations collected on the Clipboard.
When finding minimum and maximum values or displaying the results for an individual element the user can add the actual critical combination to a list stored on the Clipboard (see **6.1.1 Minimum and maximum values**). This is the icon to paste these combinations into the table.

Calculates all critical combinations based on load groups and transfers them into the load combination table.



The option *Include imperfections* is available only if the model contains an imperfection load group.

If the *Replace critical combinations* is checked all previously generated critical combinations will be deleted and replaced with the new combinations. If the *Replace only combinations of the same type* is checked only combinations from the selected ULS/SLS combination types will be replaced.

Checking *Create load combinations with descriptive names* changes the naming convention, so generated combination names will be the description of the combination (like 1.1*ST1+0.9*ST2) instead of Co. #1, Co. #2, etc.

☞ **You can also define load combinations after you have completed a linear static analysis. The postprocessor computes the results of these load combinations when required**

In case of nonlinear static analysis, AXISVM first generates the combination case, and then performs the analysis (one load combination at a time).

Automatic load combination

The program investigates all possible combinations depending on the load group parameters and the equations of the current design code.
The minimum and maximum result values of these combinations are selected as critical (design) values.
Critical combinations for Eurocode, SIA26x, Italian code are assembled according to the following schemes:

Critical combinations of internal forces (ULS)

ULS1 - Permanent and Incidental

Eurocode

AXISVM uses the combination formulas below according to EN 1990 (6.10.a) and (6.10.b). These formulas result in smaller forces and displacements making the design more economical.

$$\max \begin{Bmatrix} \sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} \Psi_{0,1} Q_{k,1} + \sum_{i>1} \gamma_{Q,i} \Psi_{0,i} Q_{k,i} \\ \sum_{j \geq 1} \xi_j \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} Q_{k,1} + \sum_{i>1} \gamma_{Q,i} \Psi_{0,i} Q_{k,i} \end{Bmatrix} \begin{matrix} 6.10.a \\ 6.10.b \end{matrix}$$

☞ **If unfavourable, the partial safety factors are multiplied by K_{FI} in order to consider the consequence class. See 4.10.1 Load cases, load groups.**

In certain countries (e.g., Austria, Germany) the national annex does not allow to use (6.10.a) and (6.10.b). In this case the following formula is used:

Eurocode(A), Italian code

$$\sum_{j\geq 1}\gamma_{G,j}G_{k,j}+\gamma_P P+\gamma_{Q,1}Q_{k,1}+\sum_{i>1}\gamma_{Q,i}\Psi_{0,i}Q_{k,i}$$

SIA26x

$$\sum_{j\geq 1}\gamma_{G,j}G_{k,j}+\gamma_P P+\gamma_{Q,1}Q_{k,1}+\sum_{i>1}\Psi_{0,i}Q_{k,i}$$

Partial factors for geotechnical limit states verification (GEO)

In case of geotechnical combinations, the program uses the above formulae, as well. The partial factors for actions are selected from the groups of values A1 and A2, which are categorized according to annex A of EN 1997-1.

Action		Symbol	Set	
			A1	A2
Permanent	unfavourable	γ_G	1,35	1,0
	favourable		1,0	1,0
Variable	unfavourable	γ_Q	1,5	1,3
	favourable		0	0

Remark: the values may be specified differently by the National Annex. The specifications of the selected NA are considered.

Partial factors for equilibrium limit state (EQU) verification

In case of verification of static equilibrium, the following partial factors are considered (annex A of EN 1997-1):

Action		Symbol	Value
Permanent	unfavourable (destabilising)	$\gamma_{G;dst}$	1,1
	favourable (stabilising)	$\gamma_{G;stb}$	0,9
Variable	unfavourable (destabilising)	$\gamma_{Q;dst}$	1,5
	favourable (stabilising)	$\gamma_{Q;stb}$	0

ULS 2 - Seismic

Eurocode, SIA26x

$$\sum_{j\geq 1}G_{k,j}+P_k+A_{Ed}+\sum_{i\geq 1}\Psi_{2,i}Q_{k,i}$$

Italian code

$$\sum_{j\geq 1}G_{k,j}+P+E+\sum_{i\geq 1}\Psi_{2,i}Q_{k,i}$$

ULS 3 - Exceptional

Eurocode and other codes

$$\sum_{j\geq 1}G_{k,j}+P_k+A_d+\Psi_{1,1}Q_{k,1}\sum_{i\neq 1}\Psi_{2,i}Q_{k,i}$$

SIA26x

$$\sum_{j\geq 1}G_{k,j}+P_k+A_d+\Psi_{2,1}Q_{k,1}\sum_{i\neq 1}\Psi_{2,i}Q_{k,i}$$

Critical combinations of displacements (SLS)

SLS 1 - Characteristic

Eurocode and other codes

$$\sum_{j\geq 1}G_{k,j}+P_k+Q_{k,1}+\sum_{i\neq 1}\Psi_{0,i}Q_{k,i}$$

SLS 2 – Frequent

Eurocode and other codes

$$\sum_{j\geq 1} G_{k,j} + P_k + \Psi_{1,1} Q_{k,1} \sum_{i\neq 1} \Psi_{2,i} Q_{k,i}$$

SLS 3 - Quasipermanent

Eurocode and other codes

$$\sum_{j\geq 1} G_{k,j} + P_k + \sum_{i\geq 1} \Psi_{2,i} Q_{k,i}$$

Eurocode(RO)

$$\sum_{j\geq 1} G_{k,j} + 0.6\gamma_I A_{Ek} + \sum_{i\geq 1} \Psi_{2,i} Q_{k,i}$$

Critical load combination method for internal forces and for displacements are selected automatically. Critical load combination method for displacements depends on the type of structure you are modeling. Click *Result display parameters* on the *Static* toolbar to set the critical combination formula.

Seismic loads: see above at Internal forces.

Italian code

Combination of seismic loads with other load types: $\sum G_K + \gamma_I \cdot E + \sum \Psi_{ji} Q_{Ki}$

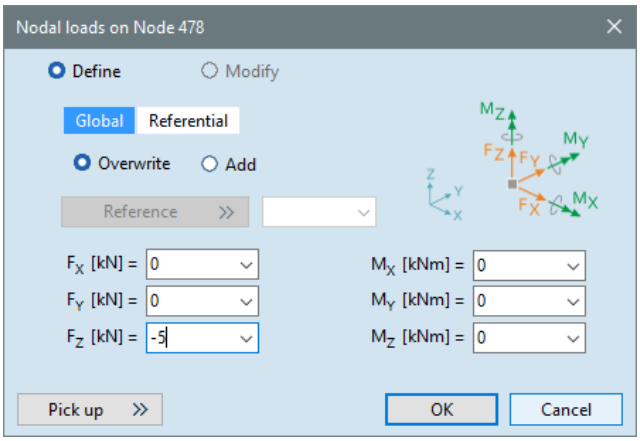
Where:

γ_I	importance factor
E	seismic load
G_K	characteristic value of permanent loads
Q_{Ki}	characteristic value of variable loads
Ψ_{ji}	Ψ_{2i}(ULS) combination factor for quasipermanent Q_i
Ψ_{0i}	(DLS) combination factor for rare Q_i

4.10.3 Nodal loads



Lets you apply forces / moments to the selected nodes. You must specify the values of the load components *(FX, FY, FZ, MX, MY, MZ)* in the global coordinate system. For a referential load (where the direction is given by the reference) enter an *Fx* and an *Mx* component.
If you apply a nodal load to a node that is already loaded, you can overwrite or add it to the existing load.



The positive directions are according to the positive directions of global coordinate axes.

Modify nodal loads

You can select, move, copy or modify the load independently of the node.

Modify position

1. Select the loads you want to move together.
2. Grab any of them by pressing the left mouse button.
3. Move them to their new position.
4. Click the left mouse button or use a command button. (Enter or Space).

Modify value

1. Select the load.
2. Click the *Nodal Load* icon on the toolbar.
3. Change the values

Nodal loads can be moved onto a beam, a rib or a domain.
Signs of the load values are calculated according to the right hand rule.

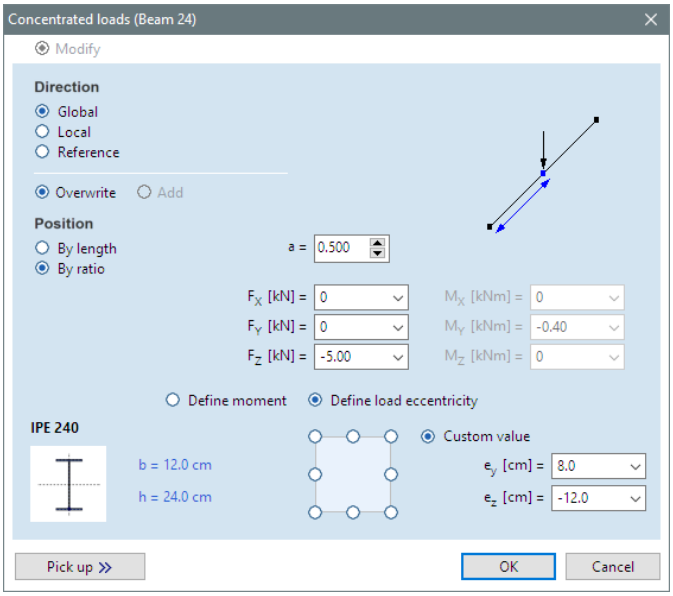
☞ **Load components applied in the direction of a constrained degree of freedom will be not taken into account in the analysis.**

The forces are displayed on the screen as yellow arrows, the moments as green double arrows.

4.10.4 Concentrated load on beam



Lets you apply concentrated forces / moments to the selected beam or rib elements. You must specify the values of the load components *(Fx, Fy, Fz, Mx, My, Mz)* in the local or global coordinate system. For a referential load (where the direction is given by the reference) enter an *Fx* and an *Mx* component.
The reference can be selected from the list or picked up from the view (if display of references is activated, see **2.16.21 Display options**) clicking on the *Reference>>* button.
Instead of defining moment components it is also possible to set the e_y and e_z eccentricity of the load in the local coordinate system.



Choose *Define load eccentricity*, then click on a radio button representing a point of the bounding rectangle of the cross-section or enter custom values.
If you apply a concentrated load to a node that is already loaded, you can *Overwrite* or *Add* it to the existing load.
Concentrated loads can be selected, moved, copied, modified independently of the beam. Modify load values like in case of nodal loads

The positive directions are in accord with the positive directions of the local or global coordinate axes.

☞ **If only some part of the structural member is selected (i.e., certain finite elements) then loads will be interpreted in the local system of finite elements. In this case the same load will be applied to all selected finite elements.**

The forces are displayed on the screen as yellow arrows, the moments as green double arrows.

Interpretation of the loads of eccentric elements

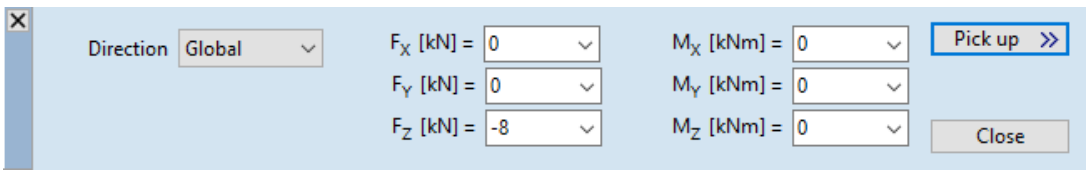
In case of eccentric beams or ribs, the actual axis of the element is not the same as the line connecting the end nodes. Load eccentricity is relative to the actual (eccentric) axis of the element.

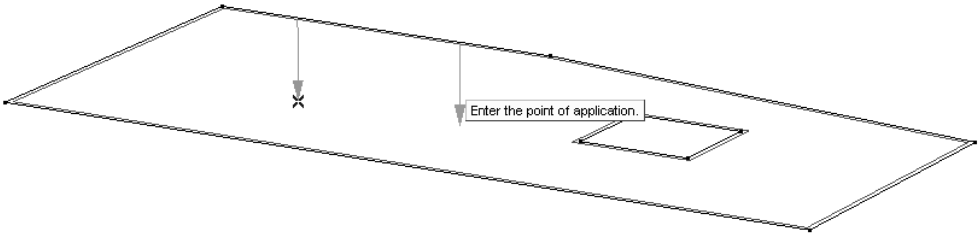
4.10.5 Point load on domain or load panel



Applies a point (concentrated) load at the location of the cursor if it is over a domain or a load panel. You can also enter the location of the load by its coordinates. Place loads by clicking the left mouse button or pressing any of the command buttons.
See **4.7.2 Entering coordinates numerically**

The direction of the load can be:
Global (with respect to the global coordinate system)
Local (with respect to the local (element) coordinate system)
Reference (with respect to a reference)





Modify point load on domain

You can modify the location and value (intensity) of the load:

Modify position

1. Select the load with the cursor (a load symbol appears beside the cursor).
2. Keep left mouse button depressed.
3. Move the mouse or enter the relative coordinates to move the load to a new location.
4. Release left mouse button to set the load in its new location.

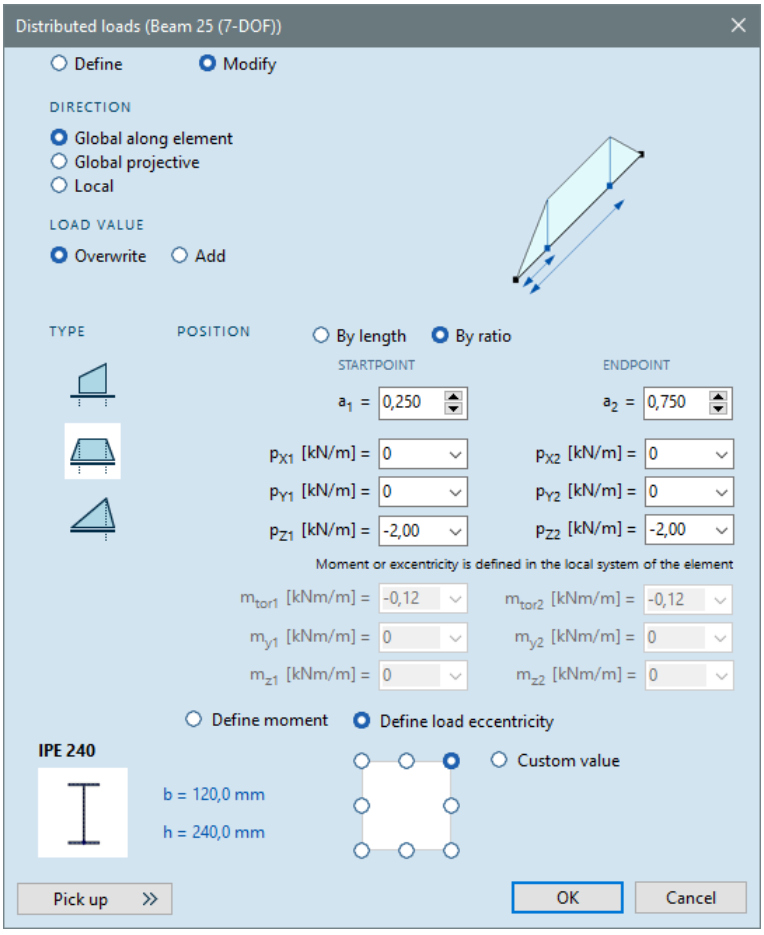
Modify value

1. Select the load with the cursor.
2. Click the left mouse button.
3. Enter the new load values in the dialog.
4. Click on the *Modify* button to apply the changes and close the window.

The load value can also be changed in the Table Browser.

☞ **Modifying domain mesh leaves the concentrated loads (applied on the domains) unchanged.**

4.10.6 Distributed line load on beam/rib



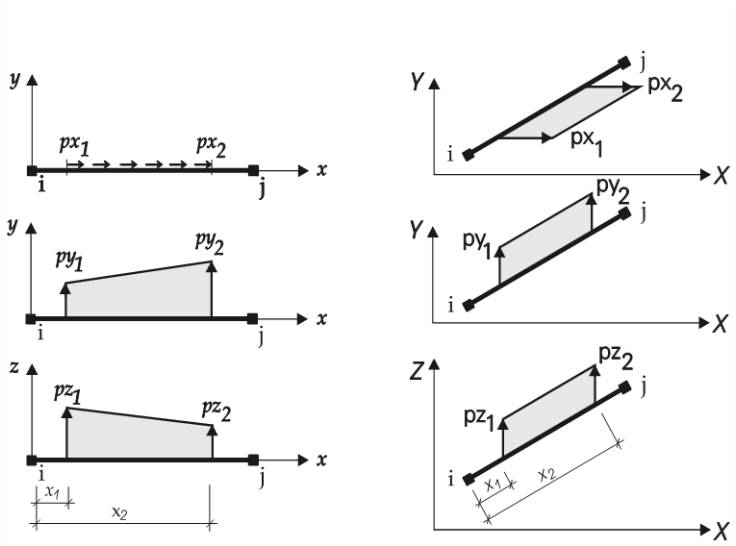
Lets you apply constant or linearly distributed forces and moments to the selected beam / rib elements. You can apply multiple distributed loads to a beam/rib in the same load case.

Instead of defining moment components it is also possible to set the e_y and e_z eccentricity of the load in the local coordinate system. Choose *Define load eccentricity*, then click on a radio button representing a point of the bounding rectangle of the cross-section or enter custom values.
Moment and eccentricity is always defined in the local system of the element even if the *Direction* of force components are set to *Global*.
Line loads can be selected, moved, copied, modified independently of the beam or rib. Modify load values like in case of nodal loads. If you apply a concentrated load to the same position that is already loaded, you can *Overwrite* or *Add* it to the existing load.

☞ **If only some part of the structural member is selected (i.e., certain finite elements) then loads will be interpreted in the local system of finite elements. In this case the same load will be applied to all selected finite elements.**

You must specify the distribution, the location and the values of the load components in the local or global coordinate system as follows:

Loads in local coordinate system *Loads in global coordinate system*



You have to specify the following parameters:

Type	 *segment*  *trapezoid*  *triangle*
Position	*By length:* between d_1 and d_2 [m], *By ratio:* between a_1 and a_2
Starting location	d_1[m] or a_1 in the local coordinate system
Starting value	p_{x1}, p_{y1}, p_{z1} [kN/m] forces, m_{TOR1} [kNm/m] torque
End location	d_2[m] or a_2 in the local coordinate system
End value	p_{x2}, p_{y2}, p_{z2} [kN/m] forces, m_{TOR2} [kNm/m] torque

By ratio: $0 \le x_1 < x_2 \le 1$, *By length:* $0 \le x_1 < x_2 \le L$, where L [m] is the length of the element

If the load is projective, the value of the load that is applied to the beam/rib is p*sin α, where α is the angle of the load direction and the beam/rib axis.

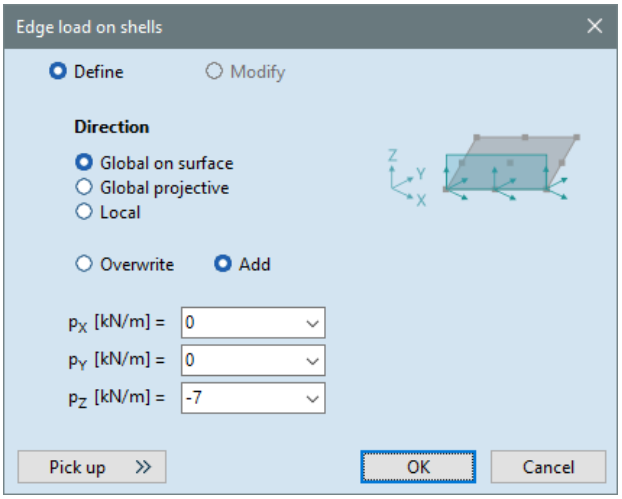
Interpretation of the loads of eccentric elements

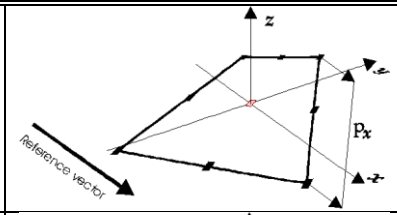
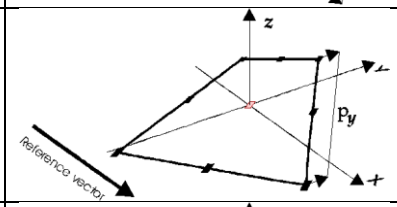
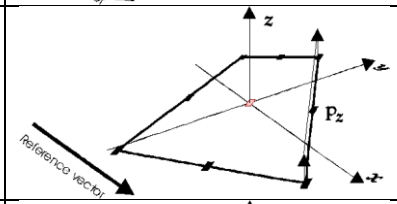
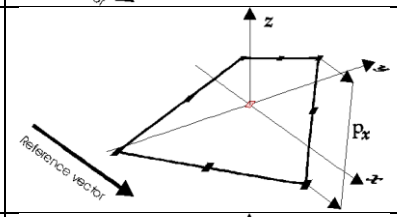
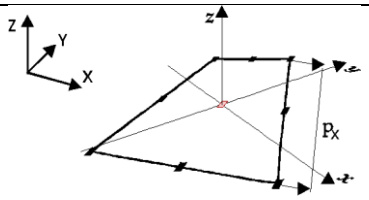
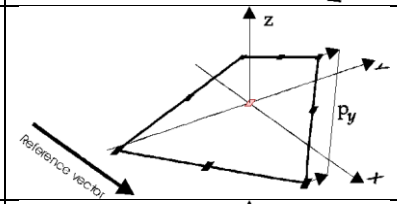
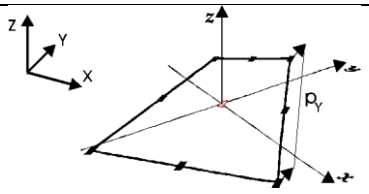
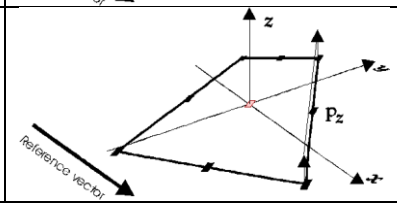
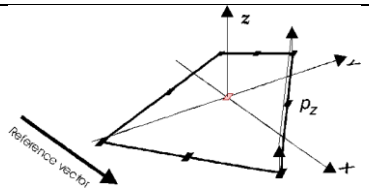
In case of eccentric beams or ribs, the actual axis of the element is not the same as the line connecting the end nodes. Load eccentricity is relative to the actual (eccentric) axis of the element.

4.10.7 Edge load



Lets you apply distributed (constant) loads to the selected edges of the selected surface elements.
If more than two finite elements are connected to the edge or they have different local coordinate systems you have to select both the edge and the finite element when you specify the local load. Load will be defined in the local system of the selected element.



Element	*Load in local directions (in local coordinate system)*		*Load in global directions (in global coordinate system)*	
Membrane	*x*		-	-
	y		-	-
Plate	*z*		-	-
Shell	*x*		*X*	
	y		*Y*	
	z		*Z*	

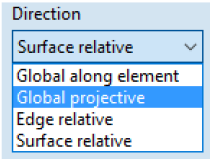
In the case of shell elements, the load that is applied in global coordinate directions can have a projective distribution. If the load p is projective, the value of the load that is applied to the shell is $p \cos \alpha$, where α is the angle of the load direction and the element plane normal.

4.10.8 Domain / Load panel line load

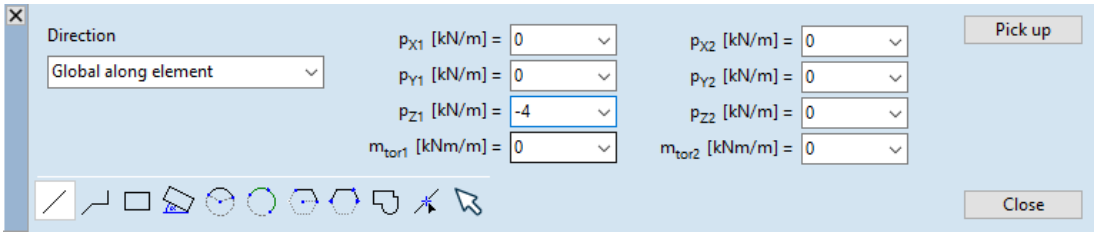


Applies a uniform or linear distributed line load over a domain or a load panel.

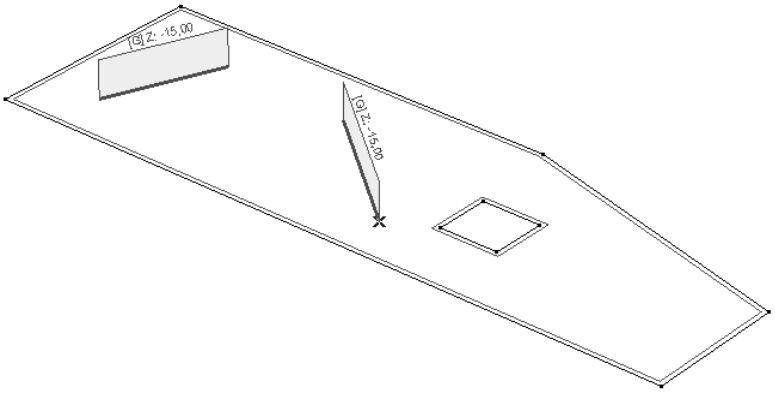
The direction of the load can be global projective, global along element, edge relative or surface relative.
m_x is always the torsional moment (around the application line of the load). Set load components and placement method then draw the load (or click the lines) to place it.



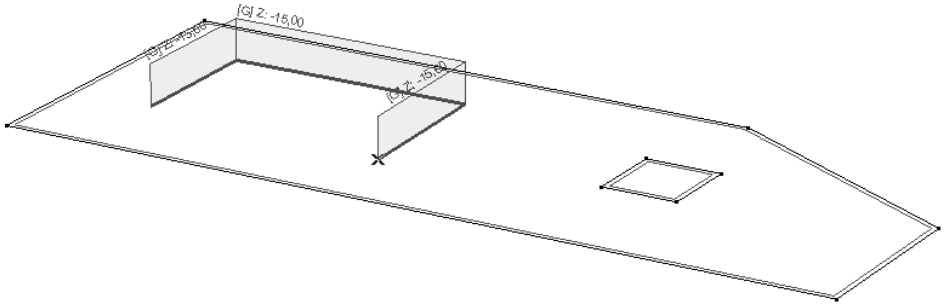
In case of *Edge relative* mode, the local coordinate system of the line load follows the direction of the specified edge (line, arc or polyline). The *x* axis is always the axis of the specified line pointing towards the drawing direction. The local *z* axis of the load is parallel with the local *z* axis of the domain. When placing a line load on an existing edge of the model, always check the direction of the resulting load, as the drawing direction of the selected line cannot be determined from node numbers.
If the line load is defined as Surface relative, then the local coordinate system of the load (x, y and z axes) follows the local coordinate system of the domain



Line load between two points



Line load along a polyline





Line load along a rectangle



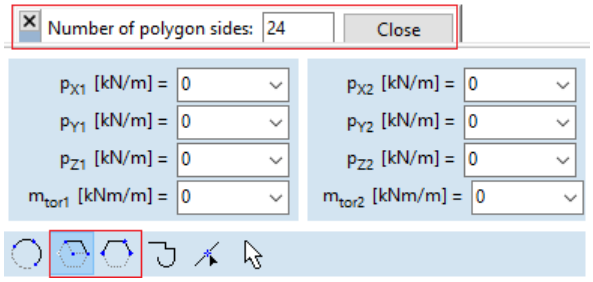
Line load along a rotated rectangle



Line load along an arc defined by its center point and two points



Line load along an arc defined by three points



Line load along an arc polygon defined by its center point and two points

Line load along an arc polygon defined by three points

Line load along a complex polygon. Complex polygons can contain arcs



During definition of a complex polygon a pet palette appears with several geometry functions. These are: drawing a line, drawing a line as a tangent of an arc, drawing an arc with center point, drawing an arc with a midpoint, drawing an arc with the tangent of the previous polygon segment, drawing an arc with a given tangent, picking up an existing line.

Distributed line load on an existing line or arc

Click any line or arc on the domain boundary or within the domain to apply the load previously defined. This type of load is associative.
Moving the boundary or the internal line moves the load as well. Deleting the line deletes the load.

Line load by selection

Similar to the previous function, but the load will be applied to the selected lines.

You can modify the location and value (intensity) and any vertex of the load polyline:

Modify location

1. Select the load with the cursor.
2. Keep left mouse button depressed.
3. Move the mouse or enter the relative coordinates to move the load to a new location.
4. Release left mouse button to set the load in its new location.

Modify shape

1. Move the cursor above the vertex (a load polyline vertex symbol appears beside the cursor).
2. Click the left mouse button
3. Drag the vertex to its new position after pressing the left mouse button.
4. Click the left mouse button.

Modify value

1. Select the load with the cursor (a load symbol appears beside the cursor).
2. Click the left mouse button.
3. Enter new load values in the dialogue window.
4. Click on the Modify button to apply the changes and close the window.

The load value can also be changed in the Table Browser.

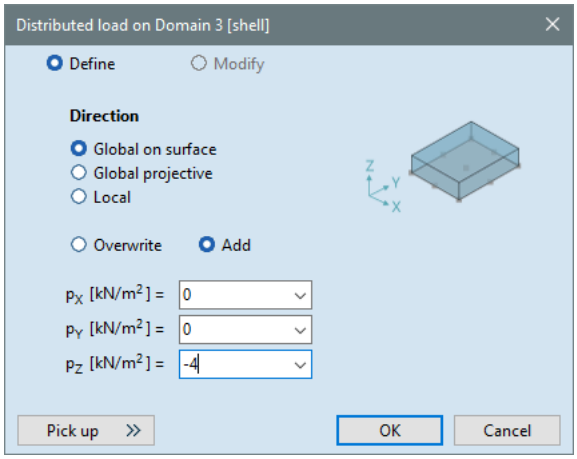
Delete Select the loads you want to delete and press **Delete**.

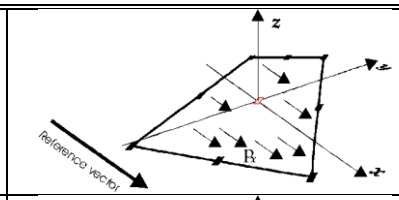
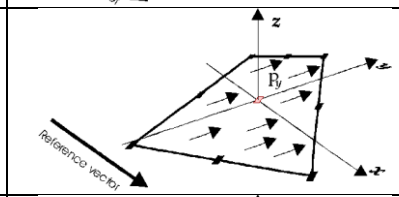
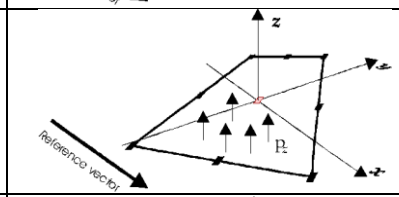
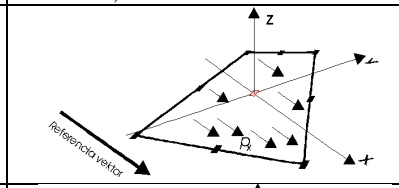
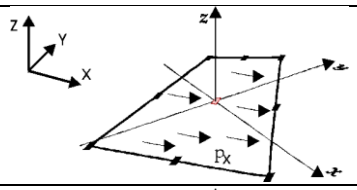
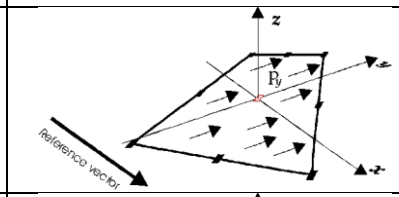
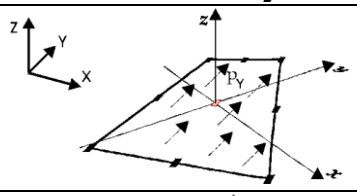
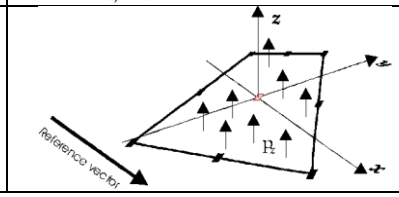
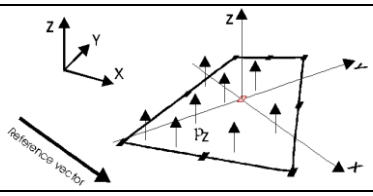
☞ **Modifying domain mesh leaves line loads (applied on the domains) unchanged.**

4.10.9 Surface load

Surface load can be applied on surface finite elements and domains.
The intensity of a distributed load on a surface element is constant.

☞ **Modifying domain mesh leaves the loads (applied on the domains) unchanged.**



Element	*Load in local directions (in local coordinate system)*		*Load in global directions (in global coordinate system)*	
Membrane	*x*		-	-
	y		-	-
Plate	*z*		-	-
Shell	*x*		*X*	
	y		*Y*	
	z		*Z*	

4.10.10 Domain / Load panel area load

Applies a mesh-independent area load to a domain or a load panel.

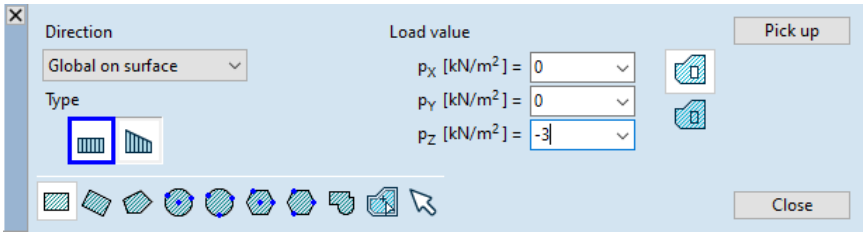
The domain element type determines the load type and direction as follows.
For a *membrane* domain the load must be in the plane of the domain. For a *plate* domain the load must be perpendicular to the plane of the domain. For a *shell* domain any load direction is acceptable. Load panels accept loads in any direction but if they transfer loads to membrane or plate domains the above limitations apply.
The load can be a global load on surface, a global projective load or a local load and the components will be interpreted accordingly.
You can select between constant or linear load intensities and set if loads disapper over holes or are distributed on the edge of the hole.

Loads disappear/ allowed on holes

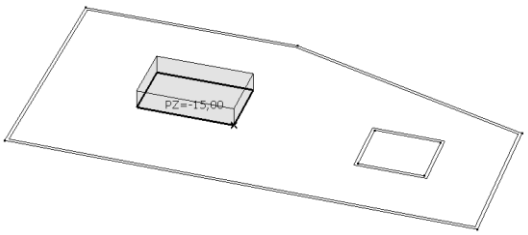
The first icon represents the option that loads over holes are not applied to the structure. The second one represents the option that loads over holes are distributed on the edge of the hole.

Steps of load definition in case of constant load:

Constant load

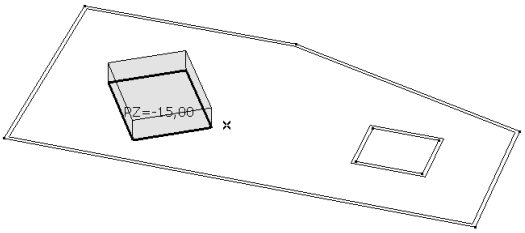


Rectangular area load



1. Enter load components (p_X, p_Y, p_Z)
2. Enter two diagonal end points of the rectangle by clicking or by coordinates.
(This function is available only on the X-Y, Y-Z and X-Z planes)

Skewed rectangle area load



1. Enter load components (p_X, p_Y, p_Z)
2. Enter three corners of the rectangle by clicking or by coordinates.
(This function is available only on the X-Y, Y-Z and X-Z planes)

Polygon load

1. Enter load components (p_X, p_Y, p_Z)
2. Enter polygon vertices by clicking or by coordinates. In this latter case press an extra Enter after specifying the last position. If you enter the polygon by clicking on the domain close the polygon by clicking on the first vertex again or by double-clicking at the last vertex. Instead of the left mouse button you can also use Space or Enter key to enter polygon vertices.

Disk or sector load defined by centerpoint and two points

Disk or sector load defined by three points

Regular polygon load defined by centerpoint and two points

Regular polygon load defined by three points

Complex polygon load



During definition of a complex polygon a pet palette appears with several geometry functions. These are: drawing a line, drawing a line as a tangent of an arc, drawing an arc with center-point, drawing an arc with a midpoint, drawing an arc with the tangent of the previous polygon segment, drawing an arc with a given tangent, picking up an existing line.



Distributed domain load

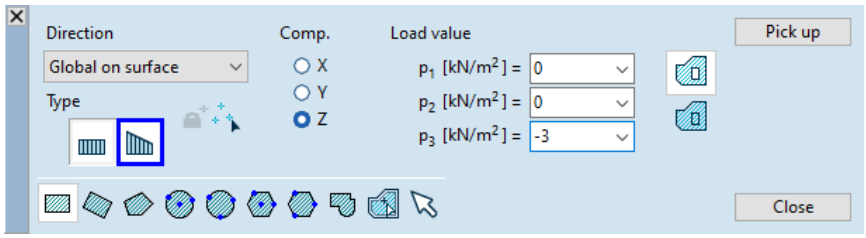
1. Enter load components (p_X, p_Y, p_Z)
2. Click on the domain

The load will be distributed over the domain. The shape of this type of load will automatically follow any change in the domain geometry. Within a load case you can apply only one load of this type on a domain. New distributed domain load definition always overwrites the previous one.



Linear load

Steps of load definition in case of linear load:



The plane of the load intensity can be specified by load intensity values (p_1, p_2, p_3) at three points [(1), (2), (3)] in the plane of the domain. These points are the load value reference points. If you want to use the same reference points and values to many loads of different shape and position you can lock the reference points and values by clicking the *Lock* button. Loads are applied by entering an area.



Define load value reference points



Lock/unlock value reference points



Rectangle area load

1. Enter load values at the reference points (p_1, p_2, p_3).
2. Enter two diagonal end points of the rectangle by clicking or by coordinates. (This function is available only on the X-Y, Y-Z and X-Z planes)
3. Enter three reference points by clicking or by coordinates.



Skewed rectangle area load

1. Enter load values at the reference points (p_1, p_2, p_3).
2. Enter three corners of the rectangle by clicking or by coordinates.
3. Enter three reference points by clicking or by coordinates.



Polygon load

1. Enter load values at the reference points (p_1, p_2, p_3).
2. Enter polygon vertices by clicking or by coordinates. In this latter case press an extra Enter after specifying the last position. If you enter the polygon by clicking on the domain close the polygon by clicking on the first vertex again or by double-clicking at the last vertex. Instead of the left mouse button you can also use Space or Enter key to enter polygon vertices.
3. Enter three reference points by clicking or by coordinates.



Sector or disc shaped load defined by center-point and two points



Sector or disc shaped load defined by three points



Arc polygon shaped load defined by center-point and two points



Arc polygon shaped load defined by three points



Complex polygon shaped load



During definition of a complex polygon a pet palette appears with several geometry functions. These are: drawing a line, drawing a line as a tangent of an arc, drawing an arc with center-point, drawing an arc with a midpoint, drawing an arc with the tangent of the previous polygon segment, drawing an arc with a given tangent, picking up an existing line.



Distributed domain load

1. Enter load values at the reference points (p_1, p_2, p_3).
2. Click on the domain.
3. Enter three reference points by clicking or by coordinates.

Within a load case you can apply only one load of this type on a domain. New distributed domain load definition always overwrites the previous one.

Modify area load

The position, shape and intensity of a mesh-independent area load can be changed.

Modify position

1. Place the mouse above the load contour (the cursor will identify the load).
2. Press the left mouse button and move the mouse.
3. Find the new load position by moving the mouse or by coordinates.
4. Drop the load by clicking the left mouse button or pressing the Space or Enter key.

Modify shape

1. Place the mouse above a vertex of the load polygon (the cursor will identify the load polygon vertex as a corner).
2. Press the left mouse button and move the mouse.
3. Find the new vertex position by moving the mouse or by coordinates.
4. Place the vertex by clicking the left mouse button or pressing the Space or Enter key. The load shape will change.

Modify intensity

1. Place the mouse above the load contour (the cursor will identify the load).
2. Click the left mouse button. The area load windows appears.
3. Change the load intensity values.
4. Click on the Modify button to confirm the changes.

Multiple loads can be selected and modified this way.

Area load intensity and shape can also be changed in the Table Browser by changing the appropriate values in the load table.

Delete

Select the loads to delete and press **[Del]**

☞ **Mesh-independent loads are not affected by removing meshes or re-creating meshes on domains.**

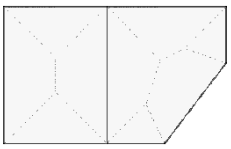
4.10.11 Surface load distributed over line elements



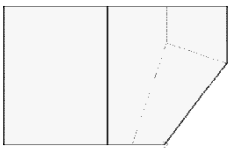
Homogenous surface load can be placed over line elements (trusses, beams and ribs).
Loads over trusses will be converted into loads on the truss end nodes.

1. Click the icon and select the load distribution range in the dialog.

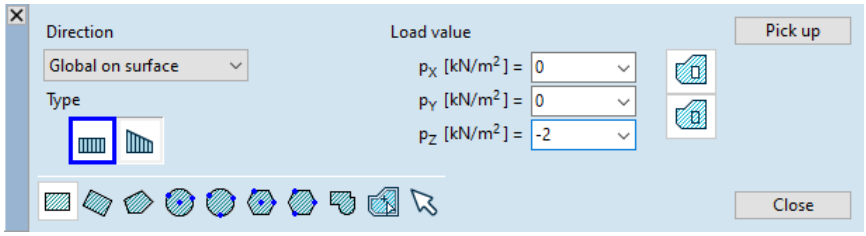
Auto distributes the load over the elements under the load. Any new truss, beam or rib defined under the load will redistribute the load.



To selected elements only distributes the load over the selected elements only. Select lines using the selection toolbar. Distribution remains the same if a new beam or rib is defined under the load.



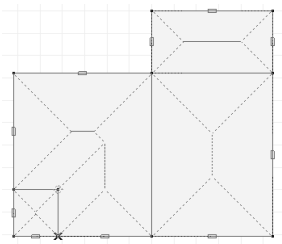
2. Define load polygon the same way as for a constant or linearly changing domain area load.



Load direction can be global on surface, global projective or local. Local directions are defined like automatic references for domains See **4.9.25 References**.

Enter load values into the edit fields. (p_X, p_Y, p_Z)

The load polygon can be a rectangle, a skewed rectangle or any closed polygon. The fourth method on the icon tollbar is to click lines of a closed beam/rib polygon. This way the load becomes associative. Moving the elements or their end nodes changes the load polygon accordingly.



Edit / Convert surface loads distributed over beams menu item converts loads created this way to individual beam loads.

4.10.12 Load panels



Load panels are used to transfer nodal loads, distributed line and surface loads to the structural elements. It is also necessary to define load panels, if snow and wind loads are generated automatically by the software (SWG module). They have no structural role, the only function of a load panel is to distribute loads over the domains, beam and rib elements. Loads can be distributed to all domains, ribs and beams automatically, that are in the plane of the panel. It is also possible to specify manually the load bearing elements within or out of the plane of the panel controlling the load transfer. Their projection to the load panel should be within the closed contour of the load panel.



Shape tools



Rectangular



Slanted rectangle



Sector or full circle by center-point and two points



Sector or full circle by three points



Sector or regular polygon by center-point and two points



Sector or regular polygon by three points



Complex polygon

Special selection tools



Clicking on a domain creates a load panel over the domain



Creating load panels for the selected domains



This function *creates load panels based on selected lines*. Load panel outlines are determined from the outlines of coplanar subsets of selected lines.



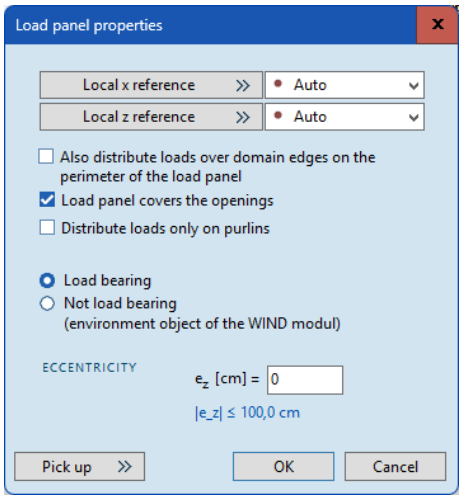
Search surface elements
A useful function when a load (e.g. wind load) must be applied on complex curved surfaces that cannot be covered with a domain or beam structures following a curved surface. The procedure generates separate load panels for each surface found for the selected lines.



Select the load bearing elements under the load panel.
The elements taken into account by the load panel during load distribution can be modified manually.



Properties.
This dialog is to modify other properties of the load panel. You can configure the local system, enable special load distribution modes and specify whether the load panel should also cover domain openings. Purlins are those line elements for which the purlin properties are defined (See **4.9.10 Line elements, 4.9.11 Modeling of cold-formed steel purlins**)
AXISVM uses two different types of load panels. One is the load-distributing *(Load-bearing)* panel, whilst the other type of load panel *(Not load bearing)* is used in the WIND module as environmental objects in wind tunnel analysis. These load panels alter the flow, but no wind load is generated on them.



Also distribute loads over domain edges on the perimeter of the load panel
Here you can specify whether, if the edge of a load panel falls on the edge of a domain, loads should be distributed over that domain edge or not.

Load panel covers the openings
It has the same function as the two buttons *Load options for openings* (see below).

Distribute loads only on purlins
In the case of beam and rib elements, the load panel distributes the load only to elements that have their purlin property enabled. See **4.9.10 Line elements**

Load distribution modes



Select this option to distribute the load on **all domains, ribs and beams** under the load panel.



Load is distributed to **all domains,** and **selected ribs and beams.**
Loads on load panels can be distributed over selected nodes as well.
Loads can be distributed over nodes and beam or rib elements out of the plane of the load panel if their projection to the load panel is within the load panel outline. In this case loads are distributed over the projected segments but will act in the element position.



Load is distributed to **selected domains and selected ribs and beams.**



Select this option to distribute the load on **domains, ribs and beams in active parts** under the load panel.

Load options for openings



If the first icon is selected the load panel applies loads over the openings (interpreting them as windows), otherwise it ignores the loads over the openings (interpreting them as actual openings).

Join load panels

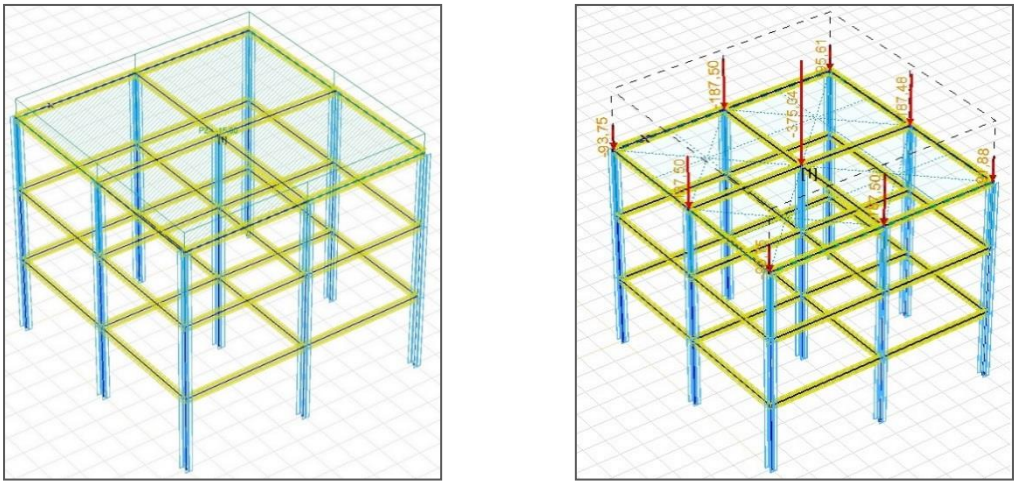


Clicking the button shown in the figure allows you to fit adjacent edges of nearby selected load panels to ensure they fit continuously without gaps. This function can be important when generating meteorological loads, as improperly fitted load panels may cause errors during the calculation of load values or when defining the load model. A maximum of three load panels can be selected at a time.

See **4.10.13 Snow load – SWG module**
See **4.10.14 Wind load – SWG module**
See **4.10.15 Simulated wind used in wind tunnel analysis – WIND module**
See **4.10.16 Generating wind load from a CFD (Computational Fluid Dynamics) file**

Loads from the selected load panels can be converted to the individual loads. See **3.2.19 Convert loads of the selected load panels to individual loads.**

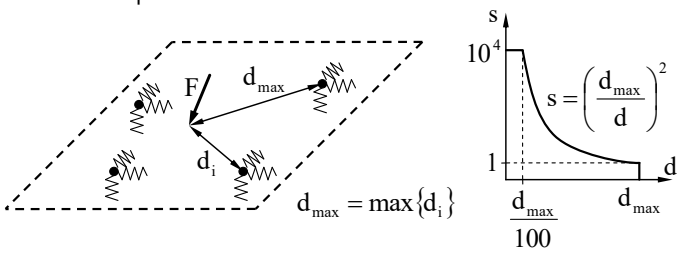
☞ **Selected beams / ribs / domains can be out of the plane of the load panel.**
If multiple domains are selected and their outlines projected to the plane of the load panel overlap, loads will be transferred only to the domain closest to the load panel.



Concentrated loads on columns from transferring a distributed load on the load panel

Method of load distribution

Distributing of a concentrated load models the load panel as a rigid sheet supported by three springs at each load distribution point. The three springs have the same stiffness inversely proportional to the square of the distance from the point of action.



Loads distributed along lines are converted to a series of point loads and the point loads are distributed with the above method. Surface loads are distributed using the method in **4.10.11 Surface load distributed over line elements.**

Defining loads

Point, line and surface loads can be placed on load panels the same way as on domains. An example:

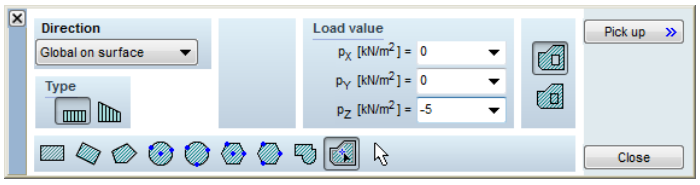


Choose the *Loads* tab and the *Distributed loads on domain* tool



Enter load value components and choose *Distributed domain load* from the palette.

Click into any number of domains or load panels. The load will be applied to them.



4.10.13 Snow load – SWG module

Design codes

Snow load is generated automatically in the program according to the regulations of several national standards and their applicable annexes. The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

Assumptions
Application limits

Applicability of the algorithm in the program is limited by the applicability of the specifications in the standards it is based on. Following is a list of such limitations for each standard in the program.



Eurocode general

- The algorithm generates snow loads on building roofs. It is not recommended for snow load generation on other types of structures, such as bridges.
- The algorithm is applicable to various roof geometries. It takes the effect of ridges and troughs on snow accumulation into account when calculating the snow effect on roof panels adjacent to the ridge or trough. It does not take into account the effect of local snow accumulation on distant (i.e., not adjacent) roof panels, therefore, it is not recommended for complex roof geometries where such effect is expected to have significant influence on the characteristic snow load. Note that neither does EC 1-3 specify snow load calculation for the latter cases.
- Building altitude shall be less than 1500 m.
- Snow impact when snow falls off a higher roof is not considered; snow load is always classified as static action.
- Ice loading is not considered.
- Lateral loading from snow is not considered.
- Exceptional snow drifts as per Annex B are not considered.
- Exceptional snow loads are not considered.
- Sliding of snow off the roof is assumed not to be prevented.
- Local snow accumulation in the vicinity of taller construction works and smaller projections on the roof are taken into account.
- Snow overhanging the edge of the roof is considered.
- Snow load on snowguards is not considered.
- The program can generate snow load corresponding to any arbitrary return period.



EC German

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well with the extensions/modifications listed below.
- The methodology in 5.3.4 for multi-span roofs is not implemented, because it leads to ambiguous load generation in 3D cases.
- Exceptional snow load with the same distribution is taken into account



NTC Italian

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.



EC Dutch

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well with the extensions/modifications listed below.
- No altitudes above 1500 m can be found in the Netherlands, thus the limit on altitude is not applicable.



EC Hungarian

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.



EC Belgian

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well with the extensions/modifications listed below.
- No altitudes above 1500 m can be found in Belgium, thus the limit on altitude is not applicable.



EC Romanian

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.



EC Czech

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.



EC Slovak

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.
- Exceptional snow load with the same distribution is taken into account.



EC Polish

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.



EC Danish

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.



EC Austrian

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well with the extensions/modifications listed below.
- The effect of solar panels as per Annex B in EC 1-3 NA is not considered.
- The influence area for snow load is assumed to be greater than 10 m^2. For structural members affected by snow from smaller areas see 9.2.1.1 in EC 1-3 NA and use an increased value of s_k for the analysis.
- For roof widths b ≥ 50 m, a load increase coefficient cA is applied in accordance EC 1-3 NA.
- The effect of short eaves (h < 0.5 m) as per 9.2.1.3 in EC 1-3 NA is not considered automatically. This effect can be considered by an appropriately extended model that includes the ground surface around building as an extension of the roof.
- The effect of graves (as per 9.2.5.4 in EC 1-3 NA) is not considered.



EC Finnish

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.



EC Norwegian

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.
- The characteristic value of snow load is calculated according to NA 4.1 even for altitudes above 1500 m.
- Exceptional snow load is not taken into account.



EC Greek

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.
- The characteristic value of snow load is calculated according to NA 4.1.



EC Portuguese

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.
- The characteristic value of snow load is calculated according to NA 4.1.
- Exceptional snow load with the same distribution is taken into account, but this type of snow load is turned off by default (see *Settings* menu/*Design codes* window on the *Loads* tab).
- Default value of C_{esl} is 2.50.



EC Spanish

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.
- The characteristic value of snow load is calculated according to NA 4.1.



EC British

- Because calculation is based on the general Eurocode, the limits presented there apply for this standard as well.
- The characteristic value of snow load is calculated according to NA.2.8.
- Exceptional snow load with the same distribution is taken into account, but this type of snow load is turned off by default (see *Settings* menu/*Design codes* window on the *Loads* tab).
- Exceptional snow drifts are not considered.



Swiss

- The algorithm generates snow loads on building roofs. It is not recommended for snow load generation on other types of structures, such as bridges.
- The algorithm is applicable to various roof geometries. It takes the effect of ridges and troughs on snow accumulation into account when calculating the snow effect on roof panels adjacent to the ridge or trough. It does not take into account the effect of local snow accumulation on distant (i.e., not adjacent) roof panels, therefore, it is not recommended for complex roof geometries where such effect is expected to have significant influence on the characteristic snow load. Note that neither does SIA261 specify snow load calculation for the latter cases.
- Building altitude shall be less than 2000 m.
- Exceptional snow loads are not considered.
- Sliding of snow off the roof is assumed not to be prevented.
- Local snow accumulation in the vicinity of taller construction works and smaller projections on the roof are taken into account.
- Snow overhanging the edge of the roof is considered.
- The software can consider the effect of snow guards.
- According to the settings, the program calculates increased snow loads of large low-slope roofs. If the *Based on maximum size option* is selected, the reference size needs to be given by the user. In this case, increased snow load is considered on every load panels. If the *By contiguous load panels* option is selected, the software analyses the size and geometry of load panel groups. Increased snow load is considered separately based on the geometry of load panel groups. This option works only for load groups that have rectangular or circular shape.

Calculation details

The logic of snow load calculation is explained below for each standard in the program



Eurocode general

- The recommended values are assumed for all coefficients unless specified otherwise by the user. None of the recommendations in National Annexes is assumed.
- The factors for combination value, frequent value and quasi-permanent value of the snow load are taken according to Table 4.1 of EC 1-3.
- The characteristic value of snow load on the ground shall be specified by the user.
- Snow load on the roof is calculated using Eq. (5.1) in EC 1-3.
- The exposure coefficient is based on the topography selected by the user. The Ce values that correspond to each topography type are taken from Table 5.1 in EC 1-3.
- The thermal coefficient is taken as 1.0 by default and shall be modified by the user if the application of a different value is justified.
- The snow load shape coefficient for roofs composed of planar panels is calculated as per Section 5.3.1 - 5.3.4 in EC 1-3.
- Snow load shape coefficients for the undrifted load case are based on μ1 in Table 5.2 in EC 1-3. Each panel has its own μ1 value that is calculated using the slope of the panel.
- Snow load shape coefficients for the drifted load case are based on μ2 in Table 5.2. μ2 coefficients for troughs are calculated as per Figure 5.4 in EC 1-3 using the slopes of the connecting roof panels in the wind direction of the given drifted snow load case. (e.g.: A horizontal trough in the X direction results in no snow accumulation when the wind blows in the X direction because the slopes of the connecting roof panels in the X direction are 0°.) When there are no troughs on a roof, the drifted load arrangements in Figure 5.3 in EC 1-3 are considered. The reduced snow intensity is always assumed on the windswept side of the roof.
- Snow load shape coefficients for cylindrical roofs are calculated as per 5.3.5. in EC 1-3. In order to achieve sufficient accuracy in the load shape, it is recommended to approximate the cylindrical roof with at least 20 planar segments.
- The effect of taller construction works and obstructions on the roof are considered as per 5.3.6 and 6.2 in EC 1-3. Their influence is only taken into account in the drifted load cases. Snow is assumed to be drifted only if the wall or obstacle is not parallel to the wind direction.
- Snow load overhanging the edge of the roof is considered as per 6.3 in EC 1-3.
- Snow load corresponding to a user-defined return period is calculated using the procedure in Annex D of EC 1-3.



EC German

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- The characteristic value of snow load on the ground is automatically calculated based on 4.1 in EC 1-3 NA using the zone and the altitude specified by the user.
- Exceptional snow load is generated. The exceptional snow load coefficient is assumed 2.0 by default, but it shall be overwritten by the appropriate value by the user for Northern Germany as per 4.3 in EC 1-3 NA.
- The effect of taller construction works is calculated as per 5.3.6 in EC 1-3 NA.



NTC Italian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground is automatically calculated based on 4.1 in EC 1-3 NA using the zone and the altitude specified by the user.
- The value of the exposure coefficient is based on 5.2 (7) in EC 1-3 NA.



EC Dutch

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground is taken as 0.7 kN/m^2 as per 4.1 (1) in EC 1-3 NA.
- The factors for combination value, frequent value and quasi-permanent value of the snow load are taken as 0, 0.2, and 0, respectively as per 4.2 (1) in EC 1-3 NA.



EC Belgian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground is automatically calculated based on 4.1 (1) in EC 1-3 NA as a function of the altitude specified by the user.
- The combination factors for frequent and quasi-permanent load combinations are based on EC 1-3 NA 4.2 (1).
- The exposure and temperature coefficients are taken as 1.0 according to the regulations in EC 1-3 NA 5.2 (7) and (8).
- When calculating the effect of taller construction works, the upper limit of μw is 2.0 as per EC 1-3 NA 5.3.6 (1).
- The effect of snow overhanging the roof is not considered (EC 1-3 NA 6.3).



EC Hungarian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground is automatically calculated based on NA1.6 in EC 1-3 NA using the altitude specified by the user.
- Exceptional snow load is generated. An exceptional snow load coefficient of 2.0 is used as per NA1.8 in EC 1-3 NA.



EC Romanian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- The characteristic value of snow load on the ground is calculated automatically based on 3.1 in CR 1-3 using the basic s_k value and the altitude specified by the user.
- Snow load on the roof is calculated using Eq. (4.1) in CR 1-3.
- The importance factor shall be selected by the user; the list of recommended values is based on Table 4.2 in CR 1-3.
- The factors for combination value, frequent value and quasi-permanent value of the snow load are taken as 0.7, 0.5, and 0.4, respectively as per Table 4.4 in CR 1-3.



EC Czech

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- The characteristic value of snow load on the ground is based on the map cited in 2.7 in EC 1-3 NA (*www.snehovamapa.cz*)
- Snow load shape coefficients for cylindrical roofs are calculated as per 2.19 and Figure NA.1 in EC 1-3 NA
- The effect of taller construction works is taken into account as per 2.20 in EC 1-3 NA with the following assumption: $b_{1,s} = 0.5b_1$.



EC Polish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- The characteristic value of snow load on the ground is calculated automatically based on 1.7, Figure 1 and Table 1 in EC 1-3 NA using the zone and altitude specified by the user.



EC Danish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground is taken as 1.0 kN/m2 as per 4.1 (1) NOTE 1 in EC 1-3 NA.
- Default snow load exposure coefficients are based on the assumption of Cs = 1.0 as per 5.2(7) in EC 1-3 NA. When a different size coefficient needs the be applied, the user shall calculate the resulting exposure coefficient and define it as custom.
- The additional snow load arrangement in 5.3.3(4) in EC 1-3 NA is not considered.
- Snow load shape coefficients for cylindrical roofs are calculated as per 5.3.5. (3) in EC 1-3 NA
- The effect of taller construction works is taken into account as per 5.3.6 in EC 1-3 NA with the following assumptions: (i) the shelter is global; (ii) bsl = 0.5b1; (iii) the shelter is on the leeward side.



EC Austrian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load should be specified by the user.
- Snow load shape coefficients for multi-span roofs are calculated as per 9.2.3 in EC 1-3 NA.
- Snow load shape coefficients for vaulted roofs are calculated as per 9.2.4.3 in EC 1-3 NA.
- The modifications for calculating the effect of taller construction works in 9.2.5.1 in EC 1-3 NA are taken into account.



EC Finnish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground shall be specified by the user based on the map in Figure 4.1 in EC 1-3 NA.
- Exposure coefficients are based on Table 5.1 in EC 1-3 NA.
- The snow load arrangement of cylindrical roofs is calculated as per EC 1-3. The unique load arrangement introduced in Figure 5.6 in EC 1-3 NA is not applied. This might lead to considerable error for shallow cylindrical roofs.
- The modifications for calculating the effect of taller construction works in 5.3.6(1) in EC 1-3 NA are taken into account. The area of the roof is assumed to exceed 6 m^2.



EC Norwegian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Drifted load arrangement for cylindrical roofs is calculated according to NA 5.3.5, but special load case for local snow loads along the eaves are not generated.
- The calculation of snow load of roofs abutting and close to taller constructions does not consider case II of NA 5.3.6(1).



EC Portuguese

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground is calculated automatically based on NA–4.1(1) using the zone specified by the user.
- Exceptional snow load can be considered, but exceptional snow drift is not available in the program.
- The default value of C_{esl} factor is 2.50.



EC Spanish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The characteristic value of snow load on the ground is calculated automatically based on the table AN.1.



EC British

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Exceptional snow load can be considered, but exceptional snow drift is not available in the program.
- The default value of Cesl factor is 2.00.
- Drifted snow load arrangement for cylincrical roofs according to NA.2.21 cannot be taken into account by the module, the general Eurocode rules are applied.



Swiss

- The recommended values are assumed for all coefficients unless specified otherwise by the user.
- The characteristic value of snow load on the ground is automatically calculated based on the reference height. The reference height (altitude) shall be specified by the user based on the map in Annex D.
- Snow load on the roof is calculated using Eq. (9) in SIA261.
- The exposure coefficient is based on the topography selected by the user. The Ce values that correspond to each topography type are taken from 5.2.4 in SIA261.
- The thermal coefficient is taken as 1.0 by default and shall be modified by the user if the application of a different value is justified.
- The snow load shape coefficient for roofs is calculated as per Section 5.3 and Figures 2 and 3 in SIA261. Each panel has its own μ value that is calculated using the slope of the panel.
- μ2 coefficients for troughs are calculated as per the second column of Figure 3 in SIA261 using the slopes of the connecting roof panels in the wind direction of the given drifted snow load case. (A trough in the X direction for example, results in no snow accumulation when the wind blows in the X direction if the slopes of the

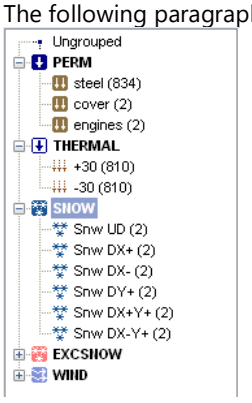
connecting roof panels in the X direction are 0°.) When there are no troughs on a roof, the drifted load arrangements in the first column of Figure 3 in SIA261 are considered. The reduced snow intensity is always assumed on the windswept side of the roof.

- Snow load shape coefficients for cylindrical roofs are calculated as per the third column of Figure 3 in SIA261. In order to achieve sufficient accuracy in the load shape, it is recommended to approximate the cylindrical roof with at least 20 planar segments.
- The effect of taller construction works and obstructions on the roof are considered as per 5.3.6 in SIA261 taking into account the recommended values for snow weight in 5.4.1 in SIA261. Their influence is only taken into account in the drifted load cases. Snow is assumed to be drifted only if the wall or obstacle is not parallel to the wind direction.
- Increased snow load is calculated for large low-slope roof according to 5.3.5 of SIA261.
- Effects of overhanging snow is taken into account according to EC 1-3 6.3.

Usage



The following paragraphs explain the usage of the automatic snow load generator module.



To apply snow loads first click on the Load cases / load groups button and define a snow load case by clicking on the snow load case button. The snow load group will be created automatically. An exceptional snow load group is also created if the checking exceptional snow is required and available in the given design code.

A temporary snow load case is automatically created in the snow load group. After the snow load parameters are specified, that load case is replaced by the generated snow load cases. The algorithm handles both undrifted and drifted load cases. Wind directions X+, X–, Y+, Y– and in 45°+*n**90° directions (where *n* = 0, 1, 2, 3) are taken into account for details and naming conventions see **4.10.1 Load cases, load groups.**

To set the snow load parameters select one of the snow loads cases as the current load case. It makes the snow load icon enabled in the *Loads* tab.

If no load panels have been created before, draw the load panels according to **4.10.12 Load panels.**

Exceptional snow load

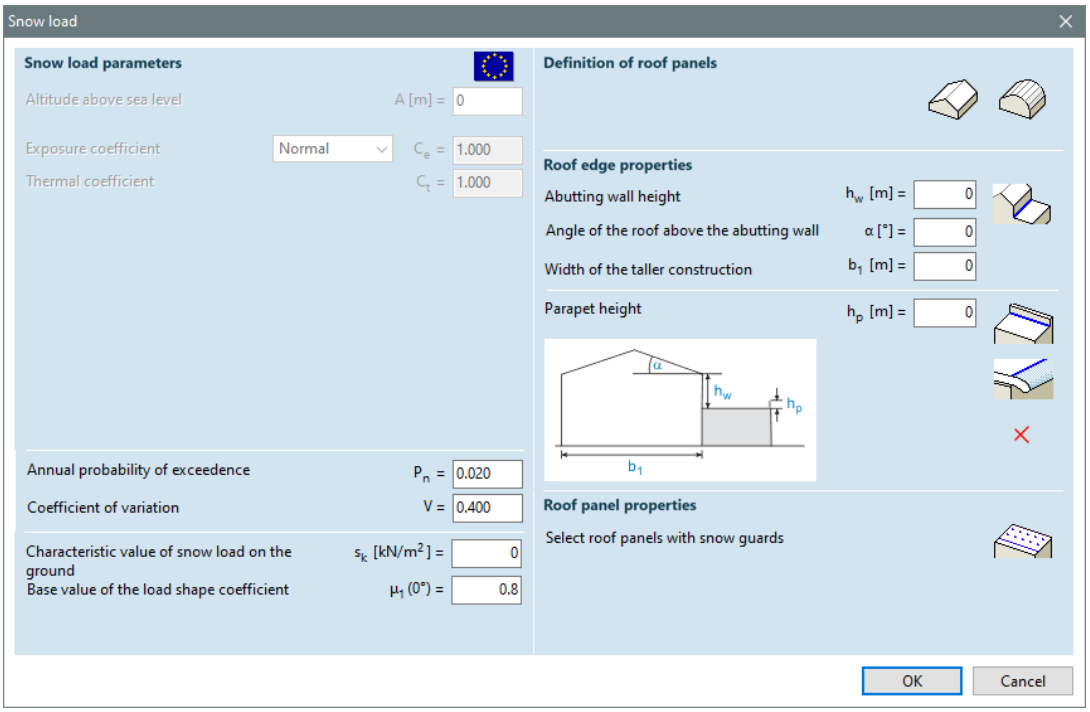
AXISVM also considers the exceptional snow load if the Eurocode national annex requires this feature. The program allows the user to neglect exceptional snow load. This can be set in *Settings / Design codes* dialog on the *Loads* tab (see **3.3.7 Design codes**). In the Spanish and British standards, the exceptional snow load is switched off by default.



To enter snow load parameters, click the snow load icon on the *Loads* tab.

The parameters dialog allows choosing pitched (or flat) roof or cylindrical roof load panels for snow load and setting the load parameters.

Snow load parameters



Two roof types can be assigned to load panels. Click on the icon and select load panels belonging to the roof.

 *Pitched (or flat) roof*

 *Cylindrical roof*

Snow load parameters:

Altitude above sea level A[m]

The C_t thermal coefficient can be set to a value other than 1.0 only if the engineer performed thermal transmittance calculations for the roof (heat loss can cause melting). In this case the program asks for confirmation then uses the custom value

Exposure coefficient C_e

In case of special circumstances an exposure factor other than 1.0 can be set depending on the topography (*windswept, normal, sheltered or other*).

A custom C_e value can also be specified. In this case the program asks for confirmation then uses the custom value when calculating snow and exceptional snow load intensity.

Thermal coefficient C_t

The C_t thermal coefficient can be set to a value other than 1.0 only if the engineer performed thermal transmittance calculations for the roof (heat loss can cause melting). In this case the program asks for confirmation then uses the custom value

Coefficient for exceptional snow loads C_{esl}

In countries where the design code requires checking exceptional snow loads the exceptional load intensity is calculated by multiplying the normal intensity by C_{esl}. Custom values can also be entered. In this case the program asks for confirmation then uses the custom value.

μ_1 (0°) Snow load shape coefficient

The snow load shape coefficient can be set manually. The default value is given by the National Annexes. (The factor only affects the snow load of flat and mono-pitched roofs.)

The factor cannot be modified in the British standard (Table NA.1 gives a value of 0.8).

Zone

In countries where the characteristic snow load depends on the geographical location the national annex divides the country into zones. The zone selected affects the characteristic snow load.

Importance factor

An importance factor can be entered depending on the classification of the building if it is required by the design code. Nonstandard values can also be used after confirmation.

Annual probability of exceedance and Coefficient of variation
Input data required to calculate snow load for a user-defined return period. Default values correspond to the standard 50-year return period.

Characteristic value of the snow load on the ground
AXISVM calculates the s_k and s_{Ad} values from the above parameters. These can be overwritten with a custom intensity but in this case changing parameters will no longer affect the load value.

AXISVM calculates snow load shape coefficients for roofs abutting and close to taller construction works or having a parapet which acts as an obstruction. It can take into account the effect of overhanging snow on roof edges.
Parameters are stored with the edges so different roof edges can have different parameters.

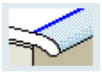
Roof edge properties



Select abutting wall edges
To define roof edges where the walls are located enter the h_w, α, b_1 parameters then click on the this icon to select the respective lines.



Select parapet edges
To define parapets enter the h_p parameter then click on the second icon to select the lines.



Roof edges with overhanging snow
Click on the icon to select the roof edges with overhanging snow.



Delete
Edge properties can be deleted by clicking the delete icon and selecting the edges.

Abutting wall height [m]
h_w height of the abutting wall relative to the roof level

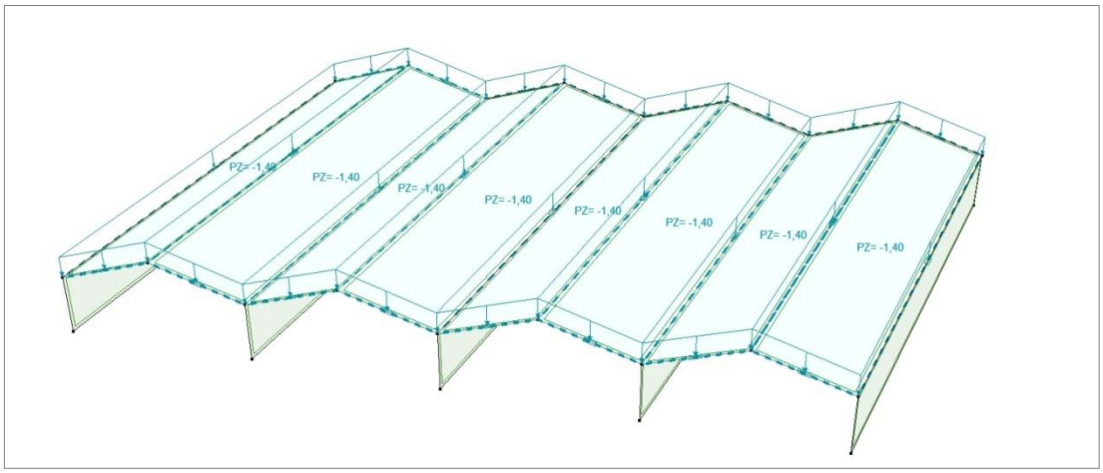
Angle of the roof above the abutting wall
α is the angle of the roof above the abutting wall. It determines the amount of snow falling from the higher roof.

Width of the taller construction
b_1 is the width of the taller construction measured perpendicularly to the wall

Parapet height
h_p is the height of the parapet or other obstruction relative to the roof level.



Select roof panels with snow guards
Click on the icon to select the roofs on which snow guards should be considered. (Cylindrical roofs cannot be selected.)



4.10.14 Wind load – SWG module

Design codes

Wind load is generated automatically in the program according to the regulations of several national standards and their applicable annexes. The design codes and national annexes in the software for which this functionality is available are listed in 3.3.7 Design codes.

Assumptions
Application limits

Applicability of the algorithm in the program is limited by the applicability of the specifications in the standards it is based on. Following is a list of such limitations for each standard in the program.



Eurocode general

- The algorithm can determine the wind load of rectangular-plan buildings, rectangular-plan canopies, free-standing walls and signboards. Buildings are structures with an internal empty space surrounded by a closed line of walls and covered by a roof.
- The following types of roofs are handled by the algorithm for buildings: flat, monopitch, duopitch, hipped, vaulted.
- The following types of canopy roofs are handled by the algorithm: monopitch and duopitch.
- The height and the width of the building shall not exceed 200 m.
- The global wind effect is determined for the structure. Hence, pressure coefficients are calculated with the assumption of at least 10 m2 of loaded area for buildings. The wind load of canopies and free-standing objects is based on force coefficients.
- The influence of neighboring structures and obstacles is assumed negligible.
- When calculating the internal pressure coefficients, the building is assumed not to have a dominant external opening. Internal pressure is only calculated for buildings.
- The structural factor c_{scd} is taken as 1.0 (corresponding applicability limits are listed in 6.2 (1) of EC 1-4).
- the influence of ice and snow on wind load is assumed negligible



EC German

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well with the extensions/modifications listed below
- building height < 300 m
- building altitude < 1100 m



NTC Italian

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well with the extensions/modifications listed below
- building altitude < 1500 m



EC Dutch

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Hungarian

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Romanian

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well with the extensions/modifications listed below
- building altitude < 1000 m (for buildings in southwest Banat and in areas with an altitude above 1000 m special consideration is required and the basic wind velocity shall be specified by the user)



EC Czech

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Slovakian

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Belgian

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Polish

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Danish

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Austrian

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Finnish

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Norwegian

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Greek

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Portuguese

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC Spanish

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



EC British

- because calculation is based on the general Eurocode, the limits presented there apply for this standard as well



Swiss

- the algorithm is only applicable to buildings with a rectangular plan; an internal empty space surrounded by a closed line of walls and covered with a roof
- building height < 200 m
- pressure coefficients are calculated as per EC 1-4; roofs of the following types are covered: flat, monopitch, duopitch, hipped, and vaulted
- wind effects are calculated for the overall load bearing structure, hence a loaded area of at least 10 m^2 is assumed
- the influence of wind friction is assumed negligible
- the building is assumed not to have a dominant face
- the influence of neighboring structures and obstacles is assumed negligible

Calculation details

The logic of wind effect calculation is explained below for each standard in the program



Eurocode general

- The recommended values are assumed for all coefficients unless specified otherwise by the user. None of the recommendations in National Annexes is assumed.
- Basic wind velocity is calculated as per Eq. (4.1) in 4.2 (2) of EC 1-4
- Terrain roughness is calculated as per Eq. (4.4) in 4.3.2 (1) of EC 1-4
- Terrain orography is taken into account with the orography factor co, but the calculation of co shall be performed by the user.
- Mean wind velocity is calculated as per Eq. (4.3) in 4.3.1 (1) of EC 1-4
- Wind turbulence intensity is calculated as per Eq. (4.7) in 4.4 (1) of EC 1-4
- Reference height is calculated as per 7.2.2 (1) in EC 1-4
- Peak velocity pressure is calculated as per Eq. (4.8) in 4.5 (1) of EC 1-4
- Pressure coefficients for walls of buildings are calculated as per 7.2.2 (2) in EC 1-4. As a conservative assumption, the lack of correlation between wind pressures between the windward and leeward side is not taken into account.
- External pressure coefficients for roofs of buildings are calculated as per 7.2.3 – 7.2.6 and 7.2.8 in EC 1-4.
- Internal pressure coefficients are calculated based on either a μ value (determined using Eq. (7.3) in EC 1-4 and specified by the user) using Fig. 7.13 in EC 1-4, or custom cpi values. Should the user decide not to specify μ nor cpi, two different cases are considered with cpi = +0.2 and cpi = -0.3 as per 7.2.9 (6) NOTE 2 in EC 1-4.
- Torsional effects are calculated as per 7.1.2 of EC 1-4.
- Pressure on the underside of protruding roof corners is considered as per EC 1-4 7.2.1 using the pressure from directly connected walls.
- Wind load of canopies is calculated as per EC 1-4 7.3. The generated distributed load is compatible with the pressure coefficients for zones A, B, C, D and produces a wind load with the magnitude and position defined by the general force coefficient.
- Wind load of free-standing walls is calculated as per EC 1-4 7.4.1-7.4.2.
- Wind load of signboards is calculated as per EC 1-4 7.4.3.
- The effect of wind friction is determined as per EC 1-4 7.5.



EC German

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- Basic wind velocity is based on Fig. NA.A.1 in EC 1-4 NA. The altitude of the building is taken into account as per A.2in EC 1-4 NA.
- Mean wind velocity and wind turbulence intensity are calculated as per Table NA.B.2 in EC 1-4 NA.
- Peak velocity pressure is calculated as per Eq. (NA.B.11) in NA.B.4.1 (4) in EC 1-4 NA
- Pressure coefficients for walls are calculated as per Table NA.1 in EC 1-4 NA.
- Pressure coefficients for flat roofs are modified according to 7.2.3 in EC 1-4 NA



NTC Italian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Basic wind velocity is calculated using the parameters in Table N.A.1 in EC 1-4 NA.
- Terrain roughness is calculated using 3.3.7 in NTC 2018.
- The wind load distribution for walls of slender buildings is not variable as per EC 1-4, but uniform according to the regulations in 3.3 in NTC 2018.
- Pressure coefficients for walls are based on Table IV. in 3.3 in NTC 2018.
- Load zone geometry and corresponding pressure coefficients for roofs are based on 3.3 in NTC 2018.
- Global pressure coefficients as per NTC 2018 are not considered.
- Automatic internal pressure calculation is based on 3.3 in NTC 2018 using $c_{pi} = \pm 0.2$.



EC Dutch

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Basic wind velocity is proposed based on the zones in Figure NB.1 and the corresponding values in Table NB.1 in EC 1-4 NA.
- Terrain roughness is calculated as per Eq. (4.5) and Table 4.1 in EC 1-4 NA
- Pressure coefficients for walls are calculated as per Table 7.1 in EC 1-4 NA. The lack of correlation between wind pressures between the windward and leeward side is always taken into account by multiplying the pressure intensities with 0.85.
- External pressure coefficients for roofs are calculated as per Tables 7.2, 7.3a, 7.3b, 7.4a, 7.4b, and 7.5 in EC 1-4 NA.



EC Hungarian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- Basic wind velocity is taken as 23.6 m/s^2 as per NA4.1 in EC 1-4 NA.
- The recommended value of $c_{dir} = 1.00$ in NA4.2 in EC 1-4 NA is taken into account.



EC Romanian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- Basic wind velocity is calculated using Eq. (2.2) and Fig. 2.1 in CR 1-4.
- The importance factor is taken into account when calculating the wind pressure intensity as per Eq. (3.1) and (3.2) in CR 1-4.



EC Czech

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- Basic wind velocity is based on the wind map of the Czech Republic in Fig. NA.4.1 in EC 1-4 NA.



EC Slovakian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- Basic wind velocity is calculated as per Table EC 1-4 NA NB1.
- External pressure coefficients for roofs are modified as per EC 1-4 NA 2.27–2.30.



EC Belgian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Basic wind velocity is based on the wind map of Belgium in Fig. 4.3 in EC 1-4 NA.
- The c_{dir} and c_{season} coefficients recommended in EC 1-4 NA can be taken into account by the user, but the default values for both parameters are 1.0.
- The turbulence factor k_I is calculated as per 4.4 in EC 1-4 NA.



EC Polish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- Basic wind velocity is calculated as per Table NA.1 in EC 1-4 NA.
- The c_{dir} coefficients recommended in Table NA.2 in EC 1-4 NA can be taken into account by the user, but the default values for the parameter is 1.0.
- Terrain roughness is calculated according to the option selected on the *Load* tab of the design code parameters dialog (**3.3.7 Design codes**). Three options are available: (1) $c_e(z)$ is calculated according to Table NA.3, (2) according to $c_r(z)$ in Table NA.3 and $I_v(z)$ in basic Eurocode, (3) $c_e(z)$ is calculated based on the basic Eurocode. By default, the terrain roughness is calculated by the first (1) option.



EC Danish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Basic wind velocity is taken as 24 m/s as per 4.2 (1)P NOTE 2 in EC 1-4 NA. Different values (such as for areas less than 25 km from the North Sea) shall be specified by the user.
- The c_{dir} and c_{season} coefficients recommended in 4.2 (2)P in EC 1-4 NA can be taken into account by the user. The default values for both parameters are 1.0.
- The pressure coefficient for zone I of flat roofs is calculated as per 7.2.3(4) in EC 1-4 NA.



EC Austrian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below
- Basic velocity pressure (q_b) shall be specified by the user based on Annex A of EC 1-4 NA.
- Peak velocity pressure is calculated as per Table 2 in EC 1-4 NA
- The simplified approaches for pressure coefficients in EC 1-4 NA are not applied. The pressure coefficients are calculated with the more accurate complex approaches at all times.
- The pressure coefficients for walls are calculated according the Table 3, 4 and 5 in EC 1-4 NA.
- Zones F and G are not applied on roofs in accordance with EC 1-4 NA if their total area does not exceed 20 percent of the total roof area.



EC Finnish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Basic wind velocity for various parts of the country is based on 4.2(1)P in EC 1-4 NA.
- The unique value for k_r in 4.3.2(1) in EC 1-4 NA is taken into consideration when terrain roughness is calculated.



EC Norwegian

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The fundamental value of basic wind velocity (v_{b0}) shall be specified by the user considering.
- The basic wind velocity is calculated with the consideration of the altitude factor (c_{alt}). The altitude factor shall be specified by the user, the default value is 1.
- The turbulence factor (k_I) can be set individually, the default value is 1. The turbulence factor is needed for the calculation of turbulence intensity $I_v(z)$.
- In the calculation of roughness factor, the module takes into account the specifications for kr, z0, zmin specified in EC 1-4 NA 4.3.2 (1).
- In the case of vaulted roofs and domes, the specifications for G and F zones of NA.7.2.8(1) are considered.
- The special cases described in NA.4.3.2(2) and NA.4.3.3 cannot be handled in the module.



EC Portuguese

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Basic wind velocity for the specified wind load zones of the country is based on EC 1-4 NA 4.2(1)P.
- In the calculation of roughness factor, the module considered the specifications for z_0, z_{min} specified in EC 1-4 NA 4.3.2 (1).

EC Spanish

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- Basic wind velocity for the specified wind load zones of the country is based on EC 1-4 NA 4.2(1)P.

EC British

- Because calculation is based on the general Eurocode, the procedure presented there is applied for this standard as well with the modifications/extensions listed below.
- The fundamental value of the basic wind velocity ($v_{b,0}$) is determined by the *OSGB co-ordinates* and the altitude according to NA2.4.
- The program can take into account four main directions for the calculation of the wind pressure.
- In the case of vaulted roofs, the program cannot take into account the local pressures in zones G, F and J as defined in note 3 of NA.2.29.

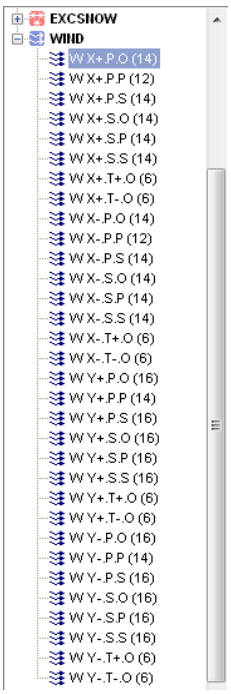
Swiss

- The recommended values are assumed for all coefficients unless specified otherwise by the user.
- The basic value of velocity pressure (q_{p0}) shall be determined by the user based on Appendix E of SIA 261
- Peak velocity pressure is calculated as per Eq. (11) in 6.2.1.1 in SIA 261
- The wind profile coefficient c_h is calculated as per Eq. (12) in 6.2.1.2 in SIA 261 according to parameters in Table 4 corresponding to the terrain category selected by the user.
- Wind pressure is calculated as per Eq. (13) (external) and Eq. (14) (internal) in 6.2.2.1 in SIA 261. The corresponding pressure coefficients are not taken from Appendix C, but from Tables 7.2.3 – 7.2.6 and 7.2.8 in EC 1-4 to provide a more generally applicable solution that is also in line with Swiss design practice.

Usage

The following paragraphs explain the usage of the automatic wind load generator module.





To apply wind loads according to Eurocode, first click on the Load cases / load groups button and define a wind load case by clicking on the wind load case button. A wind load group will be created automatically. As a first step a temporary wind load case is created in the wind load group and its name can be set. After defining load panels and setting the wind load parameters the program replaces the temporary load case with the necessary wind load cases. For details on the naming convention for wind load cases see **4.10.1 Load cases, load groups**

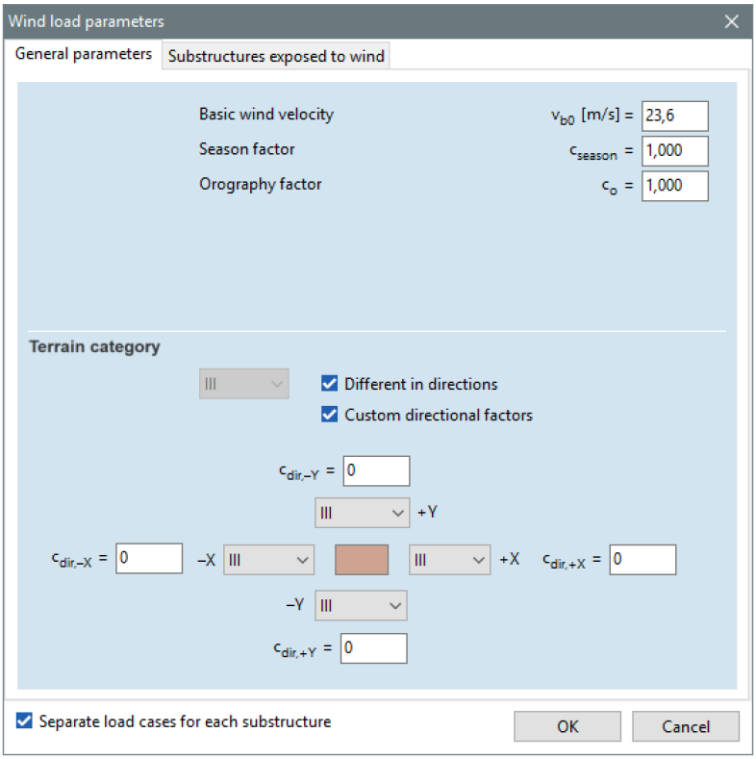


To specify wind load parameters, select a wind load case. It enables the wind load icon on the *Loads* tab. If no load panels have been created draw the load panels for walls and roofs according to **4.10.12 Load panels.** Click on the icon to open the wind load parameters dialog.

Wind load parameters

The first tab of the *Wind load parameters* dialog corresponds to the general parameters, while the second tab is used to define structure-specific parameters.

General parameters



The following parameters can be specified (certain parameters are not required by every design code)

Altitude above sea level A [m]
Altitude is one of the factors affecting the basic wind velocity in several countries.
If v_{b0} depends on *A*, v_{b0} is automatically calculated.

Importance factor
An importance factor can be entered depending on the classification of the building if it is required by the design code. Nonstandard values can also be entered with confirmation.

Zone
In countries where the characteristic wind load depends on the geographical location the national annex divides the country into zones. The zone selected automatically defines the basic wind velocity.

Basic wind velocity v_{b0} [m/s]
v_{b0} is automatically calculated from the above parameters. This value can be replaced with a custom value if desired.

Season factor c_{season}
The design code may allow reducing wind action through a c_{season} factor for temporary structures. It takes into account that the calculated wind velocity does not occur during the lifetime of the structure. The actual value is left to the designer's judgement and responsibility.

Altitude factor c_{alt}
The factor can be set only in the case of the Norwegian design code, the default value is 1. The parameter is considered in the calculation of basic wind velocity. The parameter shall be set by the guidance of the national annex.

Turbulence factor k_I
The factor can be set only in case of the Norwegian design code, the default value is 1. The parameter is needed for the calculation of turbulence intensity Iv(z). In case of the other standards, the module considers the constant values specified in the national annexes.

Orography factor c_0
The c_0 factor takes into account the effect of orography (hills etc.) to the wind velocities. The design code gives recommendations on when and how to use this factor.

Structural factor $c_s \cdot c_d$ (in SIA standard $c_{red} \cdot c_d$)
The default value of the factor is 1, and it is taken into account when calculating the wind action.

Annual probability of exceedance factor p

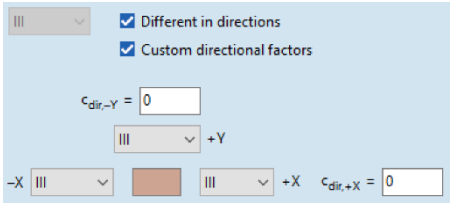
The factor is available for Eurocode standard, the default value is 0,02. The factor is taken into account in the calculation of the v_b basic wind velocity.

Terrain category

Select a terrain category from the dropdown list. Each category corresponds to the areas listed below:

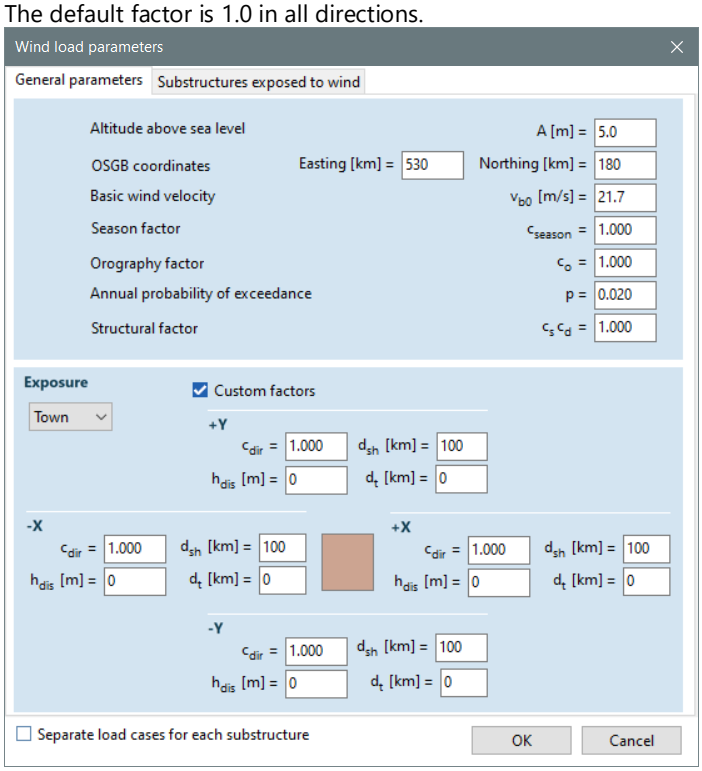
- ***0*** Sea, coastal area exposed to the open sea
- ***I*** Lakes or flat and horizontal area with negligible vegetation and without obstacles
- ***II*** Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights
- ***III*** Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, permanent forest)
- ***IV*** Area in which at least 15 % of the surface is covered with buildings and their average height exceeds 15 m

If terrains are different in directions check the *Different in direction* checkbox and set the terrain categories for each wind direction individually.



Checking the option *Custom directional factors* enables four c_{dir} directional factors which take into account a dominant wind direction on the site, thus the wind speed is not identical in all directions. The default factor is 1.0 in all directions.

Specific parameters of the British standard



OSGB coordinates [km]

Easting, Northing coordinates in OSGB system. The software uses these coordinates to determine the fundamental basic wind velocity before altitude correction.

Basic wind velocity v_{b0} [m/s]

The basic wind velocity (vb,0) is automatically determined by the given OSGB coordinates and the altitude according to NA.2.4. The value can be overwritten by the user.

Exposure

Terrain category

Country, Town

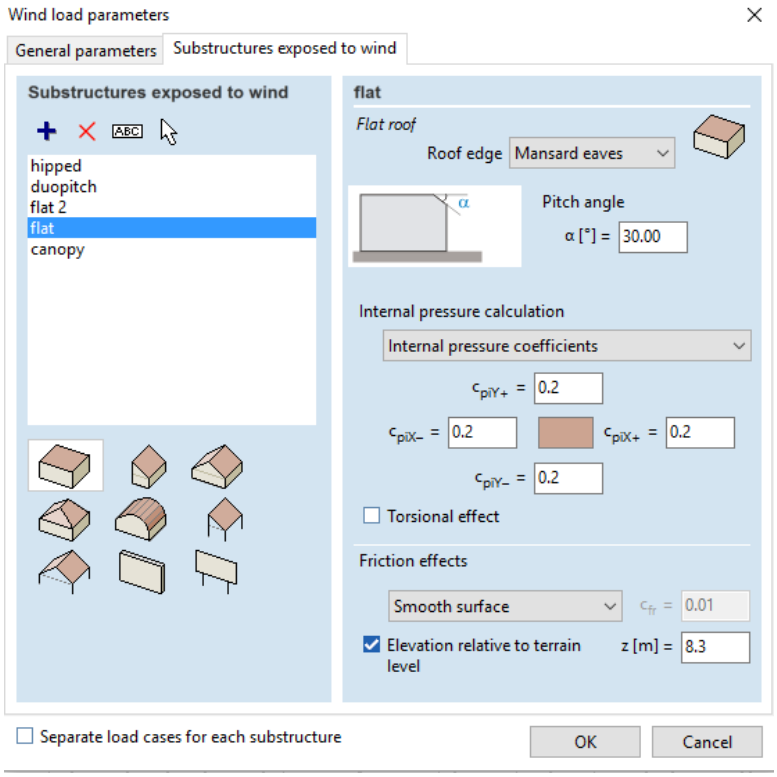
Displacement height h_{dis} [m]
Displacement height parameter allows for the sheltering effect of dense upstream urbanization.

Distance upwind to shoreline d_{sh} [km]

Distance of the building from the shoreline in wind direction.

Distance inside town terrain [km]
Distance of the building to the edge of town, it is only active for '*Town*' terrain category.

Substructures exposed to wind



Substructures exposed to wind tab allows definition of structure-specific parameters.
The program can calculate the wind load of several independent structures. It is also possible to model several parts of the same structure independently as substructures. The program can combine wind loads for such substructures into a general wind load for the whole structure. If that is not desired, check *Separate load cases for each substructure* at the bottom left corner.

Define a new substructure
First specify the name of the substructure, then select the corresponding load panels. The list of corresponding load panels is displayed as a tooltip when the mouse hovers over the name.

Delete substructure
Deletes the selected substructures.

Rename substructure
Rename the selected substructures

Modify a substructure
Modifies the load panel assignments of the selected substructure.

Buildings

Roof geometry
Select the icon describing the roof geometry that best describes the designed structure. Available types are: flat, monopitch, duopitch, hipped and barrel roof.

Flat roof *Monopitch roof* *Duopitch roof* *Hip roof* *Barrel roof*

After selecting roof geometry, the following parameters can be specified:

Roof edge for flat roof
If a flat roof is defined the roof edge has significant effect on wind load intensity. Four options are available: sharp eaves (no parameters), parapet wall (enter parapet height), round eaves (enter rounding radius), mansard eaves (enter pitch angle).

Torsional effect
Eurocode requires checking torsional winds for structures sensitive to torsion. If this option is checked additional load cases will be created for torsional winds.

Internal pressure calculation
Four options are available to determine internal pressure of buildings:
Approximative: the program uses the critical pressure and suction coefficients recommended by the Eurocode standard. Two sets of wind loads are generated in this case.
Based on area of openings: the user specifies the value of μ based on the distribution of openings in different directions and the program calculates the internal pressure coefficients as a function of these values. If μ=0 is specified, the program uses the approximative method in that direction.
Internal pressure coefficients: the user specifies the internal pressure coefficients directly.
Disabled: the program does not apply internal pressure during wind load generation.

Elevation relative to terrain level
The lowest point of the load panels selected for wind load generation is assumed to be at ground level by default. The custom terrain level option allows the user to specify a custom height for that point. This allows the user to perform advanced analyses such as consideration of wind load for only the roof of a building.

Wind friction
The effect of wind friction is considered for every structure by default. The friction coefficient corresponding to the surface of the structure shall be selected from the drop-down list. It is also possible to specify a custom value for the friction coefficient. Wind friction effects can be ignored by setting c_{fr} = 0 as custom value.

Canopies

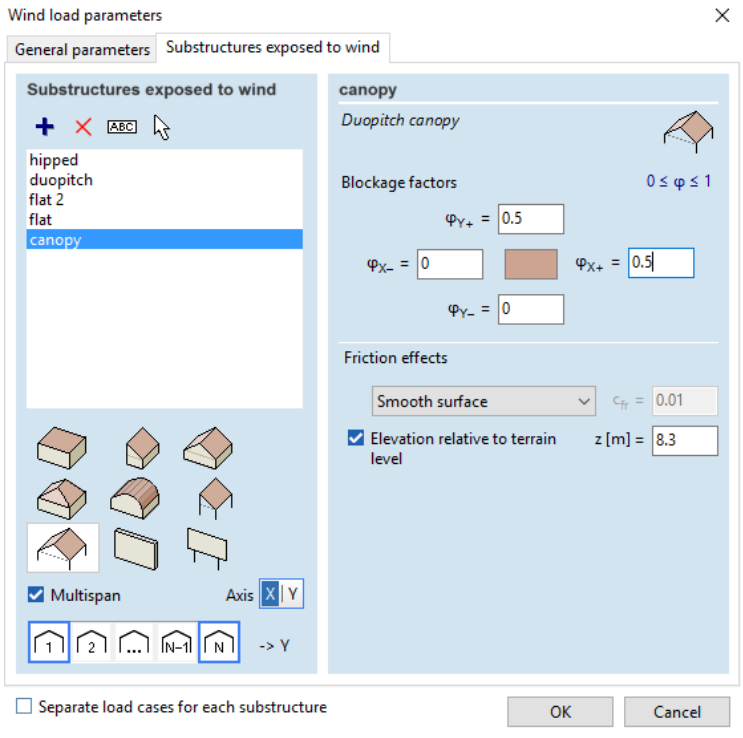
Roof geometry
Select the icon describing the roof geometry that best describes the designed structure. Available types are: monopitch and duopitch.



Monopitch



Duopitch



After the roof geometry is selected, the following parameters can be specified:

Blockage factors
The *p* blockage factor describes the number of obstacles present under the canopy roof that can block the free flow of air in the given direction. The value of 0.0 represents an empty canopy, while the value of 1.0 represents a canopy fully blocked in the corresponding direction. A blockage factor can be specified for each of the four main directions. Note that if the windward side of a canopy is fully blocked by an obstacle it is often worth considering the application of several substructures and a combined wind load. The fully blocked side in that case can be modeled as a building.

Multi-span canopies
Wind load of mutli-span canopies shall be modeled through the combination of the wind load on a set of structures. The position of each structure in the multi-span building shall be specified (end bay, second bay from either end or any of the other bays) using the icons at the bottom left. In order to get a general wind load for the whole structure, it is important to turn the *Separate load cases for each substructure* off.

Elevation relative to terrain level
If the load panels selected to describe the geometry of the canopy roof do not include any wall elements (this is the typical scenario), then the wind generator algorithm will define the terrain level as the lowest point of the roof by default. This is an incorrect assumption in most cases that can be corrected by specifying the elevation of the lowest point of the roof from the terrain level.

Free-standing structures

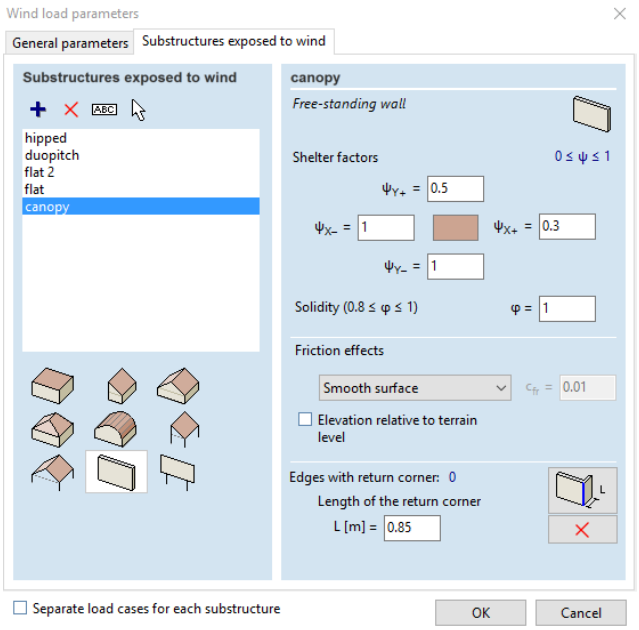
The following types of free-standing structures are available for automatic wind load generation:



Free-standing wall



Signboards



After the type of structure is selected, the following parameters can be specified:

Free-standing wall

Shelter factors
Shelter factors consider additional walls or fences upwind that can reduce the wind pressure acting on the modeled wall. Different shelter factors can be specified for the four main directions.

Solidity
The solidity ratio is proportional to the number of openings on the wall. 1.0 corresponds to a solid wall. Walls with a solidity ratio below 0.8 shall be calculated as fences and are out of the scope of the wind load generation algorithm.

Elevation relative to terrain level
Signboards that are separated from the ground by a height less than one quarter of their own height shall be modeled as free-standing walls and their elevation shall be specified here.



Edges with return corner

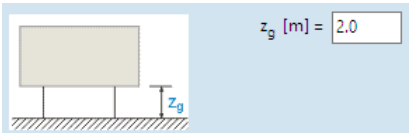
The wind load of a wall with return corners depends on the length of the return corners on each end of the wall. The length of the return corner shall be specified first and then – after clicking on the return corner icon – the edge of the wall corresponding to the return corner shall be selected. Only the length of the return corner has any effect the calculated wind load, its direction does not.



Remove all return corners by clicking the *Delete return corners from edges* button

Signboard

The calculation of wind load of signboards requires only the height of the bottom edge of the signboard from the terrain level.



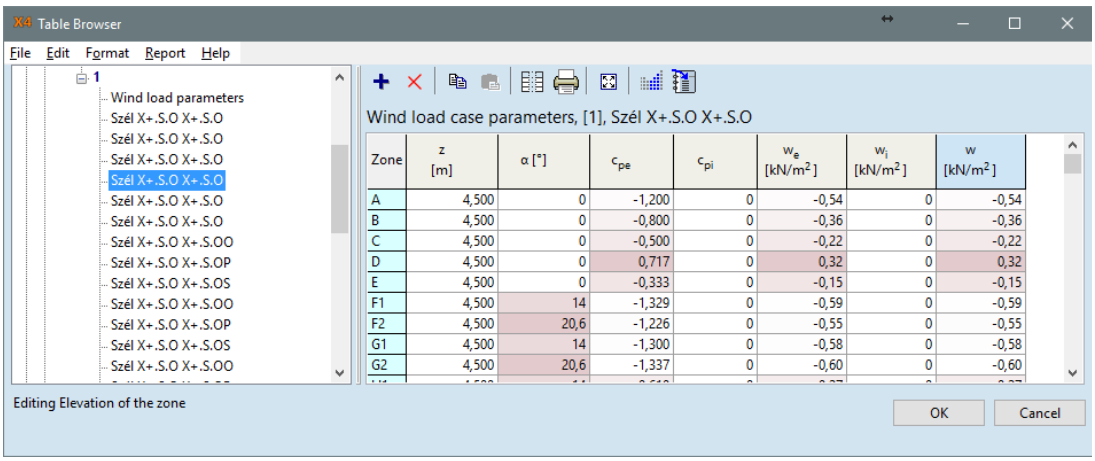
After clicking the OK button, the wind loads are generated automatically for the selected load panels.

Wind load data

After the wind loads have been generated all the data related to their calculation is available in the Table Browser under *Loads/ Wind load on substructures*.

Wind load parameters summarizes the values that are not load case specific. Default wind load settings lead to direction independent wind loading, so identical parameters in all four directions. Should the user specify direction dependent terrain categories and/or custom direction factors, these values become different in each direction.

Wind load case parameters summarizes load case specific parameters for each wind load case. These parameters are grouped by the zones generated on the selected load panels. A number after the zone letter indicates that there are more than one of the given zone types on the model in the selected load case. Wall zones (A-E) are divided to two areas for tall buildings. "1" always corresponds to the bottom, "2" to the top area for walls. Multiple roof zones of the same type are available for monopitch (F) and duopitch (F-I) roofs. Zones on the windward side of canopies are marked with "1" and zones on their leeward side are marked with "2". Zones with "S" in their name in free-standing walls are the ones where the effect of shelters is taken into consideration. The FR zone shows the effect of wind friction for every structure type.



4.10.15 Simulated wind used in wind tunnel analysis – WIND module



Wind tunnel analysis places the model in horizontal winds blowing from different directions with a speed depending on the height above the terrain level then determines the wind loads by physical wind flow analysis.

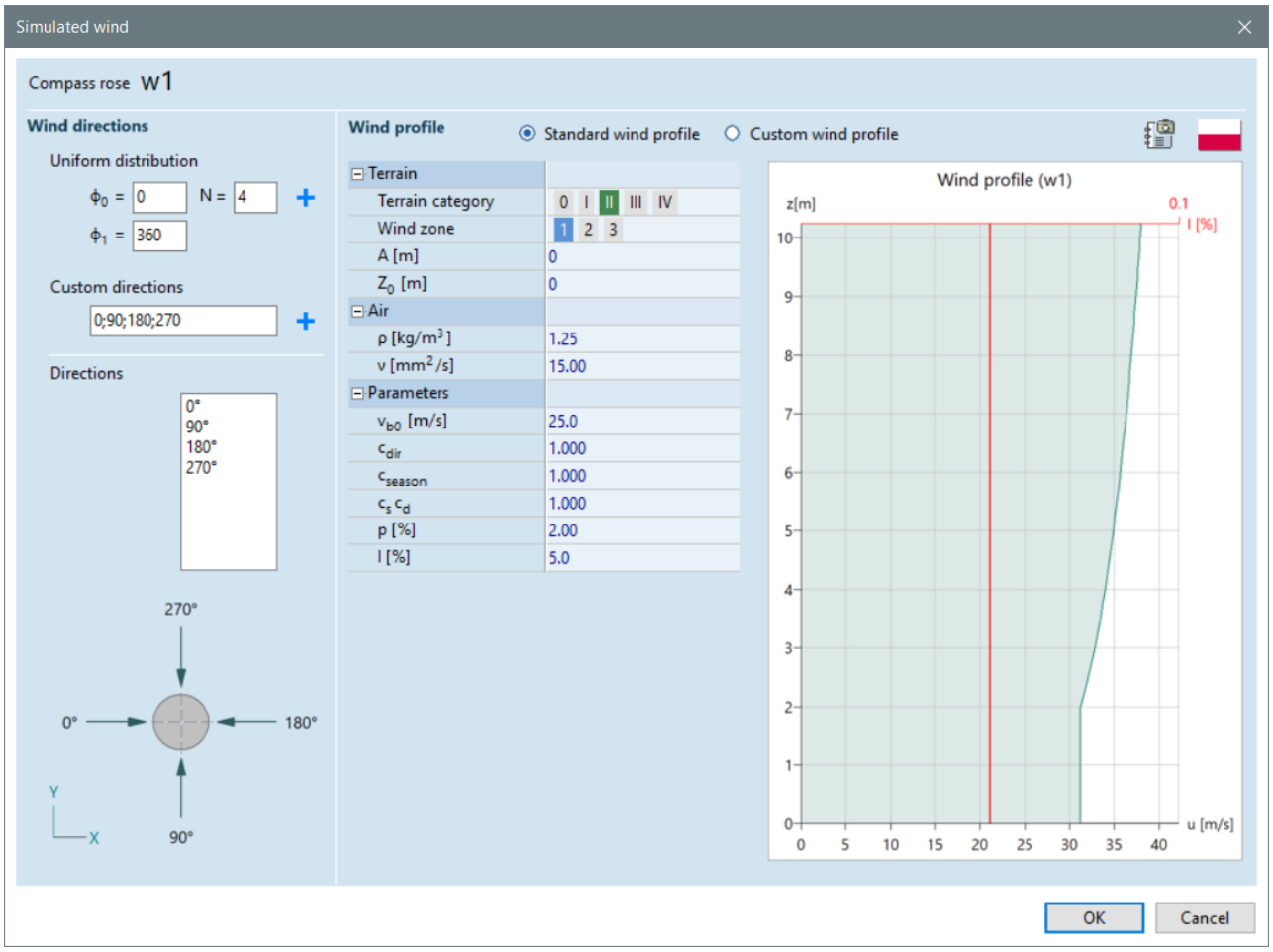
Details of the analysis is described in **4.11 Wind tunnel – WIND module** and in the WIND Guide (see **3.9.2 Guides**).

Definition of simulated wind loads allows specification of multiple wind directions with the same wind profile and parameters. A separate load case is generated for each wind direction. These cases together are called *wind roses*. Clicking on the *Simulated wind used in wind tunnel analysis* button, you can set the wind directions, the vertical wind profile, the structural factor and other parameters for terrain, air and wind load.

The WIND module supports standard wind profiles according to Eurocode 1991-1-4 and SIA261.

The national annexes used are

German EC	EC 1991-1-4:2005 DIN EN 1991-1-4/NA December 2010
Italian NTC	EC 1991-1-4:2005 UNI EN 1991-1-4/NA July 2007 NTC 2018
Dutch EC	EC 1991-1-4:2005 NEN EN 1991-1-4/NB November 2007
Hungarian EC	EC 1991-1-4:2005 MSz EN 1991-1-4/NA January 2016
Romanian EC	CR 1-1-4/2012 SR EN 1991-1-4
Slovakian EC	EC 1991-1-4:2005 CSN EN 1991-1-4/NA July 2013
Polish EC	EC 1991-1-4:2005 PN EN 1991-1-4/NA October 2008
Danish EC	EC 1991-1-4:2005 DS/EN 1991-1-4 DK NA July 2015
Finnish EC	EN 1991-1-4:2005 SFS-EN 1991-1-4/NA November 2007
Norvegian EC	EN 1991-1-4:2005 SN-EN 1991-1-4/NA 2009
Greek EC	EN 1991-1-4:2005 ELOT EN 1991-1-4
British EC	EN 1991-1-4:2010 BS EN 1991-1-4:2005+A1:2010
Swiss SIA	SIA261:2020 Einwirkingen auf Tragwerke SIA261/1:2003 Ergänzende Festlegungen



Wind directions

This panel is to specify the wind directions of the wind rose in top view, i.e. in the global X-Y plane of the model. The value to be entered is the angle of the wind direction in degrees to the X-axis of the global coordinate system.
Clicking on the + button under *Uniform distribution* N directions will be added to the list of directions distributed evenly between ϕ_0 and ϕ_1.
With the + button under *Custom directions* you can add one or more angles separated by semicolons. All angle values are interpreted in degrees.
To delete items from the list of directions select one or more values and press the *Delete* key.
Wind tunnel tests will generate winds on the inlet according to the direction assigned to the load cases. The wind speed has no Z-component and its horizontal velocity follows a vertical wind profile as a function of the altitude above the terrain level.

Wind profile

A wind profile can be defined either as a parametric profile according to the current design code or as a custom wind profile. Certain parameters must be specified regardless the type of the profile.

Terrain

Z_0 is the terrain level in the global coordinate system of the model. It means that only the part of the structure above the horizontal plane at Z_0 is placed into the virtual wind tunnel.

Air

Two parameters of the flowing air must be specified: its density, ρ and its kinematic viscosity ν. The WIND module sets the default values according to Eurocode: ρ = 1,25 kg/m^3, ν = 15 mm^2/s.

Standard wind profile

The program determines the wind profile based on the *Terrain category* and other parameters required by the current design code and national annex.

Terrain category: describes the terrain roughness upwind of the structure
I [%], turbulence intensity: describes the standard deviation of wind speed relative to the mean value

v_{b0}	Fundamental value of the basic wind velocity
c_{dir}	Directional factor
c_{season}	Season factor
c_0	Orography factor
k_I	Turbulence factor
$c_s c_d$	Strcutural factor

p Probability of exceedence

☞ **The structural factor does not appear directly in the aerodynamic simulation. The structural factor is used to modify static loads obtained from CFD simulation.**

The calculation is based on the Eurocode with the values and additions specified in national annexes.



Hungary

Terrain categories 0, I, II, III, IV

v_{b0} = 23.6 m/s according to EC 1-4 NA 4.1.
c_{dir} = 1.0 as proposed in EC 1-4 NA 4.2.
z_{max} = 200 m, maximum height above ground level



The Netherlands

Terrain categories 0, II, III

Wind zones According to EC 1-4 NA, there are 3 wind zones (I, II, III), determining the basic wind load and the distribution parameters (*K, n*). The wind zones can be defined according to EC 1-4 NA NB 1 Table 1 and map.
z_{max} =200 m, maximum height above ground level



Denmark

Terrain categories 0, I, II, III, IV

v_{b0} = 24 m/s according to EC 1-4 NA 4.2 (1)P NOTE 2. This Annex mentions that an increased wind speed must be used in the 25 km vicinity of the North Sea.
c_{dir}, c_{season} Can be modified according to EC 1-4 NA 4.2 (2) P, but their default value is 1.
z_{max} =200 m, maximum height above ground level



Germany

Terrain categories I, II, III, IV, I/II, II/III (the latter two are transitions between the terrain categories I, II and II, III according to NA.B.3.3)

Wind zones EC 1-4 NA NA Fig. A.1 specifies 4 wind zones determining the basic wind load
H_s If the altitude above sea level exceeds 800 metres, an increase factor is applied. The value of velocity for a given dynamic pressure is calculated from equation NA.B.11.
The German national annex specifies a power law wind profile opposed to the logarithmic wind profile used in other standards.



Greece

Terrain categories 0, I, II, III, IV

Wind zones Az EC 1-4 NA Fig. A.1 specifies 4 wind zones determining the basic wind load



Italy

Terrain categories I, II, III, IV, V

Wind zones UNI EN 1991-1-4 NA Fig NA1. and the table specifies 9 wind zones. wind zones determine the basic wind load and the parameters taking into account the altitude above sea level.
a_s altitude above the sea level
z_{max} =200 m, maximum height above ground level



Finland

Terrain categories 0, I, II, III, IV

v_{b0} = 21 m/s according to EC 1-4 NA 4.2 (1) P
k_r At the seashore EC 1-4 NA 4.3.2 (1) requires a value of 0.18
z_{max} =200 m, maximum height above ground level



Romania

Terrain categories 0, I, II, III, IV

Wind zones CR 1-4 Fig 2.1 and Eq. 2.2 specifies four wind zones, with wind velocities 18, 20, 22 and 24 m/s
γ_{Iw} This factor is used to modify the wind load according to the importance factor of the structure, see CR 1-4 Eq. 3.1. This factor does not appear directly in the aero-dynamic simulation. It is used to modify static loads obtained from CFD simulation.
z_{max} =200 m, maximum height above ground level



Slovakia

Terrain categories I, II, III, IV

Wind zones EC 1-4 NA Table NB1 specifies 4 wind zones
z_{max} =200 m, maximum height above ground level

Poland

Terrain categories 0, I, II, III, IV

Wind zones EC 1-4 NA Table NA.1 specifies 3 wind zones
A The altitude above sea level affects the basic wind velocity
$c_e(z)$ Terrain roughness is calculated according to the design code parameter **(3.3.7 Design codes)** in the 'Load' tab. Three options are available: (1) $c_e(z)$ is calculated according to Table NA.3, (2) according to $c_r(z)$ in Table NA.3 and $I_v(z)$ in basic Eurocode, (3) $c_e(z)$ is calculated based on the basic Eurocode. By default, the terrain roughness is calculated by the first (1) option.
z_{max} =200, 200, 300, 400, 500 m, maximum height above ground level, depends on the terrain category

Great-Britain

Terrain categories Country, Town

E, N Easting, Northing coordinates in OSGB system. The software uses these coordinates to determine the fundamental basic wind velocity before altitude correction.
A The altitude above sea level. The fundamental basic wind velocity is modified by the altitude factor based on this value.
h_{dis} Displacement height.
d_{sh} Distance upwind to shoreline.
d_t Distance inside town terrain.
☞ The software uses linear interpolation for figure NA.1 and logarithmic interpolation for figures NA.3, NA.4, NA.5, NA.6, NA.7, NA.8.

Switzerland

Terrain categories II, IIa, III, IV

Roughness The roughness for terrain categories is calculated according to SIA261 6.2.1.2 Eq. 12.
q_{p0} The reference value of the dynamic pressure according to SIA261 Appendix E
The wind velocity is calculated from the dynamic pressure as

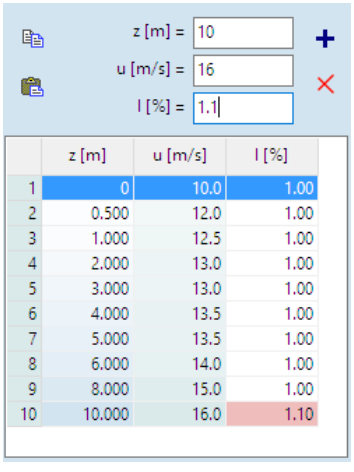
$$v = \sqrt{2q/\rho}$$

$c_{red}c_d$ A structural factor like in Eurocode. It modifies the wind loads obtained from the CFD simulation
z_{max} =200 m, maximum height above ground level

The Swiss code, like the German code, uses a power law wind profile.

Custom wind profile

A custom wind profile requires specification of the wind speed and the turbulence intensity at different *z* heights.
The + button adds *z, u, I* value triplets to the wind profile. The points can be entered in any order, the table will automatically sort them by increasing *z*.
Use the *Delete* button to delete the selected rows from the table.



	z [m]	u [m/s]	I [%]
1	0	10.0	1.00
2	0.500	12.0	1.00
3	1.000	12.5	1.00
4	2.000	13.0	1.00
5	3.000	13.0	1.00
6	4.000	13.5	1.00
7	5.000	13.5	1.00
8	6.000	14.0	1.00
9	8.000	15.0	1.00
10	10.000	16.0	1.10

Copy the table to the Clipboard. Paste it to Excel for further editing, for example.

Copy the Clipboard content to the table. Wind profiles calculated in Excel can be imported this way.

☞ **A custom wind profile can lead to a divergent or slowly convergent solution.**

4.10.16 Generating wind load from a CFD (Computational Fluid Dynamics) file

CFD interface



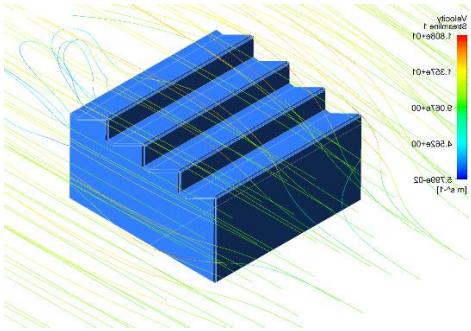
The CFD interface allows application of a pressure distribution over domains or load panels. Design codes provide guidance on pressure coefficient calculation and wind load definition only for structures with a simple geometric shape (**4.10.14 Wind load – SWG module**).
In the case of more complex or compound structures these methods are not applicable and pressure coefficients must be determined individually by a fluid dynamics simulation (CFD modeling) or a wind tunnel test. The CFD interface helps importing these results to AXISVM.
CFD is a generic interface to define pressures caused by flowing substances.

Steps of loads generation

Exporting the model to an STL file

An STL (*Stereo Lithography*) file contains data of a triangular mesh describing the surface of a body.
Choose *File / Export* from the menu then choose the STL file format. Turn on the *CFD model* radio button.
A 3D model of the structure will be generated including domains and load panels. The resulting file must be loaded into the CFD software.

Importing pressure distribution from the simulation software



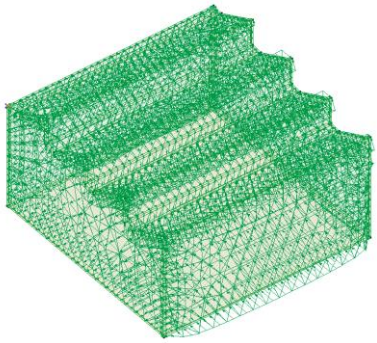
The interface imports the pressure distribution determined by computational fluid dynamics simulation and applies it to AXISVM elements as steady state wind loads.
The pressure values must be imported by processing an interface file.

The CFD interface reads pressure values from an ASCII text file. Each line of the file must contain

- **x, y, z** global coordinates
- a pressure value

The separator character between these fields can be a space, a comma or a semicolon. The units of pressure values can be chosen, but coordinates must be in meters.

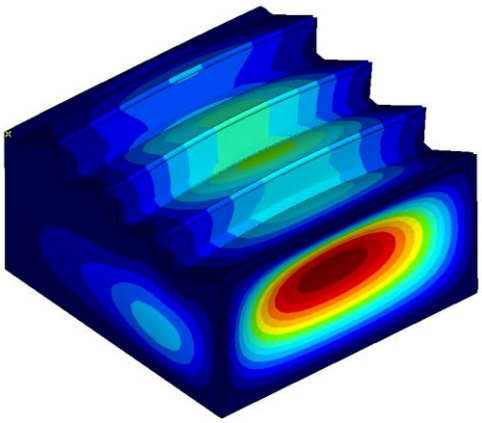
Interpolation (processing of the pressure field)



The interface file generated from the fluid dynamics simulation software can be loaded on the *Loads* tab, clicking on the above icon. Load values are obtained by interpolating the pressure field described in the interface file.
The imported pressure points are interpolated on a triangular mesh, where a linear pressure distribution is assumed on each triangle. The resultant of this linear distribution is made equal to the resultant of the original field. If no AXISVM surface could be associated to some pressure points (or the surface cannot be clearly determined) a warning message is displayed

☞ **This method ignores edge loads as surfaces cannot be unambiguously associated to edge points.**

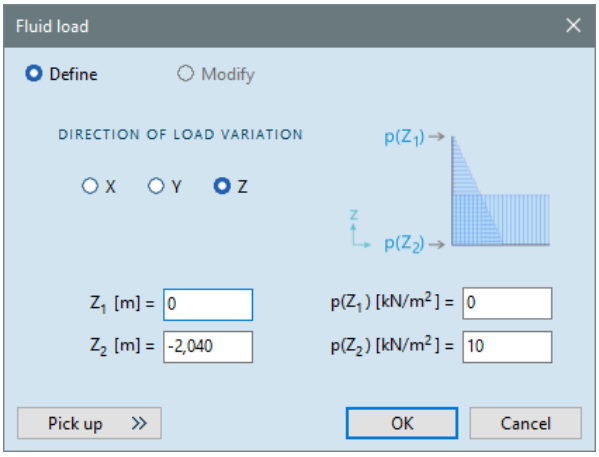
Displaying results



Deformations and forces generated by the calculated wind pressure distribution can be displayed the usual way as diagrams, isolines or isosurfaces.

4.10.17 Fluid load

Lets you apply pressure loads characteristic to fluids to the selected plate or shell elements. The actual load is calculated from values computed at the corner of the elements.
The direction of positive load is identical with the local Z axis of the domain/surface. Fluid loads created with the same definition will be handled as one load. So if you specified a fluid load on more than one element and click on the load contour on any of these elements the load will be selected on all of them and you can easily change the load parameters.



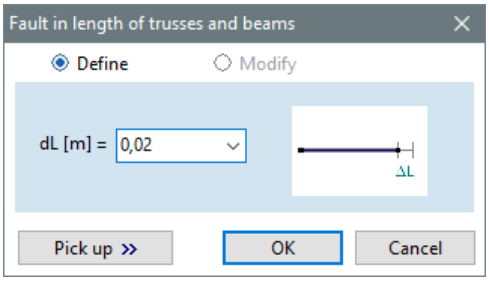
4.10.18 Self-weight

Lets you take the self-weight of the elements (that have materials assigned) and domains into account in the analysis. Self-weight is computed based on the cross-sectional, the mass density of the material, the gravitational acceleration *g*, and the length or area of the element. The load is applied as a distributed load in the direction of the gravitation vector.

A dashed line is drawn along line elements or surface/domain contours. If load intensity labels are turned on a light blue G appears.

4.10.19 Fault in length (fabrication error)

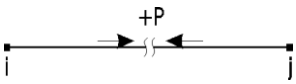
This load type is used when a structural beam element is shorter or longer than required due to a fault in manufacturing.
Lets you apply the load, which is required to force the shorter/longer beams to fit the distance of the corresponding nodes, to the selected elements. You must specify the value of the manufacturing fault, *dL [m]*. A positive *dL* means that the beam is longer by *dL*.

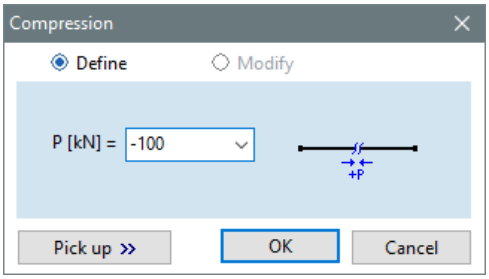


The load has the same effect as the $dT^{=} = dL/(\alpha \cdot L)$ thermal load.

4.10.20 Tension/compression

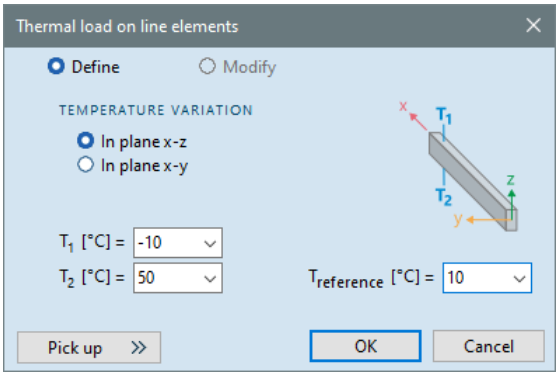
Lets you define an initial axial internal force in truss/beam elements. The load has the same effect as a $dT^{=} = -P/(\alpha \cdot E \cdot A)$ thermal load.





4.10.21 Thermal load on line elements

Lets you apply temperature loads to the selected line elements (truss, beam, and/or rib). You must specify values for the following parameters:



Truss $T_{reference}$ reference temperature (corresponding to the initial unstressed state)
T -the temperature assumed for the analysis

$dT = T - T_{reference}$ is the temperature variation that is taken into account in the analysis. A positive dT means a warm up of the truss.

Beam/Rib $T_{reference}$ reference temperature (corresponding to the initial unstressed state)
T_1 the temperature of the top cord (in the corresponding local direction)
T_2 the temperature of the bottom cord (in the corresponding local direction)

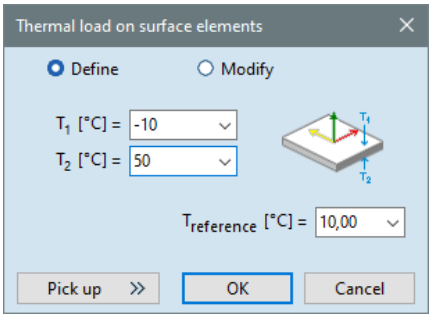
$dT^{=} = T - T_{reference}$ is the uniform temperature variation that is taken into account in the analysis, where T is the temperature of the cross-section in its center of gravity.
in local y direction: $T = T_2 + (T_1 - T_2)\frac{y_G}{H_y}$
in local z direction: $T = T_2 + (T_1 - T_2)\frac{z_G}{H_z}$

where,
y_G, z_G, and H_y, H_z are properties of the cross-section.
A positive $dT^{=}$ indicates a temperature increase of the beam.

$dT^{\neq} = T_1 - T_2$ is the non-uniform temperature variation that is taken into account in the analysis.

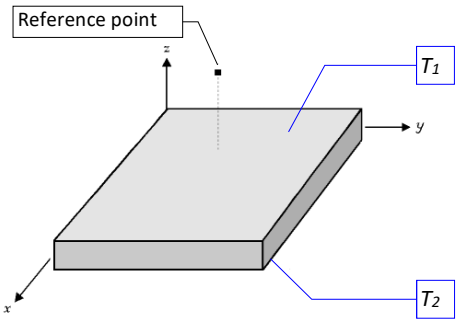
4.10.22 Thermal load on surface elements

Lets you apply temperature loads to the selected surface elements. You must specify values for the following parameters:
$T_{reference}$ reference temperature (corresponding to the initial unstressed state)
T_1 the temperature of the top cord (in the positive local z direction)
T_2 the temperature of the bottom cord (in the negative local z direction)



$dT^{=} = T - T_{reference}$ is the uniform temperature variation that is taken into account in the analysis, where T is the temperature in the center of gravity of the cross-section.

$dT^{\neq} = T_1 - T_2$ is the non-uniform temperature variation that is taken into account in the analysis.



☞ **For membranes only dT$^{=}$ is taken into account. For plates only dT$^{\#}$ is taken into account.**

4.10.23 Forced support displacement

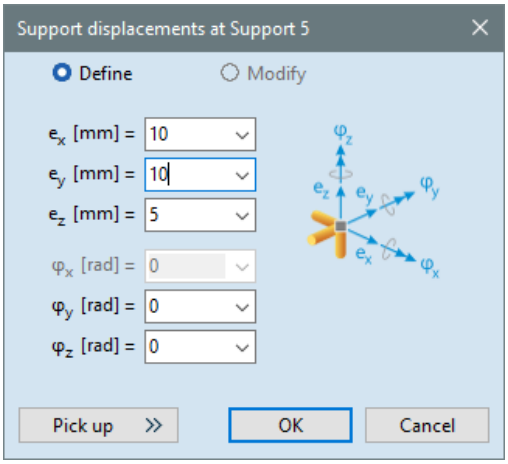
Lets you apply forced displacements to the selected support elements. You must specify the values of the forced displacement components (translational: *e[m]*; rotational: *φ[rad]*).

AXISVM approximates the problem, by applying a force $P_{support}$ in the direction of the support element so as to produce the forced displacement *e*.

$P_{support} = K_{support} \cdot e$,

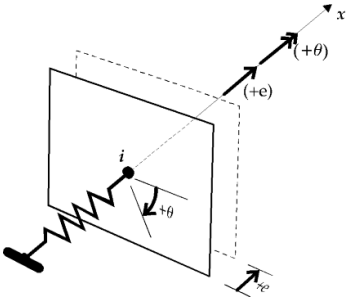
where $K_{support}$ is the corresponding support stiffness.

If a support component is zero on at least one selected support, that direction is greyed out.



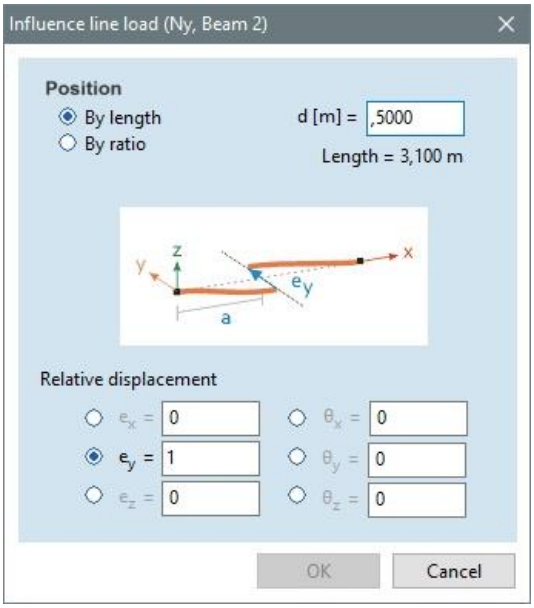
If the stiffness of the support element is large enough, the secondary deflections due to other loads will be negligible. Therefore, you may apply forced displacements only to the supports stiff enough relative to the stiffness of the structure (at least 10^3 times larger) in the corresponding direction. Check this assumption every time, by checking the displacement results and verifying the displacement at the respective node.

A positive forced displacement moves the node in the positive direction of the local axis.



4.10.24 Influence line

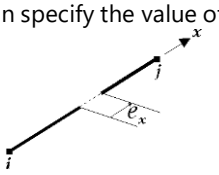
Lets you apply a relative displacement load to obtain the influence line of an internal force component, on the selected truss/beam elements. You must specify the value of the relative displacement *e* as +1 or –1.



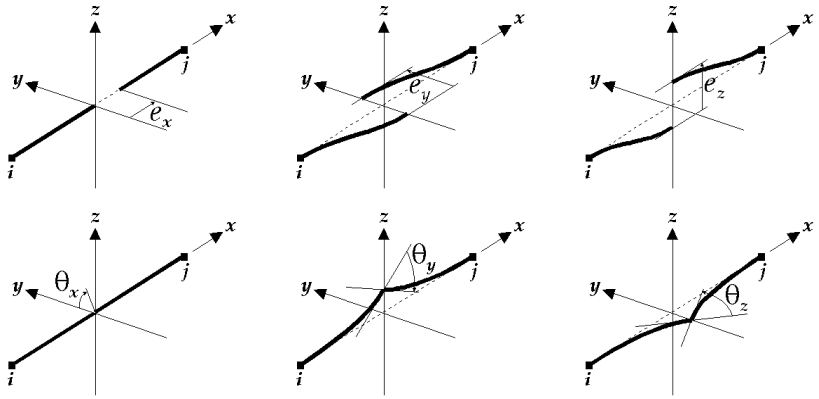
☞ **You can define influence line load, only in an influence line type load case.**

See 4.10.1 Load cases, load groups

Truss You can specify the value of the relative displacement e_x as +1 or –1.



Beam You can specify the value of the relative displacement e_x / e_y / e_z / θ_x / θ_y / θ_z as +1 or –1.



4.10.25 Seismic loads – SE1 module

The SE1 module includes several tools that facilitate the execution of Modal Response Spectrum Analysis (MRSA) in AXISVM. The tools automatically perform seismic load generation in three orthogonal directions for each vibration mode; calculation of structural response for each vibration mode and combination of modal responses into a governing seismic effect. The following guide provides information on the settings and options available in the program and explains how to use them. Due to the vast literature of MRSA it is not possible to provide a step-by-step theoretical explanation to the application of such an analysis here.

Design Codes

Calculations are performed according to regulations in the general Eurocode standards (particularly Eurocode 8) complemented by the National Annex that corresponds to the Design Code selected by the user. SIA, the Swiss national standard is similar to but not based on Eurocode. The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

MRSA procedure

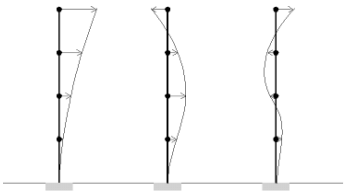
Seismic loads are generated based on the vibration mode shapes in all design codes. Loads are applied in linear static analysis and the effects of several mode shapes are combined to get the design internal forces and displacements of the structure.
Multiple seismic actions can be defined for the structure. These actions cannot be combined.

How stiffness reduction is applied

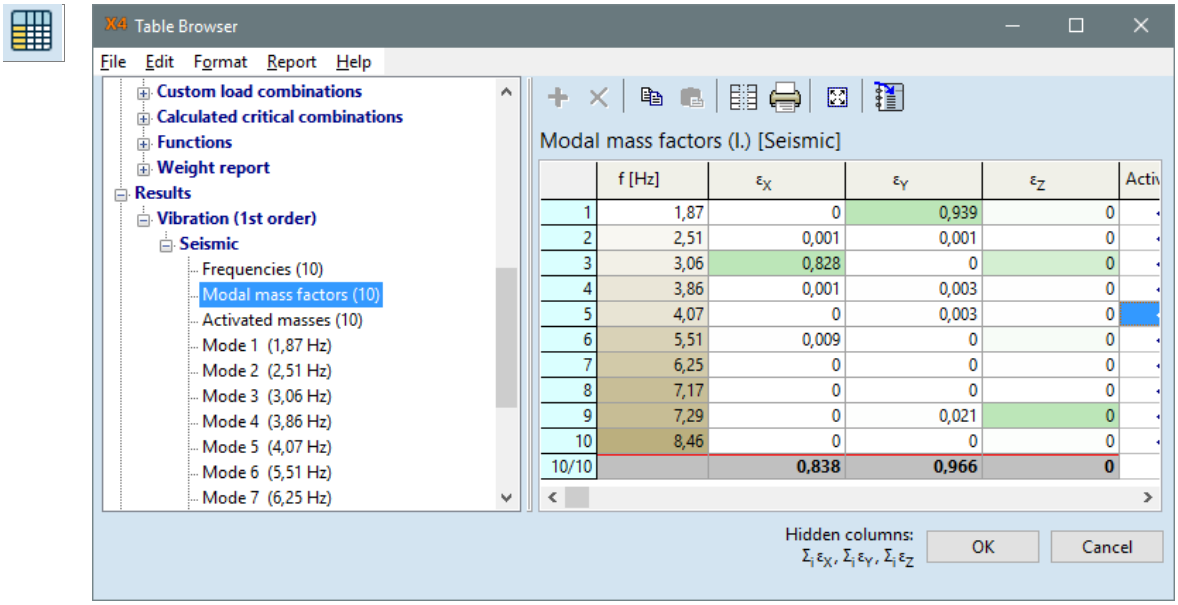
If the modal shapes used to generate the earthquake loads was obtained from a vibration analysis (**6.3 Vibration**), taking into account the stiffness reduction specified for the response spectrum analysis (**3.3.10 Stiffness reduction**) then the linear static analysis will be performed twice: one without stiffness reduction and one with the specified stiffness reduction.
The results without stiffness reduction are shown in non-seismic load cases and combinations without seismic action, while the results with stiffness reduction are shown in seismic load cases or combinations including seismic action (the same applies to envelopes and critical combinations).
If you had several vibration results calculated for different combinations, some with stiffness reduction and some without, and created multiple earthquake loads, one generated from shapes calculated with stiffness reduction and the other from shapes without stiffness reduction, then in seismic load cases or in combinations including seismic action the included seismic action will determine which results are displayed. If the seismic load case was generated from a shape obtained with stiffness reduction, then results with stiffness reduction will be displayed. Otherwise results without stiffness reduction will be displayed.
The following parts provide details on the calculation process. First, the general Eurocode procedure is presented followed by a summary of country-specific modifications.

4.10.25.1 Seismic load calculation according to Eurocode 8

1. Calculate vibration modes

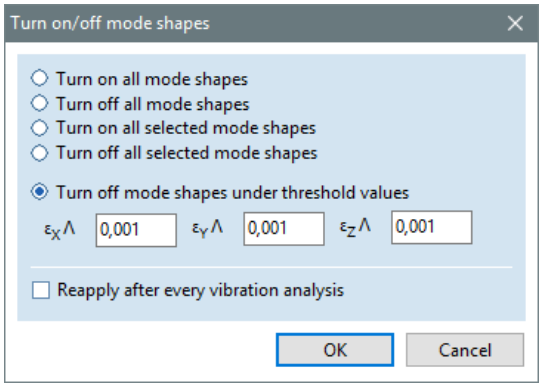


Seismic load generation requires information on undamped free vibration frequencies and corresponding mode shapes. Thus, the first step in creating seismic loads is the calculation of a sufficiently large number of mode shapes and corresponding frequencies of vibration. This analysis is performed at the *Vibration* tab by running a *Vibration analysis* (5.2).
First order vibration analysis is typically sufficient for seismic load generation. Results depend on the stiffness of structural elements and the mass distribution of the structure. Masses can be determined based on a pre-defined load combination that represents the so-called seismic mass of the structure. Alternatively, lumped masses can be placed in the structural model by the user. Stiffness reduction in reinforced concrete elements can also be taken into consideration automatically.
Use the *Modal mass factors* table in the *Table Browser* to check if the number of calculated vibration modes is sufficiently large to fulfil the requirements of the applicable standard. The general Eurocode 8 has two requirements: (1) the total considered modal mass in each direction shall exceed 90% of the total mass; (2) all mode shapes corresponding to a modal mass larger than 5% of the total mass shall be taken into consideration. The sums of modal mass factors are shown at the bottom of the table.



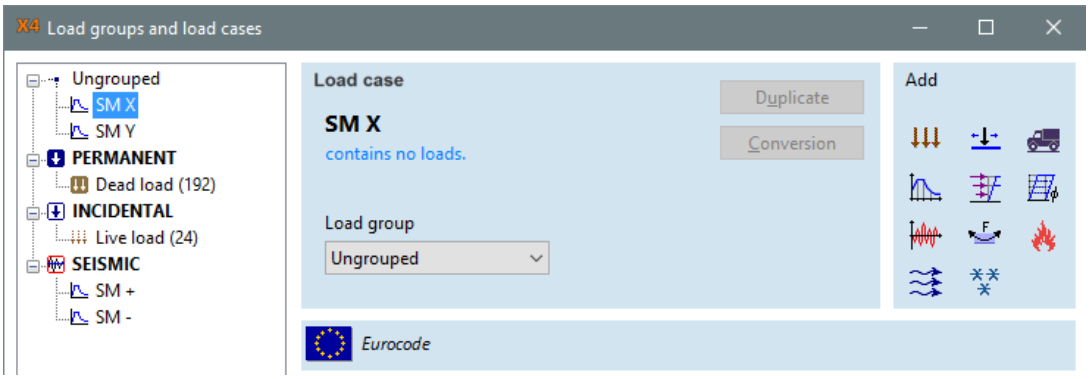
	f [Hz]	ε_X	ε_Y	ε_Z	Acti
1	1,87	0	0,939	0	
2	2,51	0,001	0,001	0	
3	3,06	0,828	0	0	
4	3,86	0,001	0,003	0	
5	4,07	0	0,003	0	
6	5,51	0,009	0	0	
7	6,25	0	0	0	
8	7,17	0	0	0	
9	7,29	0	0,021	0	
10	8,46	0	0	0	
10/10		**0,838**	**0,966**	**0**	

The user can indicate which mode shapes shall be used for seismic load generation by ticking the cell in the *Active* column of the table.
Right-clicking any cell in the *Active* column triggers a popup menu, where the *Turn on/off mode shapes* dialog can be selected. This dialog window lets the user define threshold values for automatic mode shape filtering in each direction. The checkbox at the bottom of the window allows the user the request automatic filtering after every vibration analysis.
Filtering mode shapes can significantly reduce the number of seimic load cases and consequently, the time required for static analysis.



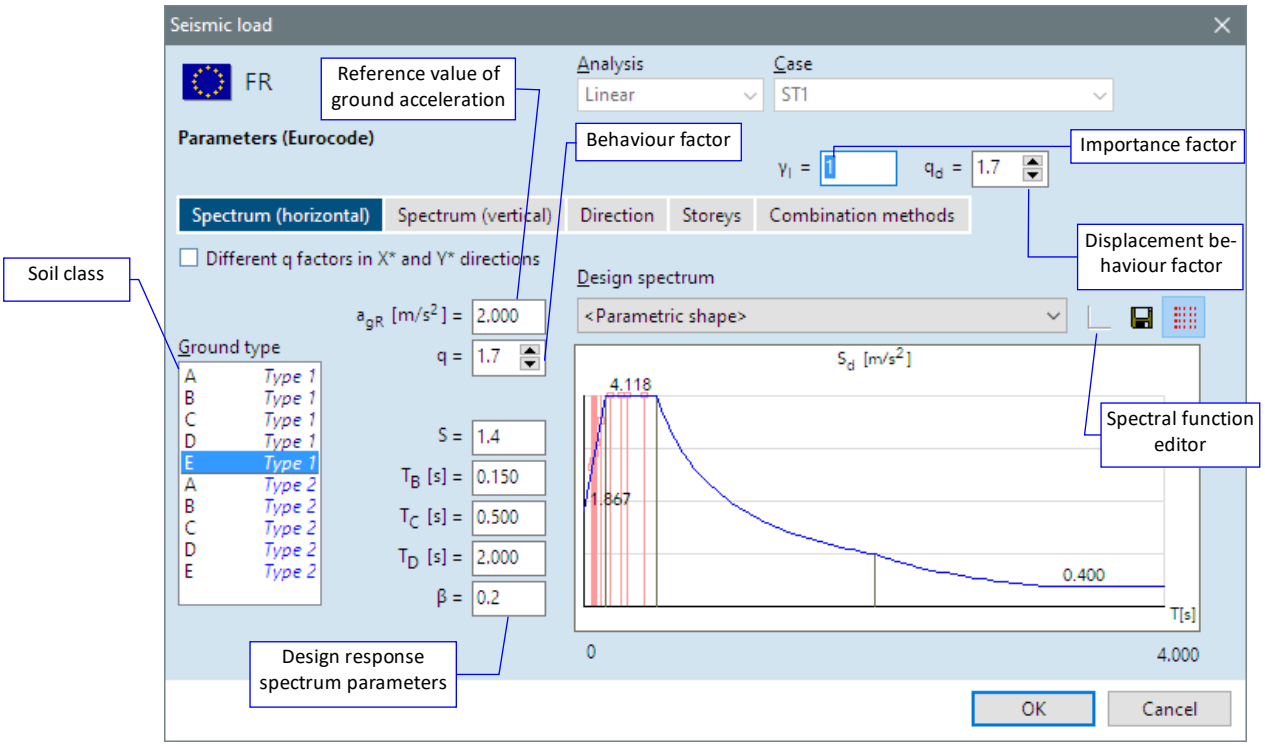
2. Create a new seismic load case

After having the mode shapes available, the seismic load case shall be created at the *Loads* tab in the *Load Cases and Load Groups* dialog window. Select the *Seismic* load case among the icons of *New Cases*. The program will automatically create several new load cases for the seismic load and these will be supplemented by others after the details of seismic load generation are set. The naming convention for load cases is explained in the static analysis part of this section.



3. Set seismic load parameters

Select any of the generated seismic load cases and click on the *Seismic load* icon in the *Loads* tab to open the *Seismic load* dialog window.



Spectrum (horizontal)

The first tab of the dialog window lets the user set details of the **horizontal response spectrum**. The horizontal response spectrum is used to calculate equivalent static loads for each vibration mode shape in two orthogonal horizontal directions. The spectrum can either be automatically generated using the functions given in EC8-1 4.2.4 and the parameters provided by the user, or a custom spectrum can be defined in the *Spectral function editor*.



Spectral function editor to create custom spectra



Save spectrum as...

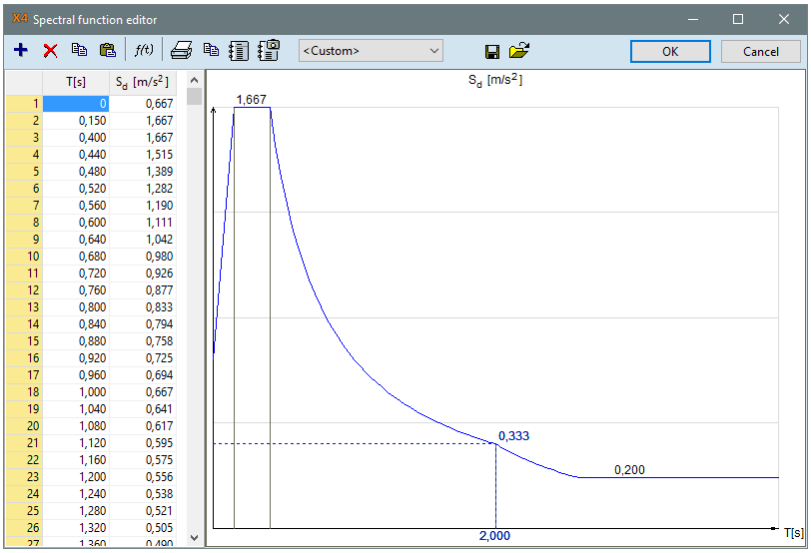


Display time period of calculated modal shapes: Turns on/off the display of time periods provided by the vibration analysis. Time periods are marked by a vertical pink line at the respective T value.

The automatically generated spectrum uses the following parameters:
a_{gR} – Reference peak ground acceleration on rock m/s^2
q – Behaviour factor for the reduction of horizontal seismic effects as per EC 8-1 3.2.2.5 (3)P. After checking *Different q factors in X and Y directions*, independent q_X and q_Y values can be defined by the user.
S, T_B, T_C, T_D – Soil-specific parameters that define the spectral shape. If the type of soil is selected from the list of soil classes, these parameters are automatically loaded according to Tables 3.2 and 3.3 in EC 8-1. Every value can be overridden by the user.
β – Lower bound factor for the horizontal design spectrum. The recommended value in EC 8-1 3.2.2.5 (4)P is 0.2.
γ_I – Importance factor as per EC 8-1 4.2.5
q_d – Displacement behaviour factor as per EC 8-1 4.3.4 (1)P

If the model contains seismic isolators a modified spectrum is used. See **3.1.17.2 Seismic isolators**.

Custom spectra are defined in the *Spectral function editor* after setting the *Design spectrum* from *Parametric shape* to *Custom*. Spectra are created in the editor by a piecewise linear approximation of the spectral shape over a list of pre-defined natural period – spectral acceleration coordinate pairs. Coordinates can be edited, copied to or pasted from other programs.

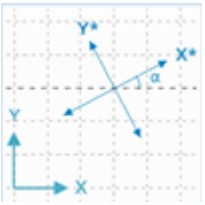


Spectrum (vertical)

The second tab of the *Seismic load* dialog window corresponds to the parameters of the **vertical response spectrum**. Note that the effect of seismic excitation in the vertical (Z) direction is only taken into account if a vertical response spectrum is defined. Vertical loads are not generated by default. Parameters and the layout of this tab are identical to the first one.

Direction

A third tab is *Direction* where the angle of the seismic action can be set. If this value is not zero seismic loads will be generated not in the global *X* and *Y* directions but in *X** and *Y** directions at the α angle to add the global directions.



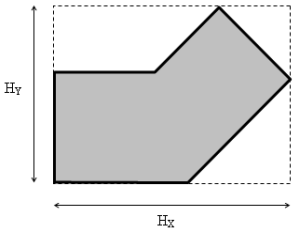
Storeys

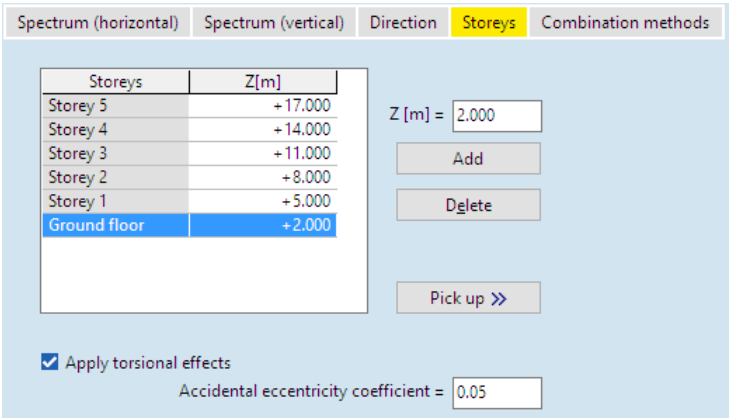
The fourth tab is *Storeys*. It lets the user decide if **torsional effects** due to accidental eccentricity shall be taken into account. If the user wishes to consider such effects, then the program calculates additional torsional moments around the vertical axis for every storey and every mode shape. The magnitude of torsional moments depends on the horizontal load and eccentricity of each storey.
Horizontal loads are obtained from load cases corresponding to each vibration mode and horizontal direction.
The size of eccentricity relative to the total size of each storey is defined by the user. An accidental eccentricity of 5% is recommended in EC 8-1 4.3.2 (1)P.

Torsional moments are calculated with the following expressions:

$$T_{X,i,j} = F_{X,i,j} \cdot (\pm e H_{Y,i})$$
$$T_{Y,i,j} = F_{Y,i,j} \cdot (\pm e H_{X,i})$$

where *F* is the horizontal force in mode *j* at the i^{th} storey; *H* is the size of the i^{th} storey in the given X or Y direction. Torsional moments are considered with either + or – signs, but always with the same sign at all storeys.

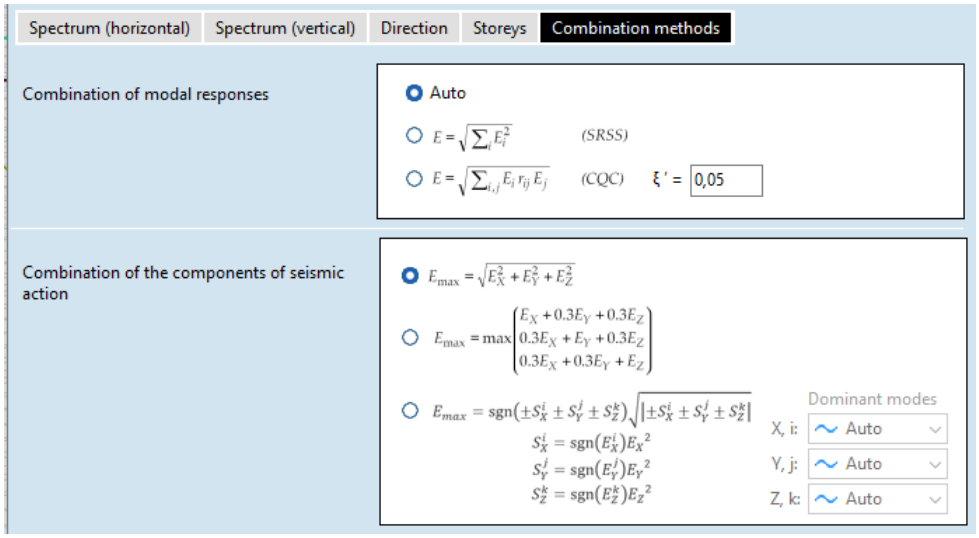




Combination methods

Combination of modal responses

The last tab of the *Seismic Load* dialog window provides control over the combination of results for individual modes and individual directions. Results from individual modes can be combined using either the Square Root of Sum of Squares (SRSS) or the Complete Quadratic Combination (CQC) method. The latter is considered more appropriate if the vibration modes of the structure are not well separated (i.e., vibration frequencies are close to each other).
Selecting the automatic option lets the program automatically decide if application of the CQC method is warranted by the vibration results. The program considers modes *i* and *j* well separated if the following condition holds: $T_j / T_i < 0.9$.



Combination of the components of seismic action

Results in the two horizontal and the vertical direction can be combined using one of the three commonly used combination methods.
The first method uses the SRSS method to combine the results of different directions, whilst the second uses the SAS (Scaled Absolute Sum) method.
The third method, the SRSS-type method based on the dominant mode, is an improved version of the classical quadratic summation. In that method, the signs are lost as a result of squaring. For this reason, to remain on the safe side all possible combinations of signs must be taken into account during dimensioning. However, some of these combinations are physically impossible. For example: under a horizontal earthquake load, it is certainly not possible for all vertical support force components to point in the same direction. The signed SRSS method excludes physically impossible combinations based on the sign of the modal response associated with each vibration mode (i, j, k) considered dominant in all three directions. In this case, the programme considers the mode with the largest mass participation factor in the given direction to be dominant.
The application of the third method requires particular care if (a) there is no dominant mode, (b) several modes with similar mass fractions arise in the building, or (c) torsional effects are significant.
The seismic effects in various directions can also be summed up manually by creating custom load combination. SMX, SMY and SMZ and load cases connected to these (see below) are also available in the table of load combinations, see **4.10.2 Load combinations.**

☞ **Warning! Multiple load cases belonging to the same direction or SMX, SMY, SMZ and SM+/- load cases must not be taken into account in the same combination collectively.**

After the load parameters are set and the *Seismic Load* dialog window is closed by clicking the OK button, the program generates a load case for each mode and direction according to the settings defined by the user. The following naming convention is used for these load cases:
Load cases ending in **01X, 02X, ...nX, 01Y, 02Y, ...nY, 01Z, 02Z, ...nZ** are the equivalent forces in X, Y or Z direction corresponding to individual mode shapes.
Load cases ending in **01tX, 02tX, ...ntX, 01tY, 02tY, ...ntY** are the torsional moments due to accidental eccentricity in X or Y direction.

4. Perform linear static analysis and evaluate results

Modal response spectrum analysis uses a combination of linear static analysis results to get the design seismic effects for the structure under consideration. After the load cases for each vibration mode in each direction are created, the next step in the MRSA procedure is to run the linear static analyses. The program will automatically calculate the effect of each mode in each direction and combine these effects according to the settings in the Seismic Load dialog window.

After running linear static analyses, there are several seismic load cases available in the list of results under the *Static* tab. We use the following naming convention for seismic load cases:
Besides load cases corresponding to individual modes, there are two types of additional results. Load case names ending in **X**, **Y**, or **Z** contain the combined response from modal results in the X, Y or Z direction. If accidental eccentricity is considered, there is an additional **a** or **b** in the name of load cases corresponding to horizontal directions. The letters a or b correspond to torsional effects with positive or negatvie eccentricity, respectively. **Ya** for instance is a combination of modal results in the Y direction considering the effect of positive eccentricity of seismic masses in the X direction.
Combination of load cases in X, Y, Z directions is performed by the program automatically. Such combination yields a single, unanimous result if there is no accidental eccentricity. If accidental eccentricity is assumed, the combination of effects from several directions is more ambiguous. There are four basic cases considered in the program depending on the direction of eccentricity in the X and Y load cases. Each of the following combinations are performed and stored in a load case with its name ending with the particular number:

1 = Xa + Ya + Z
2 = Xa + Yb + Z
3 = Xb + Ya + Z
4 = Xb + Yb + Z

In the case of SRSS (Square Root of Sum of Squares) and SAS (Scaled Absolute Sum) methods, due to the nature of the calculation, load cases contain only positive values. So results are absolute values by definition. Therefore, the load cases presented above would contain only positive values. The least favorable loading scenario is a combination of seismic effects and effects from other sources such as gravity loading. Creation of this load combination is facilitated in the program by providing two load cases for each seismic load: seismic effects with only positive and seismic effects with only negative values are identified by **a + and a – sign** at the end of the name of their load cases. Note that the absolute values of corresponding responses (internal forces, displacements, etc.) are identical in the + and – load cases.
In the case of a signed SRSS combination, we assign a sign to each earthquake load case using the dominant vibration mode. In this case, several earthquake load cases are generated (four in the case of a horizontal earthquake without torsional vibration, namely +X+Y; +X-Y; -X+Y; -X-Y), but it is not necessary to design for every possible combination of stresses with all possible signs, as the various earthquake load cases already encompass all possible combinations of stresses.

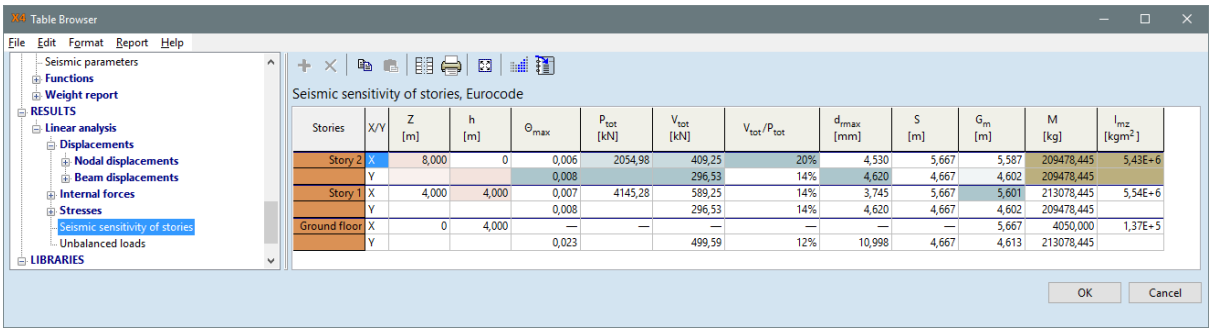
Displacement results shown in the *Static tab* are automatically scaled by the q_d factor specified in the Seismic Load dialog window.



Second order effects due to geometric nonlinearity shall be taken into consideration when evaluating the results of MRSA. The program supports such calculations with the data in the *Seismic sensitivity of storeys* table in the *Table Browser*. Independent results are available in the table for each horizontal direction.
The following quantities are calculated by the program:

θ_{max}	plastic stability index, also known as the inter-storey drift sensitivity coefficient, as per EC8-1 4.4.2.2 (2)
P_{tot}	total gravity load above the storey
V_{tot}	total seismic storey shear
d_{max}	design inter-storey displacement calculated as the relative displacement of corresponding storey centroids
S	location of the shear centre, which is defined independently of the standard earthquake analysis as the shear centre of the cross-section created by cutting walls and columns below the storey (see in detail **3.3.4.1 Storeys**)
G_m	location of the centroid
M	storey mass
I_{mz}	moment of inertia at the centroid about the Z axis

G_m, M and I_{mz} are summarised on the basis of the nodal masses and positions of the nodes belonging to the storey. The nodal masses are calculated from the load combination specified in the vibration analysis. The set of nodes belonging to the storey consist of the nodes of the slab and the nodes locating in the upper 95% of the height of the storey under the slab and in the lower 5% of the height of the storey above the slab.



Stories	X/Y	Z [m]	h [m]	Θ_{max}	P_{tot} [kN]	V_{tot} [kN]	V_{tot}/P_{tot}	d_{rmax} [mm]	S [m]	G_m [m]	M [kg]	I_{mz} [kgm^2]
Story 2	X	8,000	0	0,006	2054,98	409,25	20%	4,530	5,667	5,587	209478,445	5,43E+6
	Y			0,008		296,53	14%	4,620	4,667	4,602	209478,445	
Story 1	X	4,000	4,000	0,007	4145,28	589,25	14%	3,745	5,667	5,601	213078,445	5,54E+6
	Y			0,008		296,53	14%	4,620	4,667	4,602	209478,445	
Ground floor	X	0	4,000	—	—	—	—	—	—	5,667	4050,000	1,37E+5
	Y			0,023		499,59	12%	10,998	4,667	4,613	213078,445	

☞ ***AXISVM does not consider the influence of second order effects automatically through internal force amplification. The user shall evaluate the sensitivity of the structure to such effects. If the influence of second order effects can be taken into consideration through force amplification, then the user shall determine the appropriate value of the amplification factor as per EC8-1 4.4.2.2 (3). That amplification factor can be specified in the program as the f_{se} factor in the following design tools:***

6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module
6.5.8 Shear resistance and reinforcement calculation for plates and shells
6.5.10 Column reinforcement – RC2 module
6.5.11 Composite column design – RC2 module
6.5.12 Beam reinforcement parameters (uniaxial bending)
6.5.14.2 Fire design of reinforced concrete surfaces – RC8-S module
6.5.15 Punching analysis – RC3 module
6.5.16 Footing design – RC4 module
6.5.17 Design of reinforced concrete cores and walls – RC5 module
6.5.18 Stress-strain analysis of reinforced concrete structures – RC6 module
6.6 Steel design
6.7 Timber design
6.8 Design of CLT domains – CLT modul
6.9 Masonry wall design – MD1 module

☞ ***The fse factor scales only the internal forces from seismic load.***

4.10.25.2 Seismic load calculation according to Swiss SIA 261

The calculation procedure, dialog windows, assumptions and features are identical to those presented in the general Eurocode 8 section. The following paragraphs list only the modifications that are unique to the SIA 261 standard.

3. Set seismic load parameters

The following modifications are applied at the *Seismic Load* dialog window:

- Horizontal and vertical response spectra are based on SIA261 16.2.4.
- Soil-specific parameters are set according to Table 25 in SIA261.

4.10.25.3 Seismic load calculation according to Italian NTC 2018

The calculation procedure, dialog windows, assumptions and features are identical to those presented in the general Eurocode 8 section. The following paragraphs list only the modifications that are unique to the latest NTC standard.

3. Set seismic load parameters

The following modifications are applied at the *Seismic Load* dialog window:

- Horizontal and vertical response spectra are based on NTC 2018 3.2.3.2.
- Soil-specific parameters are set according to Tables 3.2.V-VII in NTC 2018 3.2.3.2.
- If the model contains seismic isolators a modified spectrum is used, depending on the limit state (SLO, SLD, SLV, SLC) set in the seismic parameters dialog according to NTC 2018 3.2.1. See **3.1.17.2 Seismic isolators.** As multiple seismic actions can be defined for the same model, seismic results obtained for different limit states can be compared.
- Results of the seismic isolator system are available in a separate table.

4.10.25.4 Seismic load calculation according to Romanian P100-1

The calculation procedure, dialog windows, assumptions and features are identical to those presented in the general Eurocode 8 section. The following paragraphs list only the modifications that are unique to the P100-1 standard.

3. Set seismic load parameters

The following modifications are applied at the *Seismic Load* dialog window:

- Horizontal and vertical response spectra are based on P100-1 3.1(7), 3.1(13), and 3.2(1).

4.10.25.5 Seismic load calculation according to Dutch NPR 9998:2018

The calculation procedure, dialog windows, assumptions and features are identical to those presented in the general Eurocode 8 section. The following paragraphs list only the modifications that are unique to the NPR 9998:2018 standard.

3. Set seismic load parameters

The following modifications are applied at the *Seismic Load* dialog window:

- Horizontal and vertical response spectra are based on NPR 9998:2018 3.2.2.2.3.

The $a_{g;d}$ acceleration and p, T_B, T_C, T_D parameters shall be specified by the user according to the code or the code conforming NPR 9998 web tool.

4.10.26 Pushover loads – SE2 module

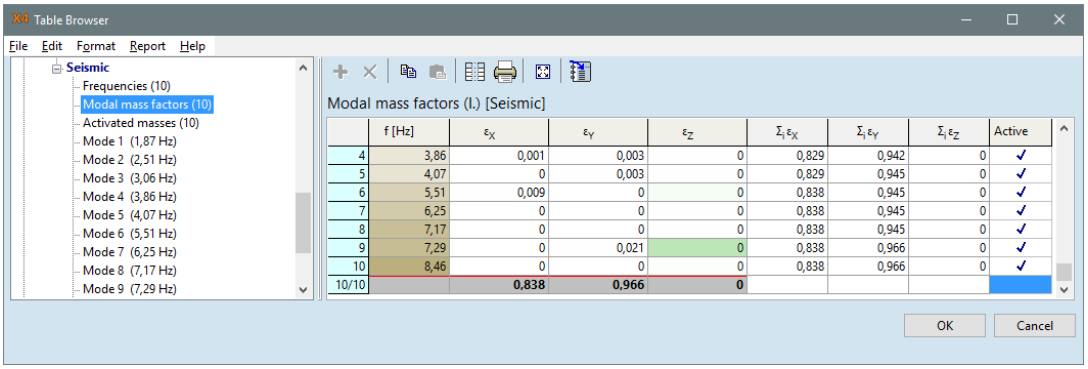
Pushover loads are generated according to the regulations of Eurocode 8 by default. The load generation uses undamped free vibration frequencies and corresponding mode shapes of the model, therefore loads can only be generated if a vibration analysis has already been performed.

Steps of pushover load generation

The following description shows how to create pushover load cases and set their properties before performing a nonlinear static analysis.

1. Calculate vibration mode shapes and frequencies.

When running the vibration analysis be sure to use the convert loads to masses option with the appropriate load case if there are loads defined that need to be considered static. Check the table of *Modal mass factors* in the Table Browser. Vibration results will appear only if the *Vibration* tab is selected.

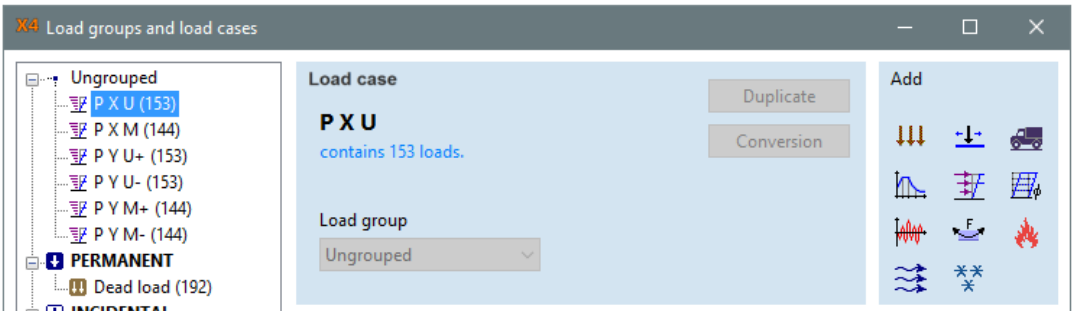


	f [Hz]	ε_X	ε_Y	ε_Z	$\Sigma_i \varepsilon_X$	$\Sigma_i \varepsilon_Y$	$\Sigma_i \varepsilon_Z$	Active
4	3,86	0,001	0,003	0	0,829	0,942	0	✓
5	4,07	0	0,003	0	0,829	0,945	0	✓
6	5,51	0,009	0	0	0,838	0,945	0	✓
7	6,25	0	0	0	0,838	0,945	0	✓
8	7,17	0	0	0	0,838	0,945	0	✓
9	7,29	0	0,021	0	0,838	0,966	0	✓
10	8,46	0	0	0	0,838	0,966	0	✓
10/10		**0,838**	**0,966**	**0**				

Although there is no requirement in Eurocode 8 for the minimum value of seismic equivalence coefficient, it is strongly advised to perform standard pushover analysis only on structures having clearly dominant mode shapes in each horizontal direction. The coefficients for each mode shape are listed in the *Modal mass factors* table. Unlike seismic loads, standard pushover load generation uses a single vibration mode shape for each load case, therefore the sum of seismic equivalence coefficients is not important. Thus, there is no need to calculate a large number of modes if the dominant ones are among the first few.

2. Create a new pushover load case.

Pushover load cases can be created, renamed and deleted in the *Load cases & load groups* dialog window. The initial configuration of four load cases is created by clicking on the *Pushover load* button.

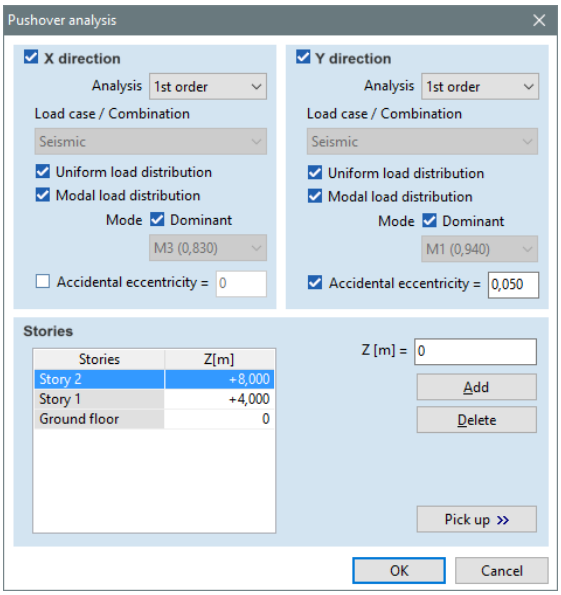


3. Setting pushover load parameters.

After creating the load cases the parameters for the loads can be set up by clicking on the *Pushover analysis* button on the toolbar of the *Loads* tab.



The parameters for load generation can be set up at the top, while the storey levels used for interstorey drift calculation are specified at the bottom part of the window (Previously defined storey data is also available here). Load generation for a specific direction can be disabled using the topmost checkboxes. This is useful in case the model is two dimensional. For each direction the vibration analysis type and the assigned load case needs to be selected first. The checkboxes below turn the uniform and modal load generation on or off respectively. The uniform load distribution option generates nodal forces proportional to the masses assigned to each node in the model. The modal load distribution uses the mode shape weighed by the masses at each node to generate the nodal force distribution. In both cases the sum of forces generated is *1 kN* in the same horizontal direction.



If modal loads are to be generated it is possible to override the dominant mode shape used for load generation. It is important to emphasize that this option is only for advanced users and Eurocode 8 requires the use of dominant mode shape for analysis. The number in parentheses by each mode number shows the corresponding seismic equivalence coefficient.

It is also possible to include the effects of accidental eccentiricities and the resultant torsional moments. AXISVM calculates the force system equivalent to the torsional moments for each storey. The sum of the signed pushover loads will be still 1 kN but the resultant force will be eccentric causing torsion. Pushover loads are generated only after closing the dialog window. Unnecessary load cases are also removed at this time.

4. Run a Nonlinear Static Analysis

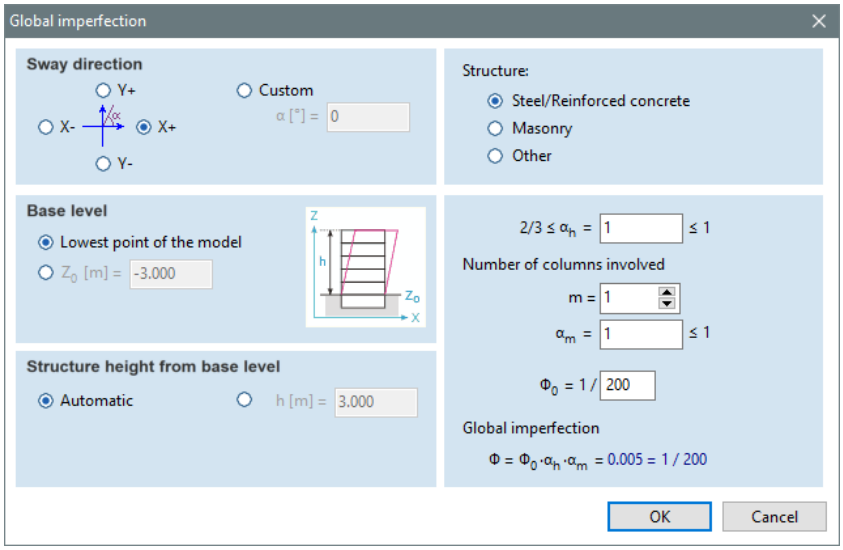
After defining loads for pushover load cases the pushover analysis shall be run using the Nonlinear Static Analysis button under the Static tab of the main window. Setting the solution control to Pushover lets the user select a parametric and a constant load case. The parametric load case is typically a pushover load case, however AXISVM does allow users to define other load cases as parametric too. The constant load case represents gravitational loads in most cases. The other settings of this dialog window are explained in **6.1 Static** chapter.

The control node shall be one of the nodes at the top of the structure. It is important to set the direction of the analysis according to the direction of the parametric load case. The stability of the analysis can be increased significantly by increasing the number of increments. Following geometric nonlinearity is recommended for pushover analyses. The analysis is started by clicking the OK button.

Generation of capacity curves and related results are explained in **6.1.5 Pushover capacity curves** chapter.

4.10.27 Global imperfection





After selecting an imperfection load case, the above icon for imperfection load becomes enabled. After setting the imperfection parameters the global imperfection is applied to the structure (its displayed amplitude is magnified to make it more visible).
Imperfection load cases can contribute to load combinations used to perform analysis with geometric nonlinearity. Nodes will be shifted from their original positions and the other loads in the combination will be applied to the distorted structure.

Parameters Global imperfection requires the following parameters.

Sway direction Defines the direction of the shift. It can act
- in global *X* or *Y* direction
- in a *Custom* α angle measured from the global X axis

Base level It is the Z_0 level where the sway begins. Two options are available
- Set it to the *Lowest point of the model*
- Set it to a custom Z_0 level

Structure height from base level The structure height is measured from the Z_0 base level. Available options are
- Set it from the highest point of the model
- Set it to a custom *h* value

Inclination Inclination is calculated on the following ways:

Steel/Reinforced concrete

$$\Phi = \Phi_0 \alpha_h \alpha_m$$

where

α_h is a reduction factor: $\alpha_h = \frac{2}{\sqrt{(h[m])}}$ with $\frac{2}{3} \leq \alpha_h \leq 1$

α_m is a reduction factor: $\alpha_m = \sqrt{0.5\left(1+\frac{1}{m}\right)}$ where *m* is the number of columns involved per level

Masonry

$$\nu = \frac{1}{100\sqrt{h}}$$

Other * Arbitrary value can be specified by the user.

4.10.28 Geometric imperfections based on buckling shapes



The only parameter of such load cases is an imperfection based on buckling shapes. Whenever a load combination contains an imperfect load case, AXISVM calculates with imperfect geometry. Though imperfection load cases based on buckling shapes can be created, deleted and edited, but load cases using imperfect shapes are created and deleted automatically according to definition and deletion of imperfect shapes.

For definition of imperfect geometry see **6.2.1 Creating imperfect geometry from buckling shapes – IMP module**

4.10.29 Tensioning - PS1 module

Tendons can be assigned to a continuous selection of beam, rib, or domain elements. After defining tendon properties and the tensioning process AXISVM determines the stress distribution in tendons immediately after anchoring and the equivalent loads at the end of tensioning (load case name-T0). After completing a static analysis, the program determines the time dependent losses of prestress and the long-term equivalent loads from the result of quasi-permanent combinations (load case name-TI). Tendon trajectory tables can be generated with user-defined steps.

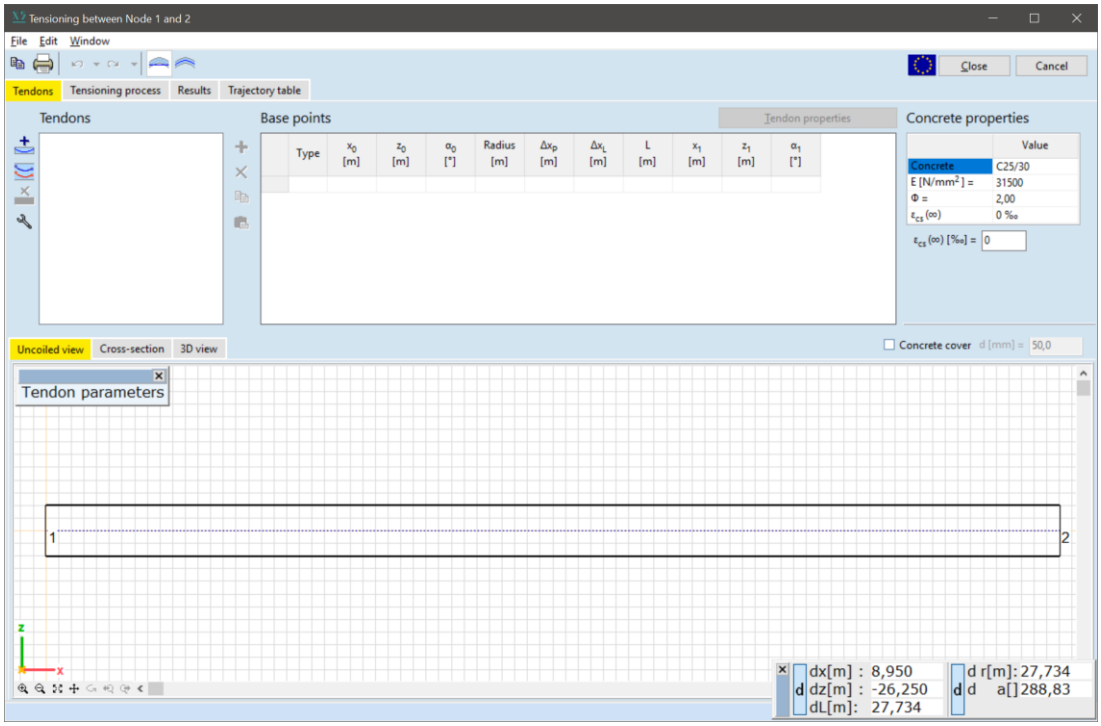
4.10.29.1 Post-tensioning of beam and rib elements



Click on this icon of the *Loads* page to assign tendons to the selected beam or rib elements.

Tendons

The first tab is to define tendon parameters and geometry.



Coordinate systems



The geometry of a tendon can be defined in a fixed coordinate system (left icon) or in a moving coordinate system (right icon).

The **fixed coordinate system** defines *x* axis on the straight line connecting the first and last node. Axis z is perpendicular to axis x and the initial cross section's y axis. While the direction of axis *y* is determined in a right-handed coordinate system.

If tensioned elements are not on a straight line but defined by an arc, or a polygon, a **moving coordinate system** can be chosen. If the beam is an arc, the coordinate system moves along this arc. If the structure is defined by a polygon, the geometry is interpolated by a cubic spline. The *x* axis of this coordinate system fits to the tangent of the generated curve, while *z* axis *z* follows the local *z* axis of the beam at each and every point. The *y* axis is defined by the right-hand rule.

The geometry of the tendon can be displayed in the following views:

Uncoiled view The position of the tendon is displayed in the working plane. Continuous lines represent the section of the working plane and the elements. Dashed lines show the center line and the projection of the beams' edges on the uncoiled working plane.

Cross-section Tendon positions are displayed in the *y-z* section (cross-section).

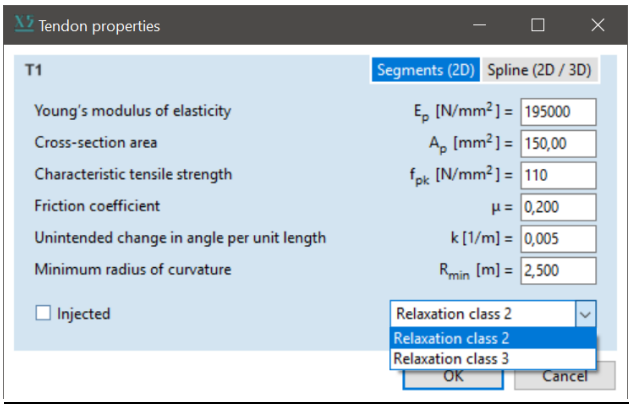
3D view The 3D view shows the cable and the elements in 3D.

The base points of the geometry and the inflection points are displayed both in the uncoiled view and 3D view.

Icons on the vertical toolbar beside the tendon list are



Add new tendon
Geometry for the new tendon can be defined using the toolbar beside the diagram. First the tendon properties must be defined.



Segments (2D) The geometry of the tendon is defined as a series of segments (straight-parabola-arc connections).

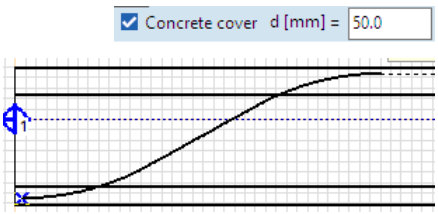
Spline (2D/3D) The geometry of the tendon is defined as a spline.

E_p	modulus of elasticity of tendon steel,
A_p	cross-section area of the tendon
f_{pk}	characteristic tensile strength of tendon steel
μ	coefficient of friction between the tendon and its sleeve
k	unintentional angular displacement for internal tendons per unit length. Shows the precision of workmanship. Usually 0.005 1/m < k < 0.01 1/m.
R_{min}	Minimum radius of curvature. Where the radius of curvature is smaller than this limit tendons are displayed in red.
Relaxation class	*Relaxation class* depends on the relaxation properties of the tendon. Wires and strands belong to *Relaxation class 2,* hot-rolled and processed bars belong to *Relaxation class 3.*
Injected	If ducts are filled with an injected material this option must be checked.

Note: To modify the properties above choose the active tendon and click on the *Tendon properties* button.

Concrete cover

If the mouse is near the top or bottom edge of the section the cursor finds the edge. If *Concrete cover* is checked the point entered will not be on the edge but inside the section as defined by d.
This function works both for segments and splines.



Drawing tendons if Spline (2D/3D) is selected

Click the icons on the vertical toolbar beside the drawing and enter base points. AXISVM determines the trajectory passing through these base points as a cubic spline to minimize curvature. For each basepoint the angles of tangent can be specified by setting the α (top view) and β (side view) values in the table. Enter values between –180° and 180°. Initial values are 0°. Existing base points can be dragged to a new position using the mouse.



Draw tendon in 2D.
Base points can be created by clicking the diagram or using the coordinate window. Double-click or right click and choose *Complete* from the popup menu to end the spline. The tendon position within the cross-section has to be specified only at the first base point. Further base points will be in the local x-z (uncoiled) plane containing the first base point.

Steps of drawing a tendon in 2D:

1. Select the *x* position of the cross-section where the tendon basepoint will be defined.
2. Place the tendon to the proper position in the cross-section view.
3. Place the other points of the tendon in the uncoiled view.



Draw tendon in 3D.
The tendon position within the cross-section has to be specified at every basepoint. To end tendon definition, click the right mouse button and select *Complete* from the popup menu.
Steps of drawing a tendon in 3D:

1. Select the position of the cross-section where you want to define a tendon basepoint.
2. Set the proper tendon position in the cross-section view.

Repeat Step 1 and Step 2 to define all basepoints.



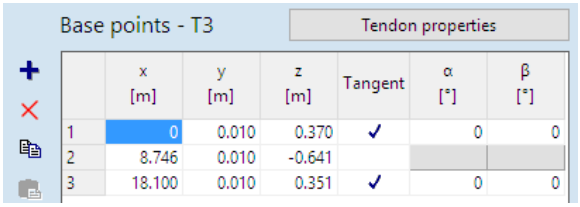
Add new base point
Click the cable to add a new base point. In case of multiple tendons this function works on the active tendon.



Delete base point
Click on an existing base point to delete it. After deleting the second base point the entire tendon is deleted. In case of multiple tendons this function works on the active tendon.

Table of base points

Base point properties can be edited in the table. *x, y, z* are the Cartesian coordinates of the base points. If *Tangent* is checked, *α* and *β* (the angle between the x direction and the tangent on the *x-y* and *x-z* plane) can be edited.





Add a new base point



Copy tendon data to the Clipboard



Delete selected lines



Paste tendon data from the Clipboard

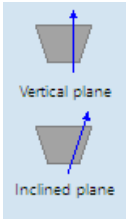
Drawing tendons if Segments (2D) is selected

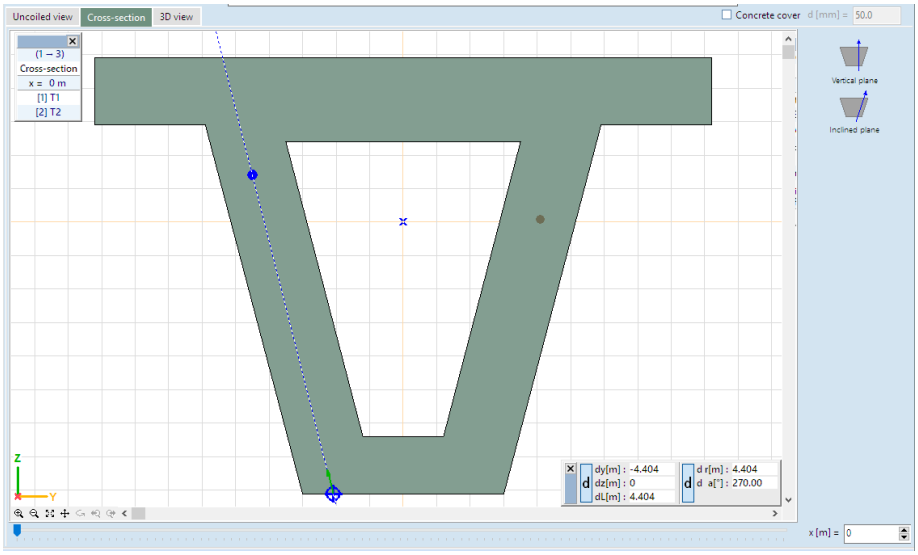
This method builds tendons from segments. Each segment is built from one, two or three basic subsegments.

Note: If the moving coordinate system was selected, *x, y* and *z* coordinates represent points in the moving coordinate system.

Setting the tendon coordinate system

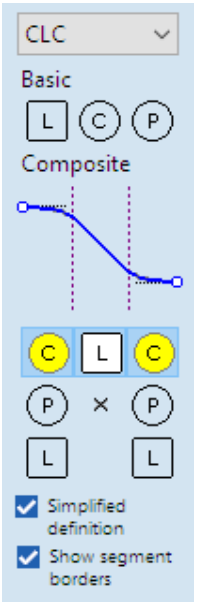
Segments are planar curves within the uncoiled plane. The position and orientation of this plane can be set in the *Cross-section* view.
Choose an *x* location along the beam by dragging the trackbar at the bottom or entering an *x* value.
Choose *Vertical plane* if tendons will run in a vertical plane. Click within the *(y, z)* section to define the *(x, y, z)* origin of the tendon coordinate system. The z axis of the tendon coordinate system will be parallel with the local z axis of the beam.
Choose *Inclined plane* if tendons will run in a non-vertical plane. Click within the *(y, z)* section to define the *(x, y, z)* origin of the tendon coordinate system and click on another point of the section to set the inclination.
The planar tendon geometry will be defined and displayed in this system in the *Uncoiled view*.





Defining segments

Each segment is built from one, two or three basic subsegments. Subsegment geometry can be chosen from the toolbar icons beside the tendon diagram. The one letter abbreviations are:



C	arc of a circle with a defined radius
P	parabolic arc
L	straight line
x	no intermediate subsegment

Options:

Basic	Click on L, C or P symbols (or select L, C or P from the dropdown list) to append a single line, circular or parabolic arc subsegment to the existing geometry with tangent continuity.
Composite	Click on the three columns of symbols (or select two or three letter codes from the dropdown list) to set a composite segment made of two or three subsegments.
Simplified definition	If a *Composite* segment is created and *Simplified definition* is checked end tangents of segments will always be horizontal. If the last subsegment of the first segment is an arc (like in an LC starting segment) all subsequent segments will be CLC. If it is a parabolic arc (like in an LP starting segment) all subsequent segments will be PLP.
Show segment borders	Segment boundaries are also displayed.

Then the points of consecutive segments can be placed.

Table of segments

The properties of the defined segments can be modified in the *Table of segments*. Editable values and functions are listed below

Segments - T2

	Type	x_0 [m]	z_0 [m]	α_0 [°]	Radius [m]	Δx_P [m]	Δx_L [m]	L [m]	x_1 [m]	z_1 [m]	α_1 [°]
1	PLP	0	0	0	-	1,000	1,000	3,000	3,000	-0,100	0
2	PLP	3,000	-0,100	0	-	1,000	1,000	3,000	6,000	0	0

x_0, z_0	the coordinates of the start point of the tendon segment,
α_0	the angle of the tendon tangent at the start point,
Radius	in the case of arc(s) the radius (radii) can be defined,
Δx_P	the x-directional length of the first parabola
Δx_L	the x-directional length of the straight line,

L length of the segment projected to the baseline

x_1, z_1 the coordinates of the end point of the tendon segment,

α_1 the angle of the tendon tangent at the end point.

Note: not all geometrical parameters are available for different segment types. For instance, the tangent at the start cannot be modified for straight line-arc segments, as the tangent is already defined by the coordinates of the start and end points and the radius of the arc.



Add a new segment



Delete selected segments



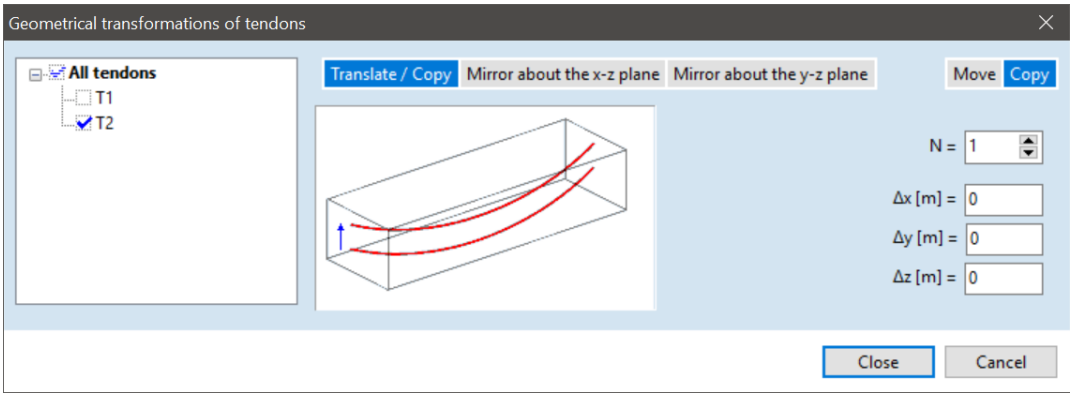
Copy tendon data to the Clipboard



Paste tendon data from the Clipboard



Geometrical transformations of tendons



Tendons selected in the tree can be translated or mirrored. Tendons can be copied or just moved. Copied tendons inherit the original parameters and the tensioning process assigned to them.



Delete tendon. Deletes the selected tendon.

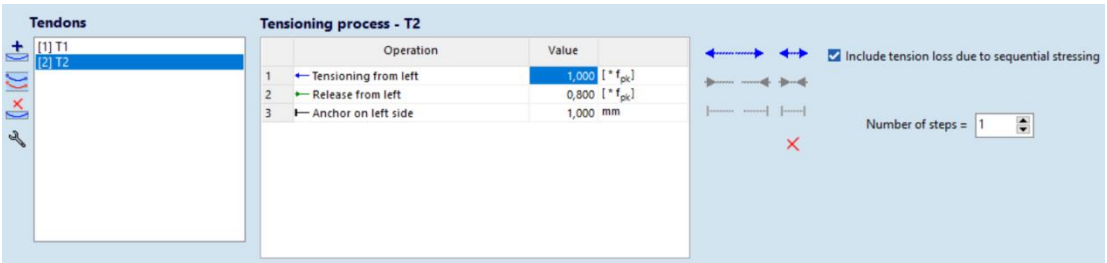


Options. Separate grid and cursor settings for the longitudinal and cross-section tendon diagram.

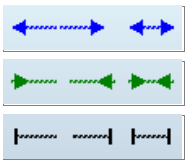
Concrete parameters Under the concrete parameters the material properties of the concrete can be checked. Moreover, $e_{cs}(\infty)$, the long-term value of the concrete shrinkage strain can be modified.

Tensioning process

The second tab is to define the tensioning process for tendons by determining the order of certain operations. Click on the proper symbol to add a new operation to the list.



Possible operations and parameters:



Tensioning from left / right / both side

Release from left / right / both side

Anchor on left / right / both side

Force as a fraction of the characteristic value of tendon steel tensile strength (f_{pk}).

Wedge draw-in of the anchorage device



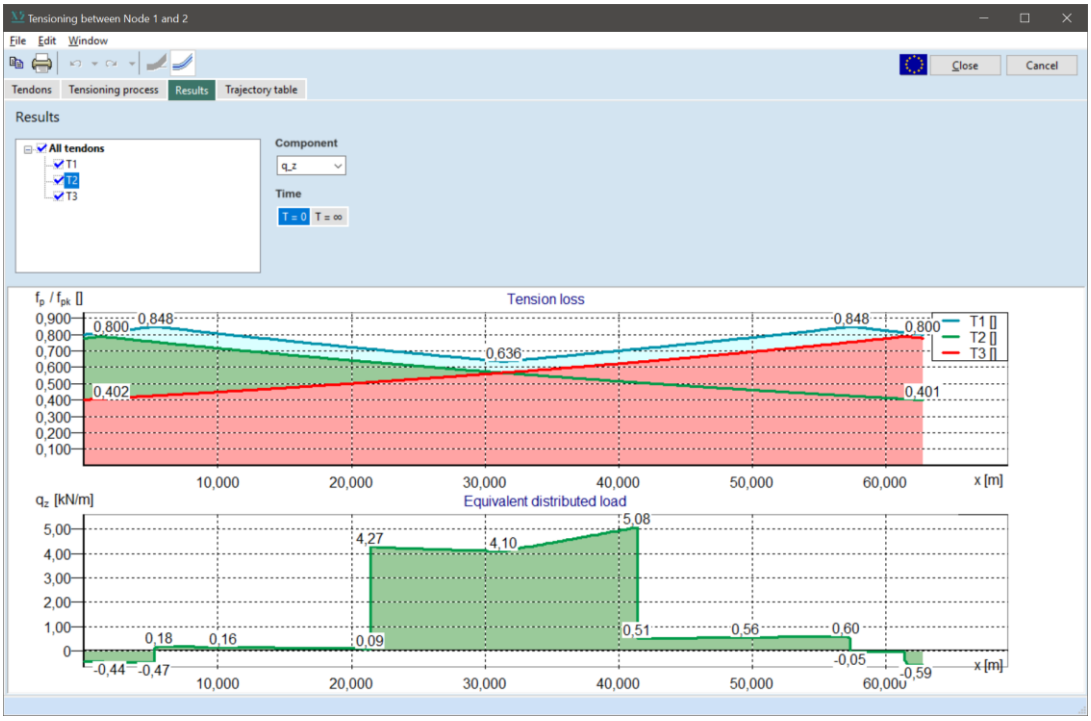
Deletes the last operation from the list.

Note: left is the starting point, right is the end point of the cable.

Include tension loss due to sequential stressing If more than one cable relates to a structural element, then these cables can be stressed at different time instant. Because of it, stress loss arises in the previously anchored tensioned cables. With this option the user can take into account this phenomenon.

Number of steps If the user takes into account the loss from sequential tensioning, then the number of the tensioning steps should be set up here.

Results If valid material parameters, geometry and tensioning processes are assigned to every tendon, result diagrams are displayed on the *Results* tab. The upper diagram shows the ratio of the actual tension and the characteristic tensile strength of the selected tendons. The lower diagram shows the chosen equivalent distributed load component for the selected tendons.



Component the component of the equivalent load system can be chosen. Three force components and three moment components are available.

Time the stresses in the cable and the equivalent load component can be illustrated right after the tensioning (choosing *T=0*) and after long term losses (choosing *T=∞*). The components, relating to *T=∞* can be chosen only after a static analysis.

Immediate losses of tension:

1. Tension loss due to friction between tendons and their sleeves at position x measured from the anchorage point along the tendon is calculated as

$$\sigma_\mu(x) = \sigma_{max}\left(1 - e^{-\mu(\Theta + kx)}\right),$$

where

σ_{max} is the maximum tension in the tendon

Θ is the sum of the absolute angular displacements over a distance *x*

2. Losses due to the instantaneous deformation of concrete are calculated as

$$\Delta P_{el} = A_p E_p \sum\left[\frac{j\Delta\sigma_c}{E_{cm}}\right],$$

where

$\Delta\sigma_c$ is the variation of stress at the centre of gravity of the cross-section

j $= (n-1)/2n$, where *n* is the number of stressing steps

E_{cm} is the secant modulus of elasticity of concrete

3. Losses at anchorage are due to wedge draw-in of the anchorage devices.

Long term loss of tension

Long term loss of force due to shrinkage and creep of the concrete and the relaxation of the tendon is calculated as

$$\Delta P_{c+s+r} = A_p \Delta\sigma_{c+s+r} = A_p \frac{\epsilon_{cs} E_p + 0.8\Delta\sigma_{pr} + \frac{E_p}{E_{cm}} \varphi \sigma_{c,QP}}{1 + \frac{E_p}{E_{cm}} \frac{A_p}{A_c} (1 + \frac{A_c}{I_c} z_{cp}^2)[1 + 0.8\varphi]},$$

where

$\Delta\sigma_{c+s+1}$ is the tension loss due to the effects above

E_{cm} is the secant modulus of elasticity of concrete

$\Delta\sigma_{pr}$ is the long-term absolute tension loss due to the relaxation of tendons

in case of 2nd relaxation class:

$$\Delta\sigma_{pr} = \sigma_{max} \cdot 0.6\rho_{1000} e^{9.1\mu} \cdot 500^{0.75(1-\mu)} \cdot 10^{-5},$$

in case of 3rd relaxation class:

$$\Delta\sigma_{pr} = \sigma_{max} \cdot 1.98\rho_{1000} e^{8\mu} \cdot 500^{0.75(1-\mu)} \cdot 10^{-5},$$

where $\rho_{1000} = 2.5\%$ is the relaxation loss at a mean temperature of 20°C at 1000 hours after tensioning

φ final value of creep coefficient

$\sigma_{c,QP}$ is the stress in the concrete adjacent to the tendons, due to self-weight and initial prestress and other quasi-permanent actions where relevant.

A_p is the total cross-section area of tendons

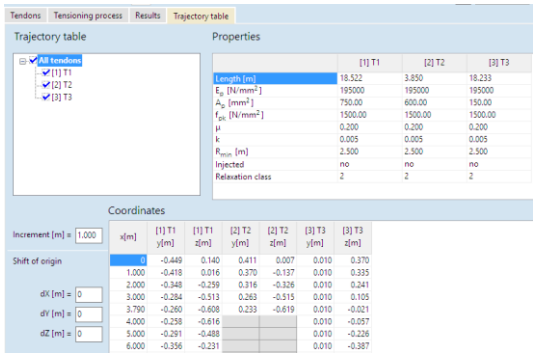
A_c is the cross-section area of the concrete

I_c is the second moment area of the concrete section

z_{cp} is the distance between the center of gravity of the concrete section and the tendons

Trajectory table

The last tab is to build a trajectory table for the selected tendons with the desired increment and optional shift of origin. The trajectory table consists of the local y and z coordinates of the selected tendons at the calculated x positions.
The defined basepoints are always displayed in the trajectory table.



Main toolbar

The main toolbar has two buttons.



Copy diagram

Ctrl+C

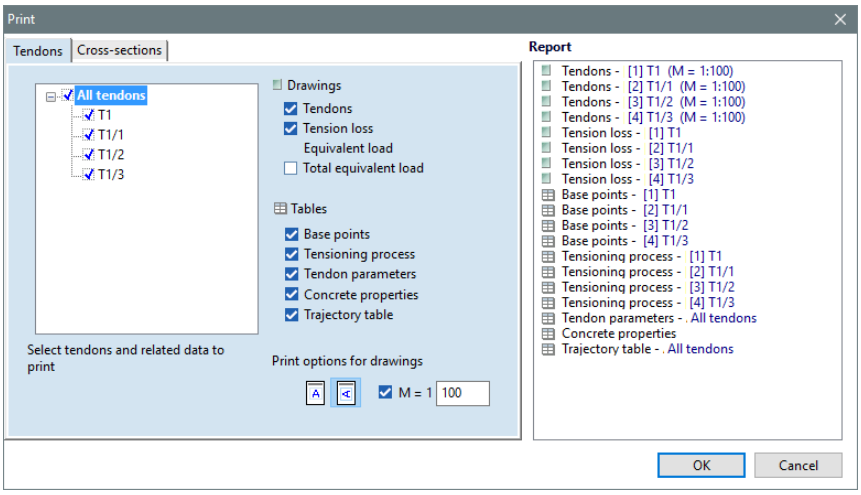
Copies the drawing on the active tab to the Clipboard as a Windows metafile. This way the diagram can be pasted to other applications (e.g., Word).



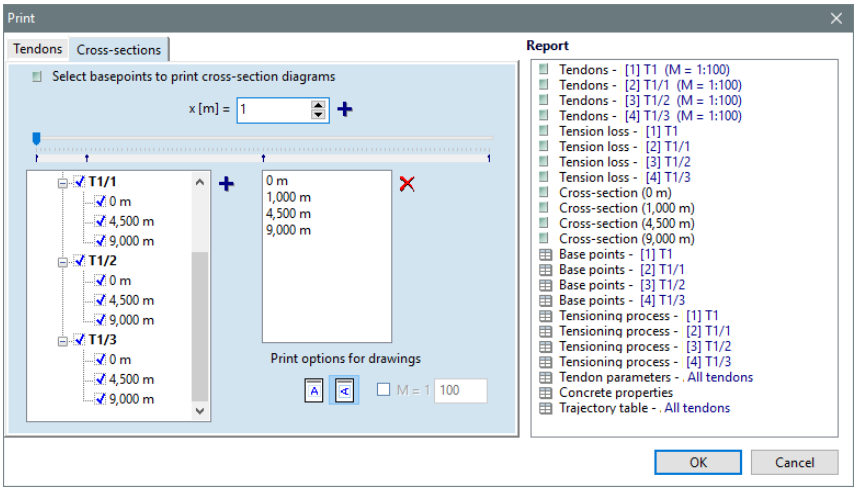
Print

Ctrl+P

Prints a report of the tensioning using diagrams and tables. Tendons and report items can be selected. You can choose the position of the drawing (landscape or portrait) and set the scale of it *(Print options for drawings).*

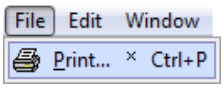


Cross-sections can be selected to print cross-section diagrams.



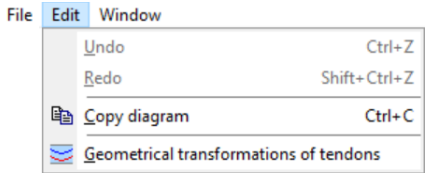
Menu *You can reach the following functions via the menu:*

File



Print **See...** **Main toolbar / Print**

Edit

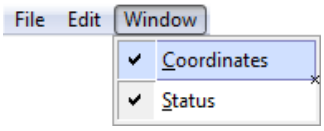


Undo/Redo Undoes the effect of the previous command / Executes the command which was undone.

Copy diagram **See...** **Main toolbar / Copy diagram**

Geometrical transformations of tendons **See...** **Tendons / Geometrical transformations of tendons**

Window



Coordinates Editing of the longitudinal and cross-section diagrams is made easier by a coordinate window. The display of this window can be turned on and off.

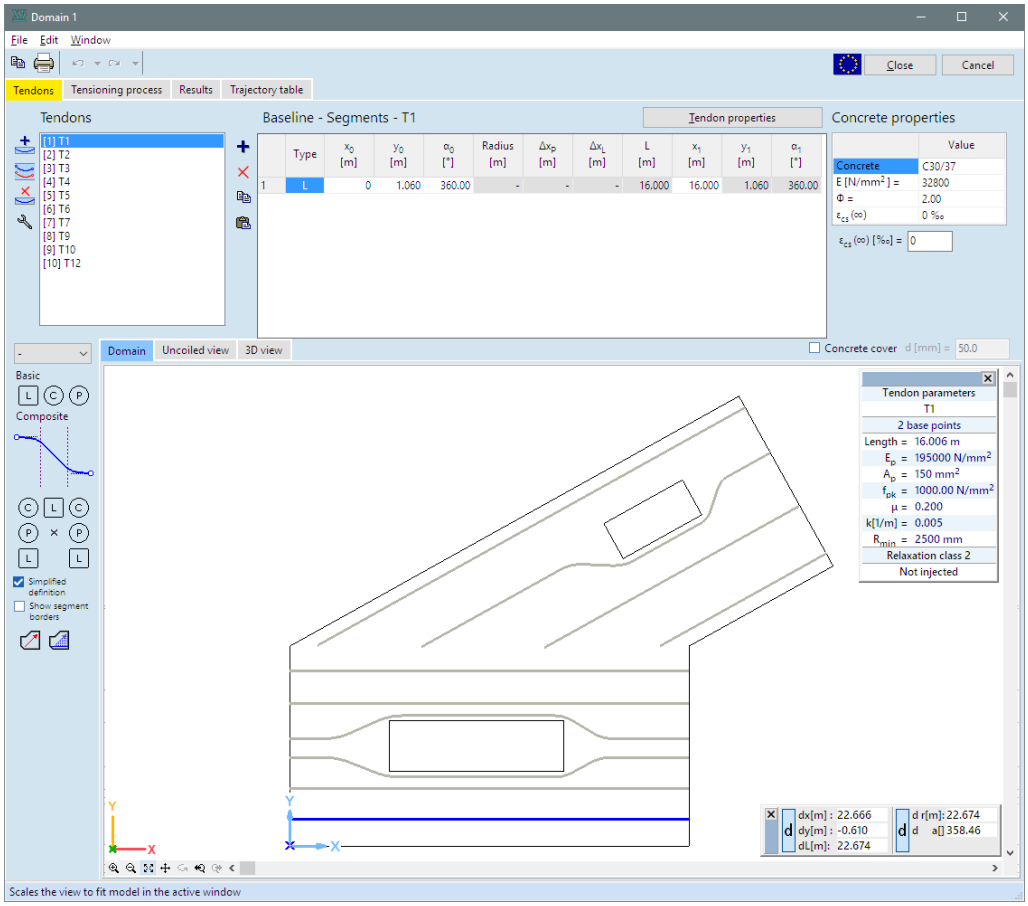
Status On diagrams an information window appears displaying diagram-specific information. Display of this window can be turned on and off.

4.10.29.2 Post-tensioning of domains



On the Loads page by clicking on this icon and selecting domains, tendons can be assigned to the selected elements.

Tendons The first tab is to define tendon parameters and geometry.



Tendon geometry can be displayed in three views.

Domain Tendons are displayed in top view showing geometry of the curve on the middle plane (baseline).

Uncoiled view Tendon elevation perpendicular to the middle plane as a function of x (the arc length position on the baseline).

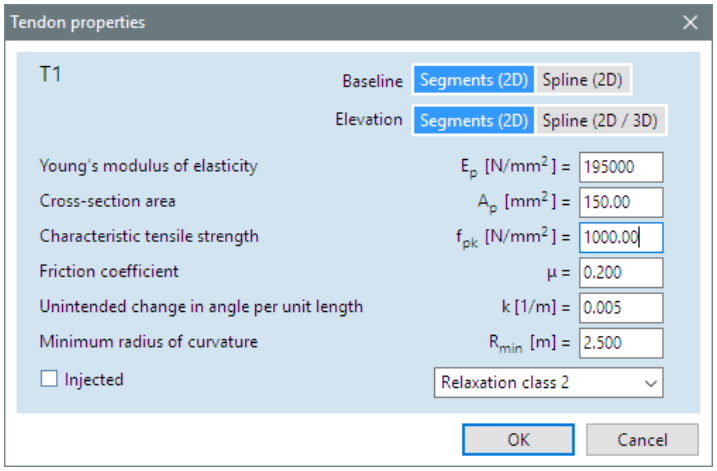
3D view The 3D view shows the cable and the elements in 3D.

Note: The base points of the geometry and the inflection points are shown both in the uncoiled view and 3D view.

Icons on the vertical toolbar beside the tendon list are:



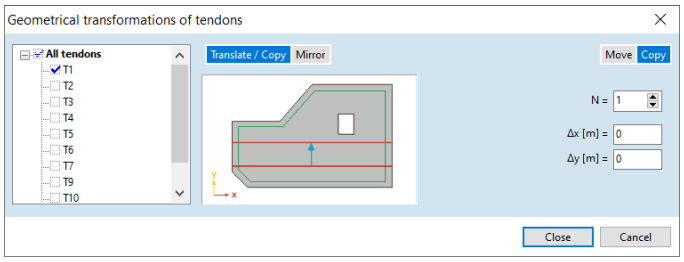
Add new tendon
Tendon properties are the same as described in the previous section.
Definition mode of tendon geometry can also be selected here.



Baseline is the perpendicular projection of the tendon to the middle plane of the domain.
Elevation is the perpendicular deviaton of the tendon from the baseline.
If *Elevation* is defined as a 3D spline the actual perpendicular projection of the tendon may also deviate from the baseline.
To modify the properties above choose the active tendon and click on the *Tendon properties* button.



Geometrical transformations of tendons



Tendons selected in the tree can be translated or mirrored. Tendons can be copied or just moved. The *Δx, Δy* coordinates are defined in the tendon coordinate system (see below). Copied tendons inherit the original parameters and the tensioning process assigned to them.



Delete tendon. Deletes the selected tendon.



Options. Separate grid and cursor settings for the longitudinal and cross-section tendon diagram.

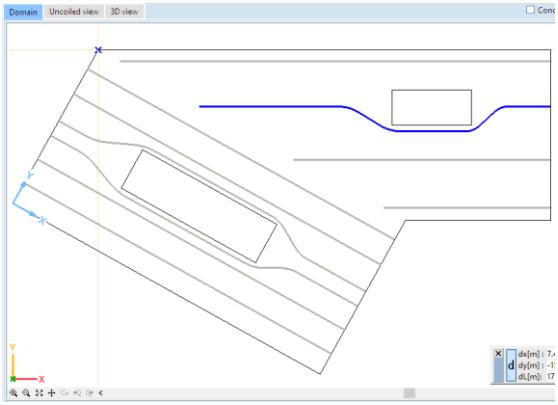
Concrete parameters Under the concrete parameters the material properties of the concrete can be checked. Moreover, $e_{cs}(\infty)$, the long-term value of the concrete shrinkage strain can be modified.

Tendon geometry The first step is to draw the baseline in top view on the *Domain* tab using the segment-based or spline-based geometry tools the same way as in case of post-tensioning of beams. As the baseline must be in the middle plane of the domain spline definition is limited to 2D. Each tendon has its own coordinate system (see below).
The second step is to draw the elevation curve along the baseline on the *Uncoiled view* tab using the definition method selected in the *Tendon properties* dialog.
Checking *Concrete cover* and entering the required value helps tendon definition the same way as in the case of beams.

Tendon coordinate system



Click on this icon to set the origin (first click) and the local X axis (second click) of the tendon coordinate sytem. The drawing of the domain will be translated and rotated accordingly. Draw the baseline the same way as in the case of beams. The Z axis of the tendon coordinate sytem is perpendicular to the plane, while Y is defined according to the right hand rule.
Redefining the local coordinate system of an existing tendon will delete the tendon and it must be redrawn in the new system.



Trajectory coordinate system



Click on this icon to set the origin (first click) and the local X direction (second click) of the trajectory coordinate sytem (one system for all tendons).

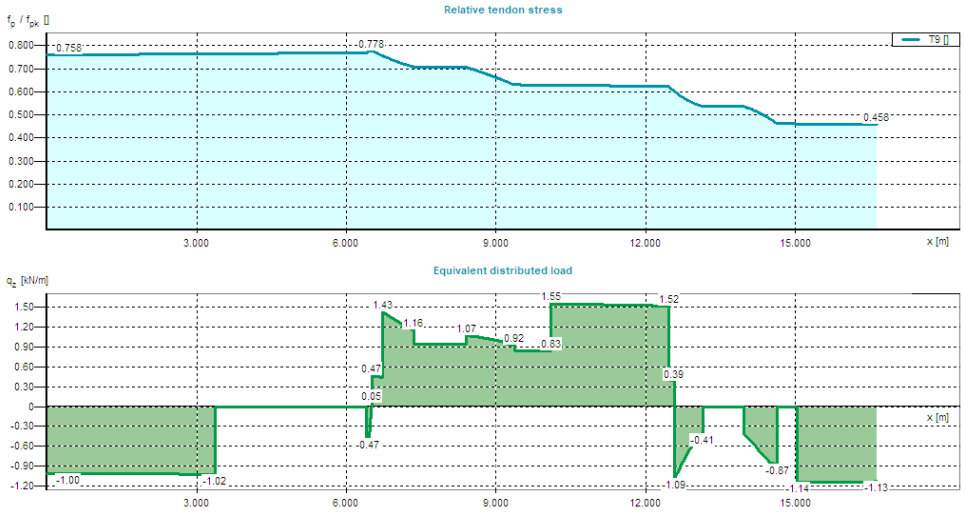
Tensioning process

The second tab is to define the tensioning process for tendons by determining the order of certain operations just like for beam tensioning.

Include tension loss due to sequential stressing — During post-tensioning of domains, the elastic loss due to sequential stressing is neglected.

Results

If all tendons have valid material parameters, geometry and tensioning process, result diagrams are displayed on the *Results* tab. The upper diagram shows the ratio of the actual tension and the characteristic tensile strength. The lower diagram shows the chosen equivalent distributed load component. In case of domains only one tendon can be chosen for post-tensioned surfaces at once.



Component — The component of the equivalent load system can be chosen. Three force components (*q_x, q_y* and *q_z*) and three moment components (m_x, m_y and m_z) are available.

Time — The stresses in the cable and the equivalent load component can be illustrated right after the tensioning (choosing *T=0*) and after long term losses (choosing *T=∞*). The components, relating to *T=∞* can be chosen only after a static analysis.

The calculation of stress loss is done the same way as it was introduced before, for post-tensioned beams. The calculation of stress losses right after tensioning neglects the elastic shrinking of concrete.

Trajectory table

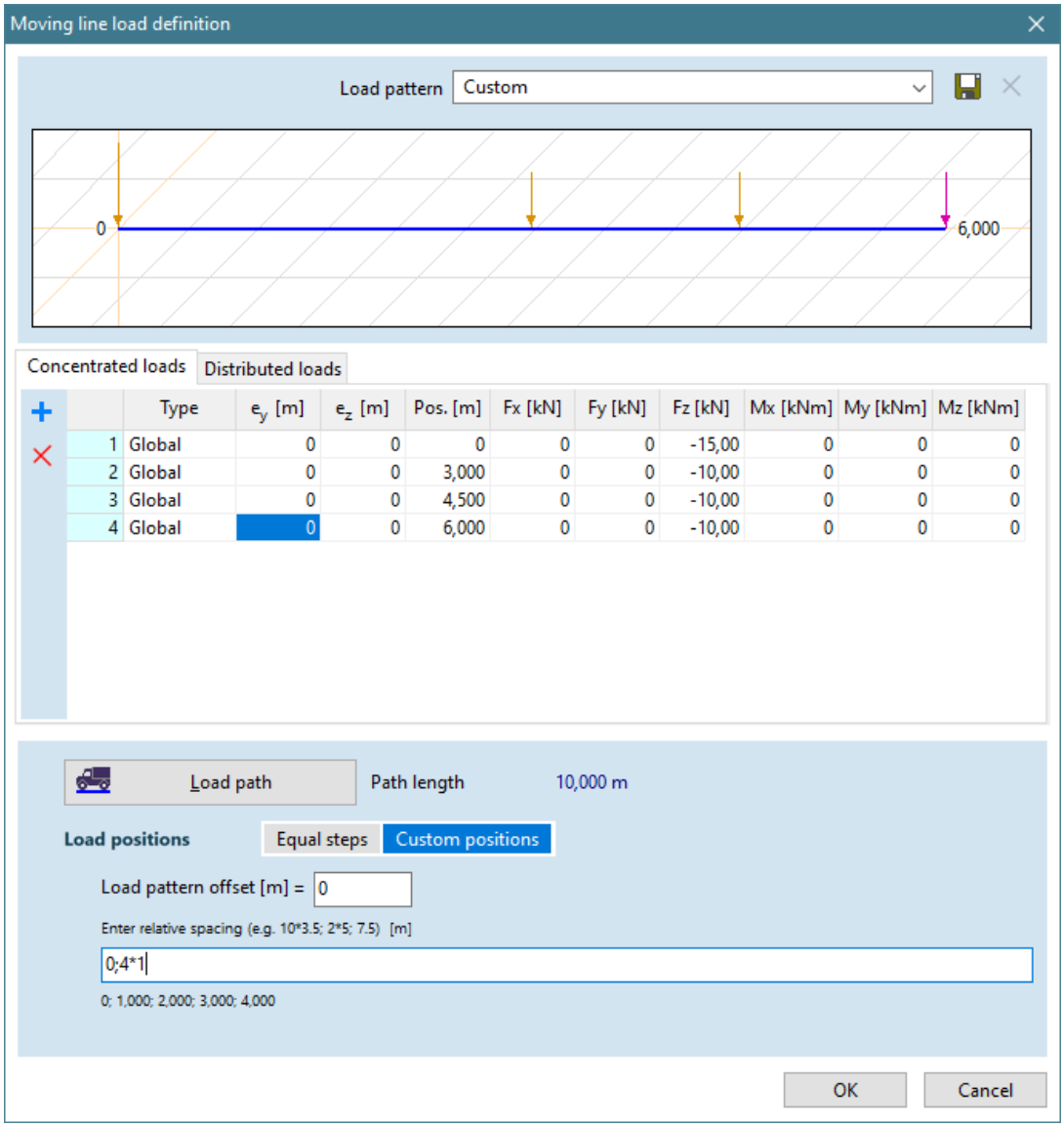
At the last page trajectory table can be created the same way like for beams. The coordinate system for the trajectories can be defined on the *Tendons* page.

For the main menu and reporting functions see **4.10.29.1 Post-tensioning of beam and rib elements**

4.10.30 Moving loads

Moving loads allow modeling of a drifting load with a constant intensity like a vehicle crossing a bridge or a crane carriage moving along its runway.
To define a moving load a moving load case must exist. It can be created on the *Loads* tab clicking the *Load cases and load groups* icon. See **4.10.1 Load cases, load groups.** Moving load icons will be enabled only if the current load case is a moving load case. After defining the load new load cases will be created automatically according to the number of steps specified. Auto-created load cases are maintained and updated automatically and they cannot be deleted or moved into another load group individually. Increasing the number of load steps will create additional load cases. Decreasing this number makes certain load cases useless. These excess load cases will be removed only before saving the model.
Moving load symbols can be displayed in two ways. The first option is to draw the current phase only. The second one is to draw other phases in grey.
Open Table Browser to see the load and the load path in tabular format. These tables can be also used for reporting purposes.

4.10.30.1 Moving loads on line elements



Moving load on line elements is a load pattern moving on a user-defined load path in *N* steps. The load pattern can contain any combination of concentrated and distributed loads.
Individual loads in the pattern can be local or global and their position, eccentricity and intensity components can be set. This way the vertical load of a crane carriage and the horizontal forces can be applied together on the runway. Load eccentricity is always parallel to the local *y* or *z* axis. Load eccentricity is always parallel to the local *y* or *z* axis and its sign is interpreted in the direction of the local system of the finite element.
Loads can be added to the pattern by clicking the plus icon and filling out the fields in the row. Selected rows can be deleted by clicking the *Delete* icon under the plus icon.
Load patterns can be saved under a name and reloaded.

After load pattern definition it is necessary to select the load path. It must be a continuous sequence of beams or ribs. After selecting the elements constituting the load path the start point and end point has to be selected. These points must be nodes along the path.

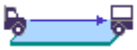
The local *z* direction of the load pattern will always be the local *z* direction of the line elements it is placed on. Lengthening, shortening or breaking a line element of the path will lead to an automatic recalculation of the load phases.

Load positions — Positions of the moving load can be specified in *Equal steps,* or as *Custom positions.*

Equal steps — The value of *N* determines the number of steps the load pattern will make evenly along the path.



Crane runway mode — In the first phase the load with the lowest coordinate in the pattern will be placed over the start point. In the last phase the load with the highest coordinate in the pattern will be placed over the end point.



Bridge mode — In the first phase the load with the highest coordinate in the pattern will be placed over the start point. In the last phase the load with the lowest coordinate in the pattern will be placed over the end point.



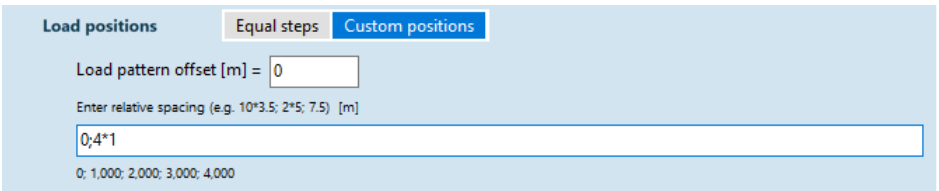
One way: Load moves from start point to end point in *N* steps.



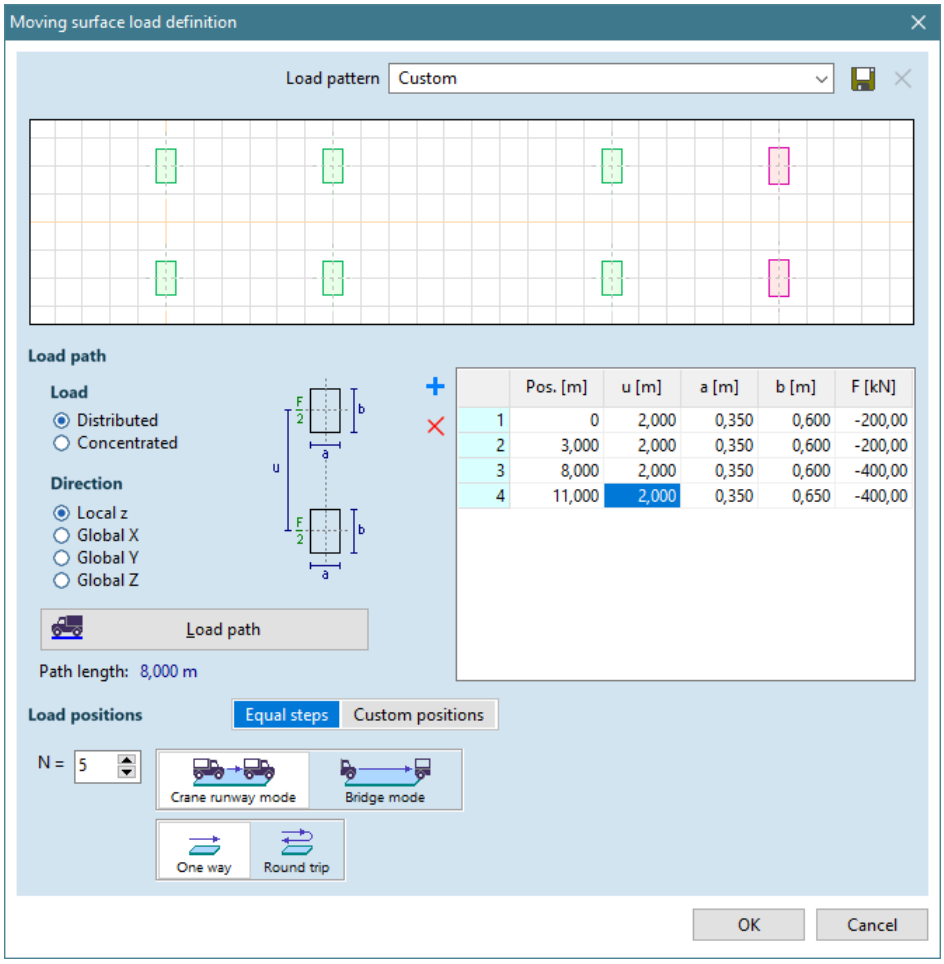
Round trip: Load moves from start point to end point and back in *2N* steps.

Custom positions — The load pattern can be moved to any position along the load path. The length of the steps can be specified like in a structural grid: the program expects numerical values or sequences specified in n*d format (where n is the number of steps, d is the length of a step), separated by semicolons.
If you want to set the load to the starting point of the path, the first value must be 0.

Load pattern offset — The load pattern can have an offset. If it is zero, the leftmost load of the load pattern will be placed over the given positions.



4.10.30.2 Moving loads on domains or load panels



This load type is convenient when vehicle loads have to be defined. The load pattern consists of concentrated or rectangular surface loads pairs representing the wheels on the axles.
u is the vehicle gauge, *a* and *b* refer to the rectangle dimensions. Axle load *F* will be distributed evenly on the two wheels. Load patterns can be saved under a name and reloaded.
The load type and direction switches on the left determines the properties of all loads entered into the table.
Loads can be added to the pattern by clicking the plus icon and filling out the fields in the row. Selected rows can be deleted by clicking the *Delete* icon under the plus icon.
After load pattern definition it is necessary to select the load path. It must be a continuous polyline running through domains or load panels.
The load path does not have to stay in the same plane and can cross holes or empty areas between domains and load panels.
Path start point and end point is the first and last point of the polyline.
Each phase will contain only the loads actually falling on a domain or load panel. The local *z* direction of the load pattern will be the local *z* direction of the domain or load panel it is placed on. In case of a path running along the edge of two or more domains in different planes only the domains in the active parts are taken into account. The local *z* direction will be chosen finding the domain with the minimum angle between local *z* and global *Z* directions.
Beside the load path button, the value of *N* can be set. It determines the number of steps the load pattern will make evenly along the path.
In the first phase the load with the lowest coordinate in the pattern will be placed over the start point. In the last phase the load with the highest coordinate in the pattern will be placed over the end point.
Changing geometry of the domain/load panel will lead to an automatic recalculation of the load phases.

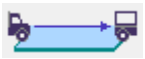
Load positions Positions of the moving load can be specified in *Equal steps,* or as *Custom positions.*

Equal steps The value of *N* determines the number of steps the load pattern will make evenly along the path.



Crane runway mode

In the first phase the load with the lowest coordinate in the pattern will be placed over the start point. In the last phase the load with the highest coordinate in the pattern will be placed over the end point.



Bridge mode

In the first phase the load with the highest coordinate in the pattern will be placed over the start point. In the last phase the load with the lowest coordinate in the pattern will be placed over the end point.



One way: Load moves from start point to end point in *N* steps.



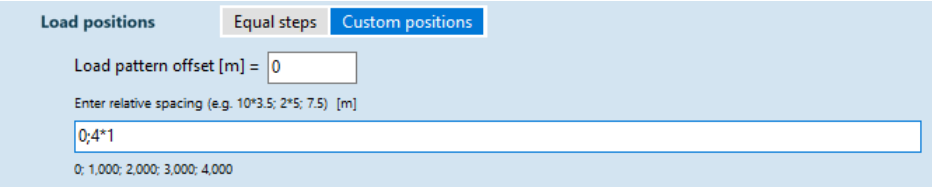
Round trip: Load moves from start point to end point and back in *2N* steps.

Custom positions

The load pattern can be moved to any position along the load path. The length of the steps can be specified like in a structural grid: the program expects numerical values or sequences specified in n*d format (where n is the number of steps, d is the length of a step), separated by semicolons.
If you want to set the load to the starting point of the path, the first value must be 0.

Load pattern offset

The load pattern can have an offset. If it is zero, the leftmost load of the load pattern will be placed over the given positions.

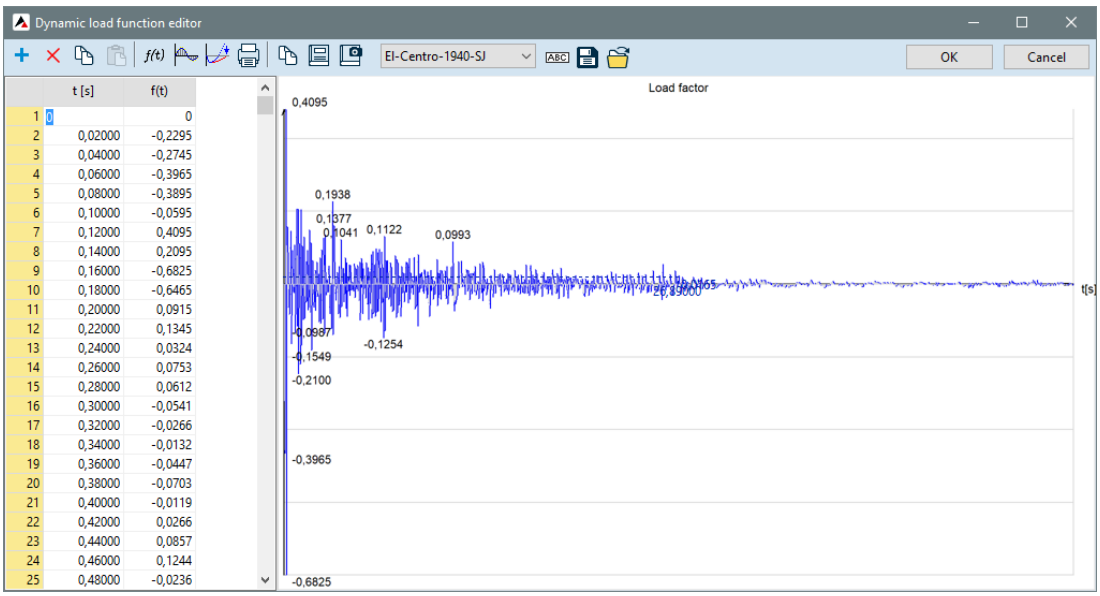


4.10.31 Dynamic loads (for time-history analysis) – DYN module

Dynamic nodal loads and acceleration functions can be defined for time-history analysis.
Acceleration functions can be used for seismic analysis. In this case it is recommended to obtain proper seismic accelerograms and assign these functions to support nodes to analyse the effects of the earthquake. This method provides more exact results than the response spectrum analysis and can be used even if nonlinear elements are defined in the model (nonlinear supports, tension-only trusses, etc.). Its disadvantage is that it cannot be combined with other load types automatically.

☞ **To define nodal loads or acceleration functions the current load case must be a dynamic load case.**
See 4.10.1 Load cases, load groups

Defining functions



Dynamic loads and accelerations are defined by functions which describe the parameter in time. Function editor is available from the dynamic load definition dialogs.

Functions must be entered as value pairs in a table. Functions are plotted automatically and can be printed. Functions can be reused. In order to make them available later, save them into the function library. Saved functions can be reloaded, edited and saved under a new name. Functions are saved into separate **.dfn* files in folder *c:\ Users \ [user name] \ AppData \ Roaming \ AXISVM \ [version number] \ dfn*.

☞ **If the total duration of the dynamic analysis exceeds the length of the function in time the parameter values beyond the end of the function are assumed to be zero.**



Table editing functions

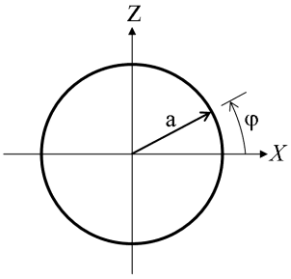
Add a new row to the table.

Delete selected rows from the table.

Copy the selected cells to the Clipboard.

Insert the content of the Clipboard into the table.

Formula editing.
The *f(t)* load function can be entered as a formula. Following operators and functions are available: *+, –, * , /, (,), sin, cos, tan, exp, ln, log10, log2, sinh, cosh, tanh, arcsin, arccos, arctan, arcsinh, arccosh, arctanh, int, round, frac, sqr, sqrt, abs, sgn, random.*
random(t) returns a random number between 0 and 1.
A machine rotating about the Y axis has a dynamic load function with the following X and Z components:

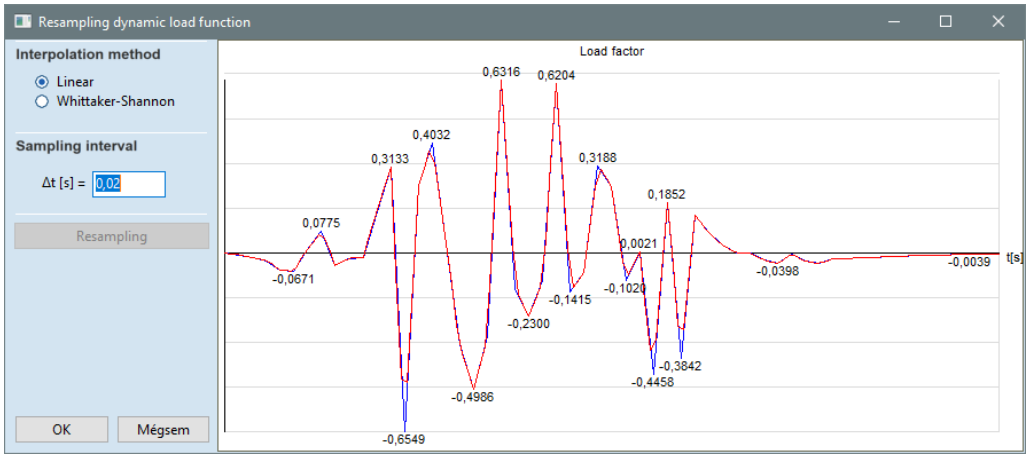


$$f_X(t) = a \cdot \cos(\omega t + \varphi) \text{ and } f_Z(t) = a \cdot \sin(\omega t + \varphi)$$

As functions are represented as a series of values a Δt step and a T_{max} total time must be specified.

Resampling the original function with equidistant sampling.
The values of the new function at sampling instants locating between the original points is obtained by interpolation. The following interpolation methods are available:

Linear
The original function is considered as piecewise linear and the values of the new function are obtained from the linear segments. In the following picture the original function (blue line) and the new resampled function (red line) can be seen together.



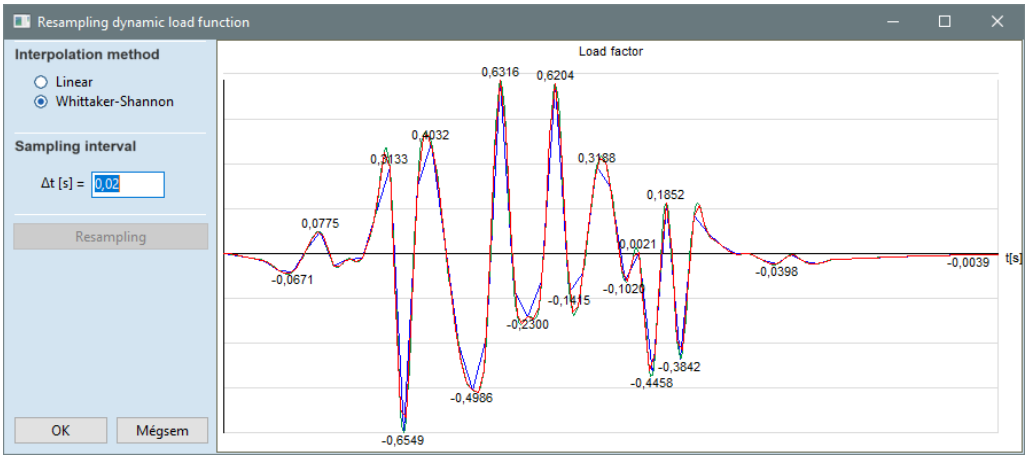
This method is recommended if the original function is really piecewise linear. If the original function is a sampled continuous function (e.g., measurement record or a function generated from Fourier spectral composition), this method cannot approximate the original curved continuous function between the sample points.

Moreover, in the preceding picture several local extreme values of the original function are lost, because the sampling instances of the new function does not coincide with the original ones. In such cases the *Whittaker-Shannon* interpolation method is recommended.

Whittaker-Shannon

The original function is considered as an equidistant sampled continuous function and the values of the new function are obtained from the reconstructed continuous function.

In the following picture the original function (blue line), the reconstructed continuous function (green curve) and the new function (red line) can be seen together. It can also be seen that the originally unsampled extreme values of the continuous function appear in the reconstructed function.



The implemented method is based on the Whittaker-Shannon interpolation formula [37]:

$$f_{new}(t) = \sum_{i=1}^{n} f(t_i) \cdot \mathrm{sinc}\left(\frac{t - t_i}{h}\right) = \sum_{i=1}^{n} f(t_i) \cdot \frac{\sin\left(\pi \frac{t - t_i}{h}\right)}{\pi \frac{t - t_i}{h}}$$

where h is the sampling interval of the original function and $t_i = hi$ is the sampling instance of the i^{th} point of the original function.

If the Nyquist criterion [38] is satisfied (the continuous function was bandlimited and the sample rate of the original record was at least two times of the bandlimit), the original continuous function can be perfectly reconstructed. Although a finite length function cannot be bandlimited the Nyquist criterion can be well approximated in practical cases. Therefore, a good estimation of the original continuous function can be obtained.

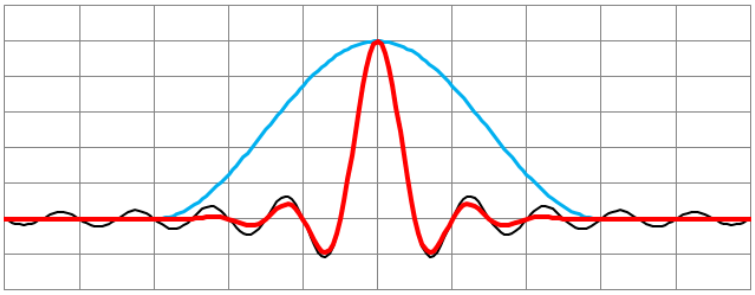
Since the Whittaker-Shannon interpolation function has infinite length the fading out end of the functions may become wavy. To avoid this phenomenon a further cosine shape factor is used in the implemented method:

$$f_{new}(t) = \begin{cases} \sum_{i=1}^{n} f(t_i) \cdot \frac{\sin\left(\pi \frac{t - t_i}{h}\right)}{\pi \frac{t - t_i}{h}} \cdot W_i(t) & if \quad |t - t_i| \le 0.5T_w \\ 0 & if \quad |t - t_i| > 0.5T_w \end{cases}$$

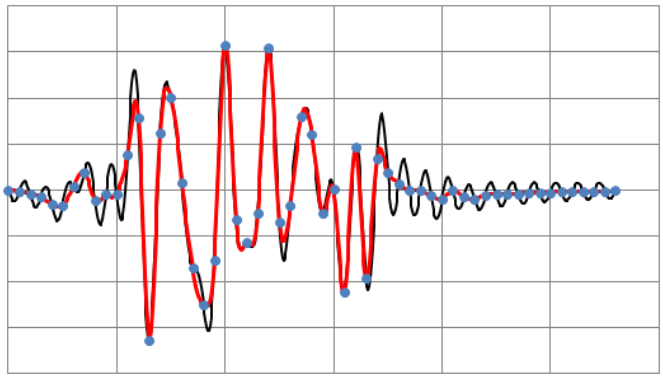
$$W_i(t) = \begin{cases} \frac{1 + \cos\left(\pi \frac{t - t_i}{0.5T_w}\right)}{2} & if \quad |t - t_i| \le 0.5T_w \\ 0 & if \quad |t - t_i| > 0.5T_w \end{cases}$$

where T_w is the length of the cosine wavelet. It is set to 10% of the length of the record.

The following picture shows the original Whittaker-Shannon interpolation function (black curve), the cosine shape factor (blue curve) and the resultant finite length interpolation function (red curve).

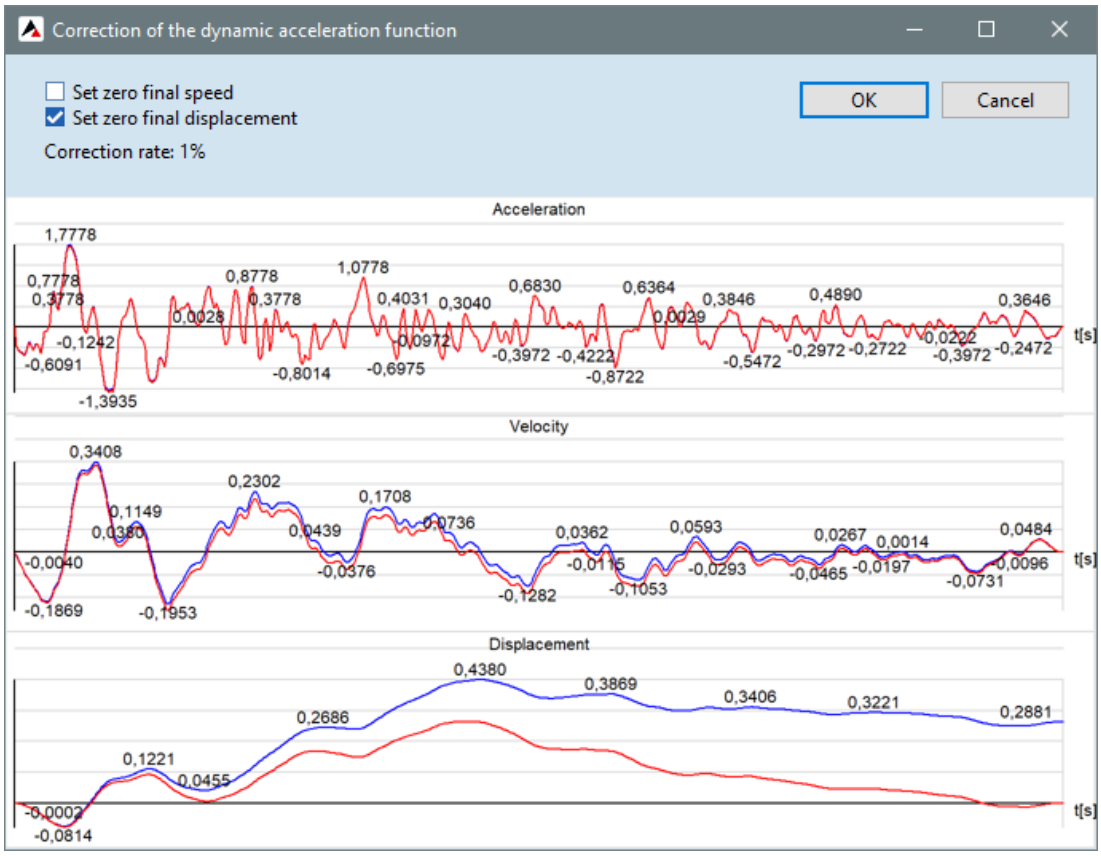


In the following picture the wavy ends obtained with the original Whittaker-Shannon interpolation formula (black curve) and smooth end with the wavelet interpolation (red curve) can be seen together. The blue circles show the sample points of the original function. It can be seen that in the middle of the large amplitude part of record the reconstructed continuous functions obtained with the different interpolation functions almost coincide. But at the ends only the red curve is smoothed.



☞ **The Whittaker-Shannon interpolation method is available only for functions obtained from equidistant sampling.**

Correcting the acceleration record for producing zero final velocity and/or zero final displacement



Measured earthquake acceleration records usually do not produce exactly zero velocity and zero displacement at the end of the record, because of measurement inaccuracies. The nonzero final velocity appears as an infinite motion of the supports with constant velocity beyond the end of the record. The nonzero final displacement causes that the supports stop out of their original position at the end of the record. Any of these inaccuracies can be eliminated by applying a small modification to the acceleration record. The type of correction can be selected by the checkboxes. The blue graph shows the original record and the red graph shows the corrected one. The *correction rate* is defined by the ratio of the RMS values of the modification and of the original record.

$$Correction\ rate = \frac{RMS\big(a_{corr}(t) - a(t)\big)}{RMS\big(a(t)\big)} = \frac{\sqrt{\frac{1}{T}\int_0^T \big(a_{corr}(t) - a(t)\big)^2 dt}}{\sqrt{\frac{1}{T}\int_0^T a^2(t)dt}}$$

where $a(t)$ is the original record and $a_{corr}(t)$ is the corrected value.
The implemented correction method is similar to that proposed by Wilson [25, Appendix J.4].

The velocity and the displacement at the end of the record are

$$V = \int_0^T a(t)dt \qquad \mathrm{U} = \int_0^T (T - t)a(t)dt$$

The corrected record is computed by

$$a_{corr}(t) = \begin{cases} \big(1 + c_1^+ h_1(t) + c_2^+ h_2(t)\big)a(t) & if \quad a(t) \geq 0 \\ \big(1 + c_1^- h_1(t) + c_2^- h_2(t)\big)a(t) & if \quad a(t) < 0 \end{cases}$$

where $h_1(t) = \frac{T-t}{T}$ and $h_2(t) = \frac{t}{T}$ are weighting functions and c_1^+, c_1^-, c_2^+, c_2^- are the correction factors.

The weighting function $h_1(t)$ focuses the correction due to the nonzero final displacement to the beginning of the record, while $h_2(t)$ focuses the correction due to the nonzero final velocity to the end of the record. The four correction factors are determined from the following four criteria. The final velocity and displacement after the correction should be zero or should be equal to their original values depending on the state of the checkboxes. The amount of the corrections applied to the positive and to the negative part of the record should be equal.

Although the preceding formulas are written as continuous functions of time the calculations are performed on the discrete time series of the accelerogram.

Diagram and report functions



Print diagram and table.



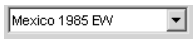
Copy the diagram and the table to the Clipboard.



Start the Report Maker.



Save the diagram into the Gallery. See **2.10.5 Gallery**



A function previously saved to the library can be loaded by selecting its name from the dropdown list.



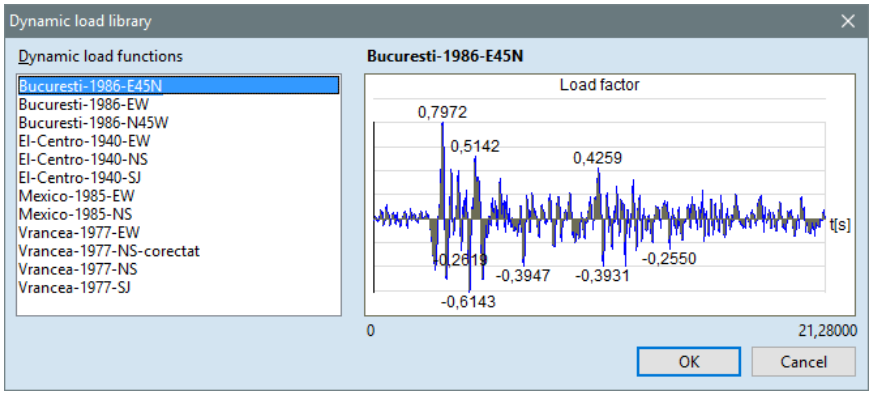
Rename the current function.



Save the current function to the library.



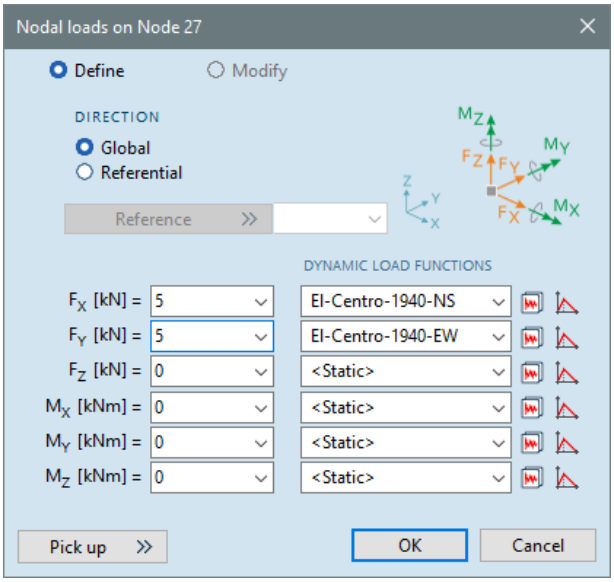
Load a function from the library.



☞ **The first point of functions must be at t=0. This value pair cannot be changed or deleted. If the load is applied only at T > 0, the function value must be zero between 0 and T.**

4.10.31.1 Dynamic nodal load

To define dynamic nodal loads, select nodes and set the parameters in a dialog. For each component you can assign an intensity and a dynamic load function describing the time-dependence of the load factor.
To use an existing function from the library, click the first icon beside the combo. To edit the load function, click the second icon.
The load directions can be the global X, Y and Z directions or the direction can be determined by a chosen reference. In this latter case there is just one force and moment component.



It is possible to define a constant (time-independent) load by selecting <*Static*> from the *Dynamic load functions* combo.

☞ **The actual value of a load component in t will be calculated as $F_i(t) = F_i \cdot f(t)$, i.e., the load intensity is multiplied by a time-dependent load factor.**

If a dynamic load is defined for a support with an existing dynamic load the existing load will be overwritten.

Modify, delete

Dynamic loads can be modified or deleted the same way as static loads.

Dynamic loads are displayed as dashed yellow arrows.

4.10.31.2 Dynamic concentrated load on domain or load panel

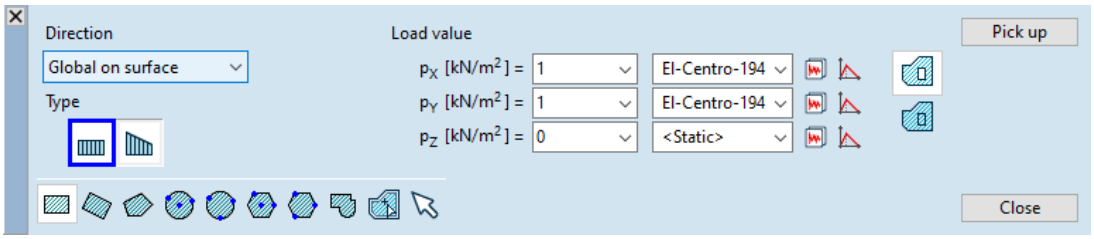


Load position and direction can be set the same way as described in **4.10.5 Point load on domain or load panel**.
A load intensity and a time dependent function can be specified for each component.
Functions can be created and edited in the Function Editor or loaded from the function library.

☞ **The actual value of the load at a given time t will be the load intensity multiplied by the time dependent function:** $F_i(t) = F_i \cdot f_i(t)$

Dynamic force is displayed as dashed yellow arrow.

4.10.31.3 Mesh-independent dynamic area load on domains or load panels



The load polygon and load direction can be defined as described in **4.10.10 Domain / Load panel area load**.
A load intensity and a time dependent function can be specified for each component.
Functions can be created and edited in the Function Editor or loaded from the function library.

☞ **The actual value of the load at a given time t will be the load intensity multiplied by the time dependent function:** $F_i(t) = F_i \cdot f_i(t)$

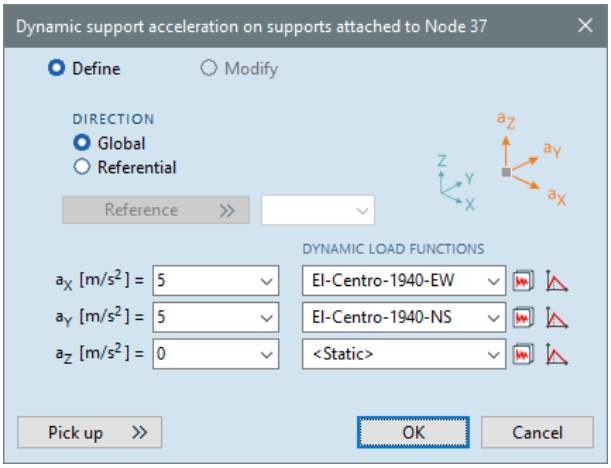
Dynamic force is displayed as a green area.

4.10.31.4 Dynamic support acceleration

Acceleration function can be assigned to any nodal support in the model. For each component you can assign an acceleration intensity and a dynamic load function describing the time-dependence of the load factor.
The actual value of the acceleration at t will be calculated as

$$a_i(t) = a_i \cdot f_i(t)$$

i.e., the acceleration is multiplied by a time-dependent load factor.



☞ **Acceleration acts at the bottom of the support string. The acceleration of the supported node can be different depending on the support stiffness.**

If acceleration is defined for a support with an existing acceleration load the existing load will be overwritten.

If multiple nodal supports are attached to a node, acceleration acts on all supports.

Modify, delete

Dynamic support acceleration can be modified or deleted the same way as a static load.

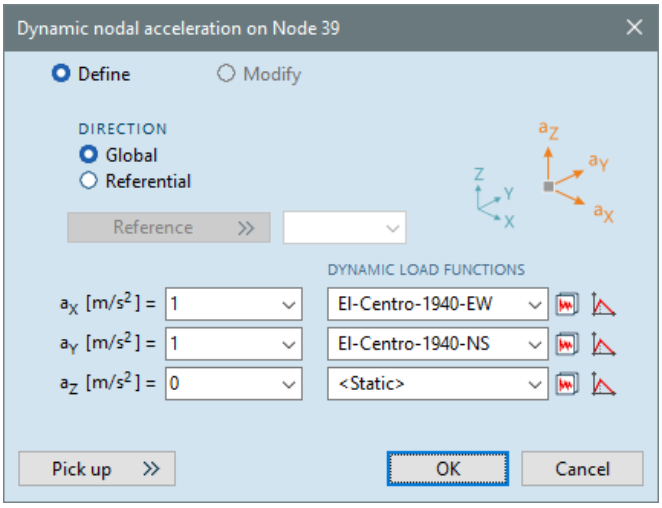
Dynamic support acceleration is displayed as a circle and a yellow arrow.

4.10.31.5 Dynamic nodal acceleration

Nodal acceleration can be assigned to any node in the model. For each component you can assign an acceleration intensity and a dynamic load function describing the time-dependence of the load factor.
The actual value of the acceleration in *t* will be calculated as

$$a_i(t) = a_i \cdot f_i(t)$$

i.e., the acceleration is multiplied by a time-dependent load factor.



☞ **If acceleration is defined for a support with an existing acceleration load the existing load is overwritten.**

To specify ground acceleration for seismic analysis nodal support accelerations must be defined.

Modify, delete

Dynamic nodal acceleration can be modified or deleted the same way as a static load.

Dynamic nodal acceleration is displayed as a circle and a yellow arrow.

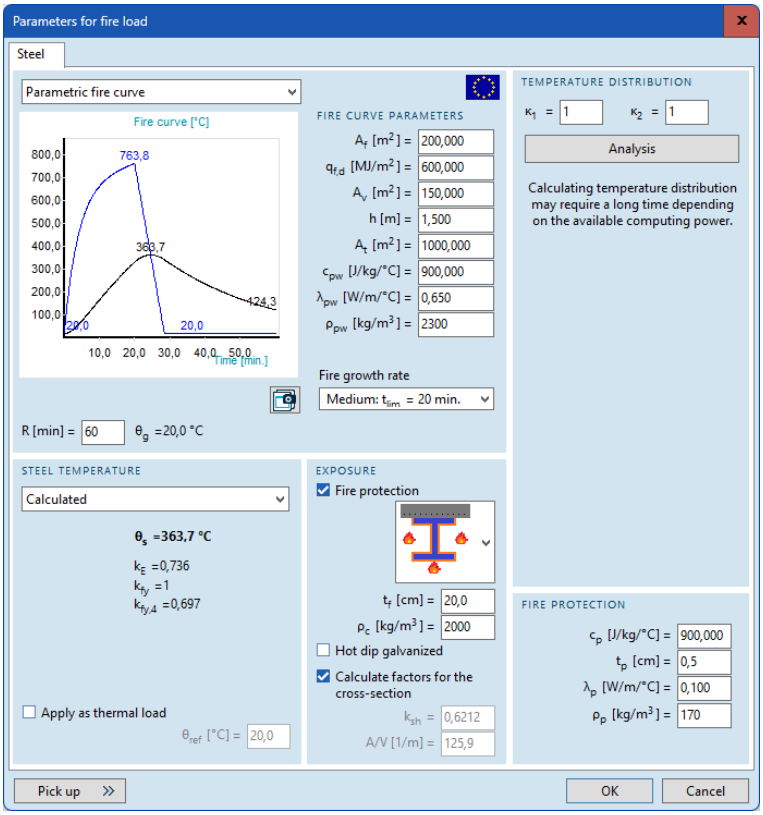
4.10.32 Fire effect on steel line elements – SD8 module

Design codes

In AXISVM, fire effects can be generated according to different national codes. The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

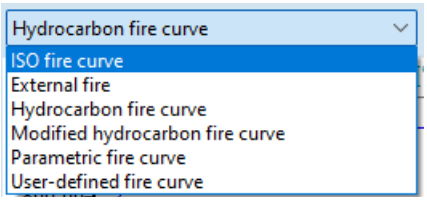
Definition of fire effect

In order to define fire effect on line elements, click on *Define fire effect on line elements* icon on the *Loads* toolbar



Choosing a fire curve

Fire effect is represented by a fire curve specifying the gas temperature in the fire compartment as a function of the time.



Standard temperature-time curve (so called ISO fire curve) (EN 1991-1-2):

$$\theta_g = 20 + 345\log_{10}(8t + 1)$$

External fire curve (EN 1991-1-2):

$$\theta_g = 660(1 - 0.687e^{-0.32t} - 0.313e^{-3.8t}) + 20$$

Hydrocarbon fire curve (EN 1991-1-2):

$$\theta_g = 1080(1 - 0.325e^{-0.167t} - 0.675e^{-2.5t}) + 20$$

Tunnel fire curve/modified hydrocarbon fire curve:

$$\theta_g = 1280(1 - 0.325e^{-0.167t} - 0.675e^{-2.5t}) + 20$$

Parametric fire curve:

$$\theta_g = 20 + 1325\left(1 - 0.324e^{-0.2t^*} - 0.204e^{-1.7t^*} - 0.472e^{-19t^*}\right) \quad t^* \le t^*_{max}$$

Cooling phase:

$$\theta_g = \theta_{max} - 625(t^* - t^*_{max} \cdot x) \quad t^*_{max} \le 0.5$$
$$\theta_g = \theta_{max} - 250(3 - t^*_{max})(t^* - t^*_{max} \cdot x) \quad 0.5 \le t^*_{max} < 2$$
$$\theta_g = \theta_{max} - 250(t^* - t^*_{max} \cdot x) \quad t^*_{max} \ge 2$$

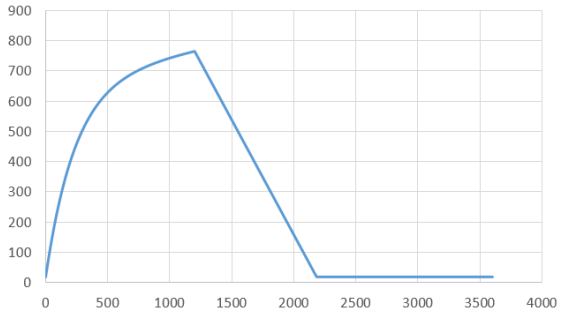
where

θ_g [°C]	gas temperature
t [min]	time
t^* [h]	modified time parameter

Parametric fire curve

Unlike the ISO-, external and hydrocarbon fire curves, parametric fire curves have a cooling phase. Using a parametric fire curve, parameters like the size and ventilation of the fire compartment, amount of the combustible materials, etc., can be considered in the calculation. These parameters considerably influence the maximum gas temperature and the duration of the fire.

Temperature [°C]



Time [s]

According to EN 1991-1-2, parametric fire curves are valid for fire compartments up to 500 m^2 of floor area, without openings in the roof and for a maximum compartment height of 4 m.

The following parameters need to be defined for the calculation of the fire curve:

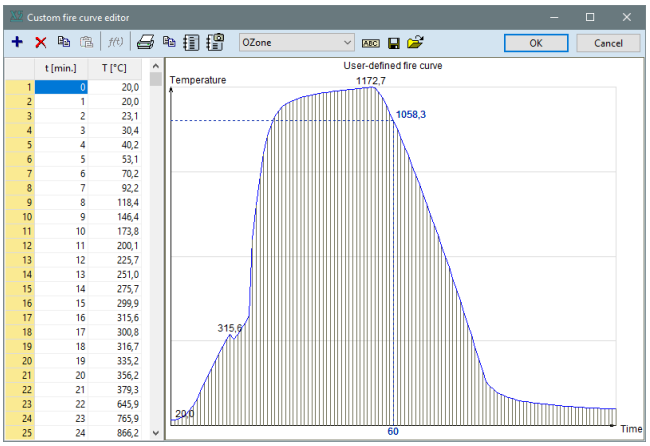
A_f [m^2]	floor area
$q_{f,d}$ [MJ/ m^2]	design value of the fire load density related to the surface area A_f of the floor
A_v [m^2]	total area of vertical openings on all walls
h [m]	weighted average of window heights on all walls
A_t [m^2]	total area of enclosure (walls, ceiling and floor, including openings)
c_{pw} [J/kg°C]	specific heat of boundary of enclosure
ρ_{pw} [kg/m^3]	density of boundary of enclosure
λ_{pw} [W/m°C]	thermal conductivity of boundary of enclosure

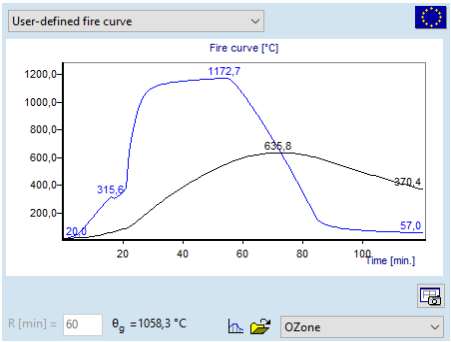
☞ **If the boundary of enclosure consists of different layers, c_{pw}, ρ_{pw} and λ_{pw} parameters should be given resulting parameter $b = \sqrt{c_{pw}\rho_{pw}\lambda_{pw}}$ equal to $b = \sqrt{c_{pw,1}\rho_{pw,1}\lambda_{pw,1}}\frac{A_1}{\sum A} + \cdots + \sqrt{c_{pw,n}\rho_{pw,n}\lambda_{pw,n}}\frac{A_n}{\sum A}$.**

☞ **In case of NEN EN standard, parametric fire curve cannot be selected, but another prescriptive fire curve, namely RWS tunnel fire curve is available.**

User-defined fire curve

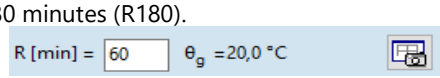
User-defined fire curve is also available to define the fire effect, thus the program allows to perform fire design considering simulation or fire test results.
It is important to note that the calculation of steel temperature based on prescriptive fire curves uses the assumption that the gas temperature is identical everywhere in the fire compartment and the steel element is exposed identical heat radiation from every direction. In case of real fires, typically, the gas temperature is lower compared to e.g., ISO standard fire curve. Accordingly, if we calculate the steel temperature with closed formula of EN 1993-1-2 standard from gas temperatures recorded in a real fire like it would be gas temperatures from ISO fire curve, we can underestimate the steel temperature because this calculation underestimates the proportion of radiated heat. The solution may be the use of the so-called adiabatic surface temperature [U. Wickström, D. Duthinh, K. McGrattan; 2007: Adiabatic surface temperature for calculating heat transfer to fire exposed structures]) (adiabatic surface temperature may be 15 - 20% higher than gas temperature according to the following study [M. Malendowski, A. Glema, W. Szymkuc; 2015: Performance based coupled CFD-FEM analysis of 3-bay high industrial hall under natural fire in ASFE 2015 proceedings]).
Similarly, to custom design seismic spectrum, the points of user-defined fire curve can be defined in function editor. The maximum number of points is 360.





Steel temperature

The steel temperature is calculated automatically from the selected fire curve and fire duration / required time of fire resistance (R15, R30, R60, etc. – where the number gives the time in minutes). This design temperature is used then in the course of steel fire design (see **6.6.3 Steel beam fire design according to Eurocode 3 – SD8 module**).
The maximum fire duration is 180 minutes (R180).



The unit of influencing parameters, fire duration and temperatures can be set manually (see **3.3.8 Units and Formats).**
The diagrams can be saved in the Similarly, Drawings Library (see **3.6.11 Save to Drawings Library**) by clicking on the icon situated below the diagram in the right. The saved diagrams may be added later to reports.

The temperature of unprotected steel elements (EN 1993-1-2; Δt = 5s):

$$\Delta\theta_{a,t} = k_{sh}\frac{A/V}{c_a\rho_a}\dot{h}_{net,d}\Delta t$$

where

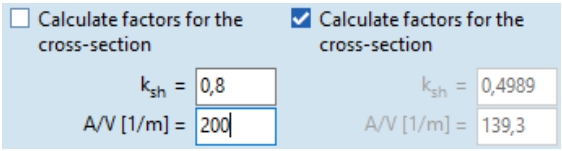
k_{sh} [-]	correction factor for the shadow effect (in case of SIA standard k_{sh} = 1.0)
A/V [1/m]	section factor for unprotected steel members
c_a [J/kgK]	specific heat of steel (according to EN 1993-1-2)
ρ_a [kg/m^3]	unit mass of steel
$\dot{h}_{net,d}$ [W/m^2]	design value of the net heat flux per unit area

Hot dip galvanized

EN 1993-1-2:2024 standard (2nd generation Eurocode) allows the use of lower emissivity factor in the case of galvanized steel elements.

Section factor and shadow effect

Correction factor for the shadow effect and section factor can be calculated automatically for the supported section types (I shape, box shape, pipe, angles, cold formed Z and C sections).



The temperature of protected steel elements is (EN 1993-1-2; Δt = 5s):

$$\Delta\theta_{a,t} = \frac{\lambda_p A_p/V}{d_p c_a \rho_a} \frac{(\theta_{g,t} - \theta_{a,t})}{1+\phi/3} \Delta t - (e^{\phi/10} - 1)\Delta\theta_{g,t}$$

$$\phi = \frac{c_p \rho_p}{c_a \rho_a} \lambda_p A_p/V$$

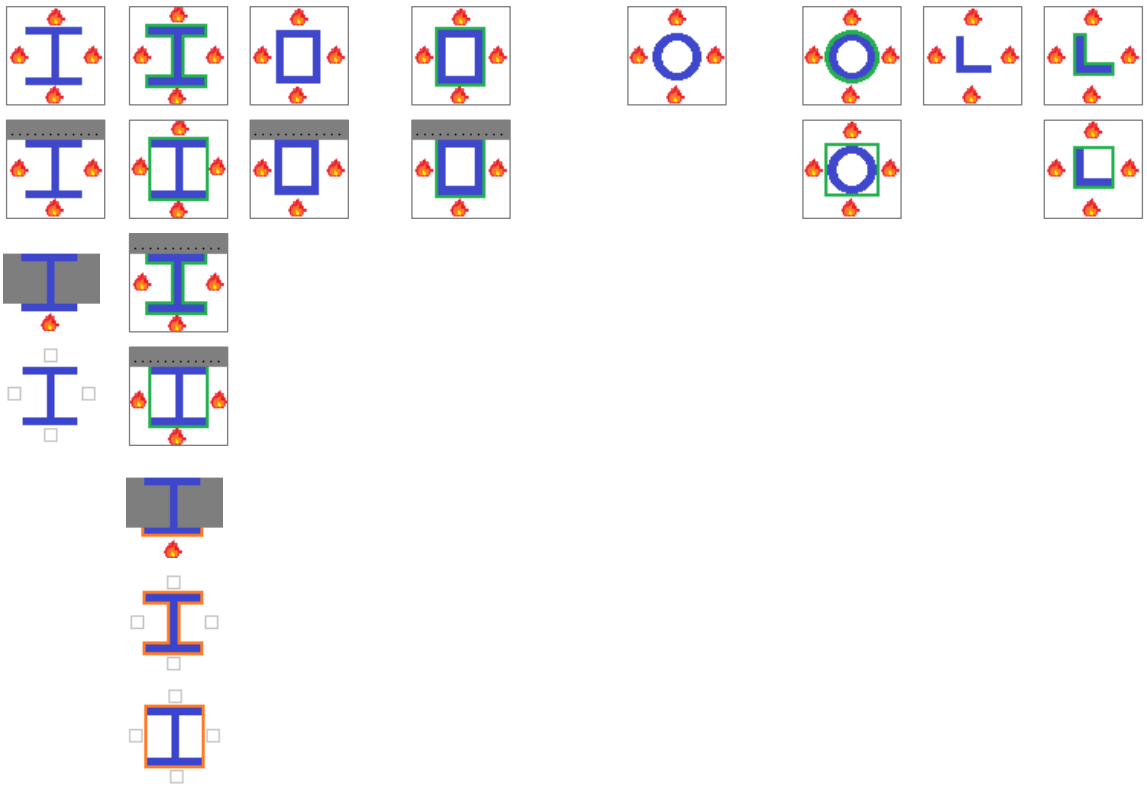
where

λ_p [W/mK]	thermal conductivity of the fire protection
A_p/V [1/m]	section factor for steel members insulated by fire protection
c_p [J/kgK]	temperature independent specific heat of the fire protection material
ρ_p [kg/m^3]	mass density of the fire protection material
d_p [m]	thickness of the fire protection material

☞ **Calculated steel temperature related to the selected fire curve and fire duration (in case of parametric fire curve the maximum temperature calculated in the interval) can be seen on the Steel temperature panel under the combo box. Under the steel temperature reduction factors related to yield strength and Young's modulus according to EN 1993-1-2 are shown.**

Exposition

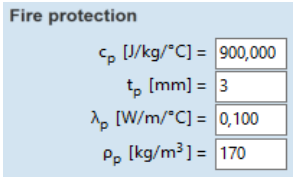
On the *Exposition* panel it is possible to set how the section is exposed to fire. Exposure type considerably influences the value of shadow correction factor and section factor. In case of supported section types, the following exposure types are available for protected / unprotected members:



For exposure types with empty boxes the exposed sides can be set by checking the respective boxes. It can be useful for columns placed against a wall or in a corner.
For unsupported sections the exposure type appears via user specified shadow correction factor and section factor (k_{sh} and *A/V*).

Fire protection

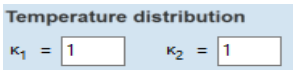
For steel structures the application of fire protection is often required due to the slender elements and the fact that steel has a high thermal conductivity. Check *Fire protection* on the *Exposition* panel to take into account a fire protection. In this case, the Fire protection panel in the bottom right is enabled. Parameters and their importance has been already shown above.



Presence or lack of fire protection has an effect on the exposition types as well.

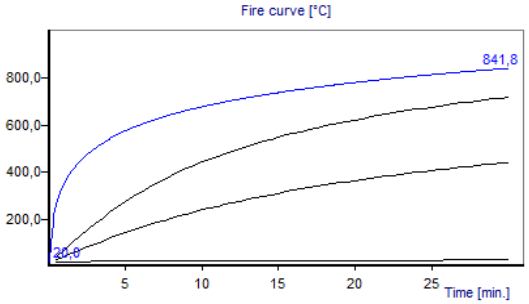
κ1 and κ2 factors

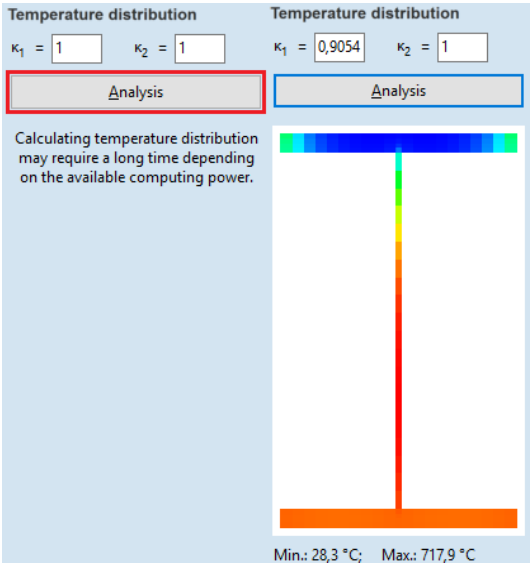
κ_1 and κ_2 are adaptation factors related to a non-uniform temperature distribution across the cross-section and along the beam, respectively, according to EN 1993-1-2. These factors can be set on the *Temperature distribution* panel.



Temperature distribution analysis

For I shapes and box sections, a more accurate temperature distribution may be calculated by clicking on the *Analysis* button on the *Temperature distribution* panel. Temperatures within the cross-section are calculated by finite difference method. In this case a two-dimensional heat conduction problem is solved.





Depending on the available computational resources and the symmetry of the problem, the calculation may take a few seconds or a few minutes. The analysis considers the temperature-dependent thermal conductivity of the steel material according to EN 1993-1-2. The diagram shows the maximum, minimum and average temperatures as a function of time.

According to the regulations of EN 1993-1-2, in case of non-uniform temperature distribution the maximum temperature within the cross-section has to be used when verifying the load bearing capacity. So if temperature distribution results are available, AXISVM uses maximum temperature in further calculations.

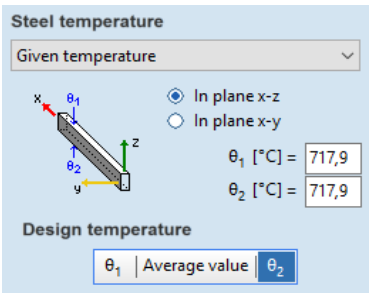
κ_1 adaptation factor related to non-uniform temperature distribution across the cross-section is automatically calculated, however it is possible to override the calculated value above Analysis button.

If the section is connected to a reinforced concrete slab on one side, temperature distribution analysis requires the values of slab thickness and concrete density.

☞ **The thermal conductivity can be set among the design parameters** (3.3.7 Design codes).

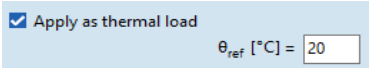
Given temperature

If *Given temperature* is selected on the *Steel temperature* panel, a uniform or linear temperature distribution can be specified. Design temperature may be selected as θ_1, θ_2 or their average. This design temperature is used later in the course of steel fire design (see 6.6.3 Steel beam fire design according to Eurocode 3 – SD8 module).



Apply as thermal load

Both calculated and given temperatures can be applied on the structural element as thermal load (see 4.10.21 Thermal load on

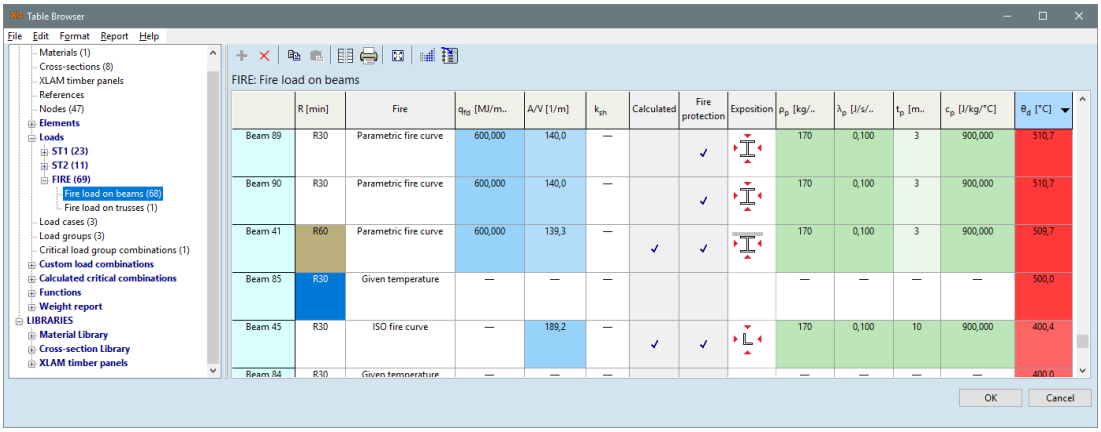


line elements) to determine internal forces due to restrained thermal deformations by a linear static analysis.
A reference temperature must be specified (in fire design it could be set to room temperature).

☞ **Using perfect model and linear static analysis significant internal forces may arise from thermal expansion since possible local stability failures are not considered. These significant forces may be released by local failures and deformations without considerably risk of global failure. According to the Eurocode, thermal forces may be neglected in certain cases.**

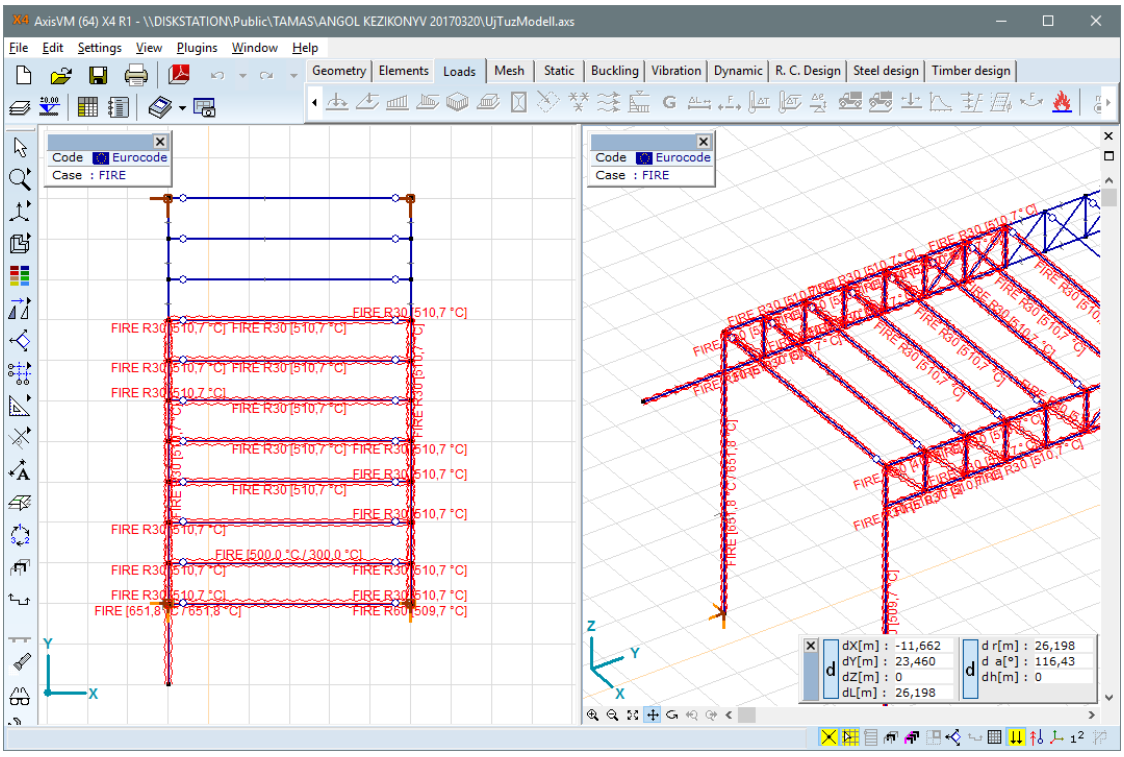
Table

Fire effect parameters for each fire load case can be checked in the Table Browser. The listed parameters are: required time of fire resistance, type of fire curve, fire load density, section factor, shadow factor, exposure type, properties of fire protection and design steel temperature. The tables can be added to reports.



Graphical symbols in the main window

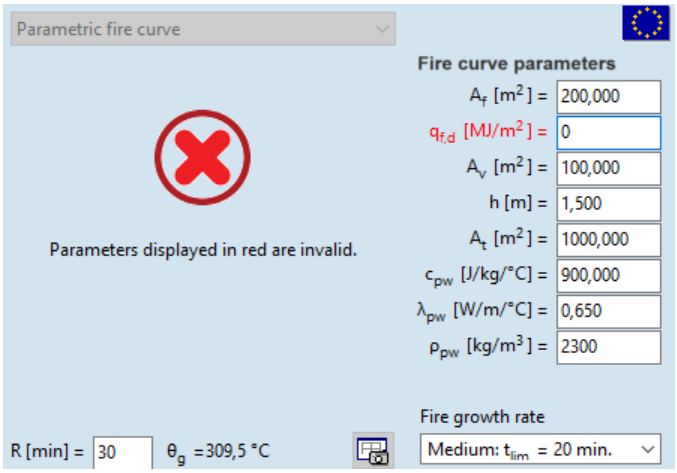
Fire effects are shown in the main window as red zigzag lines. Symbol properties (thickness, colour) can be set in *Preferences / Graphical symbols* window.



Errors

If gas or steel temperature cannot be calculated due to inappropriate parameter values an error message appears.

EN 1991-1-2 impose several restrictions regarding to initial parameters of parametric fire curves (opening factor, $b = \sqrt{c_{pw}\rho_{pw}\lambda_{pw}}$, floor area, height, etc.).



4.10.33 Fire effect on timber line elements – TD8 module

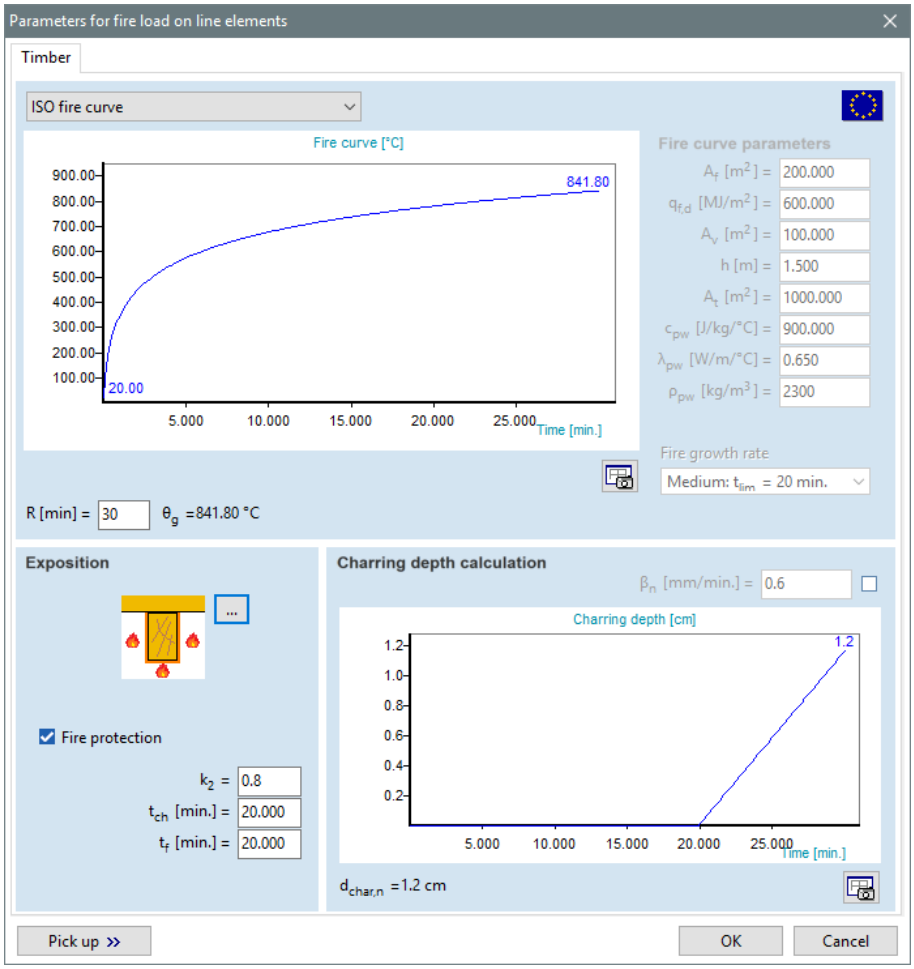
Design codes

In AXISVM, fire effects can be generated according to different national codes. The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.



Definition of fire effect

In order to define fire effect on line elements, click on *Define fire effect on line elements* icon on the *Loads* toolbar.



Fire curves

Fire effect is represented by fire curve specifying the gas temperature in the fire compartment as a function of the time. In case of timber elements, ISO standard and parametric fire curves can be selected. These fire curves are presented and discussed in the previous section (see **4.10.32 Fire effect on steel line elements – SD8 module**).

☞ **In case of NEN EN standard, parametric fire curve cannot be selected.**

Calculation of charring depth

The charring depth is calculated automatically from the selected fire curve and fire duration / required time of fire resistance (R15, R30, R60, etc. – where the number gives the time in minutes). This charring depth is used then in the course of timber fire design (see **6.7.2 Timber beam fire design – TD8 module**).
The maximum fire duration is 180 minutes (R180).
The supported cross-sections are the thick rectangle, rounded rectangle and circle sections.
The diagrams can be saved in the exposure, Drawings Library (*see* **3.6.11 Save to Drawings Library**) by clicking on the icon situated below the diagram in the right. The saved diagrams may be added later to reports.

The notional charring depth ($d_{char,n}$) of exposed timber elements is calculated as follows:

ISO standard fire curve (according to EN 1995-1-2 3.4.2.) – without fire protection:

$$d_{char,n} = \beta_n t$$

β_n [mm/min] design notional charring rate under standard fire exposure

t [min] time

Parametric fire curve (according to Annex A of EN 1995-1-2) – without fire protection:

$$d_{char,n} = \begin{cases} \beta_{par} t & t \le t_0 \\ \beta_{par}\left(1.5t - \frac{t^2}{4t_0} - \frac{t_0}{4}\right) & t_0 < t \le 3t_0 \\ 2\beta_{par} t_0 & 3t_0 < t \le 5t_0 \end{cases}$$

where

$$t_0 = 0.009\frac{q_{f,d}}{O} \qquad \beta_{par} = 1.5\beta_n \frac{0.2\sqrt{\Gamma} - 0.04}{0.16\sqrt{\Gamma} + 0.08} \qquad \Gamma = \frac{(O/b)^2}{(0.04/1160)^2} \qquad O = \frac{A_v}{A_t}\sqrt{h} \qquad b = \sqrt{\rho_{pw} c_{pw} \lambda_{pw}}$$

$q_{f,d}$ [MJ/ m^2] design value of the fire load density related to the surface area A_f of the floor

A_v [m^2] total area of vertical openings on all walls

h [m] weighted average of window heights on all walls

A_t [m^2] total area of enclosure (walls, ceiling and floor, including openings)

c_{pw} [J/kg°C] specific heat of boundary of enclosure

ρ_{pw} [kg/m^3] density of boundary of enclosure

λ_{pw} [W/m°C] thermal conductivity of boundary of enclosure

☞ **If the boundary of enclosure consists of different layers, c_{pw}, ρ_{pw} and λ_{pw} parameters should be given resulting parameter $b = \sqrt{c_{pw}\rho_{pw}\lambda_{pw}}$ equal to $b = \sqrt{c_{pw,1}\rho_{pw,1}\lambda_{pw,1}}\frac{A_1}{\sum A} + \cdots + \sqrt{c_{pw,n}\rho_{pw,n}\lambda_{pw,n}}\frac{A_n}{\sum A}$.**

Notional charring depth of protected timber elements can be calculated only for ISO standard fire curve according to Section 3.4.3. of EN 1995-1-2 standard. The following parameters are required:

k_2 [-] insulation coefficient

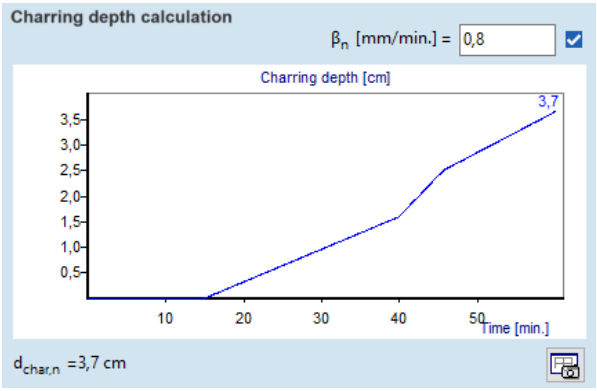
t_{ch} [min] time of start of charring of protected members (delay of start of charring due to protection)

t_f [min] failure time of protection

These parameters need to be specified based on the regulations of the code, the depth and the type of fire protection material.

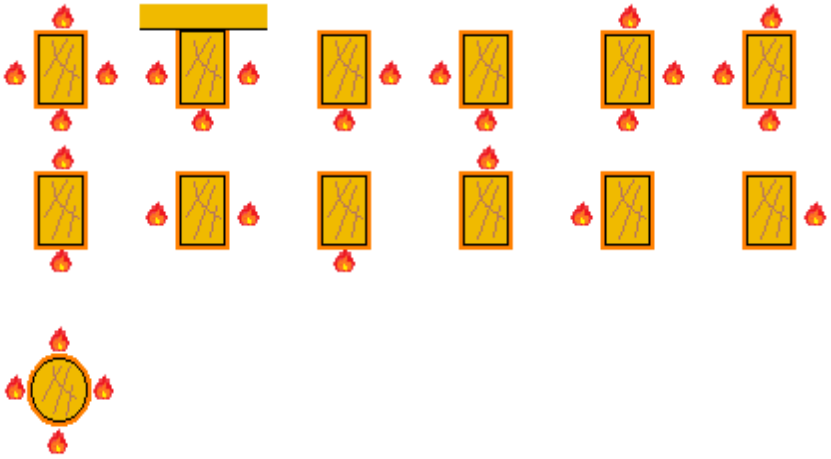
Charring rate

The software selects the notional charring rate based on the parameters of timber material of the selected member. However, the value can be specified by the user.



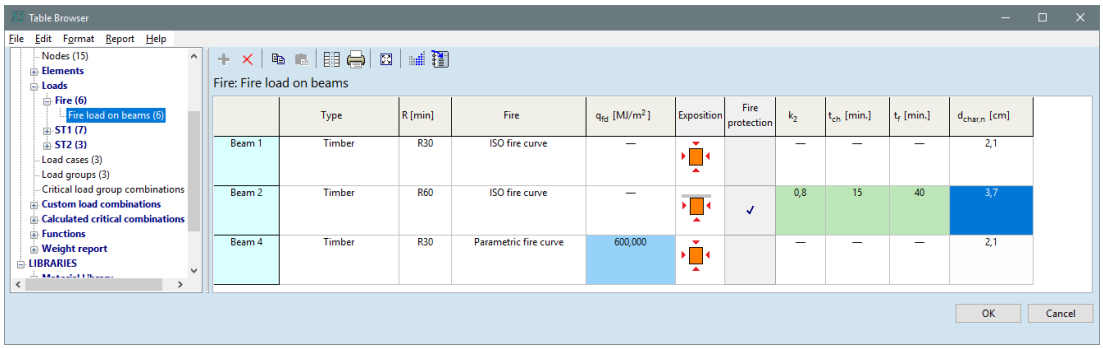
Exposition

On the *Exposition* panel it is possible to set how the member is exposed to fire. Exposure type influences only the calculation of reduced cross section during fire design (see **6.7.2 Timber beam fire design – TD8 module**). In case of supported section types, the following exposure types are available for protected / unprotected members:



Table

Fire effect parameters for each fire load case can be checked in the Table Browser. The listed parameters are: required time of fire resistance, type of fire curve, fire load density, exposure type, properties of fire protection and calculated notional charring depth. The tables can be added to reports.



Graphical symbols in the main window

Fire effects are shown in the main window as red zigzag lines similarly to steel members (see **4.10.32 Fire effect on steel line elements – SD8 module**). Symbol properties (thickness, colour) can be set in *Preferences / Graphical symbols* window.

Errors

If gas temperature or notional charring depth cannot be calculated due to inappropriate parameter values an error message appears. EN 1991-1-2 impose several restrictions regarding to initial parameters of parametric fire curves (opening factor, $b = \sqrt{c_{pw}\rho_{pw}\lambda_{pw}}$, floor area, height, etc.).

4.10.34 Fire effect on concrete line elements – RC8-B module

Design codes

In AXISVM, fire effects can be generated according to different national codes. The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

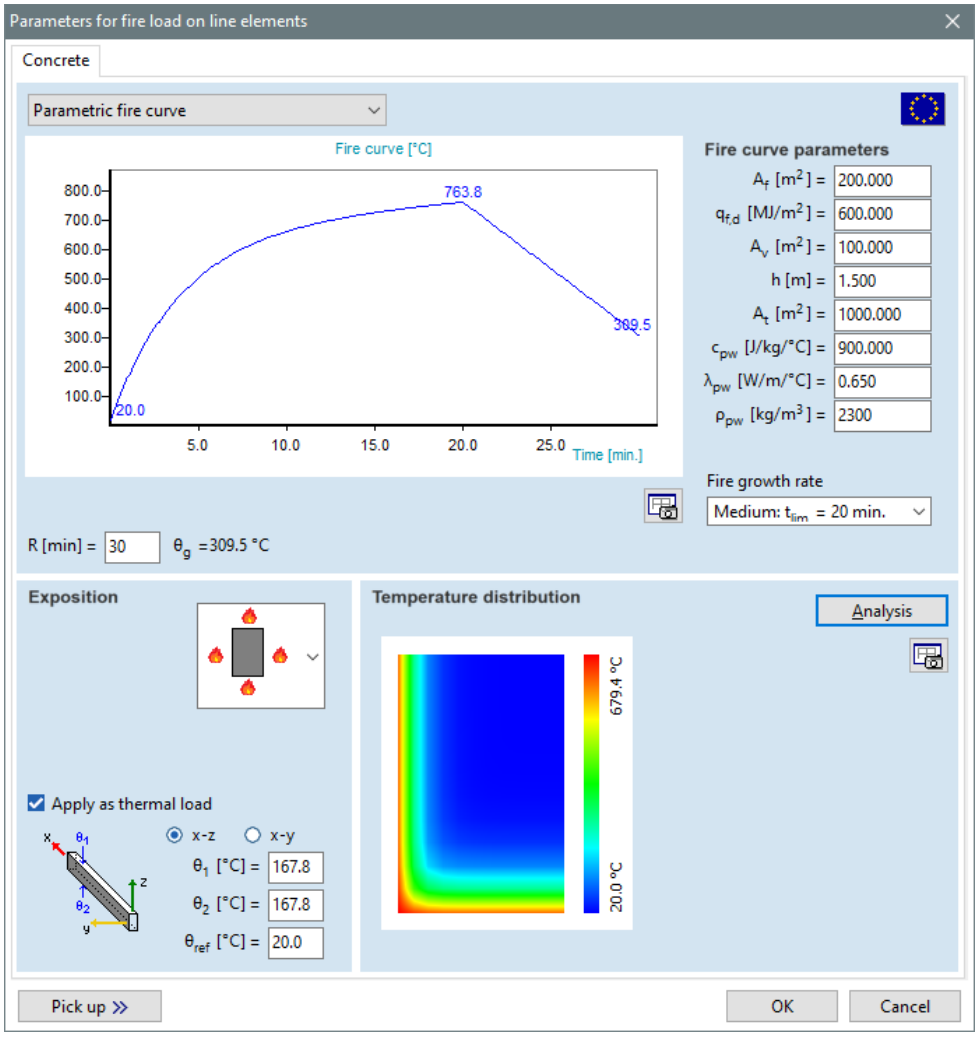
Supported cross-section types

Rectangular

Round

Definition of fire effect

Click on *Define fire effect on line elements* icon on the *Loads* toolbar to define fire effect on line elements.



Fire curves

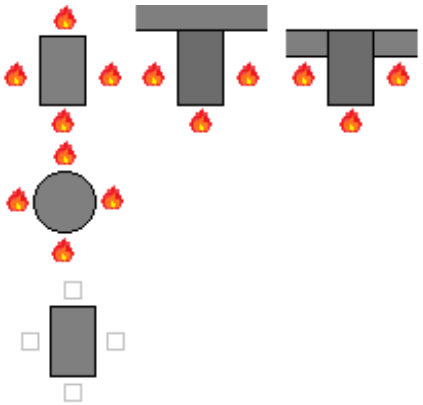
Fire effect is represented by a fire curve specifying the gas temperature in the fire compartment as a function of the time. For concrete elements, *ISO standard, external, hydrocarbon, modified hydrocarbon (tunnel)* and *parametric* fire curves can be selected. These fire curves are presented and discussed in the previous section (see **4.10.32 Fire effect on steel line elements – SD8 module**).
The maximum fire duration is 300 minutes (R300). The fire curve and gas temperatures are calculated and plotted only for the specified time period.
The units of parameters, fire duration and temperatures can be customized (see **3.3.8 Units and Formats**).

☞ **The suitability of the chosen fire curve for the design situation must be verified.**

The fire curve can be saved to the Drawings Library (see **3.6.11 Save to Drawings Library**) by clicking on the icon below the diagram in the right.

Exposition

The *Exposition* panel allows you to select how the section is exposed to fire. This information is used in the temperature analysis. The supported cases are on the right.
In case of T cross-sections, the effect of the connecting slab is taken into account. The only difference between the two options is the interpretation of the rib eccentricity.
In the case of a three-sided, rib-style exposure neither the slab thickness nor the overhang shall be less than 50 mm.



For exposure types with empty boxes the exposed sides can be set by checking the respective boxes. It can be useful for columns placed against a wall or in a corner.

Apply as thermal load

Both calculated and given temperatures can be applied on the structural element as thermal load (see **4.10.21 Thermal load on line elements**) to determine internal forces due to restrained thermal deformations by a linear static analysis.
The default value is not equal to the maximum temperature of the concrete. It is calculated regarding the difference between linear and non-linear distribution of thermal strains in order to have the same effect in the cross-section.

Temperature analysis

The verification of a reinforced concrete structure exposed to fire must take into account the loss of stiffness and strength of the concrete and steel, and the delamination of the outer, damaged concrete layer. These effects can be quantified according to design code specifications, if the temperature distribution within the structural element is known.
The software solves the 2D heat transfer problem with a finite difference method. The temperature dependent parameters are calculated according to EN 1992-1-2. Depending on the available computational resources and the symmetry of the problem, the calculation may take a few seconds or a few minutes.

$$\frac{\partial T}{\partial t} = \kappa\left(\frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) + \frac{Q}{\rho c_p}$$

The cross-sections (except circular cross-section) are discretized by a rectangular mesh of size approx. 2 centimetres. The discretization in time uses a time step of 30 s. The temperature analysis is validated using temperature profiles in EN 1992-1-2.
The results of temperature analysis are plotted in the dialog. It has to be noted that this temperature distribution corresponds to the moment of maximum temperature and that is not necessarily the end of the given time limit (in case of parametric fires there may be a cooling phase).
The temperature analysis is not performed automatically. To launch the calculation, click on the *Analysis* button. Temperature analysis results and parameters are saved. If a structural element with matching parameters is chosen the results are loaded so temperature distribution is immediately available.
If the dialog is closed without performing temperature analysis, the software will automatically calculate the temperatures if needed. However, this may slow down the design process or report generation.
A colour scale in the plot helps to identify warmer and cooler areas. If you move the cursor over the plot, a small tooltip window shows the local temperature. Double-clicking on the plot displays it in a larger window. The figure can be saved to the Drawings Library (see **3.6.11 Save to Drawings Library**) by clicking on the icon below the *Analysis* button
For an element with a variable cross-section two plots are shown. The temperature distribution within the first cross-section is on the left and the temperature distribution of the last cross-section in on the right. Temperature distributions of intermediate cross-sections are linearly interpolated between the ends.

☞ **The thermal conductivity can be set among the design parameters** (**3.3.7 Design codes**).

4.10.35 Fire effect on concrete domains – RC8-S module

Design codes

In AXISVM, fire effects can be generated according to different national codes. The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

Supported domain types

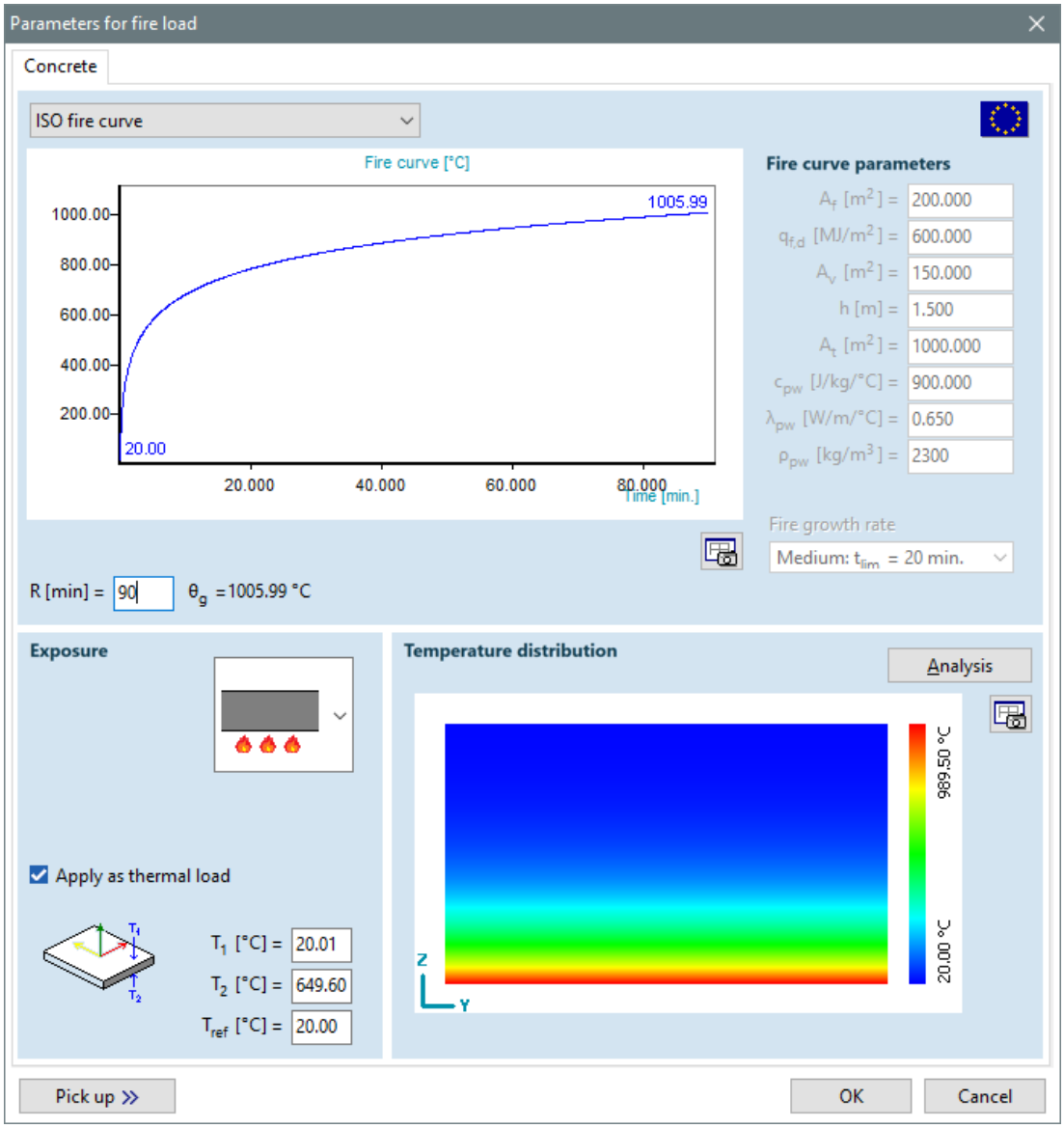
This feature is available for the following domain types (see **4.9.6 Domain**):

- normal domains,
- composite ribbed domains,
- parametric ribbed domains,
- domains with custom stiffness matrix.



Click on *Define fire effect on domains* icon on the *Loads* toolbar in order to attach fire effect to domains.

Definition of fire effect



Fire curves

Fire effect is represented by a fire curve specifying the gas temperature in the fire compartment as a function of the time. For concrete domains, ISO standard, external, hydrocarbon and parametric fire curves can be selected. These fire curves are presented and discussed in the previous section (see **4.10.32 Fire effect on steel line elements – SD8 module**).

The maximum fire duration is 300 minutes (R300). The fire curve and gas temperatures are calculated and plotted only for the specified time period.

The units of parameters, fire duration and temperatures can be customized (see **3.3.8 Units and Formats**).

☞ **The suitability of the chosen fire curve for the design situation under consideration must be verified.**

The fire curve can be saved to the Drawings Library (see **3.6.11 Save to Drawings Library**) by clicking on the icon below the diagram in the right.

Exposition

The *Exposition* panel allows you to select how the domain is exposed to fire. This information is used in the temperature analysis.



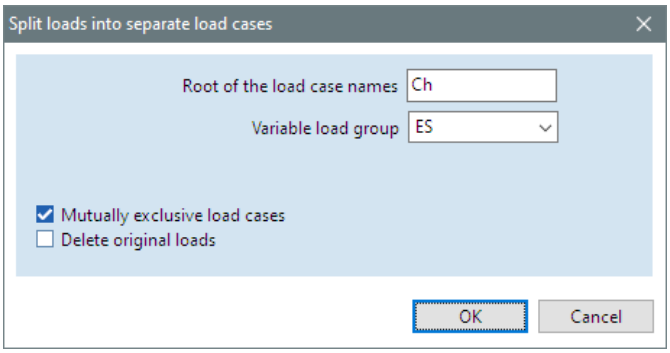
Apply as thermal load and temperature analysis

These features are presented and discussed in the previous section (see **4.10.34 Fire effect on concrete line elements – RC8-B module**).

4.10.36 Split loads into separate load cases

By elements

This function copies or moves selected loads into separate load cases (created automatically within a given variable load group). Each load will get into a separate load case.



Root of the load case names cause load cases to be named as *Root1, Root2, Root3, etc.*
Select *Variable load group* from the existing variable groups or enter a new name to create a new group.

☞ **Loads applied to the same element will get into the same load case.**

Check *Delete original loads* to move loads into the new load cases. Otherwise loads will be copied.

By loads

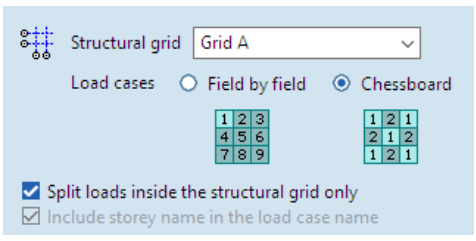
This function acts like the previous one but copies or moves selected loads into separate load cases regardless the element they are applied to. Loads applied to the same element will get into separate load cases.

By storeys

Existing loads placed into separate load cases according to the storey they are applied on.
Cut loads at storey boundaries: Loads extending to multiple storeys are splitted at storey boundaries only if this option is activated. Otherwise, these loads are placed into the load case of the lowest storey they cover.
Include storey name in the load case name: If this option is activated generated load case names will refer to the storey they belong to.

By a structural grid

This function splits existing loads along the lines of a structural grid (see **2.16.8 Structural grids**) and place the resulting loads in separate load cases. The two splitting options are:

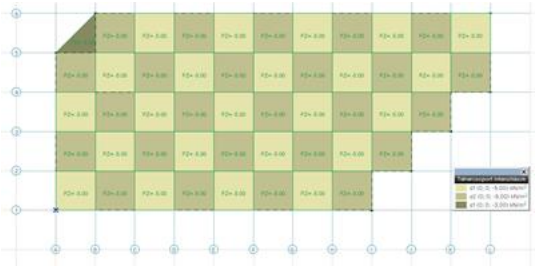


Field by field: Each field gets to a separate load case.

Chessboard: Field loads will be placed in two complimentary load cases following a chessboard pattern.

Split loads inside the structural grid only: If this option is deactivated, loads outside the boundary of the structural grid are also splitted. If the structural grid was defined for a storey load case names can refer to the storey the grid belongs to.

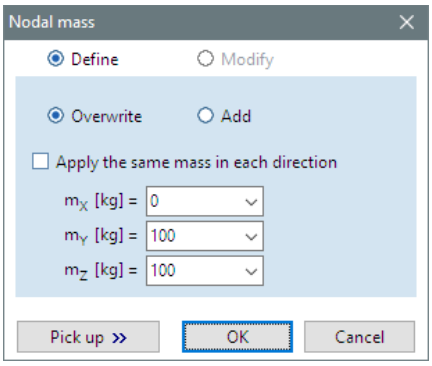
To display domain loads splitted into separate load cases choose the *Load group intensities* color coding. This way loads of all load cases within the current load group will be displayed.
See 2.16.5 Color coding.



4.10.37 Nodal mass



In a vibration analysis the masses are concentrated at nodes that you can take into account by their global components m_X, m_Y, m_Z. In second-order vibration analysis, the loads due to the nodal masses are applied on the model, as well as the masses due to the applied loads.
If mass is the same in each direction it is enough to specify one value after checking *Apply the same mass in each direction*.
In dynamic analysis nodal masses and nodal accelerations result in dynamic loads causing displacements and forces in the model.



The nodal mass is displayed on the screen as two dark red concentric circles.

4.10.38 Modify

Modify

To modify loads:

1. Press the **[Shift]** key and select loads you want to modify (or the loaded elements). You can also select by drawing a selection frame or using the Selection toolbar.
2. Click the load type icon on the Toolbar.
3. Check the checkboxes beside the values you want to change.
4. Enter new values.
5. Close the dialog with **OK**.

Immediate mode

If the *Loads* tab is active click a finite element to modify its loads. If the element has more than one load only one of them will come up. If you have placed different concentrated and distributive loads on a beam and click the beam the load nearest to the click position will come up. If more finite elements have been selected their loads can immediately be modified by clicking one of them. If you click an element which is not selected, selection disappears and you can modify the element load you clicked.

In fact, load modification is similar to the load definition, but does not assign loads to elements not being loaded and allows access to a specific load property without altering others. You can switch to the Define radio button to place loads on all the selected elements, lines or surfaces. If we select elements with loads not matching the load type we choose these loads remain unchanged.

4.10.39 Delete

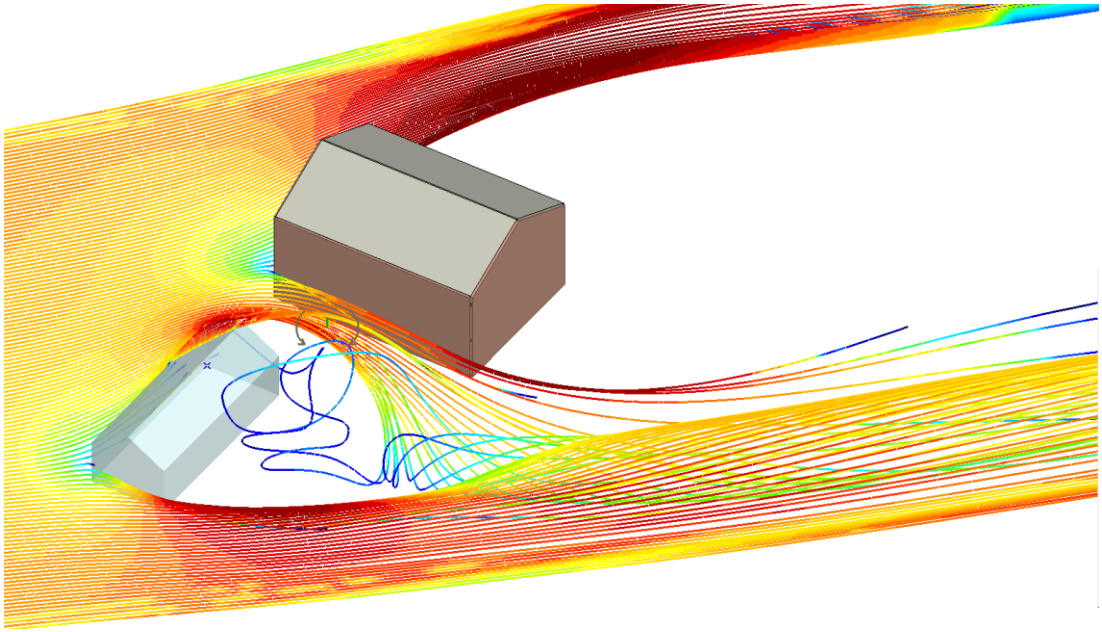
[Del]

See 3.2.9 Delete

4.11 Wind tunnel – WIND module



If the model is more complex than the cases discussed in the design code and cannot be calculated properly with the SWG module (see **4.10.14 Wind load – SWG module**) the only way to determine the real wind loads is to perform a physical wind tunnel test of a scaled model of the structure, which is quite laborious, expensive and extremely time consuming. The WIND module offers a state-of-the-art alternative: it determines the flow field around the structure and the pressure values acting on the structure by a CFD flow simulation.
Wind tunnel analysis places the model in horizontal winds blowing from different directions with a speed depending on the height above the terrain level then determines the wind loads by a CFD (an acronym for Computational Fluid Dynamics) analysis. The WIND module offers the possibility to determine wind loads for a much wider range of projects like lattice structures, partially open halls, cooling towers, wind turbines, and curved shell structures, buildings that have a complex geometry or are not free-standing providing a general solution.



Theoretical background

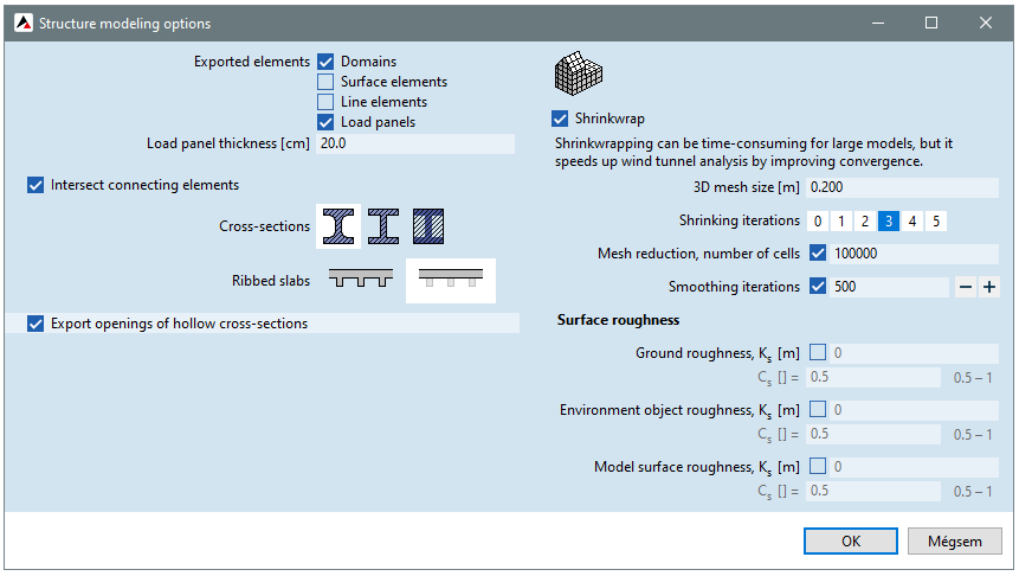
The flow analysis is performed in the background by OpenFOAM, a widely used free and open-source software developed for solving continuum mechanics problems. The WIND module is responsible for building the wind tunnel model, running the calculation, processing the results and generating wind loads. The theoretical aspects are covered in the PDF document *Guide for using the WIND module*, available from the menu under *Help / WIND Guide*.
The flow model is built in OpenFOAM by taking a rectangular space representing a wind tunnel, inserting a properly rotated model of the structure into it, filling the space around the structure with 3D finite elements. On the surface of the model, OpenFOAM creates a matching surface mesh. The face of the rectangular space where the wind enters the space is called inlet, the opposite face is the outlet.

☞ **If parts are activated in the AXISVM model, it can be decided responding to a message if the whole structure should be placed into the wind tunnel or only its active parts.**

☞ **If the geometry does not have fine details and the elements exposed to wind are surfaces, domains and load panels, it is enough to choose the Simple option.**

☞ **Before performing the wind tunnel analysis, it is advisable to review the structural modeling options.**

4.11.1 Structure modeling options



The *Structure modeling options* panel is to control the geometry of the structure model placed into the wind tunnel.

4.11.1.1 Exported elements

Exported elements is to select which types of elements should be included in the structure model placed into the wind tunnel. Load panels are elements with no thickness as their only function is to distribute loads. However, an aerodynamical simulation requires that all elements placed in the flow must have a nonzero thickness, so this value can be entered. The value of *Thickness* is relevant only if the load panel has free-standing parts.
Intersect connecting elements helps to reduce the geometry problems at intersections making meshing easier for OpenFOAM.
Under *Cross-sections* you can choose to model the cross-sections with their actual shape, without rounding radii, or only with their bounding rectangle. Simplifications allow you to reduce the detail of the flow mesh, i.e. the number of elements, and therefore the time required for the calculation.
Under *Ribbed slabs* it is possible either to model or to ignore ribs on slabs.

☞ **Wind loads will be generated only to the elements placed in the wind tunnel.**

☞ **Try to use the simplest possible geometric model. Elements that do not play a significant role in the flow should not be considered, as this would unnecessarily increase the computation time. For example, if the model is enveloped by a load panel, in most cases it is sufficient to consider only the panel, and there is no need to export additional elements.**

4.11.1.2 Shrinkwrap

In the shrink-wrap method the original model used in wind tunnel simulation – which is often highly detailed and contains minor geometric errors – is "wrapped" into a simplified, continuous outer surface. This wrapping surface seals any gaps, smooths out minute details that are irrelevant to the simulation, and creates a "watertight", meshable geometry.
Its primary aim is to simplify mesh generation, reduce the number of cells and the computational load, whilst preserving the external shape that is important when modeling the airflow.

3D mesh size The mesh size determines the edge length of the volume mesh, from which the surface to be shrunk is generated. The smaller the mesh size, the more accurately the generated geometry follows the original structure. At the same time, the goal is not to map the surface as accurately as possible, since a smaller mesh size significantly increases the computational requirements and, consequently, the complexity of the calculation. The goal is therefore to determine the largest possible mesh size at which the geometry still maintains sufficient accuracy.

☞ **The mesh size also determines the size of the openings that are closed off during the packaging process. As a rough guide, when using a mesh size of 2 metres, openings with a diameter of approximately 2 metres or less are typically closed off.**

☞ **The topological characteristics of the surface cut out from the 3D mesh, such as its genus (the number of holes), do not change during the shrinking processes.**

Shrinking iterations

It is possible to specify how many shrinkage iterations the initial wrapping surface should perform towards the target surface. During each iteration, the surface gradually approaches the target surface, whilst the mesh is refined in regions of higher curvature so that the shrunk surface follows the geometry of the target surface as closely as possible.

☞ **If the number of iterations is 0, no shrinkwrap is performed in the direction of the target surface.**

Mesh reductuon, number of cells

Once the wrapping surface has been shrunk onto the target surface, it is possible to reduce the number of triangles in the surface. At this point, you can specify how many triangles you wish to use to represent the wrapping surface at the end of the iteration.

☞ **It is not necessary to specify the number of cells, the process can be disabled by inactivating the checkbox.**

Smoothing iterations

When this function is enabled, the software smooths the wrapping surface. The number of iterations determines how many Laplace smoothing steps the process should perform. A smoother surface is easier to mesh and results in a higher-quality mesh, thereby significantly improving the convergence of the model during CFD simulation.

☞ **It is not necessary to specify the number of iterations, the process can be disabled by inactivating the checkbox.**

4.11.1.3 Surface roughness

The surface roughness settings allow the increased friction resulting from surface irregularities to be taken into account. The program allows three surface types to be handled separately: *Ground*, *Environment objects* (non-load-bearing load panels) and *Model* surfaces. For all three surface types, the option to take into account friction resulting from surface roughness can be enabled or disabled individually.

K_s Equivalent sand-grain roughness. Glass surfaces can be considered smooth (*Ks = 0* mm); for steel surfaces, *Ks = 0.05* mm; and for concrete surfaces, *Ks = 4* mm.

C_s Roughness constant (0.5–1.0): this value characterises the uniformity of the surface roughness. A lower value results in a more uniform surface roughness, whilst a higher value results in a more uneven surface roughness.

4.11.2 Wind tunnel simulation

There are two options to build a wind tunnel model: *Simple* or *Detailed*.

4.11.2.1 Setting analysis parameters (simple)

Load cases

During the definition of the simulated wind load **4.10.15 Simulated wind used in wind tunnel analysis – WIND module** load cases were created according to the specified directions of the wind rose. The flow analysis can be performed on one or more selected load cases. Load cases are grouped by wind roses. Load cases where an analysis has been performed and simulated wind loads have been generated appear in green.

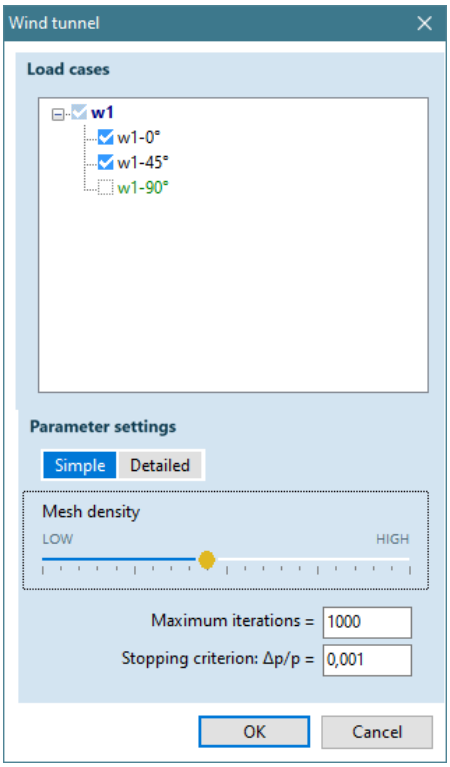
Parameter settings

In the *Simple* mode the generation of the flow analysis mesh can be controlled with a single parameter called *Mesh density*.

The meaning of the *Stopping criterion* is that if the pressure residual *Δp/p,* indicating the solution error, falls below this value, the the analysis terminates with a valid result.

If this criterion has not been met and the analysis gets to the number of iterations set in *Maximum iterations,* the analysis terminates.

☞ **If the solution is convergent, the stopping criterion is met but the values of aerodynamical forces are still changing the stopping criterion should be tightened.**



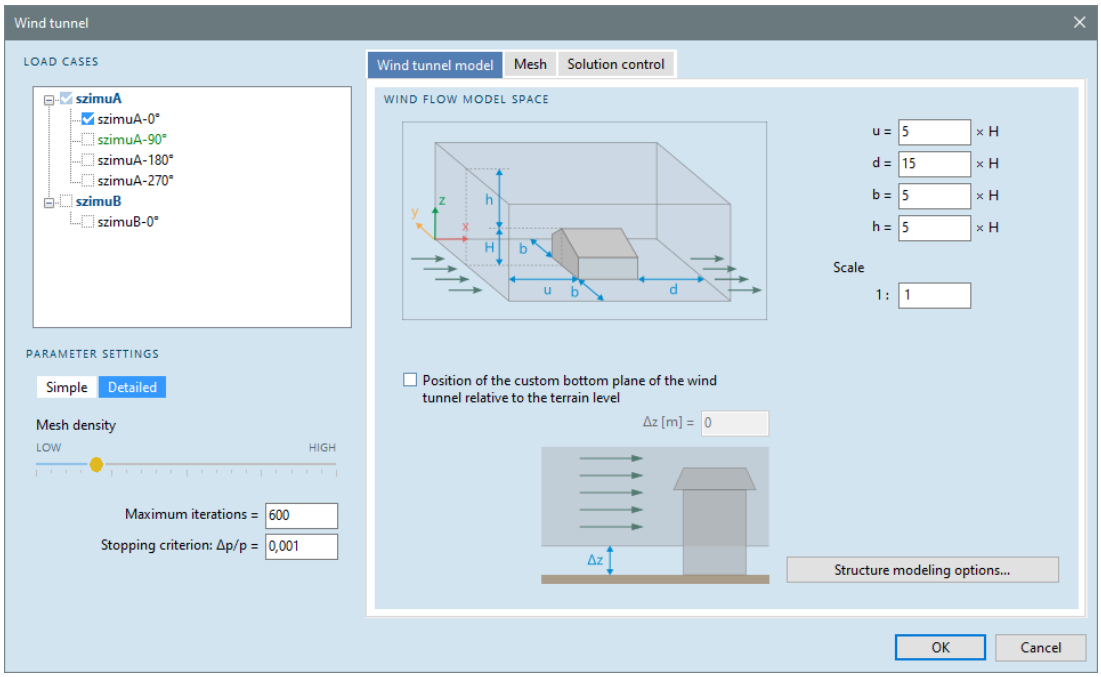
4.11.2.2 Setting analysis parameters (detailed)

Selecting *Load cases*, setting *Maximum iterations* and the *Stopping criterion* is the same as in the *Simple* mode but more options are offered to fine tune the construction and meshing of the flow analysis model (wind tunnel model + structure model), choose turbulence model and define additional stopping criteria.

4.11.2.3 Wind tunnel model

The flow analysis model is built by creating a virtual wind tunnel inside the cuboid drawn around the model according to the wind direction, then placing a model of the structure into the wind tunnel, filling the space between the structure and the walls of the tunnel with 3D finite elements using the OpenFOAM toolbox. The analysis determines the velocity and pressure fields on these finite elements

The *Wind flow model space* panel is to adjust the size of the rectangular space representing the wind tunnel. H denotes the height of the bounding box of the structure model calculated in the wind tunnel coordinate system. The space in front of, behind and on either side of the bounding box *(u, d, b, h)* must be specified. Default values are $u = 5H$, $d = 15H$, $b = 5H$, $h = 5H$.



☞ **For low and wide structures, the above height-dependent method is not the most appropriate. It must be ensured that the so-called blockage ratio (the area of the structure projected onto the inlet divided by the area of the inlet) does not exceed 3%.**

If a small model space is used, walls may affect the pressure field causing a flow that is not real. A possible error could be that the flow between the wall and the structure accelerates more than in reality and the wind suction increases significantly.
Increasing the size of the wind tunnel improves the accuracy, but also increases the computation time, so it is important to find an optimum where the computation resource requirements are still acceptable and the accuracy of the results is good.

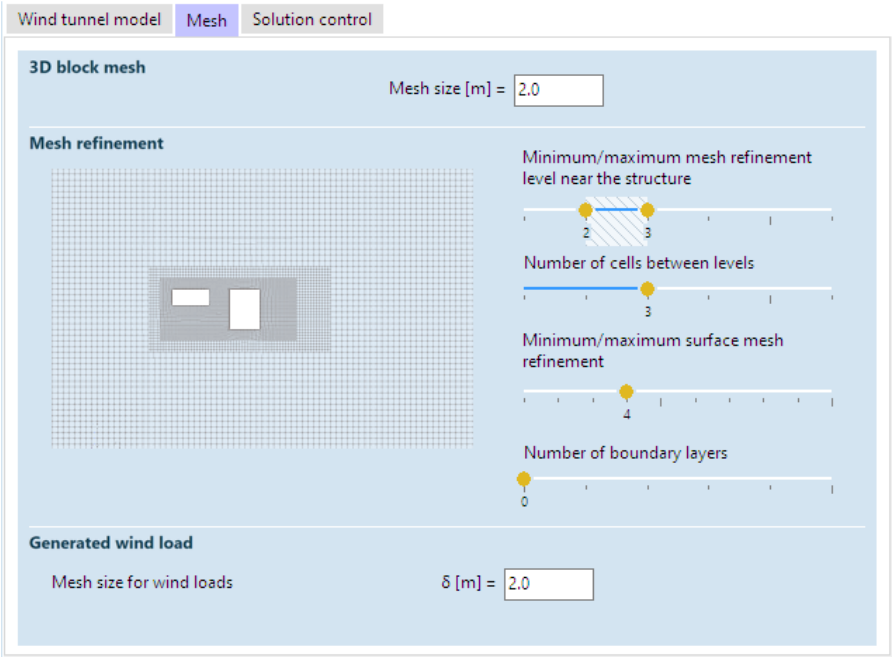
Changing the scale of the model allows reduction of the mesh density, but it is very important to note that rescaling of the model should only be used for Reynolds number independent structures, so users less familiar with CFD modelling should keep the 1:1 scale.

Setting the *Position of custom bottom plane of the wind tunnel relative to the terrain level*, the wind tunnel is truncated at a height Δz measured from ground level, and only the elements located above this height are taken into account in the calculation. A frictionless boundary condition is applied at the truncation surface.
The purpose of this setting is to ensure that, where a structure is elevated significantly above ground level, it is not necessary to analyse the full height of the flow space; instead, only the flow developing in the vicinity of the structure is modelled.

For checking structure modeling options, see **4.11.1 Structure modeling options**

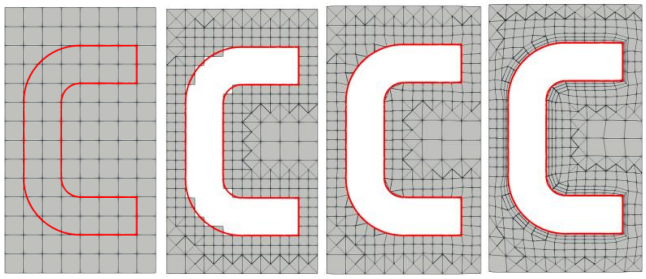
4.11.2.4 Mesh

On this tab you can control how the 3D mesh is generated in OpenFOAM for the flow simulation, and the mesh size used when generating wind loads from the pressure results.



Mesh size

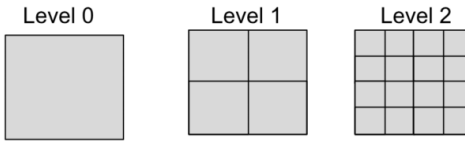
The first step is to create a so-called *blockMesh* (in OpenFOAM terminology). This is a general basic mesh for the wind tunnel space. The *Mesh size* specified controls the density of this mesh.
The second step is to take into account the model placed in the flow with a series of mesh refinements and adjustments. This process is called *snappyHexMesh* and it is the most time-consuming part.



(source: CFDSupport)

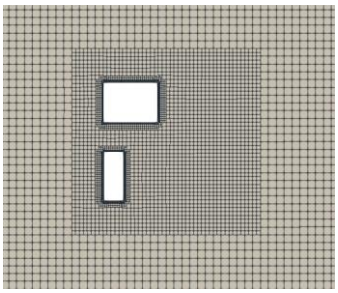
Minimum / maximum mesh refinement level near the structure.

In order to interpret the parameters shown here, we need to define the concept of "level" as used in OpenFOAM meshing. Cells of the basic mesh are on Level 0. Cutting a cell in half in three directions we get 8 small cells on Level 1. Cutting them in half again we get 64 cells on Level 2. Repeating this process *n* times we get 2^{3n} cells.

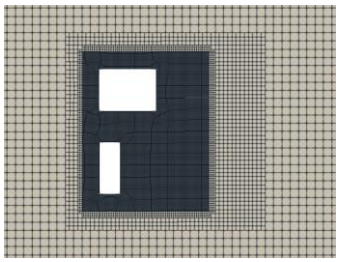


(source: CFDSupport)

The program refines the basic mesh in two steps near the bounding box of the structure. First, it evaluates 20% of the values *u, d, b, h*, and within this range, it applies the minimum mesh refinement, then it applies the maximum mesh refinement within 10% of the values *u, d, b, h*. The refinement is necessary to accurately track the behaviour of the fundamental variables over a range of rapid velocity changes.
The trackbar has two yellow sliders. If they are overlapping, the two ranges will have the same refinement level. Drag the slider to the left to set the minimum level, and to the right to set the maximum level.



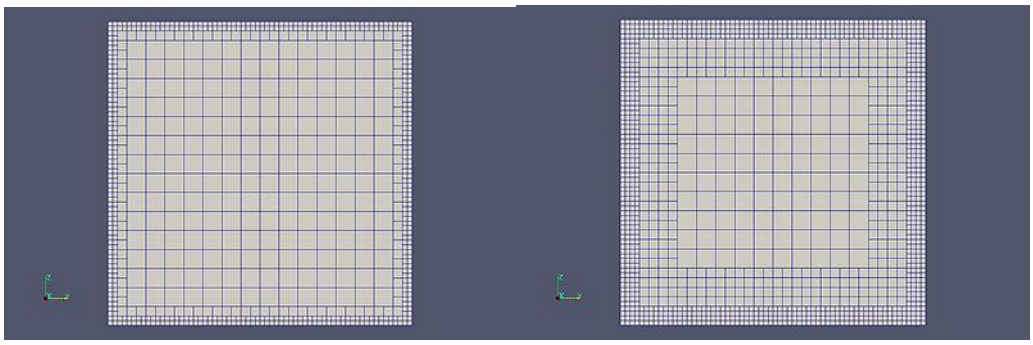
1/1 refinement



1/3 refinement

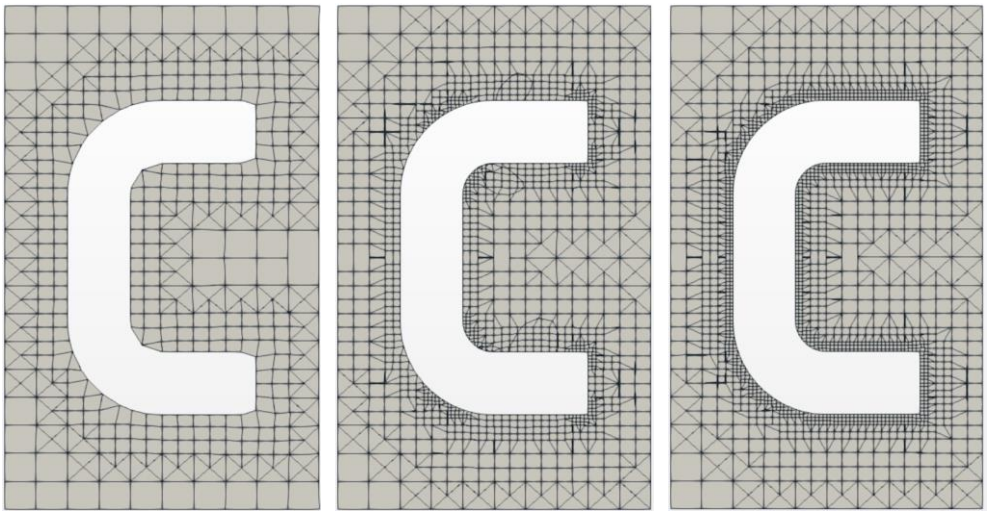
Number of cells between levels

OpenFOAM uses at least this many cell layers between the level changes



Number of cells between levels: 1 and 4 (source: OpenFOAM)

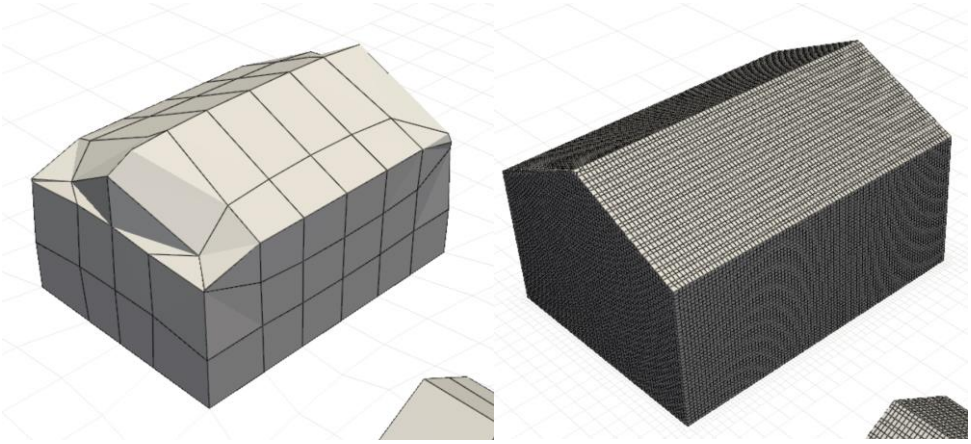
Minimum / maximum surface mesh refinement



Surface mesh refinement for different values, 1/1; 1/3; 3/3 (source: CFDsupport)

The trackbar has two yellow sliders. If they are overlapping, the two levels will be the same. Drag the slider to the left to set the minimum level, and to the right to set the maximum level.

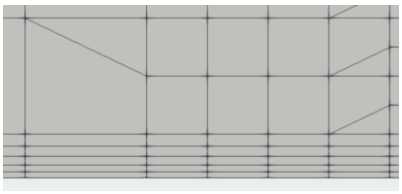
☞ **If the maximum surface mesh refinement is set too low, it may not be sufficient to describe the geometry of the structure properly:**



Example of an inadequate geometry resulting from low surface mesh refinement, and a proper geometry obtained by setting surface mesh refinement higher.

Number of boundary layers

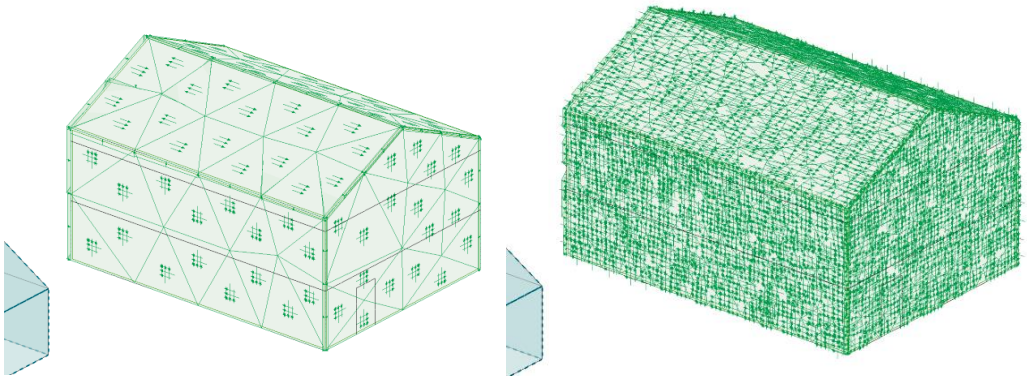
Near model surface the gradient of the velocity field is very high, so it may be necessary to refine the mesh by adding additional layers. 0 means no insertion of layers.



☞ **For curved structures, avoid adding boundary layers.**

Generated wind load

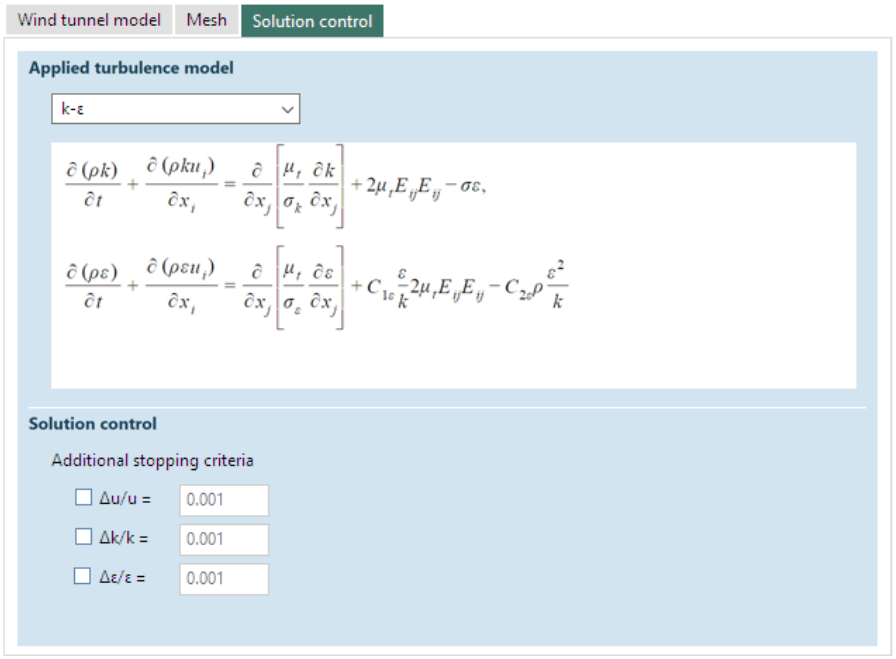
This value controls the size of the mesh used when generating wind loads.



Wind loads generated with a mesh size of 5 m and 1 m

4.11.2.5 Solution control

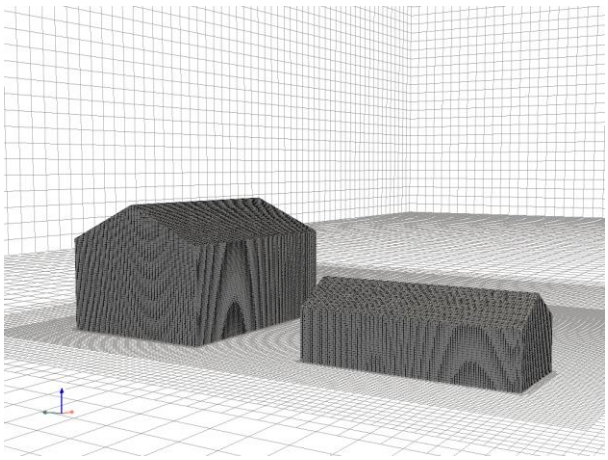
The *Applied turbulence model* can be selected from a dropdown list. The default is the *k-ε* model.
For more information on turbulence models, please refer to the PDF document Guide for using the WIND module, available under *Help / WIND Guide*.
The *Solution control* panel allows setting additional stopping criteria based on residuals of the velocity or turbulence parameters.



4.11.3 Wind tunnel analysis

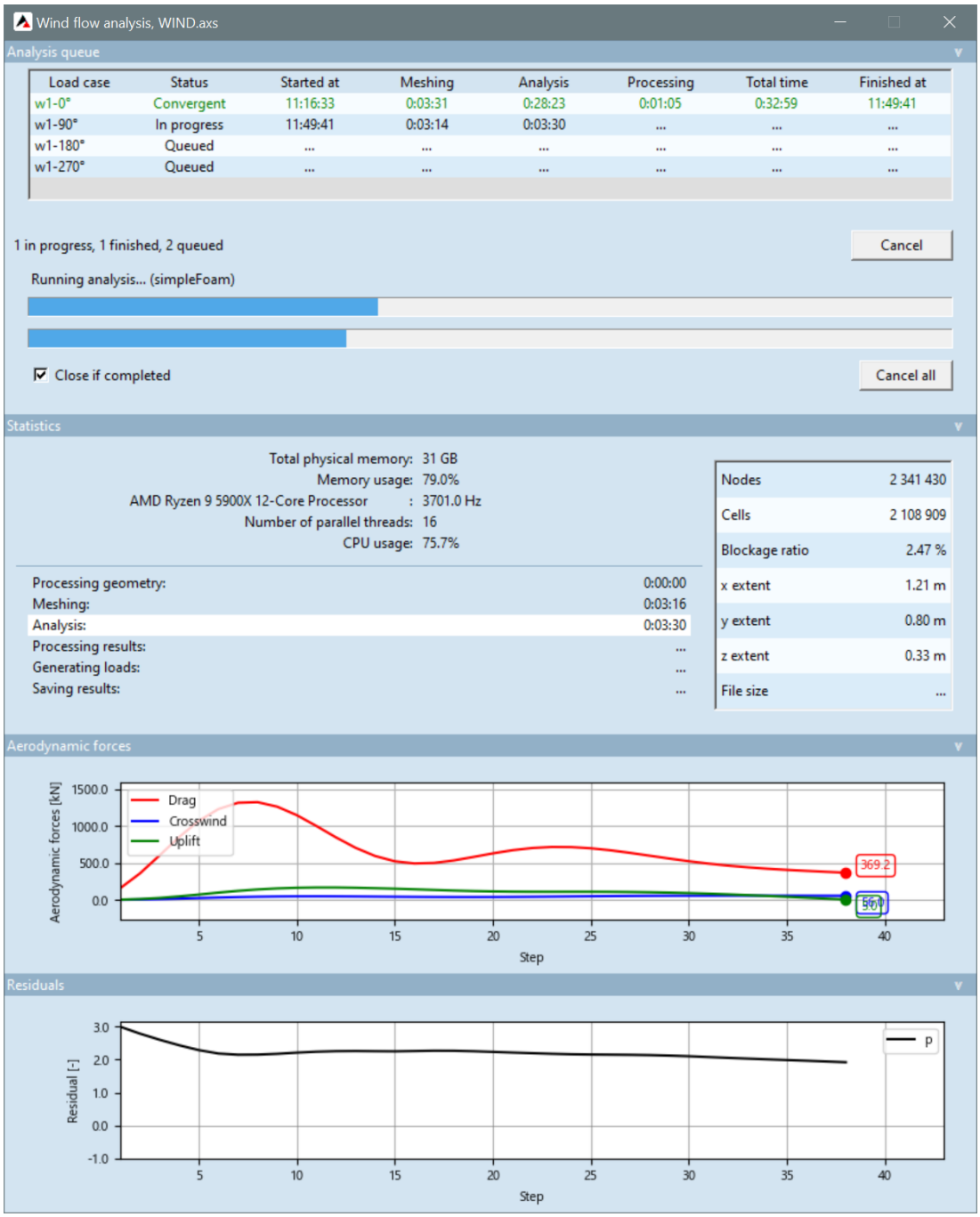
The flow analysis runs for the wind directions checked in the *Wind tunnel* dialog. The module first imports the geometry of the AXISVM model, then meshes the total volume of the wind tunnel and the model surface according to the parameters set in the dialog.

After completing the meshing phase, a 3D diagram of the wind tunnel and the mesh is displayed in a separate window without interrupting the analysis. If the geometry of the structure model is not satisfactory (e.g. mesh refinement is too low), the analysis can be stopped.



The displayed 3D visualization can be rotated around the axes of the model space by dragging, or if *Ctrl* is pressed, around the axis perpendicular to the screen. *Shift*+drag moves the model, the mouse wheel zooms in and out.

While the analysis is running the following data is displayed



Analysis queue

If more than one direction was checked in the *Wind tunnel* dialog, an *Analysis queue* appears on the top panel. It is a list indicating the process of analysis for the selected load cases, and displays the time elapsed in each sub-process. Click *Cancel* to stop the analysis of the running load case. *Cancel all* is to disrupt the entire series of analyses. Checking *Close if completed* makes the program return to AXISVM at the end of the calculation.

Statistics

Three tables are displayed: 1) the main parameters of the computer, 2) the times spent in each sub-process below, and 3) properties of the OpenFOAM mesh on the right. The *Blockage ratio* is the ratio of the area of the model projected onto the inlet plane to the total cross-sectional area of the inlet, expressed as a percentage. After finishing the analysis, the size of the result file is also displayed.

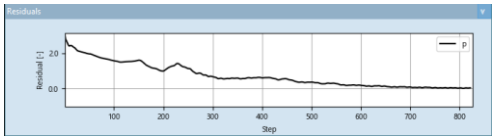
Aerodynamic forces

The resulting aerodynamic forces acting on the structure, interpreted in the local wind tunnel system, are shown here. *Drag* is the resultant force in wind direction, *Crosswind* is the horizontal force perpendicular to the wind direction, *Uplift* is the vertical component of the resultant wind force.

☞ **If the analysis has converged but the resultant forces are still changing rapidly at the end of the calculation, it is advisable to reduce the limit of the residual at the *Stopping criterion*.**

Residuals

The *Residuals* panel plots the logarithm of the ratio of the error value to the target value of the pressure and other variables that you have chosen on *Solution control* tab. The calculation converges when the residuals of all selected variables fall below zero.



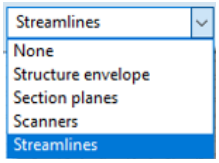
Importing results, generating wind loads

If the analysis converged, AXISVM imports the results, then generates a separate load mesh over the surface of the wind tunnel model using the load mesh size defined in the *Wind tunnel* dialog and applies distributed surface loads on the respective load panels or domains based on pressure values obtained from the simulation. These loads will be placed in the load cases assigned to the the wind roses and wind directions. Beams and ribs receive linear loads distributed along the elements.

4.11.4 Flow analysis results

Components

If the iterative process of the wind flow analysis proved to be convergent, the following results are imported for the load case selected from the dropdown list (components depend on the display mode discussed below).



Pressure

p The pressure on the model envelope *(Structure envelope),* in the model space *(Intersection planes, Scanners)* and along the streamline *(Streamlines)*

Values are negative for suction, positive otherwise.

Pressure coefficient for external surface

cpe The ratio of the pressure at a given point to the dynamic pressure at the height of that point (only in *Structure envelope* mode).

Wall shear stress

τ Shear stress on structure envelope.

Dimensionless wall distance

y+ The dimensionless wall distance is defined on the structure envelope. This parameter is used to determine whether the meshing is sufficiently fine to accurately model the flow.

Velocity

vx, vy, vz Wind speed components in the model space interpreted in the coordinate system of the wind tunnel (*x* is in wind direction, *z* is upwards vertical, *y* is perpendicular to both, so that *x, y, z* form a right-handed system) (in *Section planes* and *Scanners* modes).

vR Resultant wind speed in model space (in *Section planes* and *Scanners* modes).

v Wind speed along the streamline (*Streamlines*)

Result display modes

The following display modes are available for wind flow analysis results:

Structure envelope

Pressure or pressure coefficient displayed on the surface of the model.

Section planes

Pressure or velocity in the wind tunnel displayed in an existing section plane intersecting the 3D mesh of the model space.

Scanners

Interactively moveable section planes parallel to the base planes of the wind tunnel coordinate system. Animations can also be created.

Streamlines

Display of streamlines starting from a series of points in the inlet rectangle. See **4.11.5 Defining streamline seeds and streamline animation**

DIsplay options

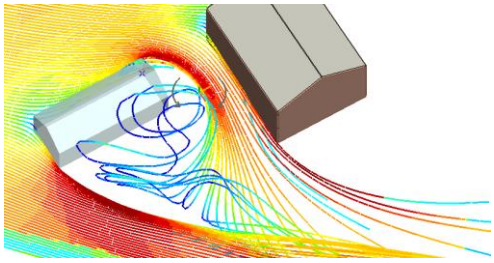


Clicking on the *Mesh* button among the speed buttons at the bottom right (**2.18 Speed buttons**) a popup window appears with several display options

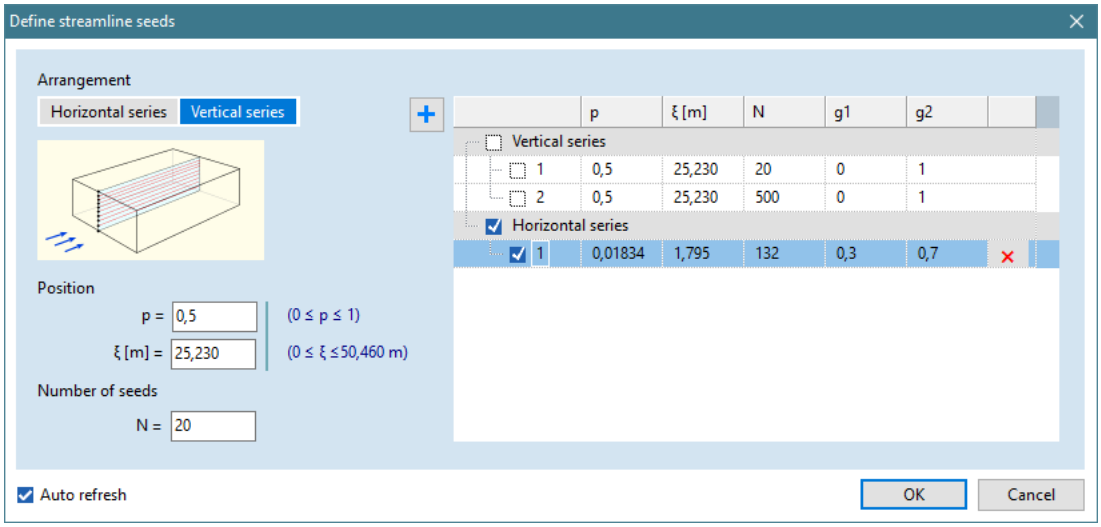
3D mesh for wind flow analysis model	Solid element mesh generated in the wind tunnel
Wind tunnel	Sides of the wind tunnel with a surface mesh
Wind flow model surface mesh	The mesh on the surface of the structural model
Wind flow model space	Sides of the wind tunnel

4.11.5 Defining streamline seeds and streamline animation

Streamlines are curves whose tangent at any point points in the direction of the velocity vector.
In a stationary, i.e. time-invariant flow, they also trace the path of air particles. Plotting them helps evaluation of the analysis results.



To plot streamlines, starting points must be defined. AXISVM allows defining one or more horizontal or vertical series of points on the inlet side of the wind tunnel called streamline seeds. Streamlines are coloured according to the local value of the current result component (velocity *v* or pressure *p*).



On the left you can set properties of a series. The arrangement of seeds can be either horizontal or vertical. For a horizontal series, the vertical position of the series, for a vertical series, the horizontal position of the series must be specified as a proportion of the corresponding size of the input side *(p)*, or as a specific distance *(ξ)*. The number of points (i.e. streamlines) is N.
By clicking on the + button, the series is added to the table on the right.
The current row in the table can be deleted by clicking on the red *Delete* icon at the end of the row.
Streamlines will be generated from the point series only if the checkbox at the beginning of the row is checked. *g1* and *g2* can be modified to indicate the position of the start and end of the point series in relation to the corresponding size of the inlet. Their default values are 0 and 1, i.e. the point line runs along the full inlet height or width.
To change a value, double-click on the respective cell of the table.

When *Auto refresh* is enabled, changes made to the streamlines seeds are applied immediately in the main window (if it is in the *Streamlines* display mode).

Streamline animation

Streamline animation is another way of visualizing the flow and changes in velocity. The length, distance and animation speed of the particles launched along the streamline can be controlled with sliders. The animation can be saved to a file.
The settings are similar to animation export settings (see **6.1.2 Animation**). Click on the *Settings* icon to set the number of frames to capture and the image size of the video output (the image in the active window is scaled to the specified size), and the additional delay between frames for GIF format.

4.12 Mesh



Clicking on the *Mesh* tab the meshing toolbar becomes available with mesh generation for line elements and domains, mesh refinement functions and a finite element shape checking.

4.12.1 Mesh generation

Automatic detection of overlapping lines and missing intersections during meshing reduces the errors in model geometry.
Support of multiple core processors can reduce the time of meshing.

4.12.1.1 Meshing of line elements



Finite element analysis uses linear elements with constant cross-section so arced and variable cross-section (tapered) line elements must be divided into parts. This is called line element meshing. The accuracy of the solution depens on the mesh density.
This mesh can be removed or modified just like a domain mesh. Removing a mesh does not delete loads and properties assigned to the line element.
A mesh can also be defined for linear elements with constant cross-section. It is useful in nonlinear or vibration analysis when it is required to divide line elements to achieve a higher accuracy.

Mesh parameters for line elements

Mesh generation can be performed according to different criteria:

Maximum deviation from arc: Chord height cannot exceed the value specified.

Maximum element size: Length of the mesh lines cannot exceed the value specified.

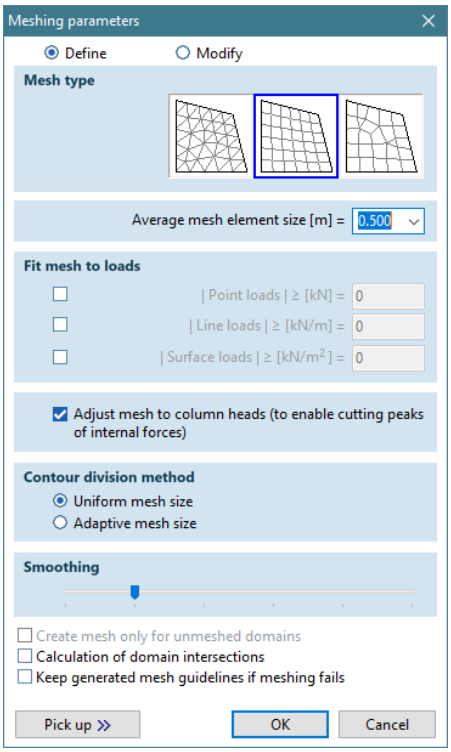
Division into N segments: Line elements are divided into N parts.

By angle: Central angle of arced mesh segments cannot exceed the value specified.

4.12.1.2 Meshing of domains



A mesh of triangular surface elements can be generated on the selected domains by specifying an average surface element side length for the mesh. Meshing will take into account all the holes, internal lines and points of the domain. Meshes optionally can follow loads above a certain intensity or be adjusted to column heads to enable cutting peaks of internal forces.



Mesh type The mesh can be a triangle mesh, a quadrangle mesh or a mixed mesh, in which most of the elements are quadrilateral with some triangles.
If lines of the domain outline including holes and internal lines can be divided into quadrangles and the quadrangle mesh is selected a better-quality parametric mesh is generated.

Mesh size An average mesh element size can be specified. The actual mesh can contain smaller and larger elements as well.

Fit mesh to loads Meshes will follow checked loads if load intensity exceeds the value specified. Point loads will create mesh nodes, line loads will create mesh lines.

Adjust mesh to column heads

The mesh must be properly adjusted to column heads to prepare cutting peaks of internal forces. Turning this option on automatically fits the mesh according to the cross-section geometry of connecting columns. All beams joining to the slab at an angle greater than 45° are identified as columns. This option must be set to enable the *Cut peaks of internal forces over columns* option of the *Display Parameters* dialog. See **6.1.13 Surface element internal forces.**

Contour division method

Uniform mesh size

Domain boundaries and inner lines will be divided according to the mesh size to ensure the given element size.

Adaptive mesh size

Adaptive meshing follows domain geometry and refine the mesh by reducing element size wherever it is necessary.

Smoothing

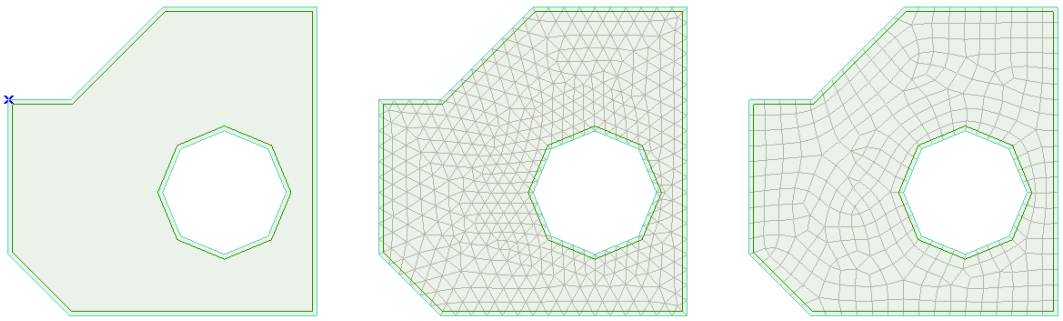
Track bar controls the smoothing of the mesh. Smoothing slows down mesh generation a bit. Moving the handle to the left end sets minimum smoothing and fast processing while the right end sets maximum smoothing with slower processing. The result of smoothing depends on domain geometry and other mesh parameters, so setting higher smoothing does not necessarily result in a better mesh quality.

If *Create mesh only for unmeshed domains* is checked no mesh will be created for domains already meshed.

If *Calculation of domain intersections* is turned on domain intersections are automatically calculated before meshing.

The progress of the mesh generation process can be monitored in a window, and can be canceled any time with the *Abort* button.

The mesh generator uses only the end points of beam elements that are in the plane of the domain, and disregards their corresponding line segments. Rib elements are incorporated with their line segments because they can be defined on surface edges as well.
If there are existing quadrilateral or triangular meshes within the domain, the mesh generator will not change these meshes, and will integrate them in the new mesh.



Before meshing *Triangle mesh* *Quadrangle mesh with triangles*

☞ **If a mesh is generated over an existing domain mesh (with a different average element side length), the new mesh will replace the existing one.**

4.12.1.3 Meshing of soil modeling domains

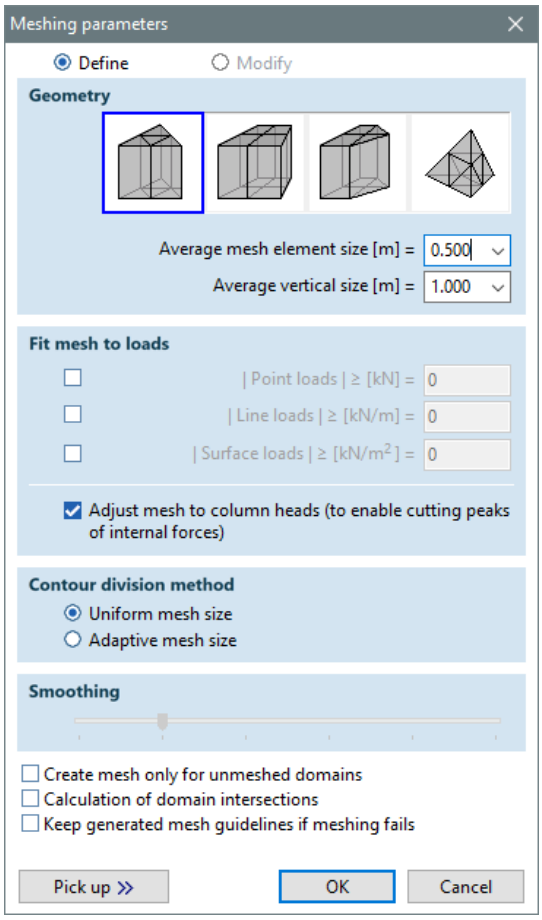


The program generates a volume mesh with the given average horizontal and vertical mesh sizes for the selected soil modeling domains considering all the holes, nodes and lines inside the domains.

Choose the type of the generated solid elements from the *Geometry* panel. The shape of the generated solid elements can be extruded triangle (wedge), extruded quadrilateral (hexahedron), a mixture of the two or tetrahedron.

The solid element mesh is created by extruding the surface mesh vertically. In the case of tetrahedral meshing, the size and number of tetrahedra are dynamically adapted to the local layer thickness. If some layers are thinning to zero, the first three meshing methods may fail. In these cases, choose tetrahedral meshing.

Average mesh element size is the average element size of the planar mesh generated for the soil modeling domain.
Average vertical size is the average vertical element size of the solid elements. The actual vertical sizes also depend on the thickness of soil layers obtained from the stratigraphic model.



☞ **The generated mesh consists of solid elements, the total depth of the model is the largest of the *d* minimum values of the contiguous soil modelling domains. See 4.9.9 Soil modeling domain – SOIL module.**

4.12.2 Mesh refinement





Lets you refine the finite element mesh of surfaces. The elements in the refined mesh have the same properties (material, cross-section / thickness, references, etc.) as those in the coarse mesh.

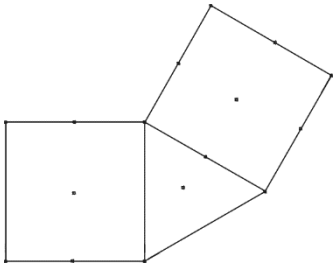
☞ **You have to manually set the nodal degrees of freedom of the newly generated mesh that were not set automatically during the process of mesh generation.**

The following options are available:

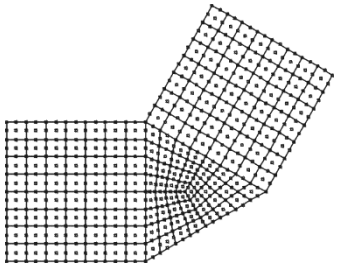
Uniform



Lets you refine the entire selected mesh. You must specify the maximum side length of a surface element in the refined mesh.



Before mesh refinement

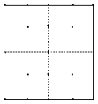


After mesh refinement

Bisection



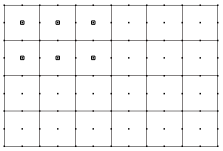
Lets you refine the selected mesh by bisecting the elements as shown in the figure.



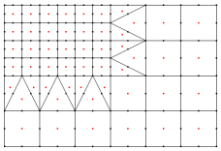
Quadrilateral element



Triangular element



before

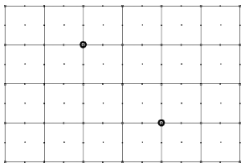


after

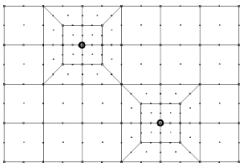
Node relative



Lets you refine the mesh around the selected nodes (locally around columns, nodal supports). You must specify a division ratio (0.2-0.8). The command refines the mesh dividing the elements connected to the respective nodes by the defined ratio.



before

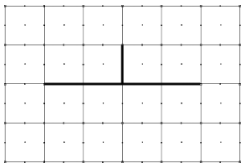


after

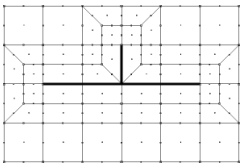
Edge relative



Lets you refine the mesh along the selected edges (locally along edge supports / loads). You must specify a division ratio (0.2-0.8). The command refines the mesh dividing the elements connected to the respective edges by the defined ratio.



before



after

4.12.3 Checking finite elements



Program checks the minimum angle of surface finite elements (α).

A triangular finite element is distorted if $\alpha \leq 15$.
A qudrilateral finite element is distorted if $\alpha \leq 30$.

4.12.4 Select free edges



This command selects all edges connected to one surface element only. This selection helps to find edges where the meshes of two domains are not connected due some editing problem.

4.12.5 Delete all meshes



Deletes all meshes from line elements



Deletes all domain meshes from domains



Deletes all (line or domain) meshes of the model.

4.13 Construction stages – STG module



The tools of this tab provide functions related to definition of construction stages, controlling the construction, modification and removal of elements in different stages, and assigning load cases to stages.

Considering construction stages in structural design

Considering construction stages in structural design is crucially important, as during the construction process the structure is subjected to loads and conditions that differ from its final (operational) state. If these are not properly accounted for, especially in finite element modeling, we may obtain a misleading picture of the structural response. The following key aspects highlight why it is essential to analyze construction stages separately:

- **Accounting for construction phases:**
  In finite element models, the entire structure is often analysed as if all structural components are present at the same time and loads are applied simultaneously. In reality, structures are built gradually, and loads – such as self-weight – appear incrementally. Ignoring this can lead to inaccurate stress and strain distributions.
- **Changing structural systems during construction:**
  The structural system evolves continuously during construction: individual elements are built at different times and it may happen that at first certain structural elements do not have their final supports. If this is not considered, the model incorrectly assumes that all elements bear the loads in their completed state.
- **Temporary loads and deformations:**
  Deflections and deformations that occur during construction may be critical, particularly for long-span or tall structures. The self-weight applied incrementally can cause permanent deformations, which affect the final structural state. A finite element model that does not simulate these intermediate phases will fail to capture the realistic structural behavior.
- **Time-dependent material behavior:**
  This is especially important for concrete structures, where the material does not achieve its final strength and stiffness immediately.

Simulation of contruction stages in finite element analysis

To model the real construction process more accurately, a **stage-by-stage analysis** should be implemented. In this approach:

- structural elements are activated, inactivated, strengthened or **weakened according to the real construction sequence**,
- loads, such as self-weight, are introduced **gradually over stages**,
- intermediate deformations and changes in the structural system are considered.

By simulating construction stages, we ensure that the design accounts for real loading and deformation conditions, leading to a more accurate and reliable structural analysis.
The module can be effectively used to model entire construction processes, starting e.g. from the erection of elements like analysis of a steel truss roof from lifting by crane to as it operates in the final state of the structure in its final position. Additionally, its versatility makes it applicable for a wide range of scenarios beyond typical construction simulations, including:

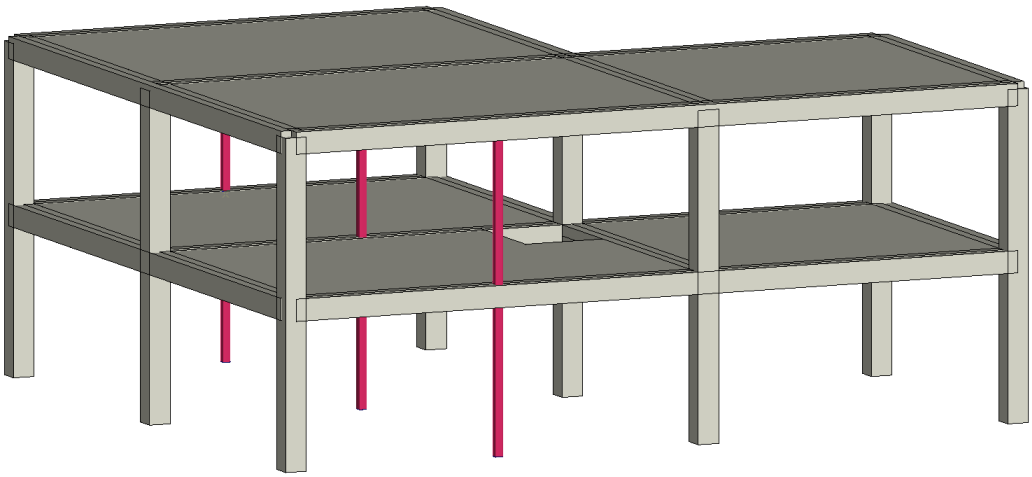
- **Modeling demolition processes:**
  The module can also simulate the step-by-step demolition of buildings and other structures, capturing the changing load paths and structural behavior as elements are removed. This is particularly useful for ensuring the safety and stability of the remaining structure during partial demolition.
- **Analysis of partially demolished structures:**
  In cases where structures are only partially dismantled, the module allows for the structural analysis of the remaining framework to assess its load-bearing capacity and ensure stability under the applied loads.
- **Verification of structural strengthening:**
  The module is well-suited for validating structural strengthening (e.g., steel plates welded to steel sections, additional RC columns added to strengthen masonry walls or replacing weak masonry columns, etc.) and evaluating the resulting changes in structural behavior.

- **Modeling structural damage:**
  The module is also capable of analyzing damaged or failed structures. By simulating the effects of deterioration, material failure, or accidental damage, it enables engineers to assess the extent of the damage, the residual load-bearing capacity of the structure, and the redistribution of forces within the damaged system. This is crucial for planning repairs, retrofitting, or even determining whether a damaged structure can remain in service.
- **Robustness analysis:**
  One of the most advanced applications of the module is the robustness analysis of structures. By selectively deactivating critical elements, the model can simulate scenarios where key structural components fail and assess the resulting structural response. This approach enables engineers to determine whether the structure, with its altered load path and changed structural framework, can withstand the applied loads without collapse.
    - Simulating key element removal: The module can identify how the loss of one or more critical elements impacts the global stability and redistributes forces within the structure.
    - Resilience evaluation: It helps evaluate the structure's ability to remain functional or at least stable under such scenarios, offering insights into progressive collapse prevention and the design of redundant load paths.
- **Bridge launching process:**
  In cases involving bridge launching (incremental bridge construction), construction stages are particularly critical. The bridge structure is moved incrementally into its final position, and during this process, the load paths, support conditions, and resulting stresses change dynamically. Each intermediate position of the bridge may create temporary stresses and deformations that do not exist in the final state. Ignoring these effects can lead to incorrect assessments of the structural performance, especially at sections with temporary support conditions.

By leveraging these capabilities, the module serves as a comprehensive tool for simulating and analyzing a variety of complex structural processes, from construction and demolition to strengthening, damage evaluation, and robustness assessment. This ensures safer, more reliable designs and improved performance under both expected and unexpected conditions.

4.13.1 General overview of the concept of construction stages

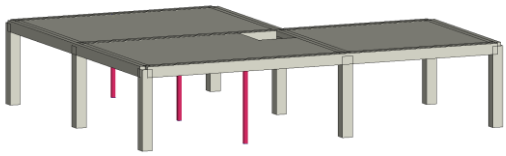
This section gives a general and practical overview about the terms and the implementation of construction stage modeling in AXISVM.
Most importantly, there are no parallel models for each construction stage edited and managed separately. The first step is to create the so-called master model. It needs to be created the same way as a simple final state model in normal situations, but the master model must include all structural elements created during the construction process regardless of whether they are built temporarily or permanently, and all the loads applied to the structure temporarily or permanently during the construction and in the final state.
The following series of images help to understand the concept of a master model.



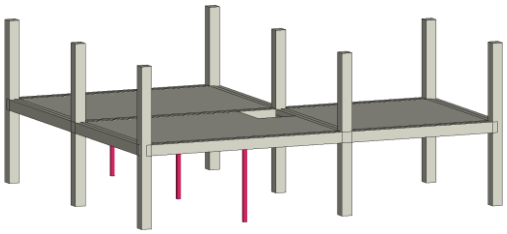
Master model



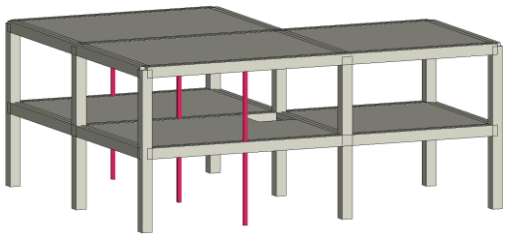
1st construction stage



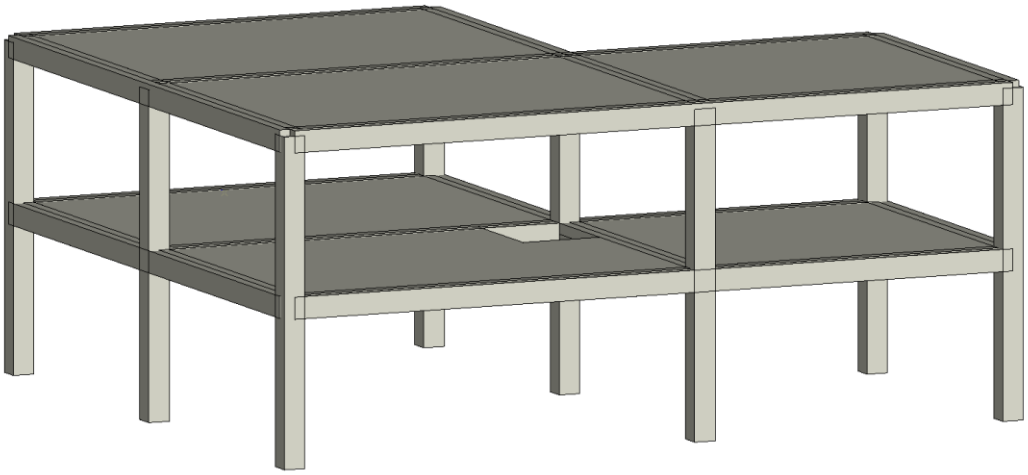
2nd construction stage



3rd construction stage



4th construction stage



5th construction stage (final state)

Construction stages are similar to user-defined parts (See **2.16.17 Parts**) as they group together the elements active in a given state of construction. Some element properties may be modified in stages as described in the next sections.

See 4.13.2 Defining stages and 4.13.3 Editing stages
Due to this logic if we delete an element on the *Elements* tab, it is deleted from the master model, therefore it will disappear from every construction stage, because individual stages are created by activating, inactivating or modifying elements of the master model.
Newly defined elements of the master model (defined in a dialog, drawn directly or created as a copy of an existing element) do not appear in any of the stages. They must be added to a stage in a separate operation on the *Construction stages* tab.
It is recommended to build the master model first including all temporary and permanent structural elements and all temporary or permanent loads. The next steps are: 1) defining stages 2) setting which elements are present in each stage 3) setting which load cases are present in each stage.
In relation to the construction stages, we distinguish permanent and temporary loads. We can have multiple load cases or load groups assigned to the construction stages. This process is discussed later, see 4.13.5 Defining stage loads. The difference between permanent and temporary loads is that permanent loads (e.g. self-weight) are calculated gradually and their effect summarized linearly over stages. In contrast, temporary loads may be active only in a specific stages. Both dead and live loads may be considered as temporary loads (e.g. weight of stored construction materials, live load caused by workers and their tools).

Final state

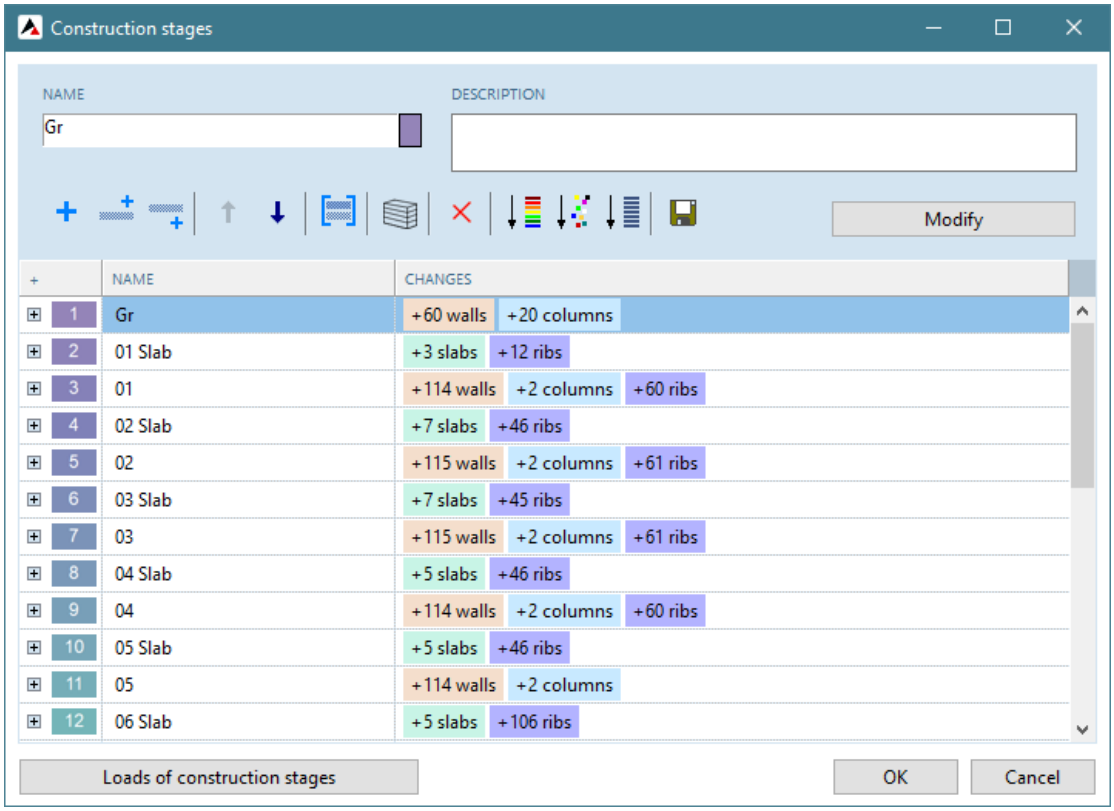
Another important term is the final state of the model.
The elements present in the final state are the same as in the last stage. However the loads applied are very different. The loads applied in the last stage are the permanent and temporary loads assigned to that stage. But in the final state all load groups and load cases defined in the master model will be applied to the structure, except for those assigned to the stages as temporary load cases.

☞ **Certain load types, like meteorological loads or seismic loads can be applied only to the final state (to do so they must be defined in the master model).**
Certain design modules work only on the final state.
In stages only the linear analysis is available.

The following sections will discuss the definition of stages, activation, inactivation of elements, modifying element properties and assigning load cases to stages.
For further details see 5.1.1 Construction stage analysis – STG module and 6.1.31 Results of construction stages regarding the analysis, results and design modules.

4.13.2 Defining stages





This dialog is to define, generate, rename or delete construction stages and their load cases. It does not assign elements from the master model to construction stages. You can simply type names into the *Name* field and press Enter or click on the blue plus sign icon. This way you can create several consecutive stages. Click on the color sample at the right edge of the edit field to change the color assigned to the next stage you add. Fill out the *Description* field to help others to interpret the stage.
To change the name, color, or description of an existing stage, select it, edit the respective field or click on the color sample to change the color, then click on the *Modify* button.

The functions of toolbar buttons:



Add a new construction stage to the list
Creates a new construction stage with properties according to the fields of the upper panel.



Insert a new construction stage before the selected stage
Inserts a new construction stage with properties according to the fields of the upper panel *before* the selected stage.



Insert a new construction stage after the selected stage
Inserts a new construction stage with properties according to the fields of the upper panel *after* the selected stage.



Bring forward
Brings forward all operations in the selected construction stage by one stage.



Postpone
Postpones all operations in the selected construction stage by one stage.



Merge construction stage with the next one
Merges operations of the selected construction stage and the next stage.



Generate construction stages from storeys defined in the master model
If the master model contains storeys, this function generates construction stages based on that information. The process may not provide a satisfactory final result in all situations, but it can speed up the work by separating the elements and creating stages automatically.
If the option *Slabs in separate construction stages* is checked in the warning dialog the algorithm finds slabs and ribs or edge supports attached to the slabs and creates a construction stage named *Storey name, Slab*. Then it creates another stage named *Storey name* for all other elements of the storey.
If this option is unchecked the algorithm creates one stage per storey.



Delete construction stage
Deletes the selected construction stage. If elements have been added to the stage (i.e. they are to be built in that stage), these elements will also be removed from all consecutive stages.



Color gradient
A color gradient can be selected for the stage colors.



Random colors
Assigns random colors to the stages.



Set a common color
Assigns the same color to all stages.



Exporting a stage as a separate model

Two options are available:

Only this construction stage:
There will be no construction stages in the exported model. Only elements and load cases present in that stage will be exported.

With all preceding stages
The exported model will have the same master model, same construction stages with the same elements and load cases until (and including) the selected stage.

Loads of construction stages

This button opens the dialog to assign load cases or groups to construction stages. This function is also available from the toolbar or the *Construction stages* tab. See **4.13.5 Defining stage loads**
Stages are displayed as tree items. Expanding the tree at a construction stage reveals additional information about the permanent and temporary loads assigned to stages.

The first stage can be optionally set as an *Initial state*. It means that this stage is used to calculate an initial state of the model, where we can decide to keep only the strain or only the stress in elements. It is useful when a soil model was built in the master model and we would like to determine the initial stress distribution in the soil (choosing *Only stress*) without taking into account the strain, the deformation of the soil as in the initial state the soil has already gone through all deformations and is considered to be undeformed.

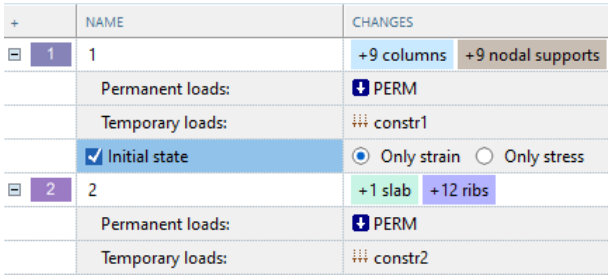


Table A summary table of the construction stages is available in the Table Browser.

4.13.3 Editing stages

Build selected elements in this construction stage
Adds the selected elements to this construction stage. Elements can be selected from the master model by clicking on the asterisk icon of the toolbar activating the display of the entire master model.
The selected elements of the master model are to be built in this stage and will continue to exist unless they are explicitly removed in a later stage.

Remove selected elements in this construction stage
Removes selected elements from this construction stage and from all subsequent stages. Elements can be selected from the master model by clicking on the asterisk icon of the toolbar activating the display of the entire master model or from any other display mode described in the next chapter.

Bring forward the construction of selected elements by one stage
If an element was built in the *N*th stage, it will be added to *(N–1)*th stage.

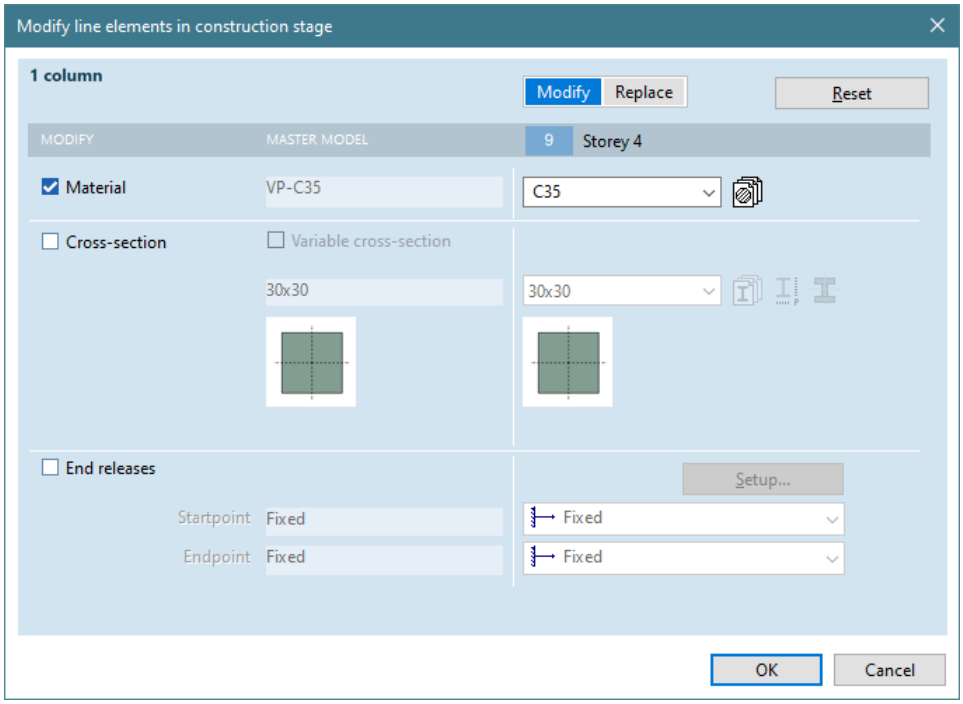
Postpone the construction of selected elements by one stage
If an element was built in the *N*th stage, it will be removed from that stage and added to *(N+1)*th stage.

Modify selected line elements from this stage onwards
Allows changing material, cross-section, and end releases of the selected line elements. Only constant cross-sections can be chosen. Changes will affect the current stage and all consecutive stages, but the element can be modified again or it can be *Reset* to the properties defined in the master model in a later stage.
The stresses in the cross-section need to be calculated in each stage and summarized similarly to internal forces repsectively to the stress points of the cross-section. *Modify* and *Replace* settings are introduced to control stress calculation.

Modify Stress values from a previous stage are summarized with stress values calculated in present stage.

Replace As the element is replaced (i.e. removed and a new element with different properties is inserted) the stress values from the previous stage are reset to zero.

For further information about the modification and replacement of elements, see **5.1.1 Construction stage analysis – STG module**



☞ **If the selection contains ribs, they can be activated or deactivated in a stage (which deactivates them in the subsequent stages as well). It has the same effect as constructing / removing the rib. Constructing / removing standalone line elements makes the geometric line appear / disappear, but this does not apply to ribs attached to domains where the domain edge remains part of the stage even if the rib is not present.**

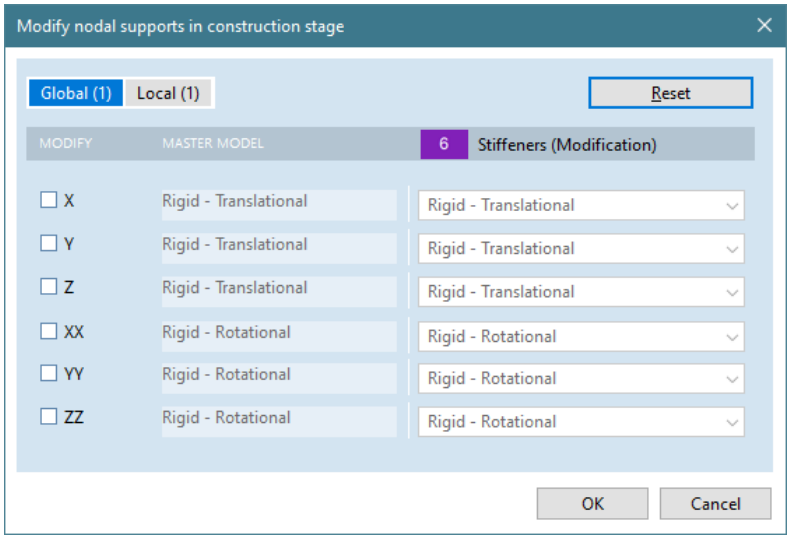




Modify selected domains from this stage onwards
Allows changing material or thickness of a domain the same way as in the case of line elements.



Modify selected nodal supports from this stage onwards
Allows changing stiffness of nodal supports. If stiffness components of selected supports were defined in different systems (in the global or a local coordinate system), these values can be set separately for global and local components.





Modify selected line supports or elastic foundation of beams and ribs from this stage onwards
Allows changing stiffness of line supports. If stiffness components of selected supports were defined in different systems (in the global or a local coordinate system), these values can be set separately for global and local components.

Modify selected surface supports from this stage onwards
Allows changing stiffness of surface supports the same way as in the case of line supports.

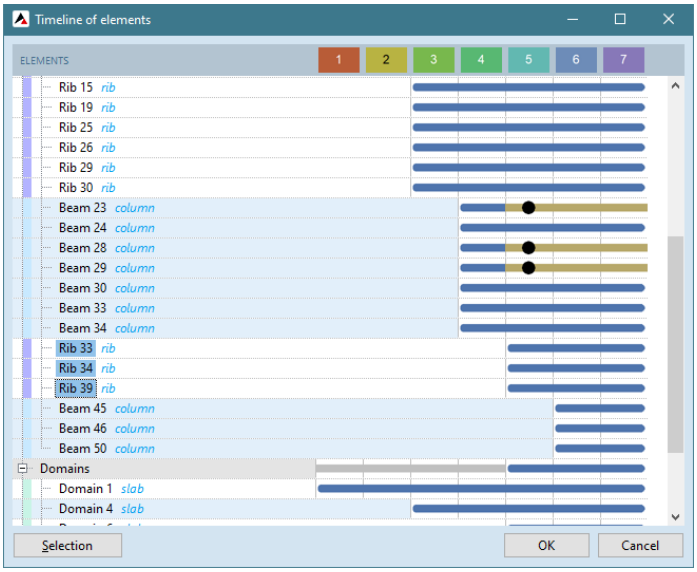
Modify selected edge hinges from this stage onwards
Allows changing spring characteristics of edge hinges the same way as in the case of nodal supports.

Modify selected line-to-line link elements from this stage onwards
Allows changing stiffness of line-to-line link elements the same way as in the case of line supports.

Reset selected elements to master model properties from this construction stage onwards
Restores the properties specified in the master model on the selected elements.

☞ **If some elements are built while others are removed or modified in one stage that stage is not properly defined as the outcome of these changes may highly depend on their order of application. Therefore, these complex stages will be broken apart into stages with only one operation so that their actual order could be set. For more information see chapter 5.1Static analysis.**

Timeline of elements
Displays a timeline for the selected elements.
This table is to check when the elements are constructed (where the blue line starts), removed (if the line ends before the last stage) or modified (golden line, with a black dot in the stage where the modification was introduced).
This information can also be checked in the tooltip of elements if you are on the *Construction stages* tab.
Multiple element names can be selected in the list. Click on the *Selection* button to select these elements in the active view. All other selections will be removed.



4.13.4 Display options for stages



Elements present in the construction stage
Displays all elements that are constructed in the current stage or have been constructed in one of the preceding stages (and not removed).

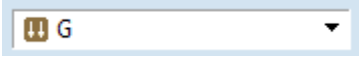
Elements built in the construction stage
Displays only the elements that are constructed in the current stage.

Elements modified in the construction stage
Displays only the elements modified in the current stage.

Elements removed in the construction stage
Displays only the elements removed in the current stage.

Master mode
Displays the master modell, making it possible to select elements that should be built in a certain stage.

4.13.5 Defining stage loads



Assigning permanent and temporary load cases to construction stages can be done by clicking on cells of a table. The row represents a load case or load group, the column represents a construction stage. Choose the load type *(Permanent loads* or *Temporary loads)*.

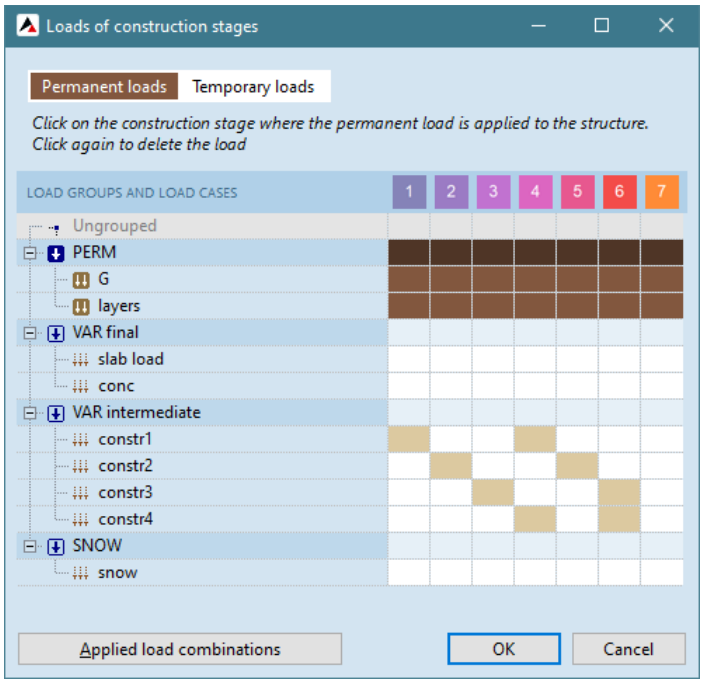
Permitted load types

Only static, moving and self-weight load cases can be assigned to construction stages. Other load cases are calculated only on the final state model (e.g. meteorological and seismic loads).

☞ **Exceptional load groups and their load cases are listed only when *Temporary loads* is selected.**

Permanent loads

Click on the construction stage where the permanent load is applied to the structure. Rows that have already been assigned as temporary loads cannot be clicked on. Clicking on a cell activates the load case in all subsequent stages, or, if the cell was activated, it will be inactivated in all subsequent stages.
Clicking in a row representing a load group activates all load cases in the group in all subsequent stages. In this mode load case rows cannot be set separately.



Temporary loads

Click on the construction stage where the temporary load is applied to the structure. Rows that have already been assigned as permanent loads cannot be clicked on. Clicking on a cell activates the load case in the given stage only. Clicking on an activated cell deactiveates the load case.
Clicking in a row representing a load group activates all load cases in the group in the given stage. In this mode load case rows cannot be set separately.

Post-tensioning loads

If the member and the tensioning load are active, only the T0 load case will be calculated in construction stages. In the final state only the TI load case is calculated.

Applied load combinations

Click on this button to control which load combinations should be applied in construction stages.
Three options are available:

None.
No load combinations will be taken into account.

Where all load cases of the load combination are applied in the construction stage.
Only load combinations where all load cases are present in the construction stage are applied.

Where at least one load case of the load combination is applied in the construction stage.
Load combinations are applied if at least one of their load cases is present in the construction stage. Load cases not applied in that stage will have a zero contribution.

Critical combinations and envelopes

Critical combinations and envelopes are also available for construction stage results.
Critical combinations in stages include only the permanent and temporary load cases that are attached to the selected construction stage.
Load cases, that were used as temporary construction loads in stages, are not present in the final state and not included in critical combinations.

The envelopes work according to the same logic as they do in the case of a regular model. Only the active load cases will be considered.
Results of the construction stages and the final state take into account only the load cases applicable during the construction stage and in the final state.

Envelope of critical combinations of construction stages

AXISVM uses the critical combination results for each construction stage and the final state to generate an envelope of these, which can be regarded as the design result for the construction process as a whole.

5. Analysis

AXISVM lets you perform linear and nonlinear static, linear and nonlinear dynamic, vibration and buckling analysis. It implements an object-oriented architecture for the finite element method.
The instructions included in this User's Manual assume a preliminary knowledge of the finite element method and experience in modeling. Note that the finite element analysis is only a tool, not a replacement for engineering judgment.

Each analysis consists of the following steps:

1) Model optimization
2) Model verification
3) Performing the analysis
4) Result file generation

Details of the analysis can be displayed by expanding one or more category panels. The *Messages* panel shows the analysis message log. The *Statistics* panel shows memory requirements, hardware information, model details and calculation times.

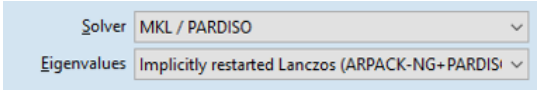
Special categories:

Nonlinear analysis: *Tracking* displays the movements of the tracked node. *Convergence* shows the convergence of the iteration process.
Vibration analysis: *Frequencies* displays how the frequencies converge. *Convergence* shows the convergence process.
Buckling analysis: *Eigenvalues* displays how the eigenvalues converge, *Convergence* shows the convergence process.
Dynamic analysis: *Time steps* display the movement of the tracked node, *Convergence* shows the convergence process.



Parameters of the latest analysis is saved into the model file and can be studied in the Model Info dialog. See 2.16.23 Model info

The solver engine can be selected in *Settings / Preferences / Analysis*. See 3.3.11.14 Analysis



When using the highly optimised MKL solver, the calculation runs with minimal preparation. The description below outlines only how the standard solver works:

Model optimization

To reduce analysis time and memory footprint AXISVM optimizes node order. If the total number of degrees of freedom is over 1000, it creates an internal three-dimensional graph from the model geometry and begins to partition the system of equations using the substructure method. The system is stored as a sparse matrix. The parameters of the optimized system of equations appear only at the end of this process. This process results in the smallest memory footprint and fastest calculation time but it assumes that the biggest block fits into the available memory. If it doesn't, AXISVM stores the system as a band matrix and begins to reduce the bandwidth of the system by iterative node renumbering. If the two longest rows fit into the available memory the system can be solved. Changes in the memory requirements for the band matrix is displayed real-time. The duration of the optimization process and the final memory footprint depends on the size of the system and the available memory.

☞ **The system of equations can be solved the most efficiently if the whole system fits into the physical memory. If the system does not fit into the physical memory but its largest block does, the running time will be moderate.**
If the largest block does not fit into the physical memory, the necessary disk operations can slow down the solution considerably.

Model verification

The input data is verified in the first step. If an Error is found a warning message is displayed and you can then decide whether to cancel or continue the analysis

Performing the analysis

AXISVM displays the evolution of the solution process by two progress bars. The bar on the top displays the current step performed, while the other displays the overall progress of the analysis process.

The equilibrium equations in the direction of constrained degrees of freedom are not included in the system of equations. Therefore to obtain support reactions you must model the support conditions using support elements.

The Cholesky method is applied to the solution of linear equilibrium equations.
The eigenvalue problems are solved with the Subspace Iteration method.

Error of the solution

Solution error is calculated from the solution of a load case with a known result. It is a good estimation of the order of errors in displacement results for other load cases.
Info palette shows this error as E(EQ).
If the value of E (Eq) is greater than 1E-06 the reliability of the computed results is questionable. It is expected, that the Error of the displacements is of the same order.

Result file generation

During the processing of the results the program sorts the results according to the original order of the nodes and prepares them to graphical display.

In the following chapters we 'll show the setting of the parameters of the each calculation methods.

5.1 Static analysis

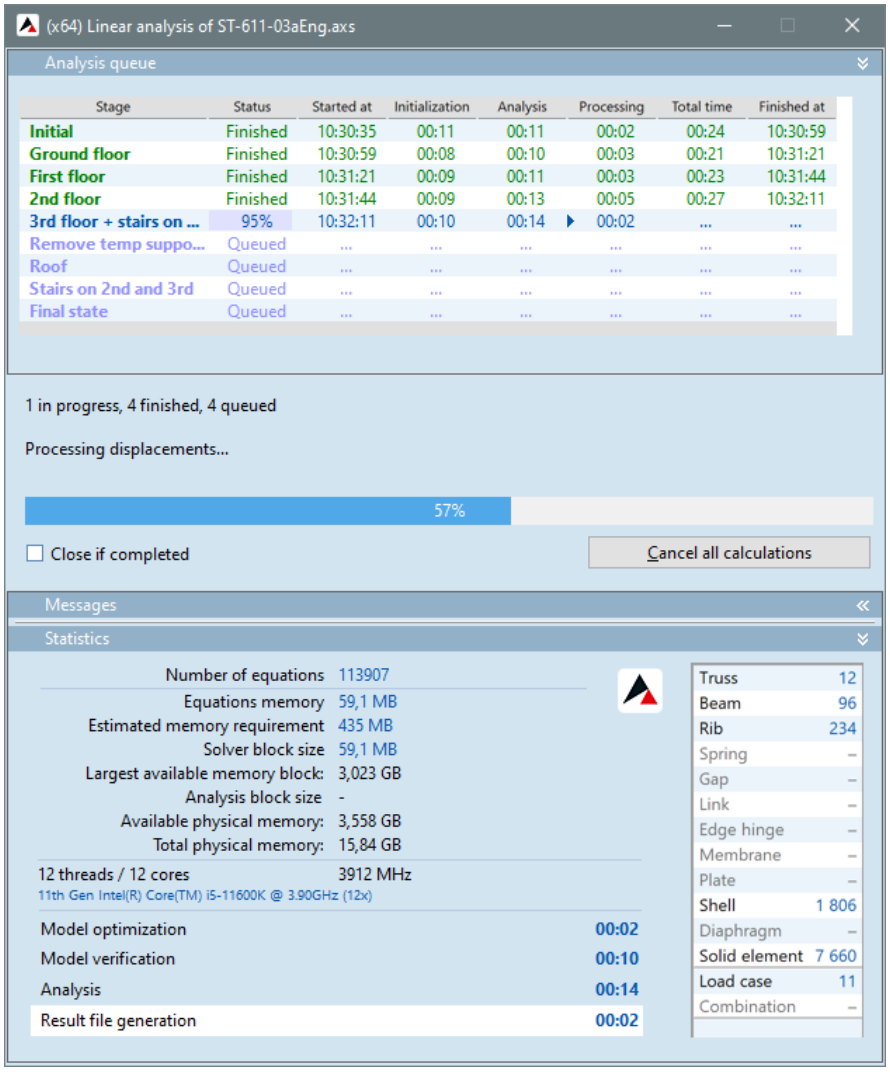
The term *static* means that the load does not vary or the variation with the time can be safely ignored.

Linear static

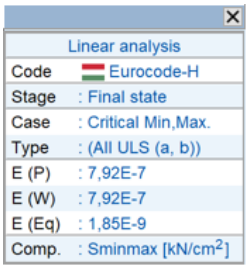


Performs a linear static analysis. The term *linear* means that the computed response (displacement, internal force) is linearly related to the applied load.
All the load cases are solved in the analysis. Through the geometric linearity, it is assumed that the displacements remain within the limits of the small displacement theory. Through the material linearity, it is assumed that all materials and stiffness characteristics are linear-elastic. The materials assigned to surface elements can be othotropic.

☞ **See the description of the truss, gap, support, seismic isolator and spring elements in Chapter 4, on how to use these elements in a linear analysis.**



The relative errors at the end of the iteration process appear in the info window.

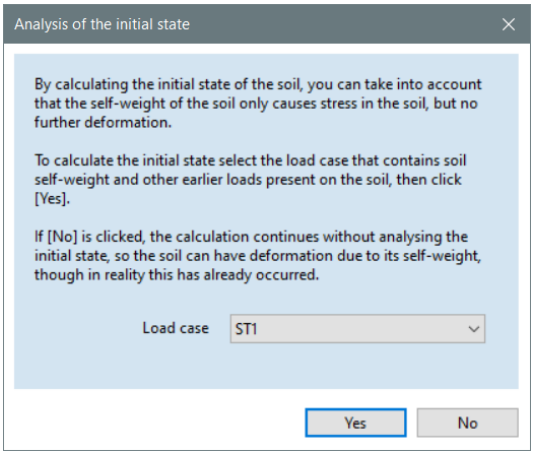


E(P): relative error of the force convergence
E(W): relative error of the work convergence
E(Eq): condition of the system of equations

Values indicating instability appear in red.

Initial state of soil modeling domain

AXISVM provides the possibility to take into account the initial state of soil (the soil is under stress due to its self-weight and other prior loads present on the soil, but the deformation is supposed to have already occurred and is considered to be zero). This function is available if at least one soil modeling domain is defined (requires SOIL module). See **4.9.9 Soil modeling domain – SOIL module)**.

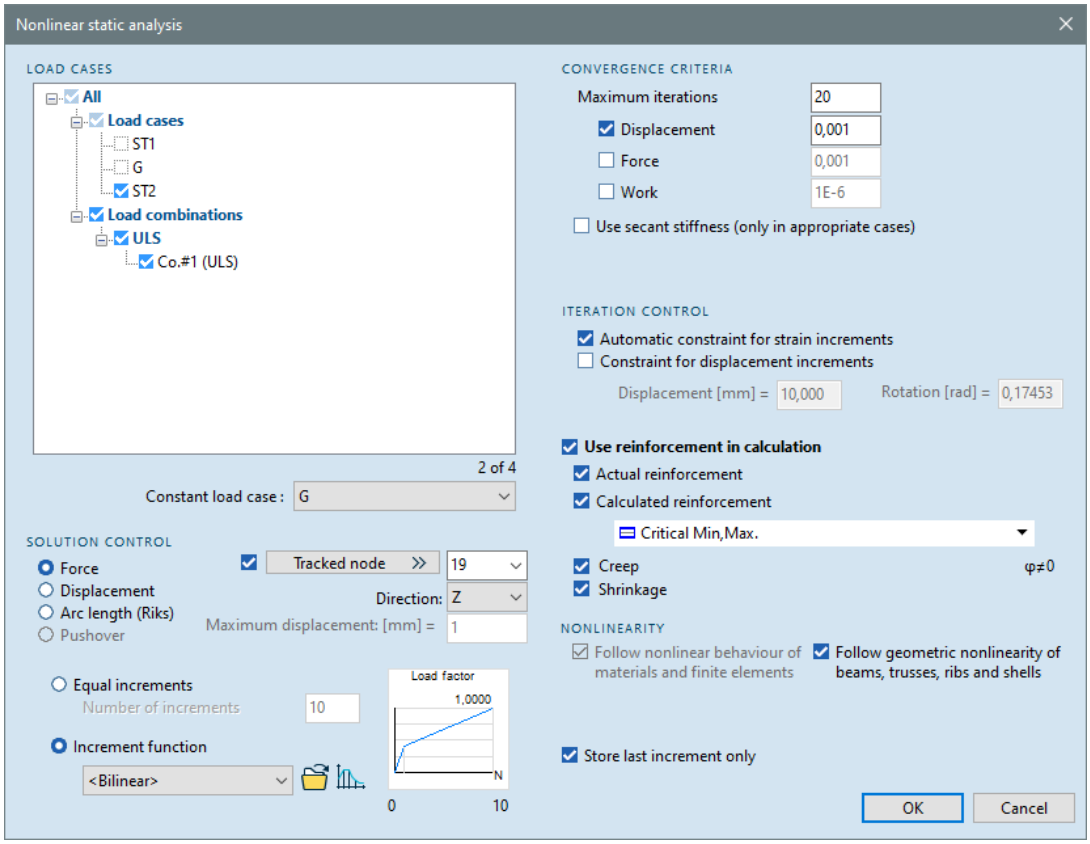


If the model contains construction stages defined (STG module is required), the calculation of the initial state of soil is automatic.
If the model does not have construction stages, but has a soil modeling domain, a dialog box pops up before the analysis.
A load case must be selected, which contains the self-weight of the soil and any other earlier loads acting directly on the soil modeling domain in the initial state.

Nonlinear static



Performs a nonlinear static analysis. The term *nonlinear* means that the computed response (displacement, internal force) is nonlinearly related to the applied load. This can be due to the use of e.g.,gap, link, non-linear support elements, line/surface elements with nonlinear metarial, or taking into account the geometric nonlinearity.
Select load cases or combinations in the tree view.



AXISVM will perform nonlinear analysis for the selected load cases / combinations and shows a progress dialog.

Constant load case

In case of force- and displacement-control the use of a constant load case is also allowed. The loads contained in this load case are applied with the first increment of the parametric load case, immediately with their full intensity. Although pushover load cases can also be selected for increasing load cases, these types of analyses do not provide pushover results. Those are available only with pushover-controlled analysis as described below.

☞ **If the model has construction stages, nonlinear analysis is always performed on the final state of the model.**
See 4.13.1 General overview of the concept of construction stages

Solution control

Force

When Force control is selected, the increments are applied as fractions of the loads (as one parameter load). It is possible to track the displacement of a node in a given direction. A graph of this displacement versus increments will be plotted during the analysis.

Displacement

When displacement control is selected, the increments are applied as fractions of the displacement component of the node specified.

Arc length (Riks)

When arc length control is selected, the increments are applied as fractions of the arc length on the force-displacement diagram of the selected degree of freedom of the selected node. This results in more displacement in the flatter parts of the characteristic and more force-like increments in the steeper parts. The target value must be defined as a displacement. The displacement in the first increment and the initial stiffness of the structure determines an initial arc length. AXISVM tries to keep the arc length close to this initial value in the subsequent increments using the algorithm proposed by Riks [44].

Pushover

Pushover control is a special type of displacement control that allows the use of a constant load case while having another parametric load case that is increased incrementally. This is essential for pushover analyses to model P-Δ effects appropriately.

After selecting pushover control, the top of the dialog changes to accommodate the drop-down boxes for parametric and constant load cases. See **4.10.26 Pushover loads – SE2 module** for details on load definition and recommended analysis settings.

Tracked node, Direction, maximum displacement

In case of displacement or pushover control a control node and a degree of freedom must be selected. Maximum displacement is the maximum allowed displacement of the control node in the given direction.

Number of increments

There are two methods to define the number of increments:

1. *Equal increments*. Specify the number of increments. The default value is 10. When highly nonlinear behavior is analyzed, you may specify a greater value in order to achieve convergence.
2. *Increment function*. Loads are not increasing in a linear way but follow a predefined function. Using an increment function it is possible to reduce the number of increments where the behaviour of the structure is linear and increase the number of increments where the behaviour is nonlinear. Saved functions can be reloaded, edited and saved under a new name (see **4.10.31 Dynamic loads (for time-history analysis) – DYN module).** Functions are saved into separate **.inc* files in folder *c:\ Users \ [user name] \ AppData \ Roaming \ AXISVM \ [version number] \ inc*.

Convergence criteria

Based on the convergence tolerances you specify, AXISVM will determine if the nonlinear solution has reached the required accuracy (convergence). Therefore it is important that the convergence tolerances to be set properly. During the iteration process, the norm of the unequilibrated load and/or of the iterational displacement increment vector must vanish (to approach zero).

Maximum iterations

You can set the maximum number of the iterations based on the specifics of your model, and of the incremental solution parameters. By default the value is set to 20. If the convergence is not achieved within the maximum number of iterations, no results will be obtained.

Displacement / Force / Work / Convergence criteria

In case of a nonlinear calculation, you can specify multiple criteria, in terms of load, displacement, and work, for monitoring the convergence of the nonlinear solution. At least one criteria has to be selected. The criteria expressed in terms of work can be adequate for most problems. However, you may encounter a small Error in your unequilibrated load while the Error in displacements is still large, or vice-versa.

Factors of convergence criteria has the following default values: 0.001 for displacements, 0.001 for force, and 0.000001 for work.

The relative errors at the end of the iteration process appear in the info window.

E(U): relative error of the displacement convergence
E(P): relative error of the force convergence
E(W): relative error of the work convergence
E(EQ): condition of the system of equations

Use secant stiffness (in appropriate cases only)

If this option is selected beam end releases will be represented by their secant stiffness instead of tangent stiffness. This improves convergence but considerably slows down the calculation. It is recommended only if convergence cannot be achieved by increasing number of increments and iterations.

Iteration control

The weak convergence of a nonlinear calculation is often caused by nonlinear elements that have sharp changing of stiffness in their stress-strain characteristic. Examples of such elements are gaps, only tension or only compression trusses, supports having limit force and reinforced concrete structural elements. If the stiffness of several elements has to be change in the same increment, infinite loop of iterations can occur without reaching the convergence. To avoid this situation iteration control can be used. The implemented control methods constrain the size of the displacement increments in the critical iterations instead of decreasing the size of the load increment. With this strategy the convergence can be achieved but a lot of iterations would be necessary in the critical increment, because of the constrained displacement increments.

The program performs the following calculation steps in each iteration:

1. Calculates the displacement increment for each DOF caused by the unequilibrated loads.
2. Controls the DOF displacement increments if any of the control options is turned on.
3. Calculates further results...

The iteration control contains the following checks depending on the options selected.

Automatic constraint for strain increments

On the basis of the DOF displacement increments the program calculates the elemental strain increments. If excessive strain or stress increment or excessive changing in the stiffness occures in any of the elements, the program calculates an allowed strain increment that does not cause any of the opreceding problems. After cheking all elements the program scales (decreases) the displacement increments so that excessive changing does not occur in any of the elements.

Constraint for displacement increments

With this option custom limit values can be defined for translational and rotational displacements. The program scales (decreases if necessary) the DOF displacement increments so that the maximum of them should be equal to the limit.

Use reinforcement in calculation

When analysing reinforced concrete plates, it is possible to take into account:

- the actual reinforcement
- or the required amount of reinforcement calculated for a load case, load combination, envelope or critical combination
- or the combination of previous two options

When considering both the calculated and actual reinforcement (both checkboxes are active), the program determines the reinforcement amounts' envelope (maximum value) separately for each direction and reinforcement position. The program does not consider if the reinforcement position is specified differently for the actual reinforcement, and in the reinforcement parameters, only the amount of reinforcement matters.
This feature is available only for surfaces/domains. Only the actual reinforcement will be taken into account for reinforced concrete beams and columns.
Displacements and internal forces of reinforced concrete plates are calculated according to the moment-curvature diagram of the reinforced cross-section of the plate. These results show the actual plate deflection and forces in the plate (see **6.5.6 Nonlinear analysis of RC surfaces**).

In case of the analysis of reinforced concrete columns and beams, it is also possible to take the reinforcement into account. Internal forces compatible with strains are calculated through the integration of fiber stresses at Gauss integration points based on ε normal strains, κ_y and κ_z curvatures considering the actual reinforcement, the concrete and nonlinear material behaviour (see **6.5.5 Nonlinear analysis of reinforced concrete beam and column elements**).

The analysis can be performed either with consideration of creep/shrinkage or without. Further information related to the consideration of creep and shrinkage can be found in the above referred sections.

Nonlinearity

Follow nonlinear behaviour of materials and finite elements

This option is enabled if the model contains elements with nonlinear behaviour (e.g., tension-only trusses, compression-only supports, surface elements lying on soil elements) or elements with nonlinear material characteristics (trusses, beams, ribs, membranes, plates, shells). If left unchecked, all elements will respond in a linear way.

Follow geometric nonlinearity of beams, trusses, ribs and shells

The equilibrium is established with respect to the deformed line elements. Depending on the magnitude of displacement second or third order analysis is performed. The loads are conservative, they keep their initial direction during the deformation. Geometric nonlinearity can be taken into account only for truss, beam, rib and shell elements. If there are no elements with nonlinear characteristic in the model this options is checked by default. If the model contains elements with nonlinear characteristic this option is left unchecked but can be activated.

Stability warning messages

In geometric nonlinear calculations a stability problem may occur in some increments. It can be a local or global loss of stability or a snap-through phenomenon. Euler's critical load may also be exceeded in any beam. In such cases the following investigation procedure is recommended

- Checking of deformed shapes
- Buckling analysis to compare the critical load parameters and the load factors in the problematic increments.

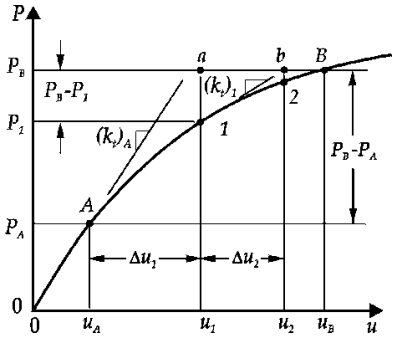
A warning message appears in the info window identifying the increments concerned.
See **2.19.2 Info window**

☞ **Beam elements must be divided into at least four parts when geometric nonlinearity is taken into account.**

Store last increment only

Allows you to reduce the size of the results file when an incremental nonlinear analysis is performed with multiple increments (load or displacement) when just the results of the last increment are of interest to you. You can enable this checkbox when you do not need the results of previous increments.

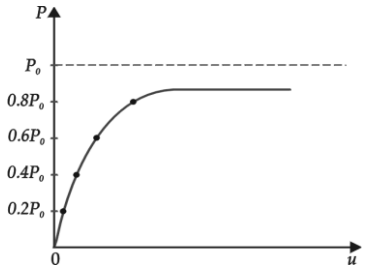
☞ **You should disable this checkbox if you want to trace the load-displacement or other (nonlinear) response of the structure.**



AXISVM applies a Newton-Raphson iteration technique to the iterational solution of each increment. The technique is known in different variants, depending on the update of the system (stiffness) matrix.
In AXISVM *n = 1* (default), the system stiffness matrix is updated in each iteration. The method is known as the classical Newton-Raphson technique.

Displacement control

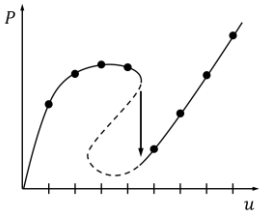
The so-called *snap-through* phenomenon cannot be analyzed with load controlled increments. You must apply a *displacement control* to pass through the peak points.
This figure shows a force control applied to a nonlinear system. The incremental solution fails in the 5th increment. To find the peak value of the load-displacement characteristics of the system, you must apply a displacement control technique.



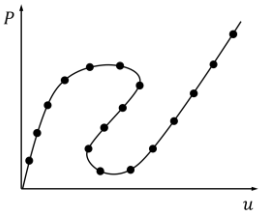
Arc length control

The so-called *snap-back* phenomenon cannot be analyzed even with displacement controlled increments. You must apply an *arc length control* to track the backward section of the characteristic. AXISVM uses the form of the arc length method proposed by Riks [44].

The figures show a force-displacement characteristic with a backward section. That part of the curve cannot be tracked with displacement control. But applying the arc length control the entire curve can be traversed with a nearly uniform step size.



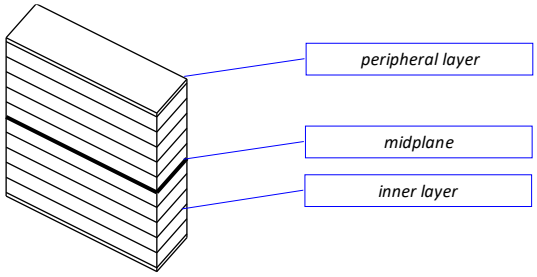
Displacement control



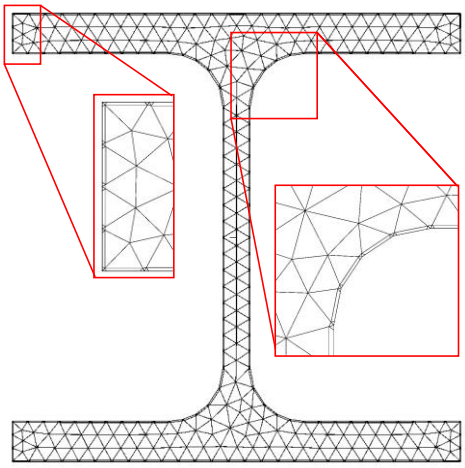
Arc length control

Finite elements with nonlinear material

Beam, rib and surface elements made of nonlinear (elastic or plastic) material are modelled with a discretized section model. Plates and shells are represented by layers, cross-section of beams and ribs are meshed. Stress distribution is determined using Navier's hypothesis (plane sections remain plane) and applying the nonlinear material model on each sub-element.



Stress components are the same as in the linear material model. For surface elements results are obtained in top, center and bottom plane, for beam and rib cross-sections in the stress points and along the outline.
To obtain accurate results in the extreme fiber the thickness of the layers is not equal. In case of plates and shells two thin peripheral layers are used, of which thickness is one tenth of an inner layer. The number of inner layers is ten. The cross-sectional mesh of beams and ribs also contains a thin peripheral layer, of which thickness is one tenth of the edge length of the inner elements. Stress results along the outline are obtained directly from the peripheral elements.



Seismic isolators

See 3.1.17.2 Seismic isolators

Compression-only connection between soil and base plate

If material nonlinearity is enabled, the connection between the soil and the surface elements lying on the soil is modeled with a compression-only connection. In the numerical model a surface-to-surface spring layer is inserted between the soil and the base plate. Its planar stiffness components *Kx* and *Ky* are linear, but its *Kz* stiffness component perpendicular to the surface is nonlinear and has a compression-only behaviour. The deformation of the spring layer in the active directions is negligible. The value of the stiffnesses:

$$K_x = K_y = 1000 \cdot \frac{E_{50}}{2(1+\mu)} \; ; \quad K_z = 1000 \cdot E_{50}$$

where E_{50} and μ are the secant modulus and the Poisson's ratio of the top soil layer. Due to the compression-only *Kz* component the surface is free to rise from the ground but the connection in the planar *Kx* and *Ky* directions still remains stiff.

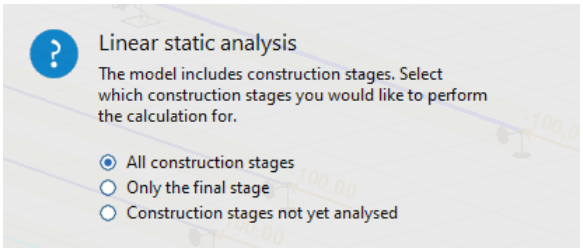
☞ **Nonlinear link layer is created only under surface elements lying on the ground. No such layer is created under elements connected to the soil along a line or at a node.**

5.1.1 Construction stage analysis – STG module

In this section, the construction stage analysis method is presented. It has to be noted that multiple different methods are available for simulation of construction processes in the literature. The STG module implements a method performing a series of linear static analyses following the construction/demolition process incrementally using different model variants and utilizing the linear superposition principle. Since it is a linear method, it has limitations in case of structures that are subjected to large deformations or characterized by inelastic behaviour.



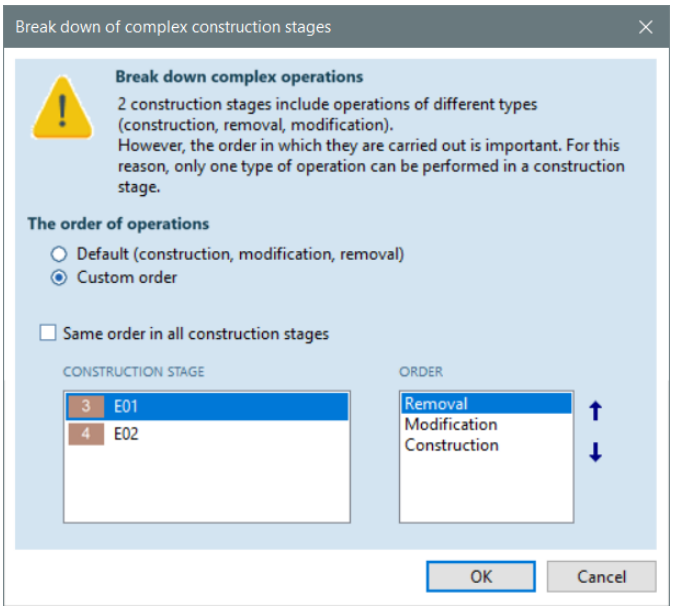
There is no separate construction stage analysis button. Click on *Linear static analysis* to perform construction stage analysis. If there are construction stages defined in the model, the following options will be displayed.



You can run an analysis for all construction stages in sequence, or a conventional static analysis on the final state. There may be construction stages that have not been analysed yet due to interruption or failure of the sequential analysis. In this case the third option is useful to obtain valid results for all stages. This may also happen in cases when parameters of a construction stage are modified, elements were built, removed or modified in a stage or a new construction stage was inserted or an existing one was deleted. In such situations, only invalid results are deleted and the software keeps the valid results of the previous stages.

Break down of complex construction stages

If the model has construction stages, and some elements are built while others are removed or modified in the same stage, that stage is not properly defined, as the outcome of these changes may highly depend on their order of application. Therefore, these complex stages must be broken apart into several stages so that there is only one operation in each one.
In such cases, a warning window pops up to set the order of operations before the analysis. The user can choose to be automatically break down of construction stages, or choose the right order for all stages or for each stage individually.

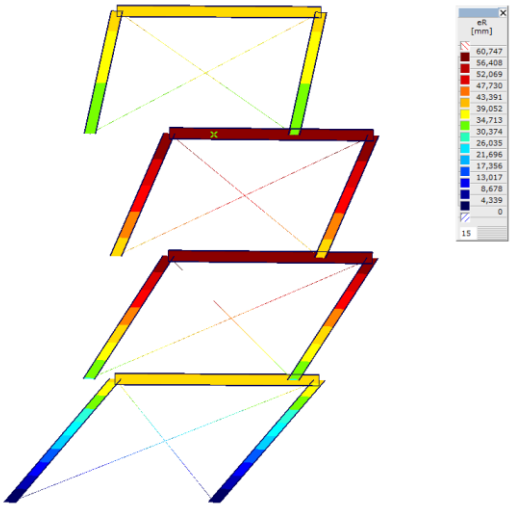


Cumulative displacements

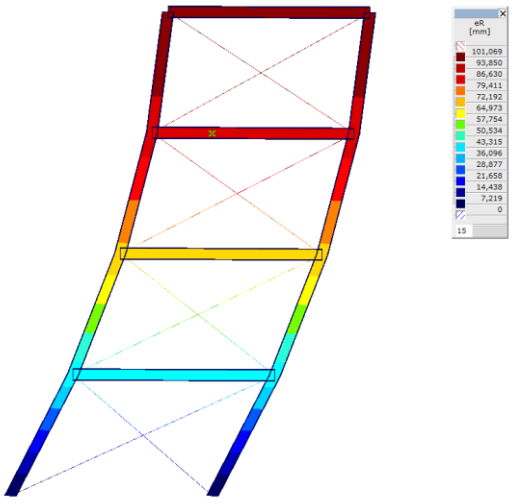
The consecutive construction stages are always analysed only for the difference in loads as the effect of previous loads are already on the structure. Then the static results are superimposed. Due to the nature of the calculation, displacement results contain discontinuities between construction stages, as the displacements obtained in each stage are relative to the deformed shape of the preceding stage. This does not always cause issues; however, in certain cases, it complicates the interpretation of results (e.g., during the construction of multi-story buildings with significant top displacements).
In order to achieve the deformed shape according to conventional engineering principles, the displacement results of a construction stage need to be aligned/cumulated. This alignment can be controlled according to three displacement degrees of freedom: X, Y, and Z displacements. AXISVM aligns the displacement results of stages only in the selected directions.



For multi-storey buildings it is recommended to enable the alignment of horizontal displacements in X and Y directions. However, it is not advisable to align the Z (vertical) displacements, as one of the purposes of analysing construction stages is to reflect the fact that the self-weight of the lower floors does not cause deformations on an upper floor as each floor is built compensating the deformed shape of the previous stage.



Displacements without cumulation of stage displacement results



Cumulative displacements in X direction

Final state

Static analyses are performed for the specified constructions stages and the so-called final state. In terms of modeling the final state is identical to the last construction stage. But the loads applied are different. Seismic loads, fire loads, SWG meteorological loads, etc. and loads not applied previously in construction stages (**4.13.5 Defining stage loads**) will be applied only in the final state, while temporary loads assigned to stages will be ignored in the final state analysis.
The results of construction stages and the final state can be displayed and listed separately (**6.1.31 Results of construction stages**)

Replacement of a structural element

The replacement of a structural element can be carried out in several ways. Perhaps the simplest example is a structural strengthening process where an insufficient masonry column is replaced with a steel column. This example illustrates different methods and the possibilities provided by the STG module.

1. The first option for replacing the column involves demolishing the masonry column with no support, allowing deformations to occur, and then installing a steel column cut to the appropriate length in its place. In most cases, like this one, this would result in unacceptable deformations. However, in certain situations, for other structural elements with a different purpose, this method might be applicable. In such a case, the following steps can be taken: starting from one construction stage, add two additional stages. In the first stage, remove the structural element (in this case, the column), i.e. deactivate it in that stage. In the second stage, reactivate it, but with the new properties.
2. The second option is to use the replacement function of the dialog modifying line elements or domains (see **4.13.3. Editing construction phases**). In this case, AXISVM performs a simplified process and only the final result is what you get. The process is: 1) A perfectly rigid bracing is created parallel with the structural element, between the connection points. 2) Keeping the braced state of the remaining structure, the structural element is removed from the model. 3) The new structural element is inserted into the structure without any gaps and prestress, while the structure is still held in the braced state. 4) Finally the bracing is released allowing further deformations to occur and the newly installed element to be loaded.
3. The third option is to temporarily support and uplift the slab in the vicinity of the column to prevent deformations during the removal of the column. This method requires *four* new construction stages, and the master model must contain the additional supports or other supportive elements as well. The modeling process is as follows: 1) In the first stage, the temporary supports are activated. 2) In the second stage, the column is demolished. The uplifting forces must be applied to the structure in this construction stage (when the column is removed). 3) In the third stage, the column is reactivated but with new section and material properties. 4) In the fourth stage, the temporary supports are removed. Alternatively, the stiffness at the upper node of the column may be reduced (e.g., by introducing hinges), ensuring that the column does not exert force against the lifting process. In this model the uplifting forces may be applied in the first stage.

The replacement of a structural element always involves removal of material under stress and addition of unstressed material. Accordingly, the actual process must be modeled in **multiple steps and construction stages** to achieve sufficiently accurate results. This step-by-step approach ensures that all intermediate states and the redistribution of forces are appropriately followed, leading to realistic and reliable results.

Modification of a structural member

Modifying a line element or a domain may involve changing material properties, cross-sectional properties, or both. As a result, the member may stiffen or soften, strengthen or weaken. The increase in material stiffness, as well as the incorporation of new material to increase the cross-sectional area, takes place without inducing stress, adapting to the deformed state of the structural element to be modified; therefore, it does not result in an increase in stress. Conversely, a reduction in material stiffness, as well as material removal modelled by a reduction in cross-section, results in the removal of stress from the removed section.

☞ **If the specified changes do not fulfil the conditions below, it is considered to be a replacement.**



In case of line elements the modification must meet these criteria:

1. The type of the material does not change.
2. The Poisson's ratio is unchanged.

If these conditions are met, AXISVM determines, for each load case, whether strengthening or weakening has occurred in that particular load case. To do this, it calculates – whilst taking into account the deformation and stress state established during the previous construction phase – whether the deformation energy associated with each load type increases or decreases solely as a result of changes in the material and cross-section. The table below shows which material and cross-sectional characteristics play a role for element types and load cases.

Internal force	*Truss*	*Beam*	*7 DOF.beam*	*Rib*
Tension	$E \cdot A$	$E \cdot A$	$E \cdot A$	$E \cdot A$
Torsion	-	$G \cdot I_x$	$G \cdot I_x$, $E \cdot I_\omega$	$G \cdot I_x$
Bending-shear	-	$E \cdot \begin{bmatrix} I_y & -I_{yz} \\ -I_{yz} & I_z \end{bmatrix}$	$E \cdot \begin{bmatrix} I_y & -I_{yz} \\ -I_{yz} & I_z \end{bmatrix}$	$E \cdot \begin{bmatrix} I_y & -I_{yz} \\ -I_{yz} & I_z \end{bmatrix}$ $G \cdot A_y$, $G \cdot A_z$

The calculation of bending and shear is carried out independently along the two principal axes in the principal direction coordinate system. If the change in second moments of area is such that the principal directions also change, the programme performs the calculation in a coordinate system aligned with their bisectors, as an approximate principal direction coordinate system. In special cases (for example, a symmetrical cross-section) or where certain characteristics remain constant (for example, symmetrical reinforcement), the relationships are simplified.
These options allow for the modelling of strengthening, jacketing as well as surface and internal corrosion effects.



In case of domains the criteria are:

1. The type of the material does not change.
2. The Poisson's ratio is unchanged.
3. The Ex and Ey moduli of elasticity and the thickness is changing in such a way that their combination has clearly either a strengthening or a weakening effect:

Strengthening: $E_{x2} \geq E_{x1}$, $E_{y2} \geq E_{y1}$, $\nu_2 \geq \nu_1$
Weakening: $E_{x2} \leq E_{x1}$, $E_{y2} \leq E_{y1}$, $\nu_2 \leq \nu_1$

These options allow modeling the hardening of concrete, the placement of a new layer of concrete, creep and both surface and internal corrosion of reinforced concrete slabs.

Modification of other elements

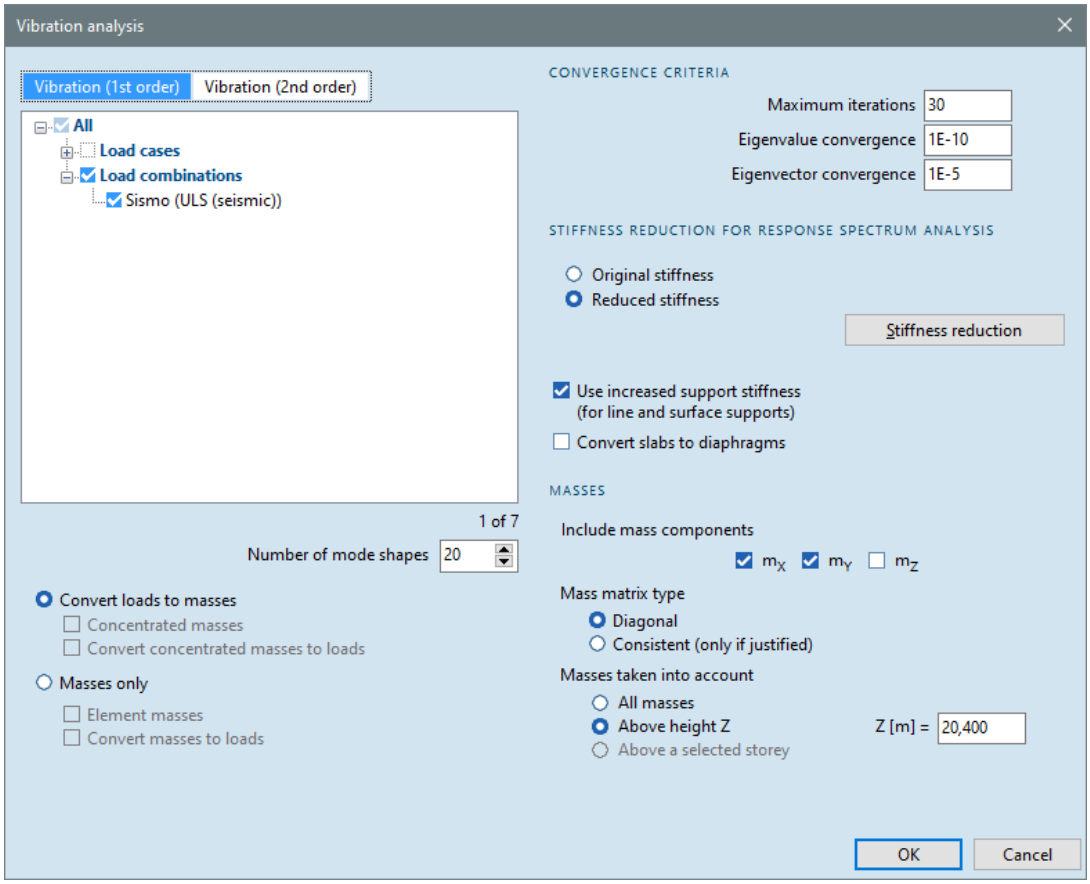
In the case of other elements (supports, line-to-line link elements, edge hinges) AXISVM determines whether changes have a strengthening or weakening effect for each component.

5.2 Vibration



Lets you determine the lowest natural frequencies and mode shapes corresponding to the free vibration of an undamped linear structure when no externally applied loads are computed. AXISVM verifies whether the required number of the lowest eigenvalues has been determined.
The system mass matrix has a diagonal structure and includes only translational mass components.
Select load cases or combinations in the tree view. AXISVM will perform vibration analysis for the selected load cases and shows a progress dialog.

☞ **The solution technique applied to the associated generalized eigenvalue problem is designed to find the lowest real and positive eigenvalues. It is not suitable to find eigenvalues that are zero or nearly zero.**



☞ **If the model has construction stages, vibration analysis is always performed on the final state of the model. See 4.13.1 General overview of the concept of construction stages**

Solution control

Lets you specify the parameters of the incremental solution process:

First order

The solution does not include the effect of axial forces of truss/beam elements on the system stiffness.

Second order

The solution include the effect of axial forces of truss/beam elements on the system stiffness. Tension axial forces have a stiffening effect, while the compression axial forces have a softening effect. These effects influence the free vibrations of the structure.

Case

Lets you select a case. The loads are converted into masses. If a second-order analysis is selected, the results of a linear (first-order) static analysis, that precedes the vibration analysis, will be accounted too.

Number of mode shapes

Lets you specify the number of the vibration mode shapes you want to evaluate. The default value is 9, maximum number of shapes is not limited. The value specified here can not be larger than the number of the system's mass degrees of freedom.

Convert loads to masses

You can enable the conversion of the load components parallel with the gravitation into masses, and take these concentrated masses into account in the vibration analysis. The generated nodal masses can be checked in the Table Browser (*Results / Vibration / [load case or combination name] / Nodal masses*).

Concentrated masses: If nodal masses have been defined (see **4.10.37 Nodal mass**) they can be included in or excluded from the analysis.

Convert concentrated masses to loads: If nodal masses have been defined they can be converted to loads in the vibration analysis.

Masses only

You can analyze models without loads, but with masses, and take *Element masses* into account. You can even *Convert nodal masses to loads.*

Convergence criteria

Based on the convergence tolerances you specify, AXISVM will determine if the calculated eigenvalues and eigenvectors have the required accuracy. Therefore it is important that the convergence tolerances be set properly.

Maximum number of iterations

You can set the maximum number of the iterations based on the specifics of your model, and the number of eigenvalues requested (more iterations for more eigenvalues). By default the value is set to 20. If the convergence is not achieved within the maximum number of iterations, no results will be obtained.

Eigenvalue convergence

Lets you specify the convergence tolerance for the eigenvalues.
The default value is 1E-10.

Eigenvector convergence

Lets you specify the convergence tolerance for the eigenvectors.
The default value is 1E-5.

Diaphragm

When running a vibration analysis with the option *Convert slabs to diaphragms* checked, all slabs (horizontal plates) will be temporarily replaced by diaphragms.
The running time is reduced if the model contains only columns and slabs. If structural walls are included, the number of equations will be reduced but the bandwidth will be increased. The resultant running time may be greater than without diaphragms.

Plates having surface support or lying on soil will not be replaced by diaphragms.

Stiffness reduction for response spectrum analysis

Further information in chapter **3.3.10 Stiffness reduction.**

Use increased support stiffness

Supports act in a different way during vibrations. Using increased support stiffness (10^7 kN/m/m for line supports, 10^7 kN/m^2 for surface supports) may help to get more realistic vibration results. For nodal support, spring and edge hinge components the vibration stiffness of the spring characteristics is always applied regardless of the state of the switch *Use increased support stiffness*. See **4.9.14 Nodal support.**

Masses

Include mass components

Only checked mass components will be used in the analysis. It is useful when calculating modal shapes only Diaphragm in a certain direction.

Mass matrix type

Diagonal: smaller mass matrix but without centrifugal intertias
Consistent (only if justified): complete mass matrix with centrifugal intertias

Masses taken into account

Here the masses taken into account during vibration analysis can be configured. Beside *All masses* it is possible to reduce the masses to those above a given *Z* height or a given level (if the model contains levels). An example could be the exclusion of the basement from the vibration analysis.

For details about the behaviour of seismic isolators see **3.1.17.2 Seismic isolators**

☞ **The program uses a diagonal mass matrix by default. Due to the lumped mass modeling technique to achieve the required accuracy the elements must be divided into more elements (by refining the mesh). Usually at least four finite elements must correspond to each half wave.**
A good rule-of-thumb is that beams must be divided into at least eight elements.

The mode shapes are normalized with respect to the mass, $\{U\}^T[M]\{U\} = 1$

5.2.1 Vibration response factor



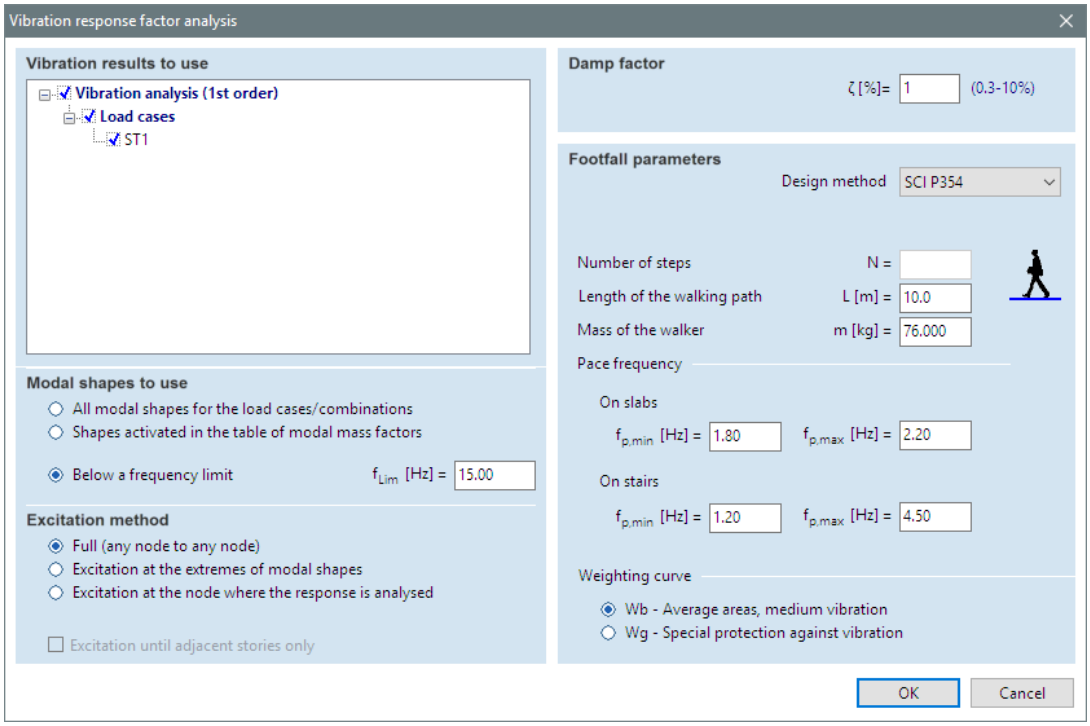
The problem of human-induced vibrations on floors is increasing with more efficient design utilizing stronger materials leading to lighter structures. Walking can cause vibration uncomfortable or unacceptable for occupants of the buildings or make delicate operations impossible (e.g., operating theatres, precesion laboratories). Footfall analysis considers walking as an external excitation force and determines the vibration response factor, which is proportinal to the maximal acceleration at the considered node. Footfall analysis is available only if the configuration includes the **FFA** module.

In order to analyse the response of a structure to human induced vibration two conditions must be fulfilled. Model must contain domains and the angle between their normal and the direction of the gravity should be less then 70°. A domain is considered to be a slab if the angle between its normal and the direction of gravity is less than 10°, if this angle is between 10° and 70° the surface is considered to be a staircase. Domains with larger angles between their normal and the gravity are considered to be walls. AXISVM does not analyse and does not excite nodes of walls as vibration of walls don't cause discomfort.

Vibration results to use

The second prerequisite is a vibration analysis with first and/or second order vibration results. Select the vibration results of load cases or combinations that should be taken into account (load cases or combinations without the appropriate number of mode shapes appear in red).

Choose an excitation method, set the damping ratio and further parameters of the dynamic load. The result of the analysis is a vibration response factor (R) for the selected excitation method on slabs and staircases. R must be checked against the recommended values (for details see the ***FFA Guide*** (see *Help / FFA Guide)*.



It is also important that the boundary and continuity conditions must be set up to reflect the stiffer behaviour of the structures for dynamic loads (see **5.2 Vibration,** *Use increased support stiffness*). Furthermore, during the calculation of the modal shapes it is suggested to take into account live loads with quasi-permanent combination (ψ_2), which influences the results favourably due to the significant increase in the structure's modal masses.

Modal shapes to use

All modal shapes for the load cases/combinations: All shapes of the selected load cases/combinations will be taken into account.

Shapes activated in the table of modal mass factors: Shapes can be made inactive in the table of *Modal mass factors* (see **4.10.25.1 Seismic load calculation according to Eurocode 8**).

Below frequency limit let the user to set a limit for the eigenfrequencies, which will appear in the calculation. Only modes with a smaller frequency will be taken into account.

Excitation method Three excitation methods are available. Each one creates a separate R result component.
Full: All nodes of slabs or staircases can excite any other node (***R full***). This is the most robust method.
Excitation at the extremes of the modal shapes: Excitation takes place only at the two global extrema of the eigenshapes (***R extr.***).
Excitation at the node where the response is analysed: The analysed node is only excited by itself (***R self***).
Excitation until adjacent storeys only: This option is enabled if the model contains at least three storeys. If checked the analysed node can only be excited with nodes on the same or adjacent storeys.

Damping The critical damping ratio of the analysed structure can be set. Suggestions can be found in [33], [34].

Footfall parameters

Design method Two approaches are available: *CCIP-016* [33] and *SCI P354* [34]. Their algorithms are described in the ***FFA Guide*** (see *Help / FFA Guide)*. Changing the approach clears the previous results.

Number of steps (CCIP-016) *The CCIP-016* design method allows resonant accelerations to be reduced due to the fact that resonance needs time to develop. So the number of footsteps (the duration of the excitation) has effect on the results.

Length of the walking path (SCI P354) SCI P354 also takes into account the same effect but the relevant parameter is the length of the walking path. In this approach the most unfavourable length should be given.

Mass of the walker Both methods allow deviation from the predefined mass of the walker.

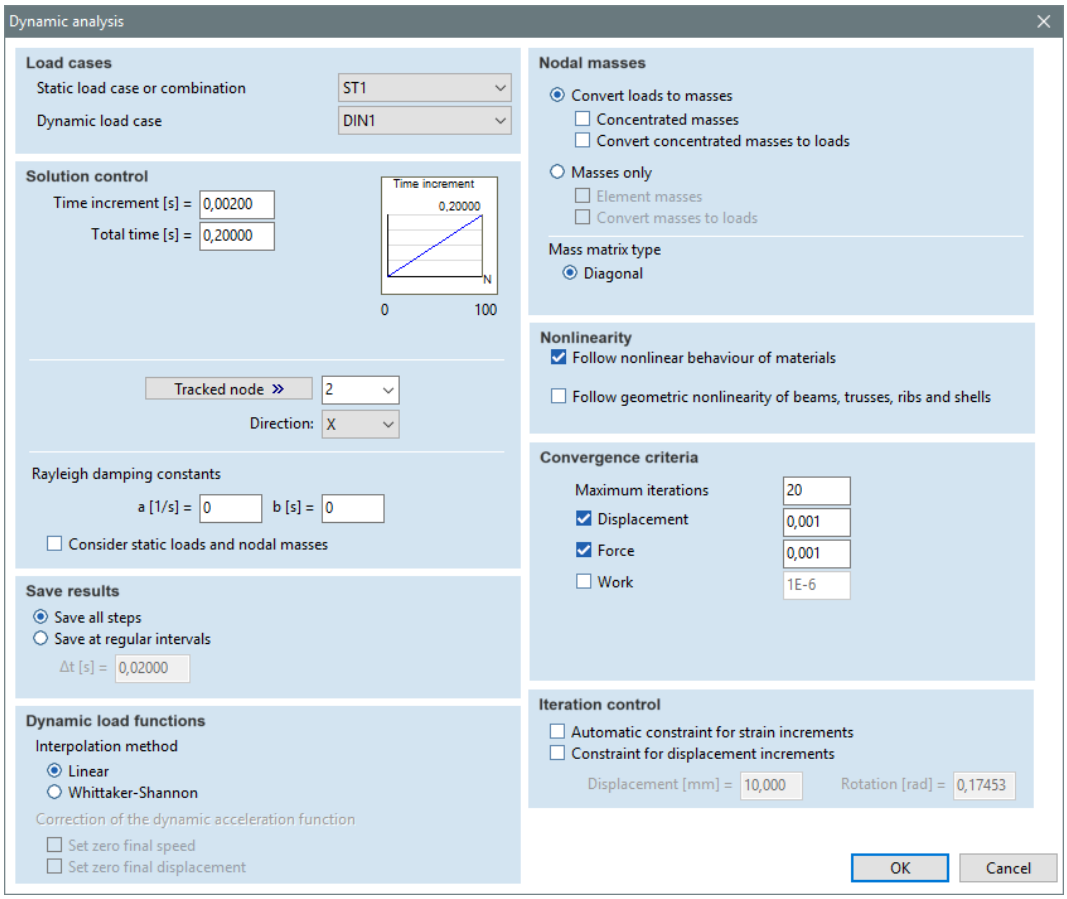
Pace frequency AXISVM analyses the effect of steps at several different pace frequencies within a given range. The lower and upper bound of the frequency range can be set here.

Weighting curve *SCI P354* allows the usage of different weighting curves. These curves take into account the fact that the human perception of vibration varies with frequency.
W_b – *Average areas, medium vibration* should be used, if general comfort must be provided in the building,
W_g – *Special protection against vibration* should be used if precise hand movements, steady vision must be ensured.
In *CCIP-016* design method only one weighting is applied.
For further details see *Help / FFA Guide*.

5.3 Dynamic analysis



Dynamic analysis determines time-dependent displacements and forces due to dynamic loads or nodal accelerations. Dynamic analysis can be performed on linear or nonlinear models where a dynamic load case has been defined and a dynamic load has been applied to the structure.



☞ **If the model has construction stages, dynamic analysis is always performed on the final state of the model. See** 4.13.1 General overview of the concept of construction stages

Load cases

Static load case or combination

Select the static load case or combination to apply during the analysis. Select 'None' to apply dynamic loads only.

Dynamic load case

Select the dynamic load case or combination.

Solution control

Analysis can be performed in equal increments or according to a custom time increment function. Predefined functions can be loaded or a new function can be created using the function editor.

If *Equal increments* is selected two parameters are required: *Time increment* and *Total time*. Analysis uses the value of *Time increment* as the increment between time steps and *Total time* defines the total time of the analysis.

Tracked node:

The displacement of the selected node in the given direction will be plotted during the analysis.

Rayleigh damping constants (a, b)

Damping matrix is determined from the damping constants, according to the following formulas:

$$\mathbf{M}\ddot{\underline{u}} + \mathbf{C}\dot{\underline{u}} + \mathbf{K}\underline{u} = P(t)$$
$$\mathbf{C} = a\mathbf{M} + b\mathbf{K}$$

If *Consider loads and nodal masses* is checked another matrix will be added to **M** representing loads and nodal masses.

Save results

Due to the considerable result file size result saving options are introduced: Checking *Save all steps* means that all result will be saved. *Save at regular intervals* saves results only at certain model time coordinates reducing file size.

Dynamic load functions

Interpolation method

The solver has to evaluate the dynamic load functions at every time step of the calculation. If there are missing sample points at any of the time steps, interpolation is necessary. This option is available only in this case. The selected interpolation method is performed for all dynamic load functions used in the selected load case. The interpolated functions are not saved into the model. They are available only for the solver. The following interpolation methods can be selected:

Linear

Simple linear sampling (see **4.10.31 Dynamic loads (for time-history analysis) – DYN module**). This method is recommended if the dynamic load functions in the selected load case are piecewise linear.

Whittaker-Shannon

This method is based on the Whittaker-Shannon interpolation formula (see **4.10.31 Dynamic loads (for time-history analysis) – DYN module**). It is recommended if the dynamic load functions in the selected load case are sampled continuous functions (e.g., measurement records or functions generated from Fourier spectral composition).

☞ **The Whittaker-Shannon interpolation method is available only for functions obtained from equidistant sampling.**

Correction of the dynamic acceleration function

The selected corrections performed for all dynamic acceleration functions belonging to the selected load case. These options are available if there are dynamic accelerations in the selected load case. The corrected functions are not saved into the model. They are available only for the solver. The correction method is described in section **4.10.31 Dynamic loads (for time-history analysis) – DYN module**

Nodal masses

Nodal masses will be taken into account like in a vibration analysis.

Mass matrix type

Dynamic analysis uses *Diagonal* matrix type.

Nonlinearity

Follow nonlinear behaviour of materials and finite elements

If nonlinear elements are defined (e.g., a tension-only truss) here you can activate or deactivate the nonlinear behaviour.

Follow geometric nonlinearity of beams, trusses, ribs and shells

If this option is activated loads will be applied to the displaced structure in each step.

Convergence criteria

If any of the nonlinearity options is checked convergence criteria has to be set and will be taken into account like in a nonlinear static analysis. Otherwise, the actual E(U), E(P) and E(W) values (their final values appear in the Info window) are compared to the reference values set here.

Iteration control

These options are described in section **5.1 Static analysis**

Solution method

Linear or nonlinear equilibrium equations are solved by the Newmark-beta method. If Δt is the time increment, in $t + \Delta t$ we get:

$$\mathbf{M}\ddot{\underline{u}}_{t+\Delta t} + \mathbf{C}\dot{\underline{u}}_{t+\Delta t} + \mathbf{K}\underline{u}_{t+\Delta t} = P(t)$$

where **C** is the damping matrix, **M** is the mass matrix, **K** is the stiffness matrix.

$$\underline{U}_{t+\Delta t} = \underline{U}_t + \Delta t \cdot \dot{\underline{U}}_t + \frac{\Delta t^2}{2}\left[(1-2\beta)\ddot{\underline{U}}_t + 2\beta\ddot{\underline{U}}_{t+\Delta t}\right]$$

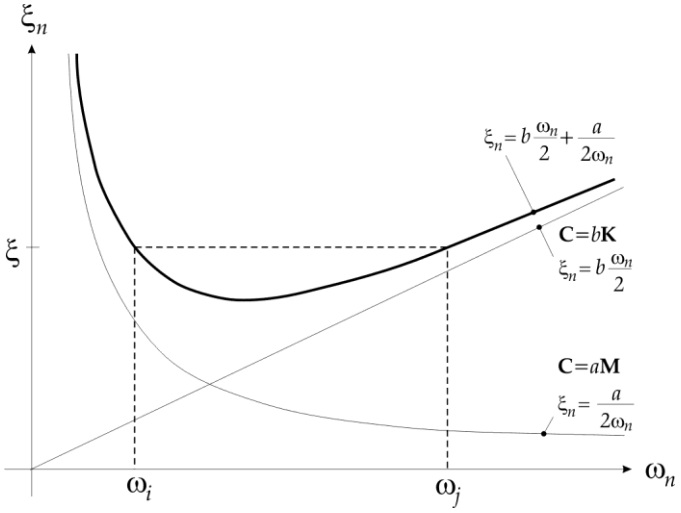
$$\dot{\underline{U}}_{t+\Delta t} = \dot{\underline{U}}_t + \Delta t\left[(1-\gamma)\ddot{\underline{U}}_t + \gamma\ddot{\underline{U}}_{t+\Delta t}\right]$$

AXISVM uses $\beta = \frac{1}{4}, \gamma = \frac{1}{2}$.

The differential equation of the motion is solved by the method of constant mean acceleration. This step-by-step integration is unconditionally stable and its accuracy is satisfying. AXISVM assumes that no dynamic effect is applied in $t = 0$. Time-limited loads appear in $t > 0$. **C** is calculated from the Rayleigh damping constants:

$$\mathbf{C} = a\mathbf{M} + b\mathbf{K}$$

Where *a* and *b* should be calculated from the damped frequency range (between f_i and f_j) and the damping ratio according to the following figure:



$$a = \xi \frac{2\omega_i\omega_j}{\omega_i + \omega_j}$$

$$b = \xi \frac{2}{\omega_i + \omega_j}$$

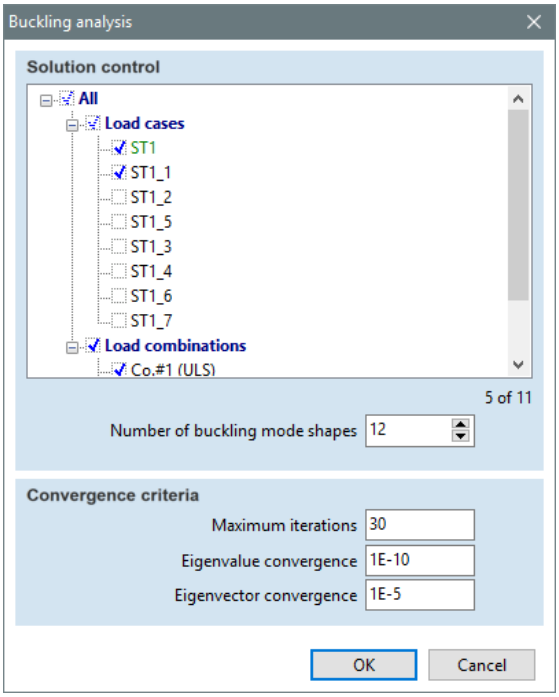
where ω_i and ω_j are angular frequencies relating to f_i and f_j:

$$\omega_i = 2\pi f_i$$
$$\omega_j = 2\pi f_j$$

For details about the behaviour of seismic isolators see **3.1.17.2 Seismic isolators**

5.4 Buckling

Lets you determine the lowest (initial) buckling load multipliers and the corresponding mode shapes.
AXISVM verifies whether the required number of the lowest eigenvalues has been determined.
The buckling load multiplier $n_{cr} = \lambda_{cr}$ is computed by solving the eigenvalue problem. λ_{cr} is the smallest eigenvalue and the corresponding eigenvector is the buckling mode shape.
The Sturm sequence check is applied to verify whether the computed eigenvalues are the lowest. $\lambda_{cr} < 0$ means that buckling occurs for the opposite load orientation and $\lambda_{cr}^{effective} \leq |\lambda_{cr}|$.



☞ **The solution technique applied to the associated generalized eigenvalue problem is designed to find the lowest real and positive eigenvalues. It is not suitable to find eigenvalues that are zero or nearly zero.**

If the model has construction stages, buckling analysis is always performed on the final state of the model. See 4.13.1 General overview of the concept of construction stages

It also possible to run buckling analysis of a substructure. See **6.2 Buckling**

Solution control

Select load cases or combinations in the tree view. AXISVM will run a linear static analysis before the buckling analysis of the selected load cases.
Lets you specify the parameters of the incremental solution process:

Case

Lets you select a case that will be taken into account. A linear (first-order) static analysis, that precedes the buckling analysis, will be performed.

Number of mode shapes

Lets you specify the number of the vibration mode shapes you want to evaluate. A maximum number of 99 can be requested. The default value is 6. The lowest positive eigenvalue is of main importance.

Convergence criteria

See **5.2 Vibration**/Convergence criteria

Shells The buckling of shell elements are fully taken into account in the analysis.

Beams/ribs The buckling of beams/ribs is considered as in-plane buckling (flexural buckling), which means that the deformed shape of the element remains in a plane and the cross-section does not warp.
For buckling analysis the beam cross-section must be defined by specifying its principal moments of inertia.

☞ **The beam elements must be divided into at least four elements.**

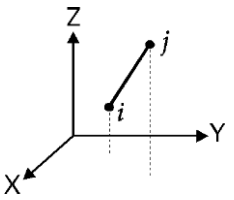
Trusses The flexural buckling of truss elements are not considered by the program. You must calculate the buckling load of each truss manually, or by modeling the trusses by four beam elements with the corresponding end releases.

☞ **If the model contains trusses the critical load parameter of global structural buckling will be computed only. Buckling of individual trusses is not analysed.**

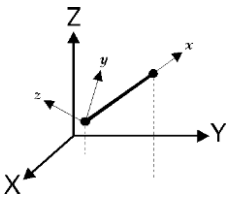
Seismic isolators For details about the behaviour of seismic isolators see **3.1.17.2 Seismic isolators**

5.5 Finite elements

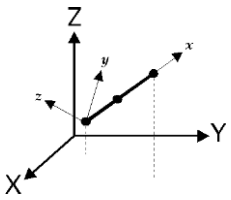
All finite elements may be used in a linear static, nonlinear static, vibration, buckling and dynamic analysis. Note that ***not*** all elements have geometric stiffness. Only the following elements have geometric stiffness: truss, beam, 7 DOF beam, rib, shell, solid, rigid element, diaphragm.



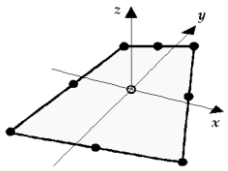
Truss



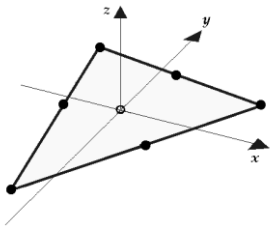
Beam



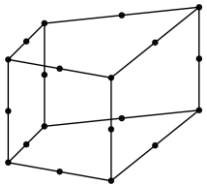
Rib



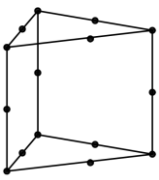
Quadrilateral membrane, plate, or shell



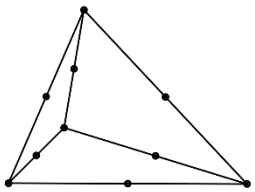
Triangular membrane, plate, or shell



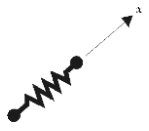
Hexahedron (20)



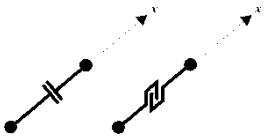
Wedge (15)



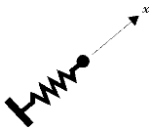
Tetrahedron (10)



Spring
(only one component is shown)

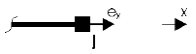
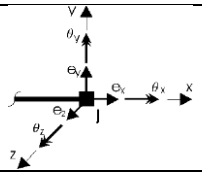
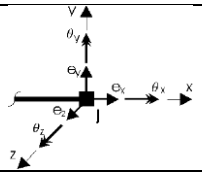
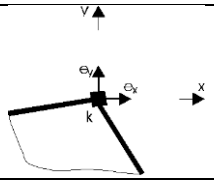
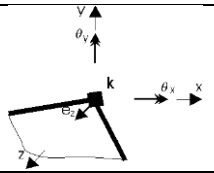
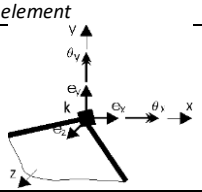
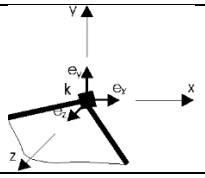
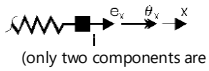
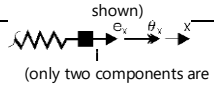


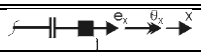
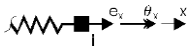
Gap
(active in compression and tension respectively)



Support
(only one component is shown)

The directions in the local coordinate system in which an element has stiffness, and the corresponding local displacement components are summarized below:

Finite element	*ex* *u*	*ey* *v*	*ez* *w*	*Θx*	*Θy*	*Θz*	
Truss	*						
	2-node, linear, isoparametric element						
Beam	*	*	*	*	*	*	
	Euler-Navier-Bernoulli type, 2-node, cubic Hermitian element						
Rib	*	*	*	*	*	*	
	Timoshenko type, 3-node, quadratic, isoparametric element						
Membrane	*	*					
	Serendipity type, 8-node, quadratic, isoparametric element						
Plate			*	*	*		
	Hughes type, 9-node, Heterosis isoparametric plate element						
Shell	*	*	*	*	*		
	Flat shell superimposed membrane and plate element						
Solid element	*	*	*				
	Hexahedron, wedge, tetrahedron						
Support	*	*	*	*	*	*	 (only two components are shown)
Spring	*	*	*	*	*	*	 (only two components are shown)

Finite element	ex u	ey v	ez w	Θx	Θy	Θz	
Gap	*						
Rigid							
Link	*	*	*	*	*	*	 (only two components are shown for a node-to-node link)

where:
u, v, w denote the deflections in local *x, y, z* directions.
Θx, Θy, Θz denote the rotations in local *x, y, z* directions.
***** element has stiffness in the respective direction.

Internal forces

The computed internal forces in the local coordinate system are:

Finite element	Internal forces							
Truss	N_x							
Beam	N_x	V_y	V_z	T_x	M_y	M_z		
Rib	N_x	V_y	V_z	T_x	M_y	M_z		
Membrane	n_x	n_y	n_{xy}					
Plate				m_x	m_y	m_{xy}	v_{xz}	v_{yz}
Shell	n_x	n_y	n_{xy}	m_x	m_y	m_{xy}	v_{xz}	v_{yz}
Solid	S_{xx}	S_{yy}	S_{zz}	S_{xy}	S_{xz}	S_{yz}		
Spring	N_x	N_y	N_z	M_x	M_y	M_z		
Gap	N_x							
Support	N_x	N_y	N_z	M_x	M_y	M_z		
Rigid								
Link N-N	N_x	N_y	N_z	M_x	M_y	M_z		
Link L-L	n_x	n_y	n_z	m_x	m_y	m_z		

Line elements

Truss
Interpolation formulation: 2 noded, linear, isoparametric Lagrange with constant cross-section.
Strains and internal forces: Tension-compression.

Beam
Interpolation formulation: 2 noded, cubic Hermite. The structural member may have a tapered cross-section, but in the computational model the finite elements of the mesh will have a constant cross-section, with stepwise changes in dimensions.
Theory: **Euler-Bernoulli**
Strains and internal forces: Tension-compresson. Bending without taking the transverse shear deformation into account. Saint-Venant torsion without taking the restrained warping into account.
Concentrated or distributed load on the element can be defined. The exact displacements and internal forces compatible with the loads are calculated and displayed along the element. Therefore mesh is not necessarily required for linear static analysis.

7-DOF Beam
Interpolation formulation: 2 noded, cubic Hermite. The structural member may have a tapered cross-section, but in the computational model the finite elements of the mesh will have a constant cross-section, with stepwise changes in dimensions. In the case of single- or double-symmetrical, single- or double-haunched I-sections with variable cross-sections, the finite element also takes the effect of the variation into account, with corrections appearing in the torsional and warping elements of the stiffness matrix.
Theory: **Euler-Bernoulli-Vlasov**
Strains and internal forces: Tension-compresson. Bending without taking the transverse shear deformation into account. Restrained warping is also taken into account.
Warping displacements and internal forces are only linearly interpolated between the nodal values. Therefore mesh is always necessary for appropriate results. Especially when concentrated or distributed load exists on the element.

Rib
Interpolation formulation: 3 noded, quadratic, isoparametric Lagrange. The structural member may have a tapered cross-section, but in the computational model the finite elements of the mesh will have a constant cross-section, with stepwise changes in dimensions.
Theory: **Timoshenko**
Strains and internal forces: Tension-compresson. Bending with taking the transverse shear deformation into account. Saint-Venant torsion without taking the restrained warping into account.
This element is compatible with the surface elements. Therefore, it can be used for modeling beams and columns connected to slabs and walls.
The displayed displacements and internal forces are only linearly interpolated between the nodal results.. Therefore mesh is always necessary for appropriate results.

Surface elements

Triangular membrane
Strains and internal forces: Tension-compression and in-plane shear.
Interpolation formulation: 6 noded Lagrange

Quadrilateral membrane
Strains and internal forces: Tension-compression and in-plane shear.
Interpolation formulation: 8 noded serendipity

Triangular plate
Theory: **Mindlin**
Strains and internal forces: Bending with taking the transverse shear deformation into account.
Interpolation formulation: 6 noded Lagrange
It is usable up to 1/100 thickness-to-length ratio.

Quadrilateral plate
Theory: **Mindlin**
Strains and internal forces: Bending with taking the transverse shear deformation into account.
Interpolation formulation: **Heterosis**
8 noded serendipity for the transversal displacements
9 noded Lagrange for the bending rotations. The middle node has only two bending rotational degree of freedom.
The reason why the Heterosis quadrilateral element is used that it exhibits improved characteristics in comparison with both the 8 noded serendipity and the 9 noded Lagrange elements. It has full rank stiffness matrix with selective reduced integration, therefore it also has good accuracy in thin plate and shell situations, up to 1/1000 thickness-to-length ratio.

Triangular and quadrilateral shell
Superimposed from the corresponding membrane and plate element and a 6th drilling DOF. The 6th DOF has only a very small stiffness, therefore it is unable to resist any direct load. (see in detail **4.9.13.3 Shell)**

Solid elements for modeling soil

Tetrahedral
Interpolation formulation: 10 noded Lagrange

Hexahedral
Interpolation formulation: 20 noded serendipity

Wedge
Interpolation formulation: 15 noded serendipity

5.6 Main steps of an analysis

1. Define the geometry of the structure, the material and cross-sectional properties of the members, the support conditions, and the loads.
2. Determine the load transfer path.
3. Determine local discontinuities such as stiffeners, gussets, holes.
4. Determine the type of finite elements that will best model the behavior of the structure. With this step the properties of structural elements will be concentraded in their neutral *axis* (point, axis, or, plane).
5. Determine a mesh type and size for the model. The size of the mesh have to correspond to the desired accuracy of the results and with the available hardware.
6. Create the model:
   a.) Equivalent geometry
   b.) Equivalent properties
   c.) Topology of the elements
   d.) Equivalent support conditions
   e.) Equivalent load (static) or masses (vibration, response-spectrum)
7. Check input data (accuracy, compatibility)
8. Run analysis
9. Select important results
10. Evaluate and check the results
    a.) Accuracy and convergence of the solution
    b.) Compatibility taking into account point 6.d.
    c.) Uncommon structures shall be analyzed with other methods and/or software as well.
11. Restart analysis with a correspondingly updated model, if in step 10 a criteria is not satisfied.
12. Evaluate the results by the means of isoline/isosurface plots, animation, tables... Draw conclusions on the structure's behavior.

Modelling

To build a model of a structure you have to accept many assumptions so you also have to keep the effects of these assumptions in view when evaluating results.

The finite element method provides an approximative solution for surface models. To make the model match the real solution you have to use finite element meshes with an appropriate density.

Making finite element meshes you have to take into account the expected stress distribution, the model geometry and the materials, supports and loads used.

The position af nodes and mesh lines (called the *topology* of the finite element mesh) depends on the geometrical discontinuities (irregular contours, line supports) and the discontinuities of loads (concentrated loads, terraced load values for line loads).

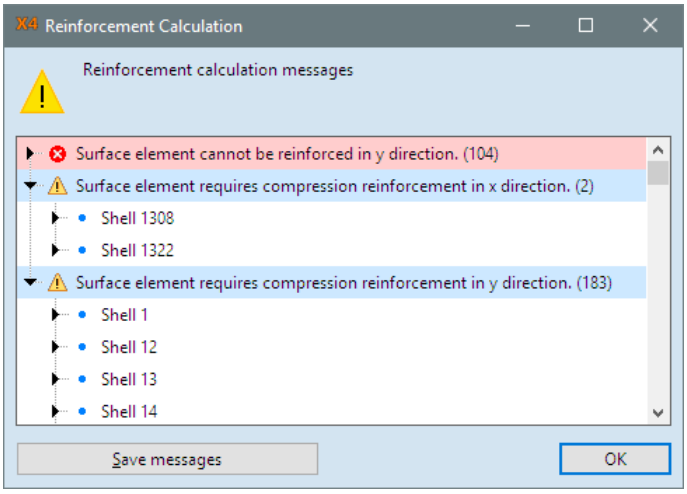
At stress concentration points (sharp corners) you have to refine the mesh. To avoid singularities due to concentrated effects you can distribute them on a small area around the point of effect.

Arc contours can be approximated as polygons. Using very small tolerance in this approximation leads to polygons with extreme small sides. The very dense mesh created on this contour may cause the model exceed the capacity of your computer.

In general if you refine the mesh you get more accurate results.

5.7 Error messages

Multiple error messages and warnings are organized into a tree.



The error messages corresponding to modeling errors are listed below:

Non-positive definite stiffness matrix

The determinant of the stiffness matrix is zero or negative due to modeling error.

Singular Jacobian matrix

Determinant of the element's Jacobian matrix is zero, due to distorted element geometry.

Excessive element distortion during deformation

The element has been excessively distorted in the current increment.

Too large rotation increment

The rotation increment of an element exceeds $\pi/4$ radian (90°). You should increase the number of load increments.

Invalid control displacement component

The displacement control is applied about a constrained degree of freedom.

Convergence not achieved

The number of iteration is too low.

Too many eigenvalues

The rank of the mass matrix is lower than number of requested eigenvalues (frequencies or buckling modes).

No convergent eigenvalue

No eigenvalue converged.

Not the lowest eigenvalue (xx)

There are xx lower eigenvalues than the lowest the one determined

Element is too distorted

The geometry of the finite element is distorted. In order to maintain the accuracy of the results you should modify the finite element mesh to avoid too distorted element geometries.

Excessive element deformation

During a nonlinear analysis excessive deformations developed the element within an increment (load or displacement). You should increase the number of increments.

No convergence achieved within maximum number of iterations

There was no convergence within the maximum number of iterations (see **5.1 Static analysis** Static /Nonlinear Static Analysis/Solution Control parameters). You can increase the number of iteration. The model may not converge at the respective load level, and you should change the Solution Control parameters accordingly.

Divergence in the current iteration

A divergence was detected in the iteration process. Increments are too large or the convergence criteria are too loose.

No stiffness at node ... in direction ...

There is a singularity in the system stiffness matrix corresponding to that degree of freedom. You should check the support and degrees of freedom (DOF) settings of your model.

Other (system) error messages

In some cases, other type of error messages may also appear. Based on the system error code, more information is provided about the error on the following link:
http://docwiki.embarcadero.com/RADStudio/Tokyo/en/Input-Output_Errors

5.8 Analysis parameters

The parameters entered for nonlinear / vibration / deflection / dynamic calculations performed are displayed in the Table Browser (see **2.9 Table Browser**) under the *Results* tab, where they can be checked and added to reports.

Analysis parameters [Vibration I., ST1]

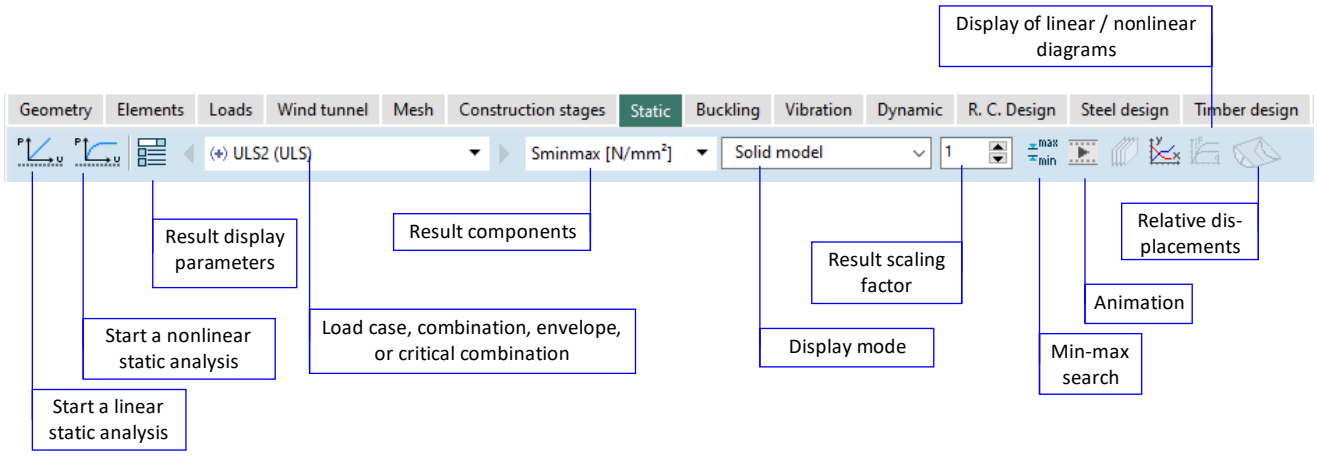
Name	Parameters
Analysis:	Vibration (1st order)
Load case:	ST1
Solution control:	
Number of mode shapes:	12
Convergence criteria:	
Maximum iterations:	30
Eigenvalue convergence:	1E-10
Eigenvector convergence:	1E-5
Stiffness reduction for response spectrum analysis:	Original stiffness
Use increased support stiffness:	—
Convert slabs to diaphragms:	—
Masses:	Convert loads to masses
Concentrated masses:	—
Convert concentrated masses to loads:	—
Include mass components:	X, Y, Z
Mass matrix type:	Diagonal

6. The Postprocessor

Static	Lets you display the results of a static analysis. (6.1)
Buckling	Lets you display the results of a buckling analysis. (6.2)
Vibration	Lets you display the results of a vibration analysis. (6.3)
Dynamic	Lets you display the results of a dynamic analysis. (6.4)
R.C. Design	Lets you display the results of a reinforced concrete design analysis. (6.5)
Steel design	Lets you display the results of a steel design analysis. (6.6)
Timber design	Lets you display the results of a timber design analysis. (6.7)
CLT design	Lets you display the results of an CLT-domain design analysis. (6.8)
Masonry design	Lets you display the results of a masonry design analysis. (6.9.1.6)

6.1 Static

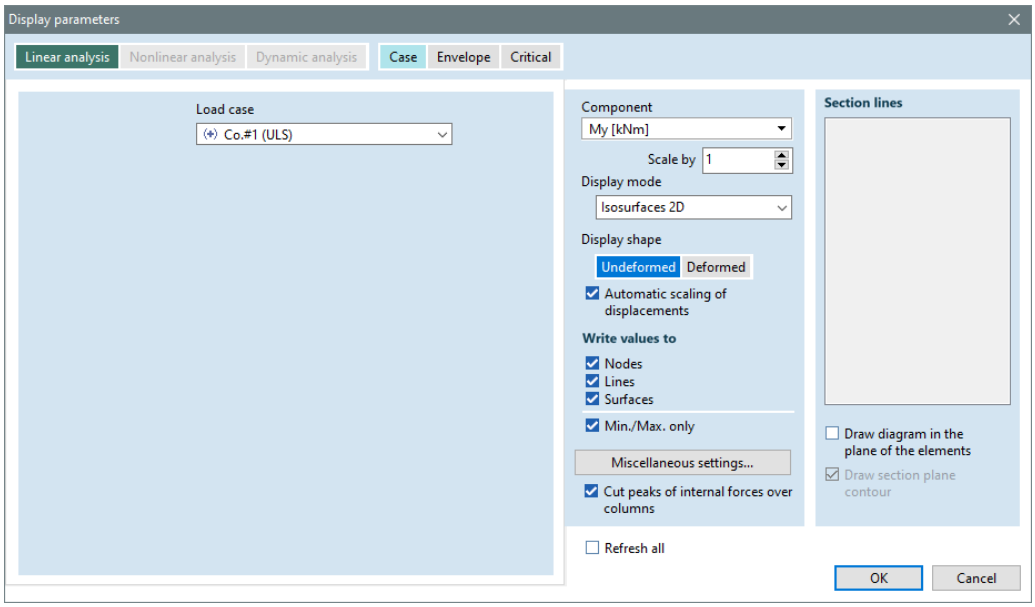
The Static menu item allows you to display the tools for displaying and interpreting the static analysis results.



Width of combo boxes and the result scaling factor edit field can be adjusted on each postprocessor tab separately. Place the mouse between the control and the next control of the toolbar. The cursor will change to ⟺and you can adjust the width of the control. To restore the default sizes click *Settings / Toolbars to default position*. See also **3.3.15 Toolbars to default position.**

Start a linear static analysis	See **5.1 Static analysis**
Start a nonlinear static analysis	See **5.1 Static analysis**
Result display parameters	Lets you set the options of the graphical display of the results. You can select the results of a load case/combination or critical load combination.





Analysis Type

Depending on the performed analysis you can select the results of a linear or nonlinear static analysis. Each analysis type can be further defined:

Case

Lets you display the results of any load case/combination.

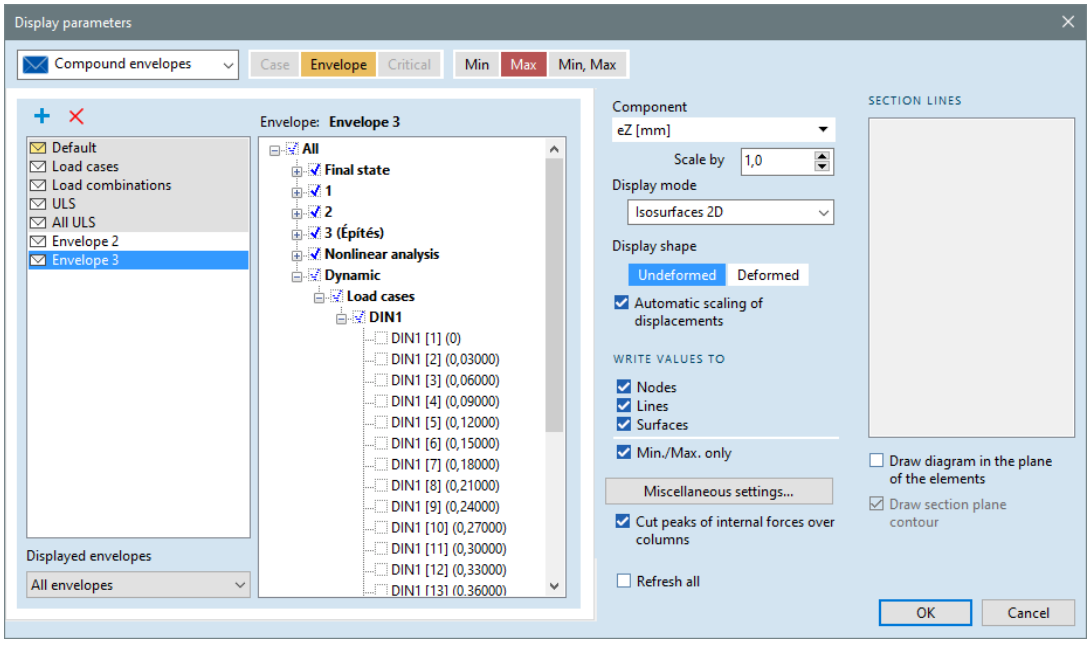
Envelope

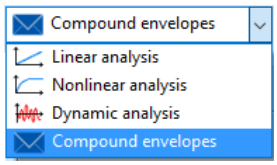
Lets you display the envelope of the results from the selected load cases and/or load combinations. The program searches for the minimum and/or maximum values at each location of the selected result component.

Critical

Lets you generate the critical load combinations, according to the load group definitions, for each location of the selected result component.

Envelope





Envelopes are derived results containing the minimum and/or maximum values at each point from the results of load cases and/or load combinations. Depending on which load cases and combinations are included, different envelopes can be generated.
In the top left-hand corner, you can select whether to view envelopes generated from the results of linear, non-linear or dynamic analyses.

By selecting 'Compound envelopes', you can generate arbitrary envelopes from the results of all analysis types (or construction stages).

AXISVM allows to define and use different envelopes with names. On the left a list of the available envelopes are listed. Certain basic envelopes are automatically created (envelope of all load cases, all load combinations or certain combination types (e.g., ULS, SLS Quasipermanent). The composition of the selected envelope is displayed in the tree of load cases and combinations. Changing the composition of an envelope results in creating a new, custom envelope. Selecting a custom envelope and clicking on its name makes the name editable.
If AXISVM main window is divided into sub-windows a different envelope can be chosen for each sub-window. The name of the selected envelope is also displayed in the status window. Drawings and tables of the report also contain and display envelope information.



Create a new custom envelope



Delete a custom envelope (only custom envelopes can be deleted)

Multiple selection is enabled in the tree of load cases and combinations. To check or uncheck a continuous range of load cases click on the first load case within the range (it will be selected) then Shift+click on the last load case of the range.

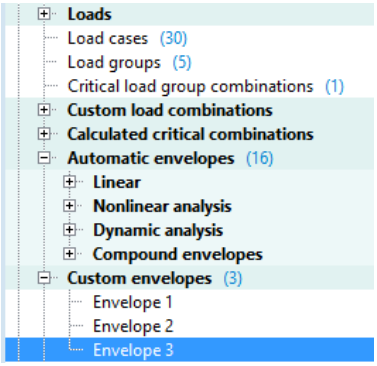
Displayed envelopes

Select the displayed envelopes from the dropdown list under the list of envelopes. This way you can control which envelopes are available for result evaluation in the dropdown list of load cases and combinations.

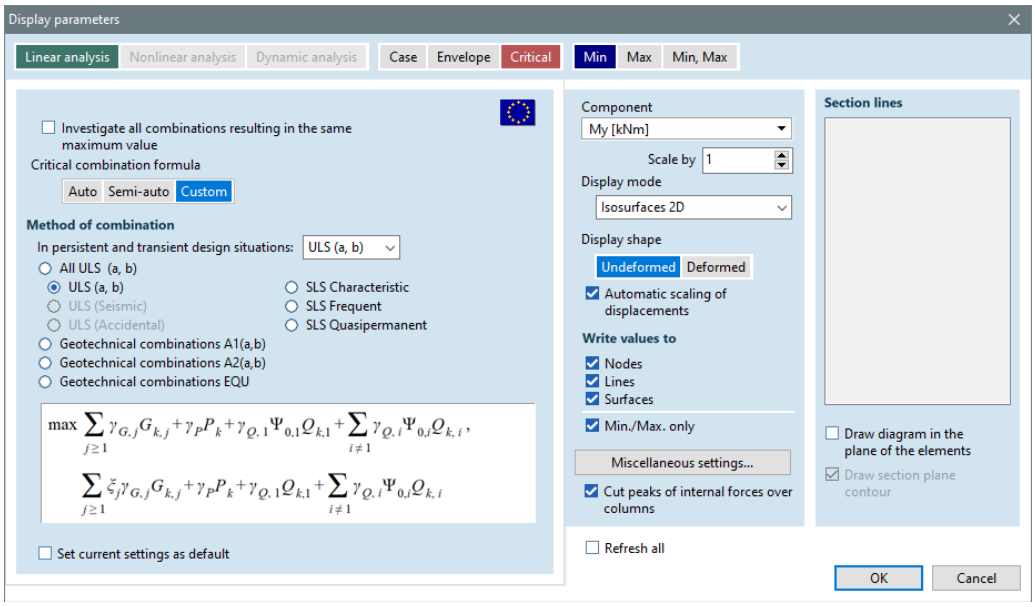
Only the selected envelope	Only one envelope will be available which is the currently selected one.
Only custom envelopes	All custom envelopes will be listed
All envelopes	All envelopes (basic and custom) will be listed.



Table Browser provides tables to list composition of the envelopes used in the model. These tables can be added to the report. See 2.9 Table Browser
Automatic envelopes are displayed in separate groups according to their result type. Custom envelopes appear in one group. The tables list the load cases and load combinations included in the envelope.
Envelopes for non-linear or dynamic results can only be compiled after the relevant analysis type has been completed.
Compound envelopes only appear if a non-linear or dynamic analysis has also been completed.
The standard operation of the Table Browser also applies here: clicking the *Add table to report* icon adds the selected table (or, if a category node is selected, all tables under that category) to the current report.



Critical

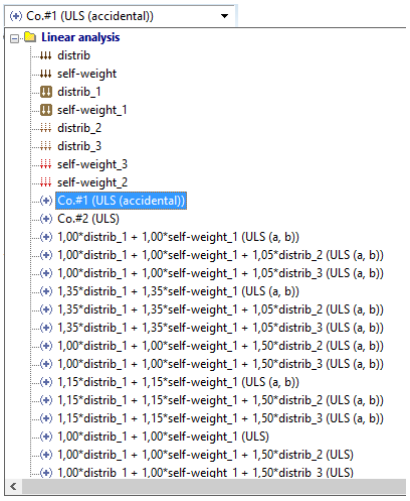


Investigate all combinations resulting in the same maximum value

By default, this option is off. AXISVM takes into account combinations resulting in an extreme for any result component. In certain design methods however a combination which produces no extremes can be more unfavorable.
In this case turn this option on. In design calculations AXISVM will build all possible combinations and check them according to the design code requirements. As the number of combinations can be extremely high this option is recommended only if the model size and the number of load cases are small.

Method of combination

If *Critical combination formula* is set to *Auto AXISVM* determines if ULS (ultimate limit state) or SLS (service limit state) combination is required based on the result component.



If *Critical combination formula* is set to *Custom Min / Max / Min, Max* results of all combination methods will be available in the load case tree regardless the current result component.
In case of Eurocode and SIA and other Eurocode based design codes the formula for creating SLS combinations can be chosen.

If the *Auto* option is selected all design calculations will choose the appropriate critical formula (e.g., SLS Frequent for cracking width calculation according to EC-HU, SLS Characteristic for displacements of a timber structure, ULS for forces and stresses).

In *Semi-auto* mode AXISVM uses ULS or SLS combination according to the type of calculation but the formula for creating SLS combinations can be chosen.

Display values

If you have selected Envelope or Critical you can choose from the following options:

Min+Max
: Displays the minimum and maximum values of the current result component.

Min
: Displays the minimum (sign dependent) values of the current result component.

Max
: Displays the maximum (sign dependent) values of the current result component.

Display Shape

Undeformed
: Displays the undeformed shape (original configuration) of the model.

Deformed
: Displays the deformed shape of the model.

Automatic scaling of displacements

Scales the real deformation by an automatic scale factor so that the maximum displacement is represented as 50 pixels. This pre-scaled deformation is the basis of the further scaling by the *result scaling factor*. Without this option the basis of the scaling by *result scaling factor* is the real deformation.

Display Mode

Diagram

Lets you display the current result component in a colored diagram form. The numerical values are displayed if a *Show value labels on* option is enabled.

Filled diagram

Similar to *Diagram* but fills it in a customizable way (see Graphic symbols, **3.3.11 Preferences**)

Diagram+average values

This display mode is available only if line support forces are displayed. If this mode is selected line support forces diagrams are enhanced with the display and labeling of the average value. Averaging is made over continuous supports. Supports are considered to be continuous if they have the same stiffness and their angle is below a small limit.

Section line

Lets you display the current result component in the active section lines and/or planes in a diagram form. The numerical values are displayed if the *Show Value Labels On* option is enabled.

Filled section line

Similar to *Section line* but fills it in a customizable way (see settings for the *Filled diagram,* Graphic symbols, **3.3.11 Preferences**)

Isoline (contour line)

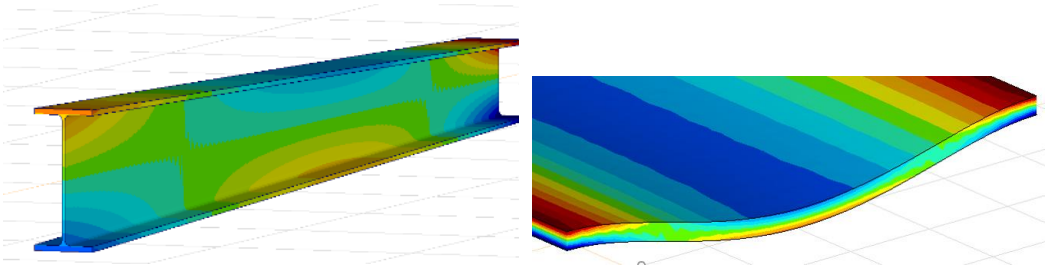
Lets you display the current result component in a line color contour plot form. The values that are represented by the isolines are specified in the Color Legend window. You can set the parameters of the Color Legend window as was described in the Information Windows paragraph. The numerical values are displayed if a *Show Value Labels On* option is enabled.

Isosurface 2D or 3D

Lets you display the current result component in a filled color contour plot form. The ranges that are represented by the isosurfaces are specified in the Color Legend window. You can set the parameters of the Color Legend window as was described in Information Windows paragraph. The numerical values are displayed if a a *Show Value Labels On* option is enabled. See **2.19.4 Color legend window**

Solid model

This mode displays results on a rendered view of the model. It is especially suitable to display the distribution of normal stress within beam cross-sections or display top and bottom values of surface stresses, reinforcement or crack width simultaneously.
For settings of the display mode see **3.3.11.11 Display**



None

The current result component is not displayed.

Section lines

Lets you set the active section lines, planes and segments. If display mode is set to Section line result diagrams will be drawn only on active (checked) section lines. Symbol of the section planes can be displayed enabling the *Draw section plane contour* checkbox. Turning on the *Draw diagram in the plane of elements* option changes the appearance of all section diagrams. To change this parameter individually use the Section lines dialog.
See **2.16.18 Sections**

Component

Lets you select the result component to be displayed.

Scale by Lets you set the scale of a diagram drawing. The default value is 1, when the maximum ordinate is represented as 50 pixels.

Write Values to ... *Nodes*

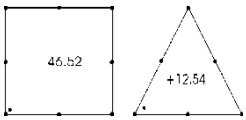
Writes the values of the current result component to the nodes.

Lines

Writes the values (intermediate values if applicable) of the current result component to the line elements.

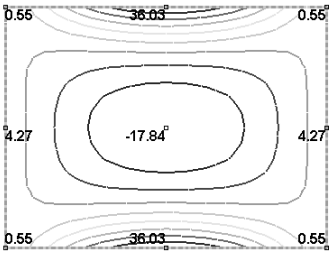
All surfaces

Writes the values of the current result component to the surface elements. The maximum absolute value of the nine values computed at the nodes of each surface is displayed, and the respective node is marked by a small black circle.

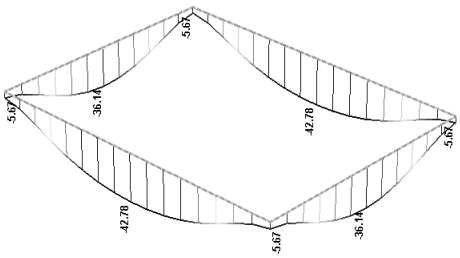


Min/max only

Writes the local min/max values only of the current result component to the nodes, lines and surfaces.



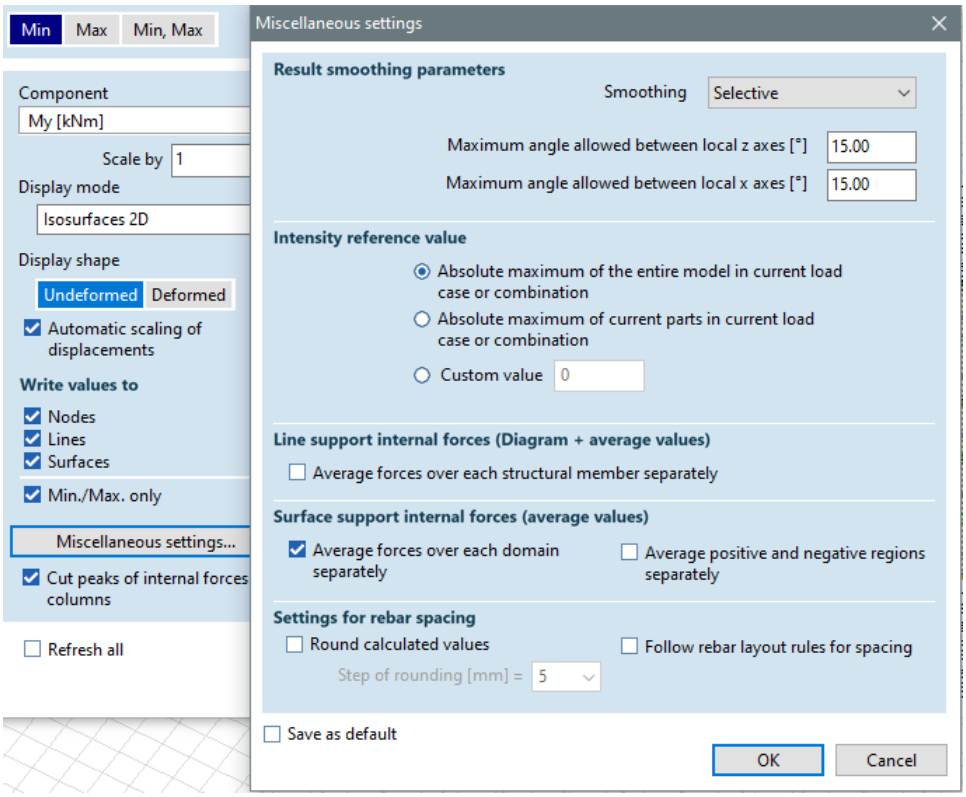
my moment component



Rz support force component

☞ **If labels overlap the drawing can be made clearer (more readable) by checking *Prevent labels from overlapping*** (2.16.21 Display options)

After clicking the *Miscellaneous Settings...* button the following options are available:



Result smoothing parameters

None

The software does not smooth the values of the internal forces of the surface elements computed at the nodes (the results related to joint nodes or edges of connecting finite elements may differ).

Selective

The values of the internal force components of the surface elements computed at the nodes are averaged in a depending on the local coordinate systems, the support conditions and the loads of the elements that are attached to a node.

The software recgonizes the typical cases, when a jump is expected in the internal forces due to the loads and support conditions (e.g., the step in the internal shear forces at inner line support). At these nodes the software does not proceed the smoothing.

If the local coordinate systems of the connecting domains are different, smoothing is performed at the nodes of the connecting edges only if the angle between the locale x or z axes is not greater than the values specified in the settings.

All surfaces

The values of all internal force components of the surface elements computed at the nodes are averaged without consideration of the properties of the elements, loads and support conditions.

Intensity reference value

Intensity variation of the current surface internal force component displays the variation as a percentage of the referenmce value set here.

See **6.1.13 Surface element internal forces**

Line support internal forces

See **6.1.14 Support internal forces**

Surface support internal forces

See **6.1.14 Support internal forces**

Settings for rebar spacing

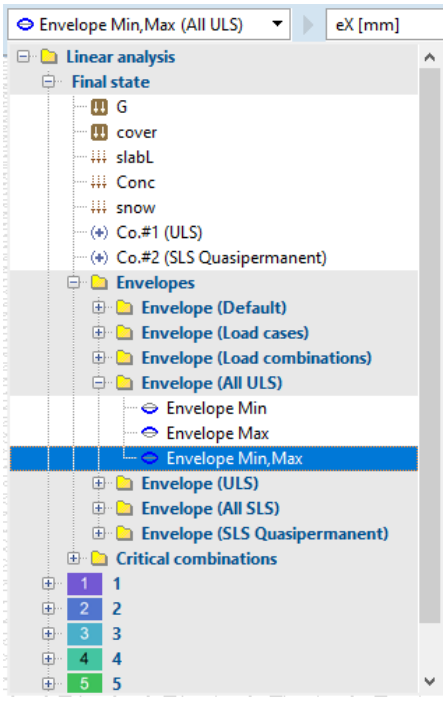
Check *Round calculated values* and step of rounding to make the calculated values more realistic. Following rebar layout rules for spacing is optional.

Cut peaks of internal forces over columns

If the mesh on a domain was created with the option *Adjust mesh to column heads* peaks of internal forces can be averaged and cut (except shear forces that are set to zero within the column perimeter) over column heads by activating this option.

See **6.1.13 Surface element internal forces**

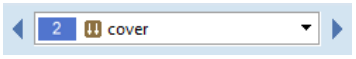
Case selector to display



You can select a case from the drop-down list to display:

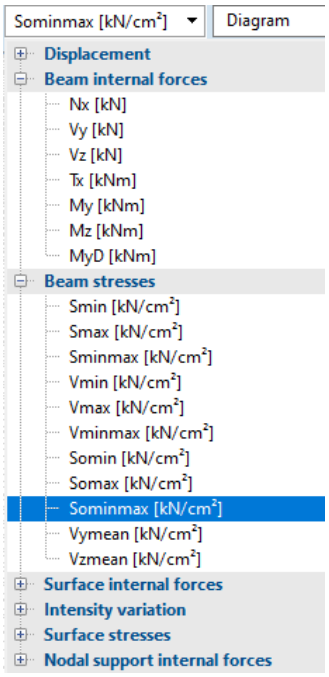
- Load case, load combination
- The k-th increment of a nonlinear analysis
- Envelope display
- Critical combination

Construction stages and the final state are displayed as independent tree nodes in the load case tree. The results of construction stages and final state can be plotted and listed separately.



Arrows to the left and right of the dropdown list switch between construction stages keeping the selected load case

Available result components



You can select a result component from the drop-down list for display.

** MyD component is calculated only in certain cases, please see section* **6.1.11 Rib internal forces**

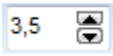
Display mode

You can select a display mode from the drop-down list:

Diagram
Filled diagram
Section lines
Filled section line
Isolines
Isosurfaces 2D
Isosurfaces 3D
None

☞ **If Min,Max envelope or critical load combination is selected, the Isoline and Isosurface 2D cannot be selected.**

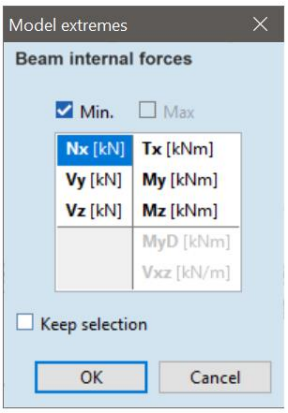
Display scaling factor



Lets you scale the display of the diagrams.

6.1.1 Minimum and maximum values

Lets you search for the minimum and maximum value of the current result component. If you are working on parts, the search will be limited to the active parts.
AXISVM will mark all occurrences of the minimum / maximum value.
Using the function in *Drone* mode (see **2.16.16 Drone**) sends the drone to the first element found.



Min, Max

You can search only the minimum or the maximum values.

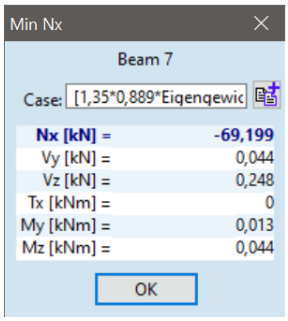
Keep selection

Turn this checkbox on if you want to keep the min/max locations selected after closing the dialog.

☞ **If parts are displayed extreme values are determined from the displayed parts only.**

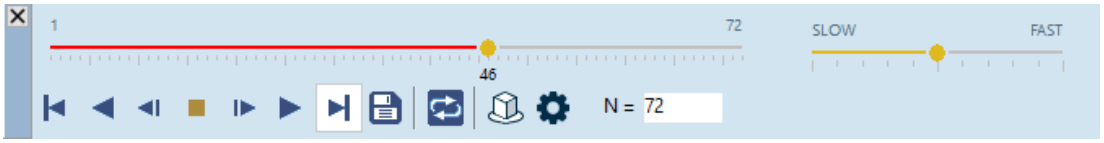


If this function is used when displaying critical combinations, the actual critical combination causing the extreme can be added to a cumulative list on the clipboard (no duplicates will appear).
The combinations in this list can be added to the load combination table.
See 4.10.2 Load combinations.



6.1.2 Animation





The animation toolbar can run result animation in the active window on the *Static, Buckling, Vibration, Dynamic* tabs and on all design tabs displaying the displacements, internal forces, and mode shapes in animated form (frame by frame). The animation of linear results consists of a sequence of frames that are generated by linear interpolation between initial values (frame 0) and the actual values of the current result component (frame *n*), according to the number of frames (*n*). Animation of nonlinear or dynamic results generates frames from the results of individual increments. After choosing *Deformed* as the *Display shape* in the *Result display options* the animation helps to see the deformation.
If the *Play* button is clicked the program starts building the frames of the animation. The generation can be interrupted by pressing the *Esc* key. The palette will then play the animation at the speed set on the yellow slider and according to the chosen playback mode.



Startpoint
Sets the position to the first frame of the animation.



Play backwards
Plays the animation frames in reverse order.



Previous frame of the animation
Moves to the previous frame.



Stop
Stops the animation.



Next frame of the animation
Moves to the next frame.



Play
Plays the animation frames.



Endpoint
Sets the position to the last frame of the animation.



Video file
It plays the animation generating the frames and saving them as a video file.
The animated GIF or animated PNG files can be played in most web browsers or converted to other formats like MP4 easily online that can reduce the file size further.



Play once
The animation stops at the final frame.



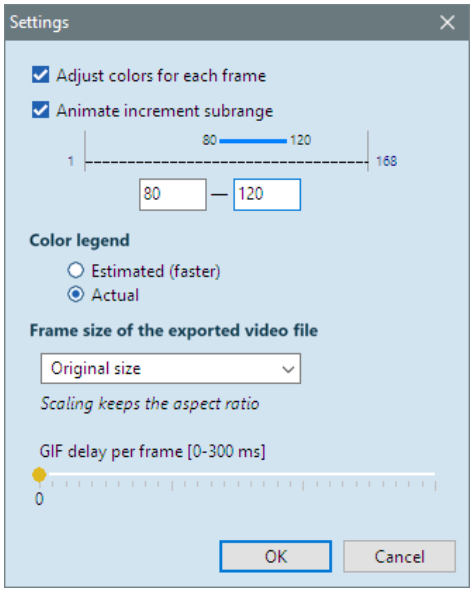
Unidirectional replay
The animation is played again when reaching the final frame.



Bidirectional replay
Plays the animation alternately forwards and backwards

Turntable
The result animation is combined with a turntable animation, rotating 360° around the Z-axis.

The content of the *Settings* dialog depends on result type as well.



Adjust colors for each frame
Updates the colors to cover the range between minimum and maximum values according to the color legend settings.

Animate increment subrange
In case of nonlinear or dynamic analysis the animation can be limited to a subrange of increments.

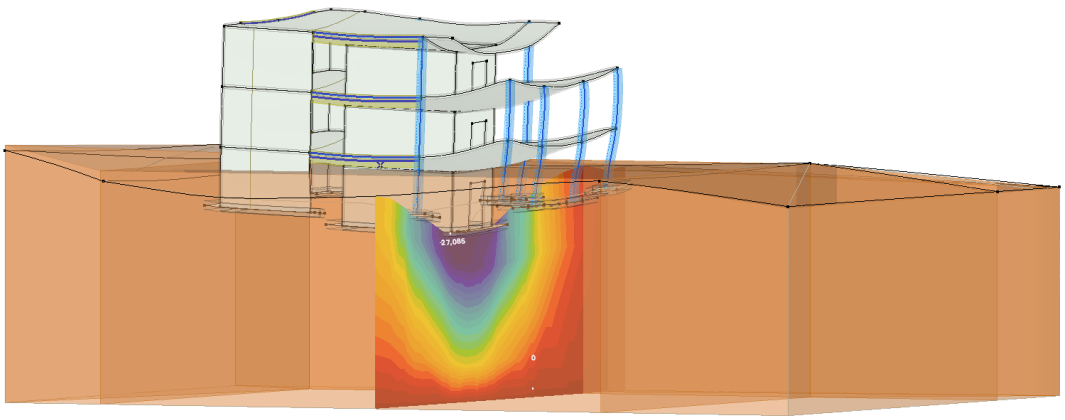
Color legend
Proper setting of colors may require the calculation of the overall minimum/maximum of the displayed result component which can be a time-consuming process for a dynamic analysis with a huge number of increments. Choosing the first option estimates the minimum and maximum from the increments used for generating frames.

Frame size of the exported video file
The available options are: the original size of the active window, scaled by a percentage (Zoom), a predefined or customized image width or height.
In all cases, scaling preserves the aspect ratio of the image.

GIF delay per frame [0-300 ms]
The playback speed of the animation saved in GIF format depends on the number of frames set. To fine-tune the playback of the animation, use the slider to adjust the delay between frames.

6.1.3 Soil section result animation

If soil deformations or stresses are displayed on section planes in *Section* result display mode (SOIL module is required), section planes can be moved interactively, animated and the animation can be saved in animated GIF or PNG format.
The animation toolbar works like in *Animation* (see **6.1.2 Animation**), but here the section plane can be selected from a dropdown list. The position of the section plane can be set by dragging the slider or editing the *x* field, where *x* is the distance from the original position of the section plane measured along its normal.

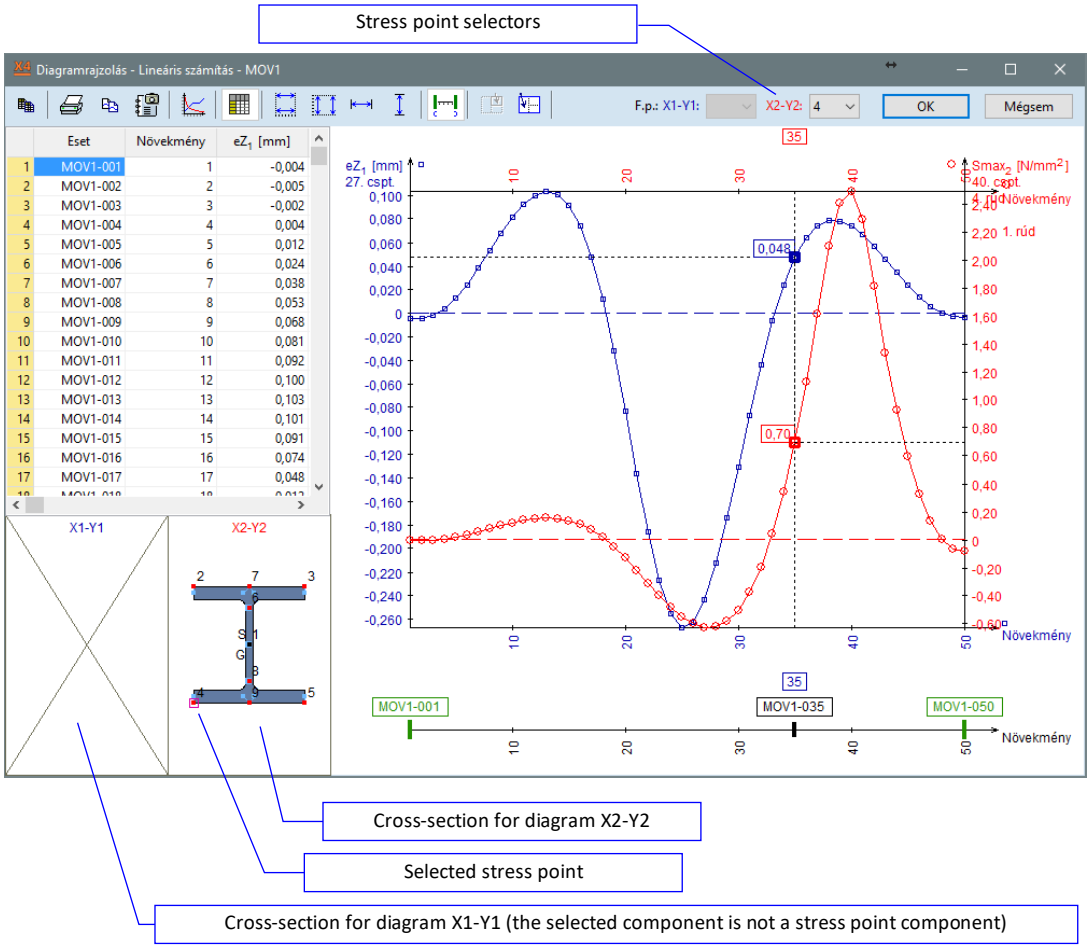


6.1.4 Diagram display

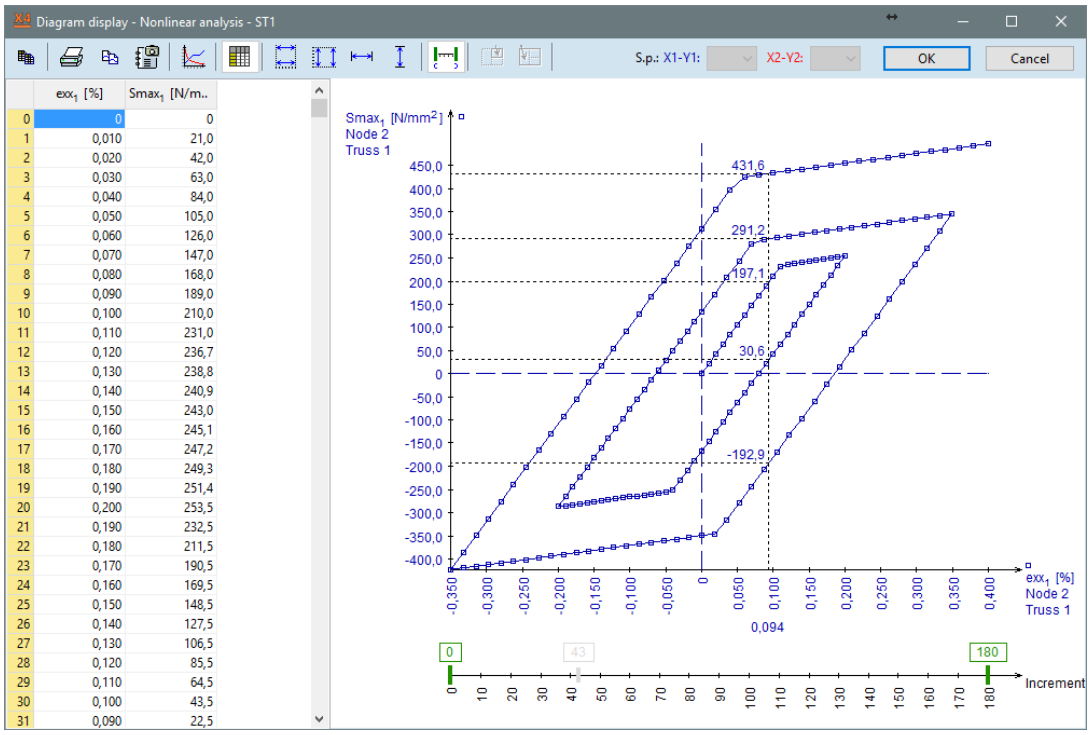


This dialog displays linear, nonlinear or dynamic results as diagrams. Two diagrams can be displayed simultaneously. Each diagram has a result component on its X and Y axis. Points representing consecutive value pairs are connected. Reading coordinates can be changed by dragging the dashed lines or the black mark of the bottom trackbar. Diagram points can be displayed as a table and exported to Excel through the Clipboard.

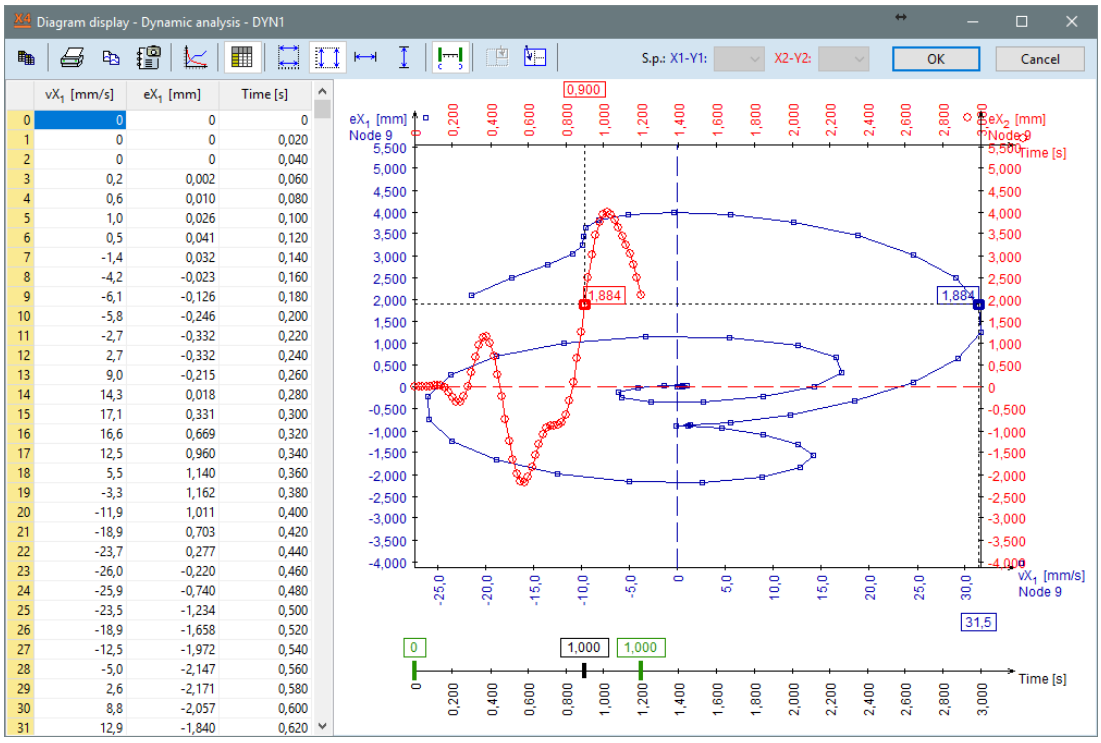
In case of linear analysis all load cases, all load combinations, all steps of a moving load or all members of an envelope are displayed depending on the chosen case in the main window.
If a result component relating to stress points is selected the cross-section is displayed and a specific stress point or the cross-sectional min/max value can be selected.



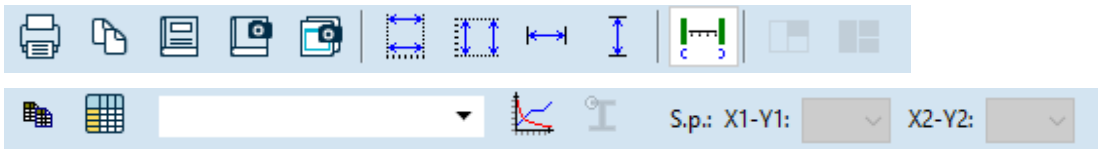
In case of nonlinear analysis all increments of an analysis or all members of an envelope can be displayed.



In case of dynamic analysis, the bottom trackbar displays time instead of increment numbers.



Toolbars





Copies selected cells to Clipboard
If the table is visible its selected cells are copied to the Clipboard.



Print drawing

Prints the diagram (and the table if it is displayed)

Copy to Clipboard
Copies the diagram to the Clipboard.

Add drawing to Drawings Library
Saves the drawing into the Drawing Library to make it available for reports.

Diagram Display Parameters
Components to be displayed can be selected from combo boxes.
If a result component is selected clicking the *Node* button allows selecting the node where the result is read.
The *x1-y1* diagram is in blue, with ticks and labels on the left and bottom axes.
The *x2-y2* diagram is in red, with ticks and labels on the right and top axes.
After turning on *Show markers,* data points are marked with small rectangles.

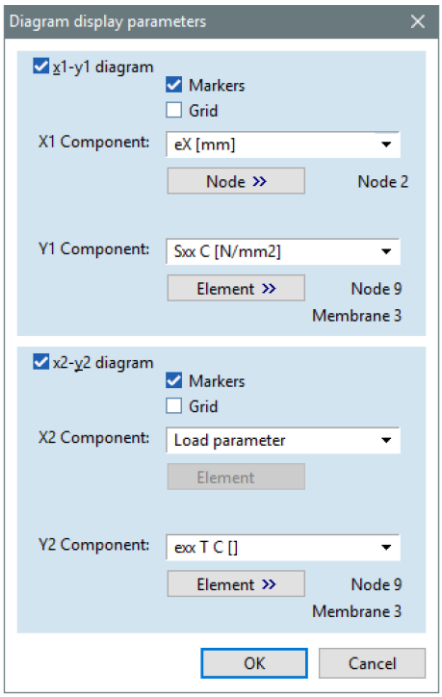


Table
Turn on/off the table displaying numerical values.

Same range on the two X-axes
If the same X-component is chosen for the two horizontal axes their ranges can be set to the same.

Same range on the two Y-axes
If the same Y-component is chosen for the two vertical axes their ranges can be set to the same.

Fit in view in X-direction
Sets the horizontal range between minimum and maximum of X values.

Fit in view in Y-direction
Sets the vertical range between minimum and maximum of Y values.

Interval controls
Turns on/off the green interval control rectangles of the bottom trackbar. Dragging them changes the displayed range of increments or time.

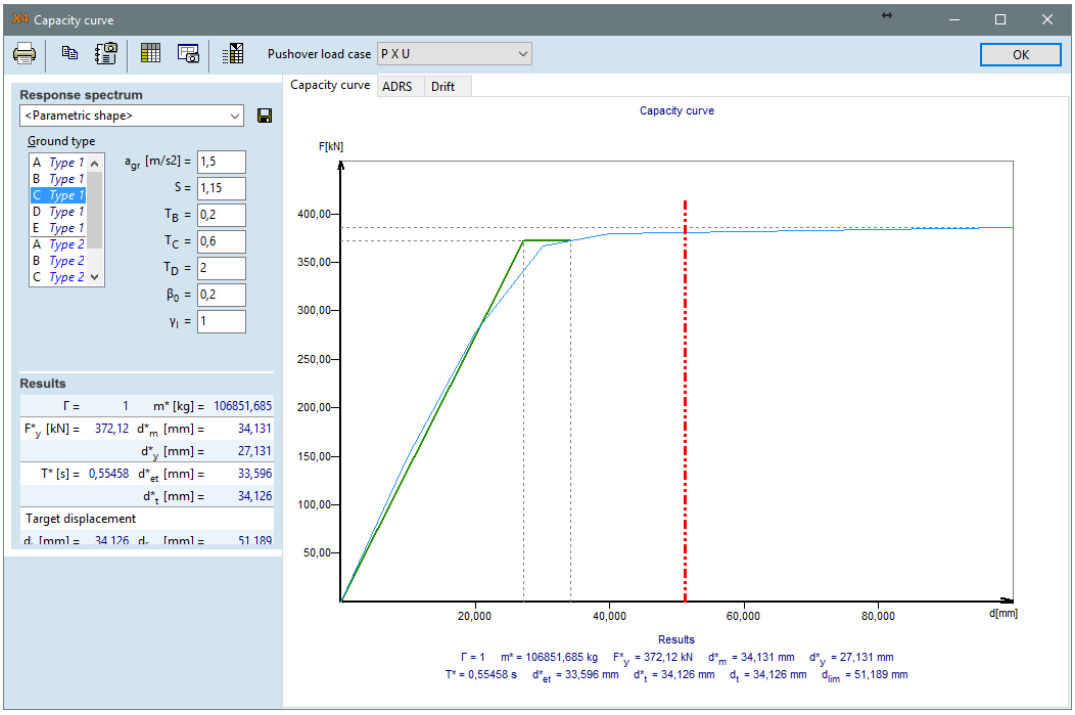
Loads the chosen load case, load combination, increment or time step to the active window.
(available in multi-window mode only)

Loads the chosen load case, load combination, increment or time step to the window.

6.1.5 Pushover capacity curves



This dialog is only active if results of pushover analysis are available and it helps the user determine the capacity curve and the target displacement depending on ground motion characteristics.
A combo box on the top of the dialog lets the user select the pushover load case to be analysed. Results are based on an acceleration-displacement response spectrum with properties specified on the left side of the dialog. These are identical to the properties of response spectra used for Seismic loads (See **4.10.25 Seismic loads – SE1 module**). Main results of the calculations are shown both on the bottom left side of the dialog and under the diagrams themselves.
The default dialog displays a capacity curve for both the Multi Degree of Freedom System (MDOF) and the equivalent Single Degree of Freedom System (SDOF).
The sky-blue curve is the capacity curve of the equivalent single degree of freedom system (SDOF). It has the same shape as the deeper blue curve for the multi degree of freedom system (MDOF). Its points are a result of dividing the corresponding force and displacement values of the MDOF curve by Γ.
Generally, the end point of both capacity curves is the point corresponding to the maximum displacement (divided by Γ for the SDOF curve) set by the user at the beginning of the nonlinear static analysis. The resulting curve on the figure below shows that the structure is capable of even more displacement, since the base shear force (vertical axis) is increasing as the displacements are increasing. The maximum value of the shear force can only be determined if a sufficiently large target displacement is specified when running the nonlinear pushover analysis.



Toolbar



Print drawing
Prints the current diagram



Copy to clipboard
Copies the current diagram to the Clipboard.



Add drawing to Gallery
Saves the current diagram to the Gallery to make it available for reports.



Table
Turns the table displaying numerical values on/off.

Add to Drawings Library
Saves the current diagram to the Drawings Library to make it available for reports.

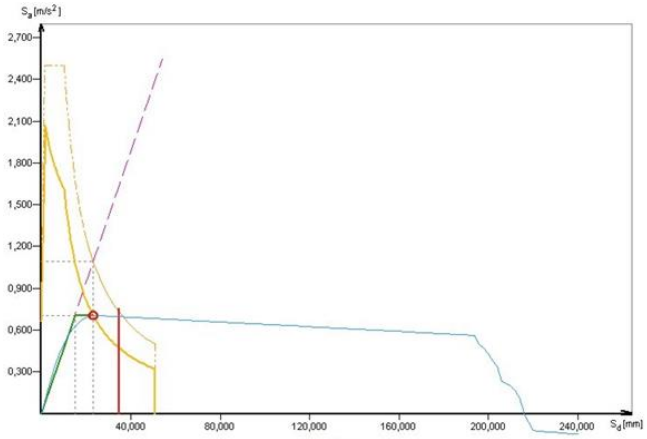
Seismic parameters
Displays a table with absolute and relative displacements of storeys and other parameters.

6.1.5.1 Capacity curves according to Eurocode 8

All of the results are based on the N2 method (see **11.32)** recommended in Appendix B of Eurocode 8. The bilinear force-displacement relationship for the SDOF system (green curve) is calculated by taking the force at the target displacement (d_t^*) as the force that corresponds to yielding (F_y^*) and defining yield displacement (d_y^*) using the equivalent deformation energy principle.
A vertical red line marks 150% of the target displacement (d_t) according to Eurocode 8 (4.3.3.4.2.3). Generally, if the deformation capacity of the structure is above this level (the line style is dash-dot) it fulfils the deformation capability requirements, otherwise (the lines style is continuous) it fails these requirements.

6.1.5.2 Acceleration-Displacement Response Spectrum (ADRS)

The Acceleration-Displacement Response Spectrum (ADRS) is shown by switching to the ADRS tab on the dialog. Both elastic (dashed yellow line) and inelastic (yellow line) ADRS spectra, SDOF (blue line) and equivalent bilinear capacity curves (green line) are shown here.
A separate line highlights the natural period corresponding to the elastic behavior of the structure. The intersection of capacity and demand corresponding to the target displacement is marked by a red circle. Purple dashed line shows the elastic approximation based on the initial stiffness of the bilinear capacity curve.

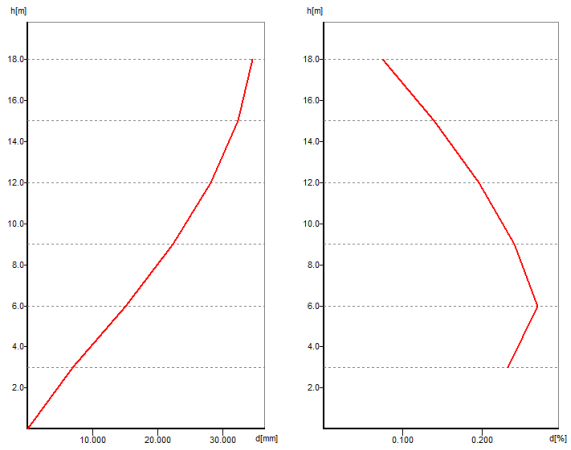


Results The variables marked by an asterisk (*) represent the SDOF system's behavior, while the others correspond to the MDOF system.

Γ	transformation factor for computing SDOF characteristics
m^*	mass of equivalent SDOF system
F_y^*	base shear force at d_m^* displacement of the equivalent SDOF system and yield force of the elasto-perfectly plastic force-displacement relationship
d_m^*	ultimate displacement of the idealized bilinear force-displacement relationship (not necessarily the ultimate displacement of the SDOF system due to the iterating procedure of the N2 method)
d_y^*	yield displacement of the idealized bilinear force-displacement relationship
T^*	natural period of the equivalent SDOF system
d_{et}^*	target displacement of the equivalent SDOF system with period T^* and unlimited elastic behavior
d_t^*	target displacement of the equivalent SDOF system considering inelastic behavior It represents the end of the green bilinear capacity curve.
d_t	target displacement of the MDOF system considering inelastic behavior

6.1.5.3 Drift

On the *Drift* tab the diagrams of absolute drifts and relative storey displacement (interstorey drift) ratios are displayed. The diagram of absolute drift shows the horizontal displacement of the centre of gravity of storeys relative to the soil. The interstorey drift ratio diagram shows the interstorey drift expressed as a percentage of the storey height. The latter diagram helps to check if the structure meets the drift limit requirements of Eurocode 8.
Clicking the *Seismic parameters* button on the toolbar the numerical values can be displayed in a table together with seismic parameters of storeys.



File Edit Format Help

Seismic parameters

Story	Z [m]	h [m]	d [mm]	Δd [mm]	Drift	P_{tot} [kN]	V_{tot} [kN]	V_{tot}/P_{tot}	Θ	S_x [m]	S_y [m]	G_x [m]	G_y [m]
3	9.0	3.0	32.255	9.786	0.33 %	100.00	18.22	18.22 %	0.0179	5.0	0	5.0	0
2	6.0	3.0	22.469	14.177	0.47 %	200.00	36.43	18.22 %	0.0259	5.0	0	5.0	0
1	3.0	3.0	8.292	8.292	0.28 %	300.00	54.65	18.22 %	0.0152	5.0	0	5.0	0

6.1.6 Result tables

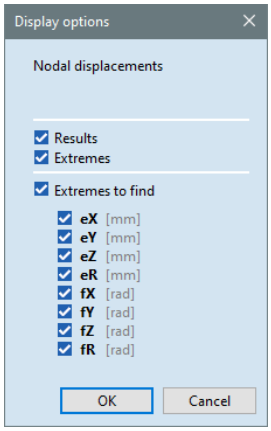


Table Browser lets you display the numerical values of the results in a table in customizable form. If you switched on parts, the table will list the values corresponding to the active parts. If you selected elements the table will list the selected elements only by default. You can change the range of listed elements by clicking the property filter button on the Table Browser toolbar.
You can transfer data to other applications via Clipboard. See 2.9 Table Browser.



Setting *Display options / Labels / Use finite element numbers* controls not only labeling but the way result tables are compiled.
For example if this option is activated you will find beam internal forces in the Table Browser under *RESULTS / Linear analysis / Internal forces / Finite elements / Beam internal forces*. Results are listed per finite element.
If it is unchecked this path is *RESULTS / Linear analysis / Internal forces / Beam internal forces* and the results are listed per structural member.

Displaying results
[Ctrl]+[R]



After calling the Table Browser you can set if you need a detailed table and/or the extremes and you can select which components you need the extremes from. This dialog can be called later from *Format / Result Display Options*.

Results Unchecking this option removes the detailed results leaving the extremes as the only content of the table.

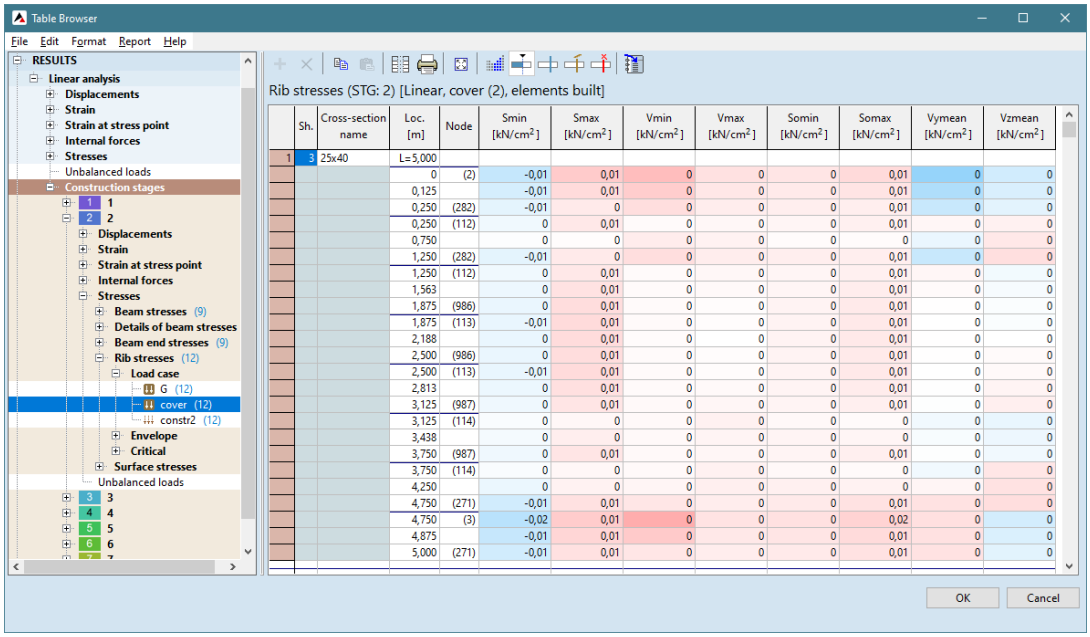
Extremes Unchecking this option removes the summary of extremes from the end of the table.

Extremes to find

The initial set of extremes to find is determined from the default column visibility of the result table. The user can set which columns (result components) should be visible by default in the result table. Only visible result components will be checked automatically.
You can set the components for which you want to find the *extreme (maximum and minimum) values*. Among the minimum and maximum values the concomitant values of the different result components are displayed if the minimum/maximum values occur in a single location or otherwise. If there are multiple locations the symbol ***** will appear, and in the Loc (location) column the first occurrence of the extreme value will be displayed.

When you display the *results of critical combinations* in addition to the minimum and maximum values, the load cases that lead to the critical values are included with the following notations:

[...] represents the results of a permanent load case.
{ ... } represents the results of an incidental load case.
(...) represents the results of an exceptional load case.



☞ **If construction stages are defined and analysed, stages are collected under separated tree nodes in RESULTS tree. The results of construction stages and the final state can be listed separately.**

Property Filtering



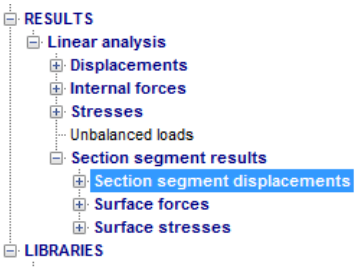
See in detail **2.9 Table Browser**

Print
[Ctrl]+[P]

Clicking the Print tool button or choosing the *File / Print* menu item the print dialog appears.
See **3.1.12 Print.**

6.1.6.1 Section segment result tables

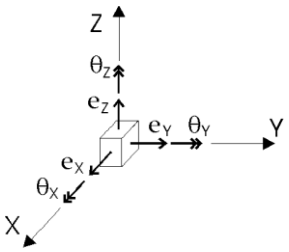
If section segments are defined in the model section segment result tables appear under the RESULTS node.
These tables list the values of result components along the active (displayed) section segment. Internal segment result points will be created where the segment plane intersected the finite element edges.



6.1.7 Displacements

Nodes

At each node, six nodal displacement components (three translations and three rotations) are obtained in the global coordinate system. The resultant values of translations *(eR)* and of rotations *(fR)* are also determined.



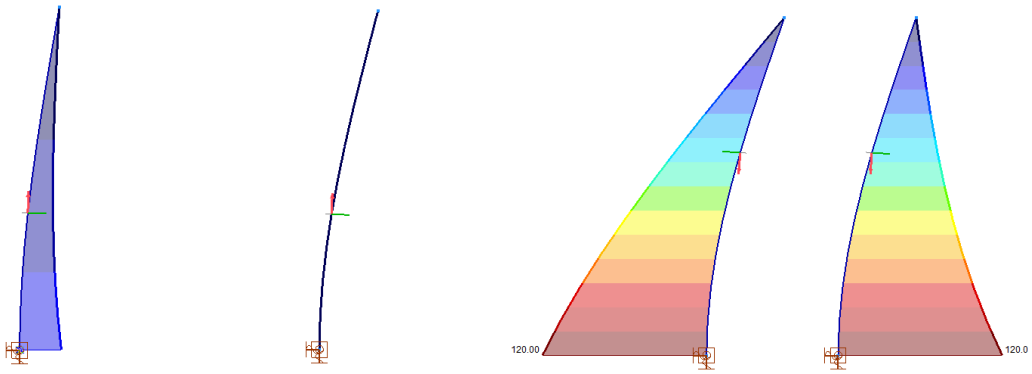
☞ **In some cases, the displacements and internal forces associated with the same critical or envelope load combination seem to be inconsistent.**

There are two reasons for this phenomenon:

- displacement components and the internal forces are calculated from different compilation of loads (eg. SLS or ULS), so the results cannot be interpreted together.
- the displacement components are obtained in the global coordinate system, but the internal forces are presented considering the local coordinate system of the elements (beam, surface, virtual beam,...). Results can only be interpreted within the appropriate the coordinate system.

Simple cases are presented below to illustrate these phenomenons. In the figures, the internal forces of the beam element is shown in deformed shape.

1. 2. 3.

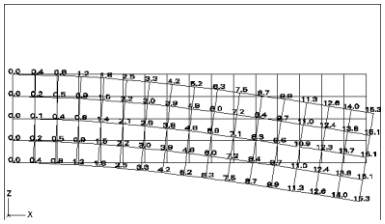


1. In critical or envelope load combinations, the tension side of the beam seems to be contradictory based on the results of the displacement and internal forces. The components come from different compilation of loads (SLS and ULS), so the results are not coherent.
2. This special case also shows the effect of different load combinations. The internal force is zero, but the deformed shape shows (significant) deformation.
3. The last case illustrates the consequences of different local coordinate systems in envelope load combination. The applied load and the calculated displacement is the same. The internal force is also identical, but the diagram of the internal force is on the opposite side due to the various local *z* axis. This may seems to be contradictory compared to the deformed shape in the interpretation of the tension or compression side).

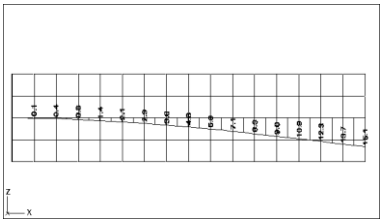
☞ **This may also occur in seismic combinations.**

Displaying the displacements of a cantilever (membrane model):

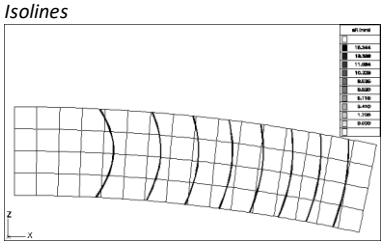
Diagram with nodal values



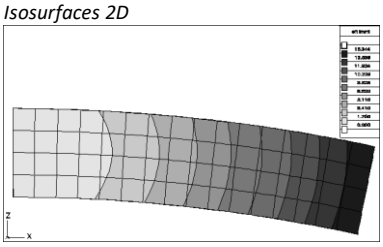
Section line with nodal values



Isolines



Isosurfaces 2D

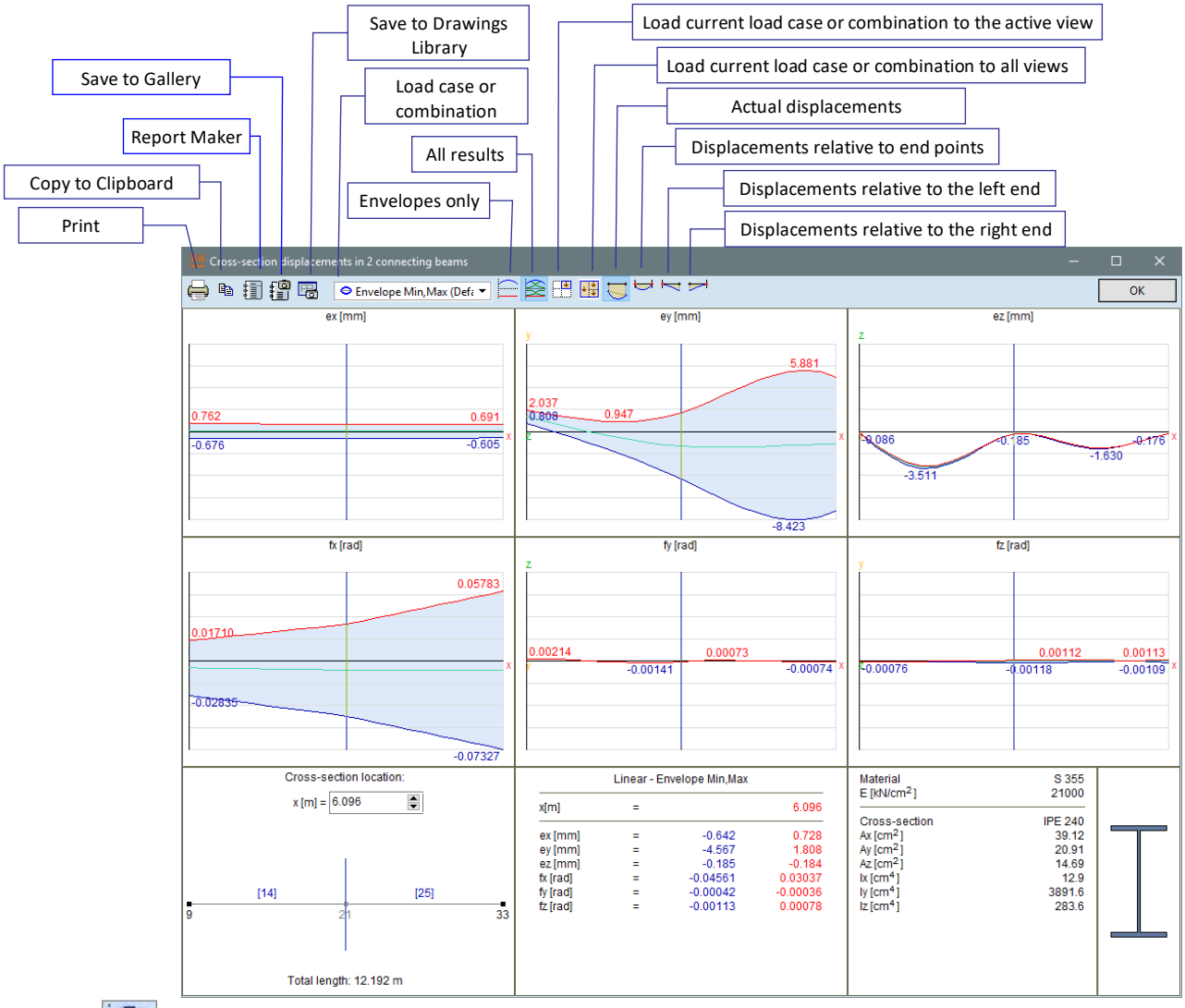


Beams

For each beam element the intermediate displacements are obtained in the local and global coordinate systems. When displaying the displacements of the structure the beam displacements are related to the **global** coordinate system. If you pick the cursor on a beam element the six beam displacement components related to the element **local** coordinate system are displayed in a diagram form.
If the model contains a 7 DOF beam (**4.9.10.3 7 DOF beam element – 7 DOF module**) an additional diagram appears for the w = dfx/dx specific rotation interpreted in the local system of the element.

You can display displacements of more than one beam element if:

a) The local coordinate system of the elements are almost or entirely identical.
See **2.16.22.3 Drawing**/ Contour line angle
b) The local x orientation is the same.
c) The elements have the same material



Display the envelope only

Display results of all load cases / combinations plus the envelope


Load current load case or combination to the active view (in case of multi-window mode the load case or combination will only be loaded to the active view)


Load current load case or combination to all views (in case of multi-window mode the load case or combination will be loaded to all views)


Actual displacements


Displacements relative to end points


Displacements relative to the left end


Displacements relative to the right end

You can display the diagrams corresponding to any load case or combination, as well as envelopes. You can turn on and off the display of envelope functions and set the position along the member where you want the results displayed.

Save diagrams to the Drawings Library



Associative diagrams can be saved to the Drawings Library. Drawings from this library can be inserted into reports. After changing and recalculating the model diagrams in the library and reports change accordingly.

Result tables

See **6.1.6 Result tables**

6.1.7.1 Nonlinear calculation of total deflection (wtot) for RC plates

Design codes

To determine the total deflection of a reinforced plate different type of vertical deflections must be calculated by running a series of nonlinear analyses for certain combinations *with* and *without* creep.

☞ ***Use reinforcement in calculation* must always be checked.**

Calculation of total deflection requires the following values

w1 *Initial part of the deflection under permanent loads*
Run nonlinear analysis for 1) permanent load cases or 2) characteristic, frequent or quasi-permanent SLS combinations without variable loads) *without* creep (φ = 0)

w2 *Long-term part of the deflection under permanent loads*
The same quasi-permanent SLS combinations must be calculated both *with* and *without* creep as *w2* is obtained as the difference between the two results.
Permanent part of the SLS combinations must include the same permanent load cases as the combination used to determine *w1*.

w3 *Additional part of the deflection due to the variable actions of the relevant combinations*
Must be calculated from the same SLS combinations as used to determine *w1* but including variable load cases as well *without* creep. *w3* is obtained as the difference between the two results.

wbij *wbij =w2+w3*

wtot *Total deflection*
wtot = w1+w2+w3

Steps of calculation

1. Generate all SLS combinations you want to take into account. Quasipermanent SLS combinations must always be included. See **4.10.2 Load combinations**

2. Run a nonlinear analysis for all combinations *without* creep. Quasipermanent SLS combinations must also be calculated *with* creep.
Use reinforcement in calculation must always be checked. See **5.1 Static analysis.**

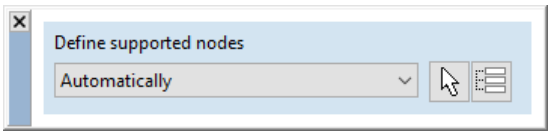
3. Select a predefined envelope of *SLS Quasipermanent* or a custom envelope created from a subset of the combinations. See **6.1 Static.**

4. If the envelope contains all the needed combinations, the program finds the set of combinations that include the same permanent load cases and determine *w1, w2, w3, wbij* and *wtot* value for each set, calculates their envelope and makes the envelope values available as *w1, w2, w3, wbij* and *wtot* result components.

6.1.7.2 Relative displacements



If the model contains domains and a linear analysis has been performed, the *Relative displacement* button on the *Static* tab is also enabled. This feature displays the displacement of domains relative to their supports.
Click on it to get to the following dialog.



Selection of domains and supported nodes

Three methods are available for defining supported nodes:

1. *Automatically*: nodes of the selected domain(s) having a nodal, linear or surface support attached will be considered to be supported if the support has a nonzero stiffness in the local *z* direction. The following nodes are also considered to be supported:
   a. nodes of the selected domain(s) with columns attached if the angle between the axis of the column and the local *z* direction is less than 45°,
   b. nodes of the selected domain(s) with walls attached if the angle between their local *z* directions is larger than 45°,
2. *Automatically (slabs only):* The above method but limited to selected slabs (a domain is considered to be a slab if its local z direction is parallel to the global Z direction).
3. *Selection:* the user can select both domains and supported nodes. Only nodes within the plane of the domain will be taken into account.

Note: one region can only have one reference plane.



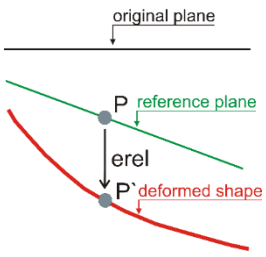
Select domains, supported nodes: click on this icon to select domains. In *Selection* mode the supported nodes must also be selected.

Analysis

The reference plane is determined from the displaced position of supported nodes. The way it is calculated depends on the number of supported nodes.

1. *No supported node:* in this case no reference is assigned to the domain and results are not calculated.
2. *One supported node:* the reference plane has the same normal as the domain and goes through the displaced position of the node.
3. *Two supported nodes:* the reference plane goes through the two displaced nodes. An in-plane vector perpendicular to the line going through these nodes is parallel to the similar vector in the plane of the undisplaced domain.
4. *Three or more nodes:* in this case the reference plane is fitted to the displaced nodes (it goes through all supported nodes for three nodes).

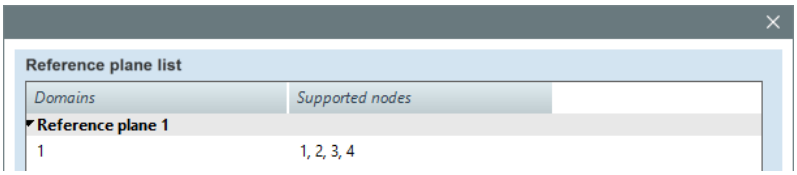
The *erel* relative displacement of a domain node is defined as the distance of P and P' where P is the intersection of the reference plane and a line perpendicular to the original plane of the domain and going through P'.



Selecting *erel* from the component list the results of the calculation can be displayed for any load case, combination, envelope or critical combination.



Reference plane list: Displays a list of reference planes with the domains where *erel* was calculated from the given reference plane and a list of supported nodes defining the reference plane. The selected reference plane can be deleted by clicking on the ***Delete*** button or by pressing *Del*.



Result tables

See 6.1.6 Result tables

Save diagrams to the Drawings Library

Diagrams can be saved to the Drawings Library and inserted into reports the usual way.

6.1.8 Nodal velocities

Dynamic analysis provides six nodal velocity components in the global coordinate system (three translational and three rotational) at each node. The resultant values of translational *(vR)* and of rotational ones *(vRR)* are also determined.

6.1.9 Nodal accelerations

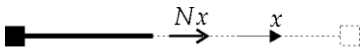
Dynamic analysis provides six nodal acceleration components in the global coordinate system (three translational and three rotational) at each node. The resultant values of translational *(aR)* and of rotational ones *(aRR)* are also determined.

6.1.10 Truss/beam internal forces

Truss

Axial internal forces (*Nx*) are calculated for each truss element.

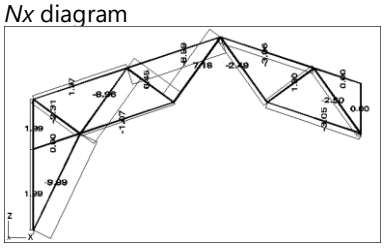
A positive axial force corresponds to tension, a negative axial force corresponds to compression.



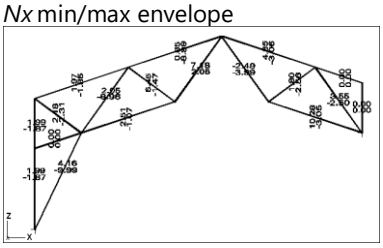
When displaying the *Envelope* and *Critical* results the minimum and maximum values can concomitantly be displayed.

Displaying the internal forces of a truss girder:

Nx diagram

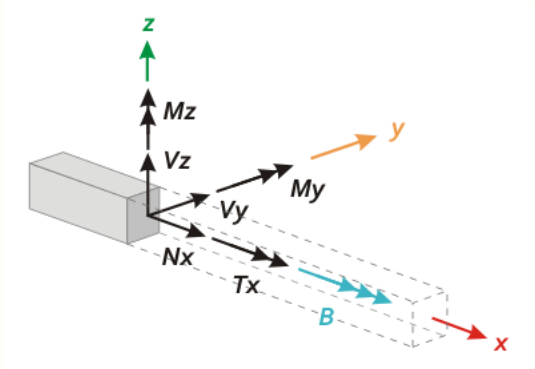


Nx min/max envelope



Beam

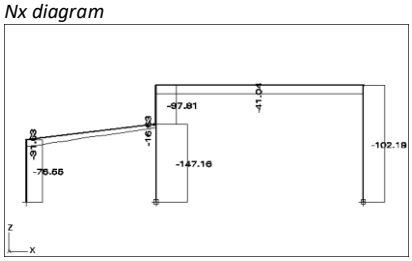
Three orthogonal internal forces, one axial and two shear forces (*Nx* , *Vy* , *Vz*) and three internal moments, one torsional and two flexural *(Tx, My, Mz)* -and for 7 DOF beams also a *B* bimoment- are calculated at the intermediate cross-sections of each element.



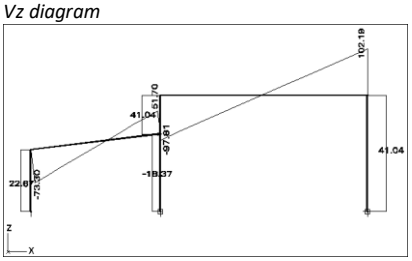
The internal forces are related to the element local coordinate system, and the positive sign conventions apply as in the figure above. The moment diagrams are drawn on the tension side of the beam elements.

Displaying the internal forces of a frame:

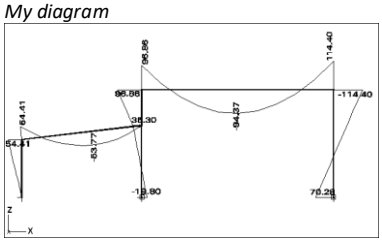
Nx diagram



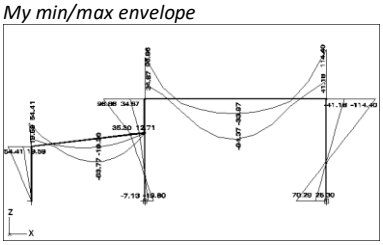
Vz diagram



My diagram

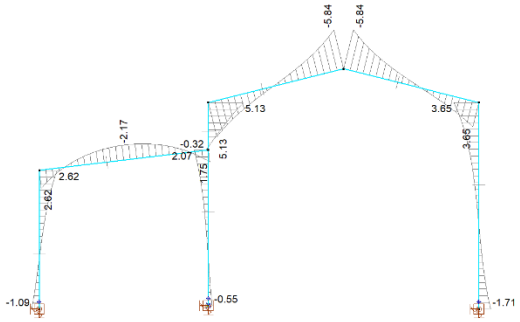


My min/max envelope

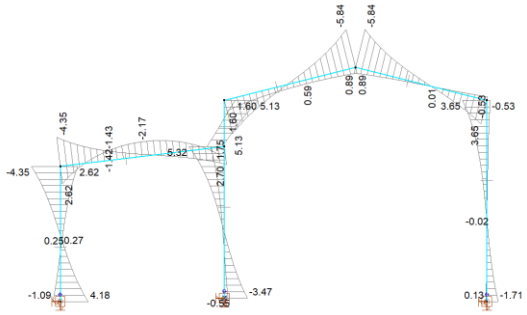


on 7 DOF beams

B diagram



B min/max diagram



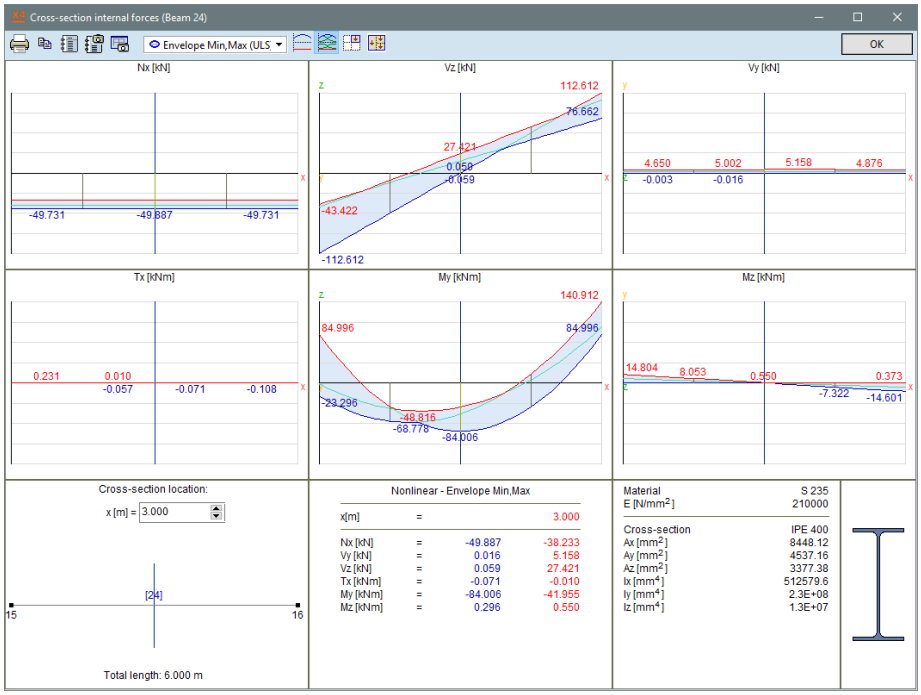
Displaying beam/truss internal forces

You can display internal forces of more than one beam/truss element if:

a) the angular deviation of local x and z axes does not exceed the respective maximum values specified in the Result display parameters / **Miscellaneous settings** dialog.

b) The elements have the same material.

If you click a beam element all six beam internal force components are displayed in a diagram form.

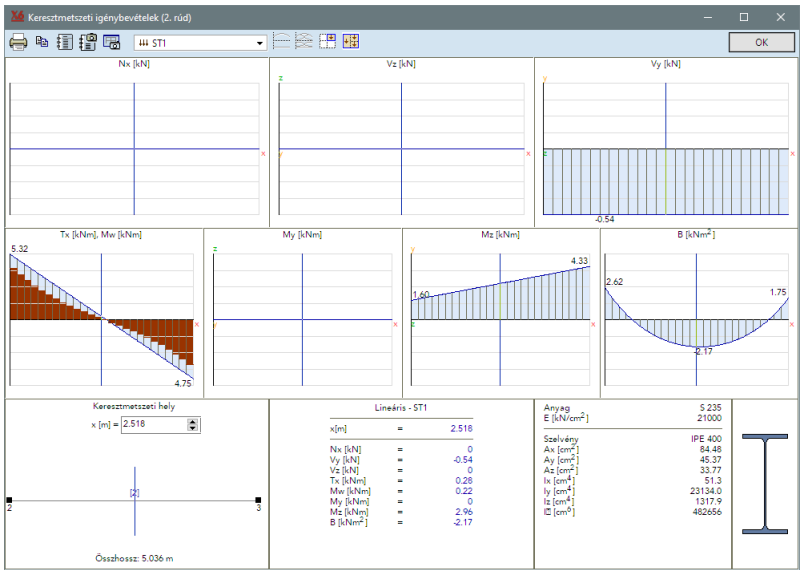


On selecting envelope or critical load combination, the selected beam internal force minimum and maximum values of the intermediate cross sections will be displayed.

You can display the diagrams corresponding to any load case or combination, as well as envelopes. You can turn on and off the display of envelope functions and set the position along the member where you want the results displayed.

Diagrams for 7 DOF beams

Besides the *Nx, Vy, Vz, Tx, My, Mz* forces and moments a diagram of the *B* bimoment also appears. In load cases and load combinations the *Tx* diagram also displays the *Mw* warping torsion.



Result tables

Three different types of the table are available.

Beam internal forces:	displays internal forces along the beam
Beam end internal forces:	displays internal forces only in start and end point of the beams
Forces for connection design:	lists internal forces of connecting ribs / beams / trusses per node

See **6.1.6 Result tables**

Save diagrams to the Drawings Library



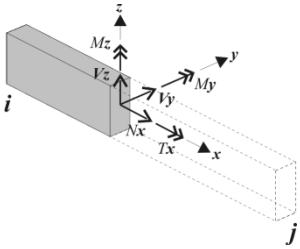
Associative diagrams can be saved to the Drawings Library. Drawings from this library can be inserted into reports. After changing and recalculating the model diagrams in the library and reports change accordingly.

If the min/max values occur in a single location the concomitant values of the afferent internal force components are displayed, or the symbol * (if there are multiple locations). An occurrence of such a location is displayed.

See **6.1.6 Result tables**

6.1.11 Rib internal forces

Three orthogonal internal forces, one axial and two shear forces *(Nx, Vy, Vz)*. and three internal moments, one torsional and two flexural *(Tx, My, Mz)* are calculated at the nodes of each element. The rib can be used independently (not connected to a surface element), or connected to a surface element.



The internal forces are related to the element local coordinate system positioned in the center of gravity of the cross-section, and the positive sign conventions apply as in the figure below. The moment diagrams are drawn on the tension side of the beam elements.

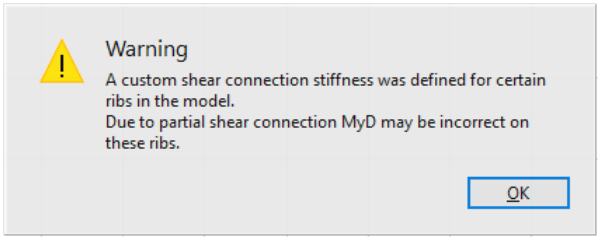
If the rib is connected eccentrically to a shell element, axial forces will appear in the rib and in the shell. In this case the design moment can be calculated as follows: $M_{yD} = M_y + e_z \cdot N_x$

Assumptions for the calculation of MyD component

The design moment M_{yD} is always calculated by the program, but the above calculation method gives good results only under the following assumptions. In all cases, the user is to check their fulfilment.

- there is a fully rigid shear connection between the rib and the shell,
- there is no significant normal force in the rib except from the bending,
- the rib is rigid/large enough compared to the slab so neglecting bending moment of the slab does not cause considerable error (bending moment of the slab is not included in M_{yD}).

M_{yD} is also calculated and displayed when elastic shear connection is defined between the slab and rib, but in this case the program gives a warning message when M_{yD} result component is selected (it appears only once after selecting this component for the first time after the analysis).



Conditions of the calculation of v_{xz} component

In case of bottom and top ribs, if *Elastic* type shear connection (see **4.9.10 Line elements/Ribs**) is set then the specific longitudinal shear force - v_{xz} result component - is also calculated. Using *Rigid* shear connection this type of result component is not available).

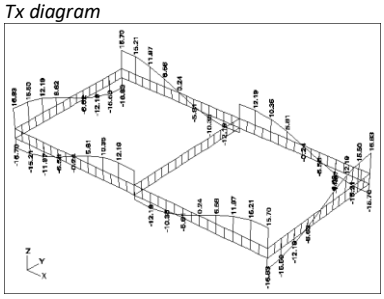
Displaying rib internal forces

You can display internal forces of more than one rib element if:

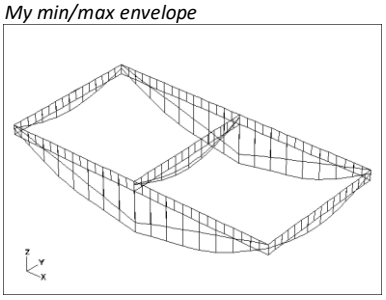
a) the angular deviation of local x and z axes does not exceed the respective maximum values specified in the Result display parameters / **Miscellaneous settings** dialog.
b) The elements have the same material.

Displaying the internal forces of a ribbed plate:

Tx diagram



My min/max envelope



Result Tables

See **6.1.6 Result tables**

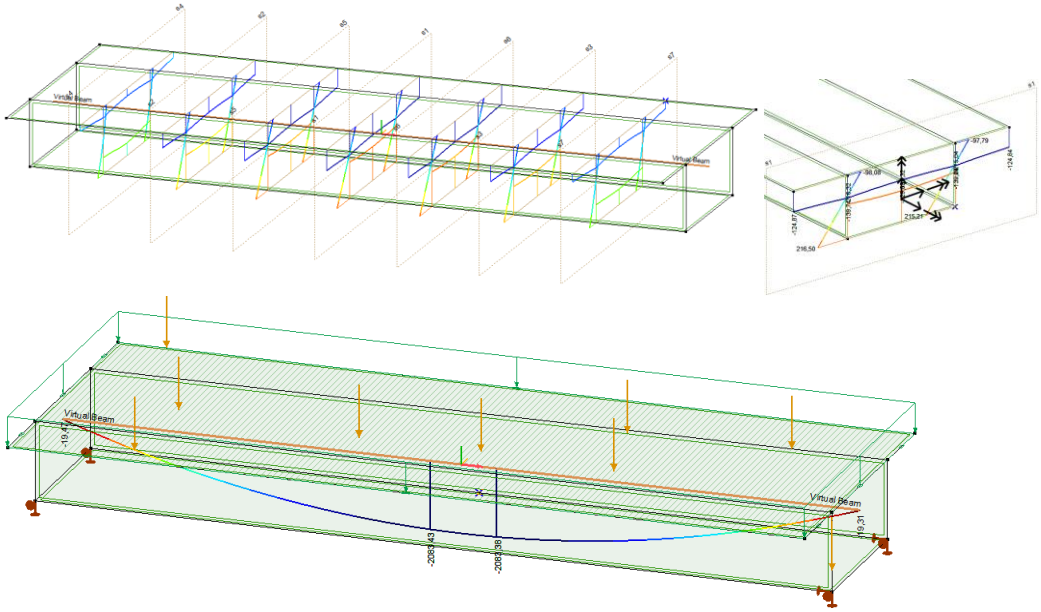
A potential error

MyD assumes that the rib is connected to surface elements. If a rib is on the edge of more than one domain, the user must ensure that the rib is connected to a domain having surface elements along the rib. Select one domain (or two domains) before defining a rib to register the rib to that domain. But if you define a rib registering it to a domain which has a hole along the rib, it is an error.
The domains registered to the rib can be checked in the element tooltip popping up if the *Elements* tab is active and the mouse hovers over the element.
Domain registration also takes into account the visible parts. If a rib is on the edge of multiple domains, and only one of these domains is visible (the parts the other domains belong to are turned off), the situation becomes clear, and the rib will be registered to the only visible domain.

6.1.12 Virtual beam internal forces

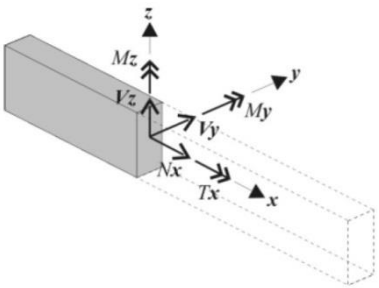
Calculation

After defining virtual beams (see **2.16.19 Virtual beams**), the program calculates center of gravity in each section to determine the centre line, on which a finite number of sections are taken, depending on the geometry, connecting elements and the density of the finite element mesh. In each section, the section forces are reduced into the intersection of the centre line and the plane of the section. After this procedure, the results of the virtual beam are plotted over the centre line.



Components of internal forces

Three orthogonal internal forces, one axial and two shear forces *(Nx , Vy , Vz)* and three internal moments, one torsional and two flexural *(Tx, My, Mz)* are calculated at the intermediate cross-sections of each element.
Internal forces are related to the local coordinate system, and the positive sign conventions apply as in the figure. Moment diagrams are drawn on the tension side of the beam elements.



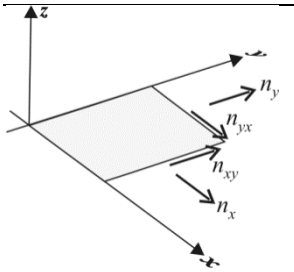
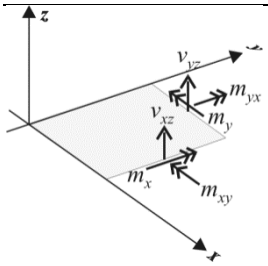
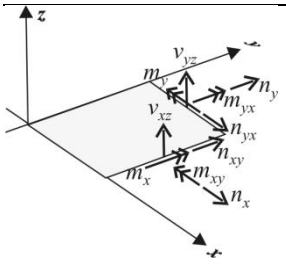
Result tables

The results are listed in all sections, with the global coordinates of the center of gravity.

6.1.13 Surface element internal forces

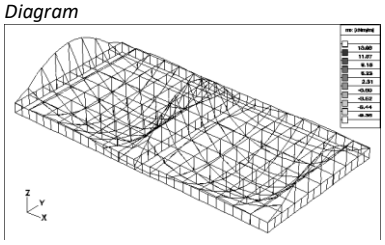
Internal forces

The internal forces and the positive sign conventions of each surface element type are summarized in the table below.

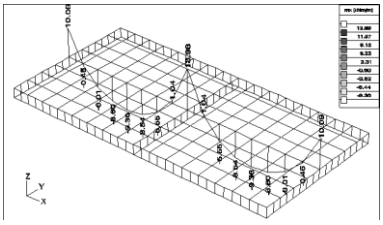
Surface elements					
Membrane	n_x n_y n_{xy}	*Plate*	m_x m_y m_{xy} v_{xz} v_{yz}	*Shell*	n_x n_y n_{xy} m_x m_y m_{xy} v_{xz} v_{yz}
					

Displaying the internal forces of a ribbed plate:

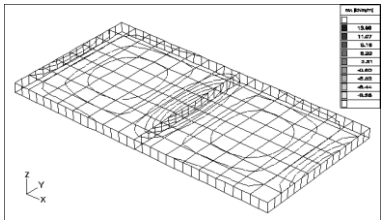
Diagram



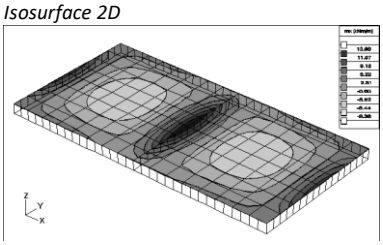
Section line



Isoline



Isosurface 2D



☞ **The x and y index of the plate moments indicates the direction of the normal stresses that occur due to the corresponding moment, and not the rotation axis.**

So, the *mx* moment rotates about the *y* local axis, while the *my* about the *x* local axis.
The moment diagrams of plate and shell elements are drawn on the tension side. On the top surface (determined by the local *z* direction) the sign is always positive, on the bottom surface it is always negative.

Intensity variation

The finite element method is an approximate method. Under *normal circumstances* the results converge to the exact values as the mesh is refined.
The refinement of the mesh (the number of the elements used in the mesh), the geometry of the elements, the loading and the support conditions, and many other parameters influence the results. Therefore some results will be relatively accurate whereas other results require the user to determine if they meet the conditions of accuracy that he expects.

The intensity variation values are intended to give you help in identifying the regions in your model (mesh) where it is possible that the accuracy of the results is not satisfactory, without performing an additional analysis. This method does not show that the results are good, but will highlight intensity variations with high magnitudes, where you may want to check and/or refine your mesh.
The allowable values of the intensity variation can be determined based on practice.

Averaging/cutting peaks of internal forces over columns

If we model columns connecting to slabs as nodal supports, peaks of internal forces will appear over the supports. If we use a denser mesh these peaks increase due to the nature of the finite element method. A more realistic model takes into account the fact that columns have a nonzero cross-section area. Knowing the column cross-section peaks can be averaged and cut. If we checked the option *Adjust mesh to column heads* (**4.12.1.2 Meshing of domains**), the mesh already follows the column cross-section. After turning on *Cut peaks of internal forces over columns* on the *Display parameters* dialog (**6.1 Static**), moment and membrane force diagrams will be averaged and the shear force diagram will be set to zero within the perimeter of the columns.

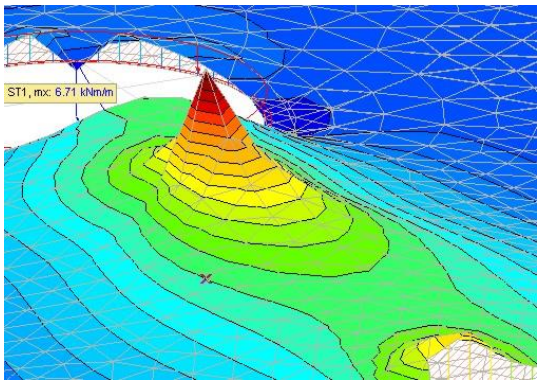


Diagram without cutting moment peaks

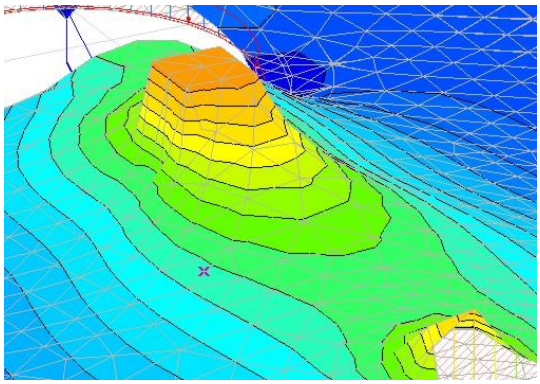


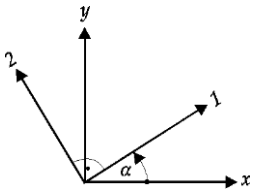
Diagram with moment peaks cut

Result Tables

See **6.1.6 Result tables**

Principal forces

The *n1, n2 and m1, m2,* principal internal forces are computed with their direction: *αn and αm.*



The direction of principal forces 1 and 2 can also be queried individually choosing *αn1, αn2 and αm1, αm2* result components.

Above these, the resultant shear force *vRz* is also calculated with its direction: *αvRz.*

The sign conventions are as follows:

$m_1 \geq m_2,\ n_1 \geq n_2,\ -90° < \alpha \leq +90°$ (relative to the local ***x*** axis)

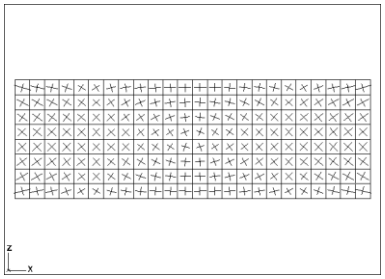
	Shell	
	Membrane	*Plate*
n_1	$n_1 = \frac{n_x + n_y}{2} + \sqrt{\left(\frac{n_x - n_y}{2}\right)^2 + n_{xy}^2}$	—
n_2	$n_2 = \frac{n_x + n_y}{2} - \sqrt{\left(\frac{n_x - n_y}{2}\right)^2 + n_{xy}^2}$	—
α_n	$\mathrm{tg}(2\alpha_n) = \frac{2n_{xy}}{n_x - n_y}$	—
m_1	—	$m_1 = \frac{m_x + m_y}{2} + \sqrt{\left(\frac{m_x - m_y}{2}\right)^2 + m_{xy}^2}$
m_2	—	$m_2 = \frac{m_x + m_y}{2} - \sqrt{\left(\frac{m_x - m_y}{2}\right)^2 + m_{xy}^2}$
α_m	—	$\mathrm{tg}(2\alpha_m) = \frac{2m_{xy}}{m_x - m_y}$
vRz	—	$vRz = \sqrt{v_{xz}^2 + v_{yz}^2}$
αvRz	—	$\alpha vRz = arctg\left(\frac{v_{yz}}{v_{xz}}\right)$

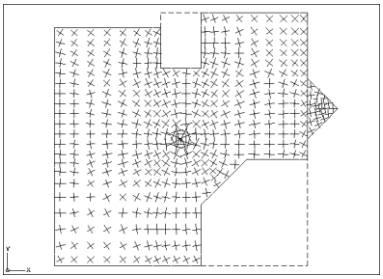
☞ **In the case of plane strain membrane elements, $n_z \neq 0$ and is not determined.**

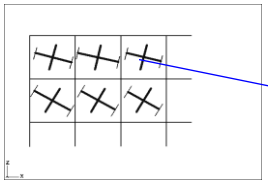
The internal forces can be displayed in diagram, section line, isoline or isosurface style.

The principal directions *(αn, αn1, αn2* and *αm, αm1, αm2* and *αvRz)* can be displayed only as diagrams+++. The color and the size of the direction vector are determined based on the value of the respective principal internal forces.

If the principal internal force is negative the corresponding direction vector is bounded by two segments perpendicular to it.







Result Tables

See **6.1.6 Result tables**

Reinforcement forces

For surface elements *nxv, nyv, mxv, myv* reinforcement (design) forces and moments are also calculated according to the following rules:

$$n_{xv} = n_x \pm |n_{xy}|, \qquad n_{yv} = n_y \pm |n_{xy}|$$
$$m_{xv} = m_x \pm |m_{xy}|, \qquad m_{yv} = m_y \pm |m_{xy}|$$

The reinforcement design forces can be displayed in diagram, section line and iso-line / surface colored form.

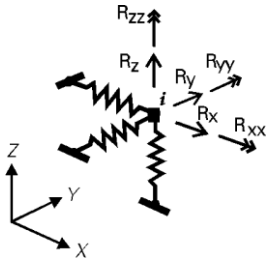
Intensity variation

dnx, dny, dnxy, dmx, dmy, dmxy, dqx, dqy are dimensionless result components showing the intensity variation within one finite element. The difference between the maximum and minimum value within the finite element is divided by a reference value. The reference value can be set to any of the following:

- Absolute maximum of the respective force component over the entire model
- Absolute maximum of the respective force component over the active parts
- Custom value

The reference value can be selected in the *Result display parameters* dialog (see **6.1 Static**).

6.1.14 Support internal forces



Positive displacement and rotation components cause positive internal force components. Elongation of the support spring elements causes tensile force, while shortening of them causes compressive force.

The internal forces can be displayed in diagram or colored form. In the case of nodal supports, when displaying in diagram form, the internal force components are represented as vectors.

Resultant internal forces

The resultant internal forces R_{eR},$R_{\Theta R}$ are computed as follows:

$$R_{eR} = \sqrt{R_x^2 + R_y^2 + R_z^2}, \qquad R_{\Theta R} = \sqrt{R_{xx}^2 + R_{yy}^2 + R_{zz}^2}$$

Rxyz and _Rxxyyzz_ result components refer to a special display mode where the individual force or moment components are displayed simultaneously as three arrows pointing in the respective local direction.

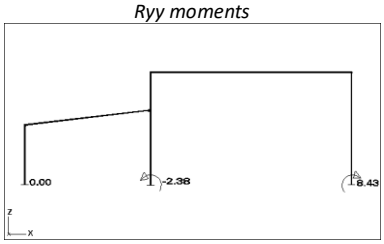
Ratio of horizontal and vertical forces

A result component showing the ratio of horizontal and vertical forces is calculated as

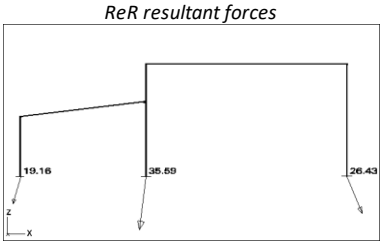
$$\alpha_R = \frac{1}{R_z}\sqrt{R_x^2 + R_y^2}$$

Displaying the internal forces of supports in a frame and a shell structure:

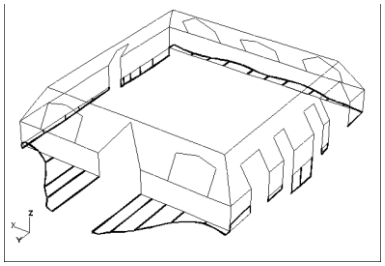
Ryy moments



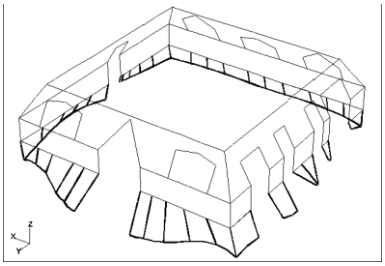
ReR resultant forces



Ry edge forces



ReR edge resultant forces

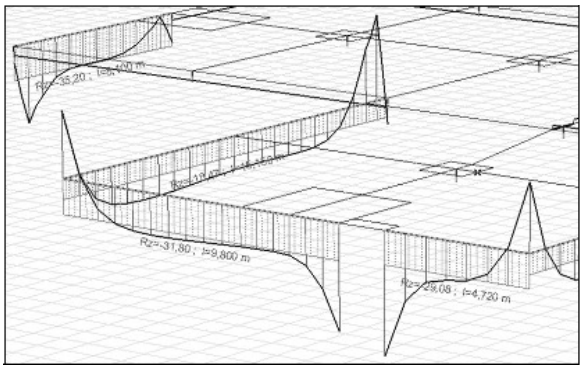


Result Tables

See **6.1.6 Result tables**

Diagram + average values

When displaying line support forces a special display mode *(Diagram + average values)* is available. If this mode is selected line support forces diagrams are enhanced with the display and labeling of the average value. Averaging is made over continuous supports. Supports are considered to be continuous if they have the same stiffness and their angle is below a small limit. Labels also show the length of the averaging segment.



Optionally results can be averaged over each structural member separately. The method of averaging can be selected in the *Result display parameters* dialog (see **6.1 Static**).

Average value

For surface support forces a separate display mode is introduced displaying the average values of *Rx, Ry* or *Rz*. Optionally results can be averaged over each domain separately. The method of averaging can be selected in the *Result display parameters* dialog (see **6.1 Static**). It is also possible to calculate average for positive and negative regions separately.

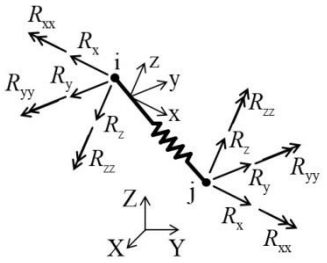
6.1.15 Internal forces of line to line link elements and edge hinges

Internal forces

AXISVM determines the *nx, ny, nz* forces and *mx, my, mz* moments for line-to-line link elements and edge hinges. If any stiffness component is set to zero the related result component is zero and not displayed neither in the component combo nor in result tables.

6.1.16 Spring element internal forces

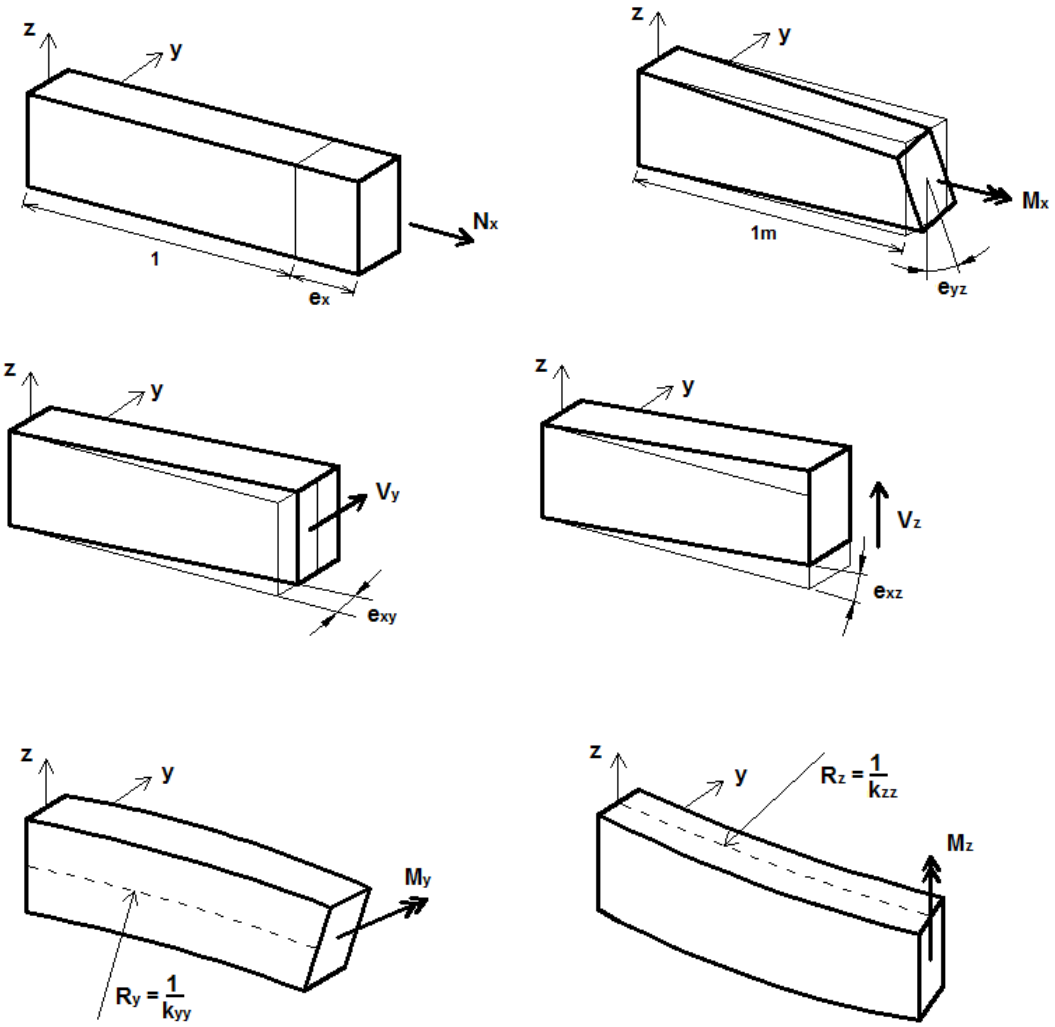
Internal forces are related to the local coordinate system. The positive sign conventions apply as in the figure.



6.1.17 Truss, beam and rib element strains

The strain results are only available in case of materially nonlinear analysis.

Strain components	
exx	Axial strain in local *x* direction
exy	Shear strain in local *xy* plane
exz	Shear strain in local *xz* plane
eyz	Torsional strain in local *x* direction
kyy	Curvature in local *xy* plane
kzz	Curvature in local *xz* plane



The following strain components are available for the line elements:

Strain component	*Truss*	*Beam*	*Rib*
exx	*exx*	*exx*	*exx*
kyy		*kyy*	*kyy*
kzz		*kzz*	*kzz*
eyz		*eyz*	*eyz*
exy			*exy*
exz			*exz*

6.1.18 Truss, beam and rib element strains at stress points

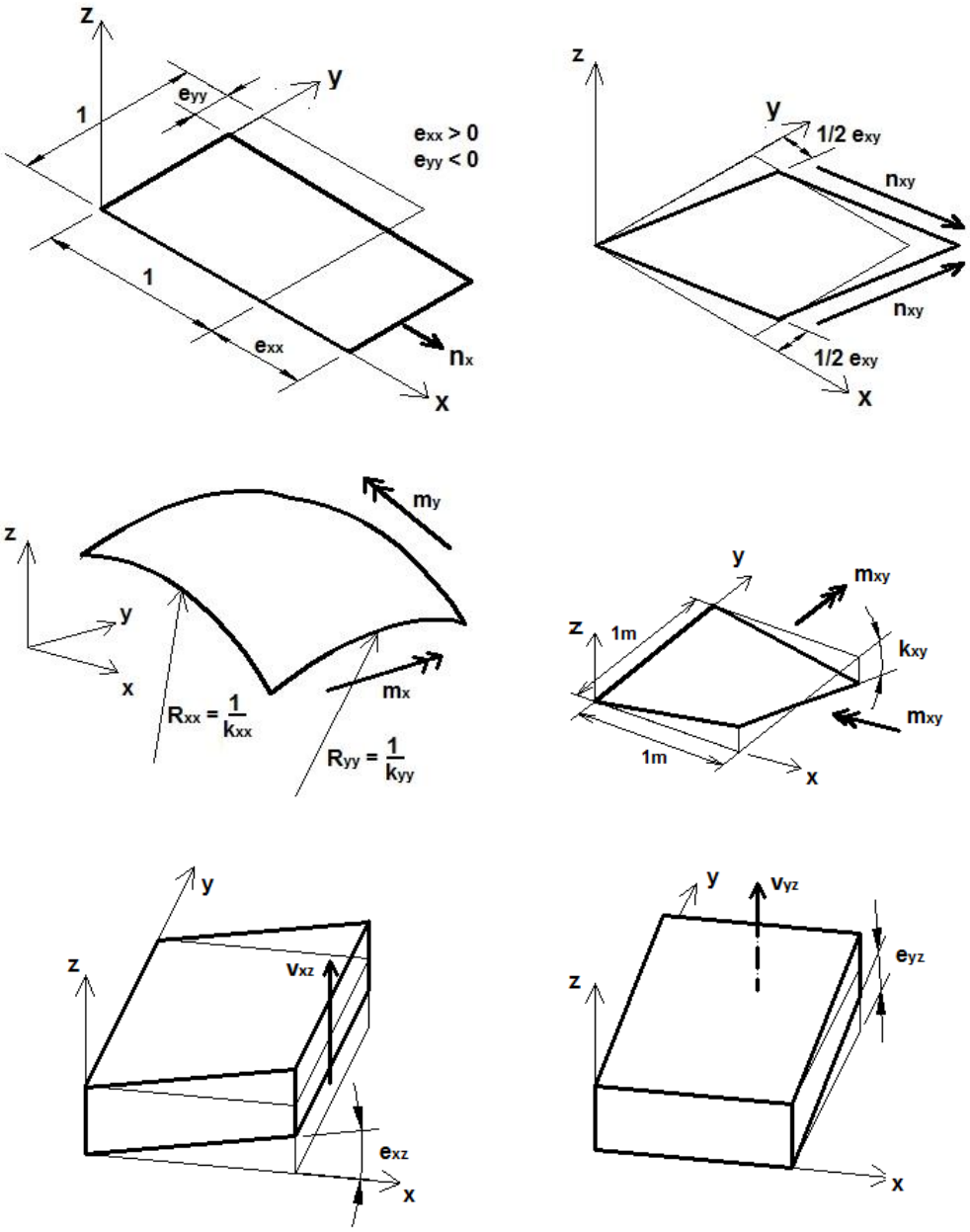
The strain results at stress points are only available in case of an analysis with material nonlinearity. The following strain values are calculated for truss elements and in each stress point of each cross-section of the beam/rib elements:

Strain components at stress points	
exx T	Axial strain in local *x* direction, total
(NLP) exx E	Axial strain in local *x* direction, elastic
(NLP) exx P	Axial strain in local *x* direction, plastic
(NLP) eeff	Effective plastic strain
(NLP) deeff	Effective plastic strain increment
(NLE) eeff	Effective strain $(NLE)\ e_{eff} = \lvert e_{xx}\ T \rvert$
Notation	(NLE) – Nonlinear elastic material (NLP) – Nonlinear plastic material T - Total E - Elastic P - Plastic

6.1.19 Surface element strains

The strain results are only available in case of materially nonlinear analysis.

Strain components	
exx	Axial strain in local *x* direction
eyy	Axial strain in local *y* direction
exy	Shear strain in local *xy* plane
kxx	Curvature in local *xz* plane
kyy	Curvature in local *yz* plane
kxy	Distortional curvature
fzz	Rotation about local z axis (drilling)
exz	Shear strain in local *xz* plane
eyz	Shear strain in local *yz* plane
eSz	Resultant shear strain normal to plane of the element $e_{Sz} = \sqrt{e_{xz}^2 + e_{yz}^2}$
e1	Maximum principal strain in local *xy* plane
e2	Minimum principal strain in local *xy* plane
ae	Angle of the principal strain direction
k1	Maximum principal curvature
k2	Minimum principal curvature
ak	Angle of the principal curvature direction



The following strain components are available for the surface elements:

Strain component	*Membrane*	*Plate*	*Shell*
exx	*exx*		*exx*
eyy	*eyy*		*eyy*
exy	*exy*		*exy*
fzz	*fzz*		*fzz*
kxx		*kxx*	*kxx*
kyy		*kyy*	*kyy*
kxy		*kxy*	*kxy*
exz		*exz*	*exz*
eyz		*eyz*	*eyz*
eSz		*eSz*	*eSz*
e1	*e1*		*e1*
e2	*e2*		*e2*
ae	*ae*		*ae*
k1		*k1*	*k1*
k2		*k2*	*k2*
ak		*ak*	*ak*

6.1.20 Surface element strains at stress points

The strain results at stress points are only available in case of materially nonlinear analysis. The following strain components are calculated at each node of the element in the top, center, and bottom fiber:

Strain components at stress points	
exx T, (NLP) exx E, (NLP) exx P	Axial strain in local *x* direction
eyy T, (NLP) eyy E, (NLP) eyy P	Axial strain in local *y* direction
exy T, (NLP) exy E, (NLP) exy P	Shear strain in local *xy* plane
e1 T, (NLP) e1 E, (NLP) e1 P	Maximum principal strain in local *xy* plane
e2 T, (NLP) e2 E, (NLP) e2 P	Minimum principal strain in local *xy* plane
ae T, (NLP) ae E, (NLP) ae P	Angle of the principal strain direction
(NLP) eeff	Effective plastic strain
(NLP) deeff	Effective plastic strain increment
(NLE) eeff	Effective strain

$$e_{eff} = \sqrt{\frac{1}{(1-2\nu)(1+\nu)}\left[(1-\nu)\left(e_{xx}^2+e_{yy}^2+e_{zz}^2\right)+2\nu\left(e_{xx}e_{yy}+e_{xx}e_{zz}+e_{yy}e_{zz}\right)+\frac{1-2\nu}{2}e_{xy}^2\right]}$$

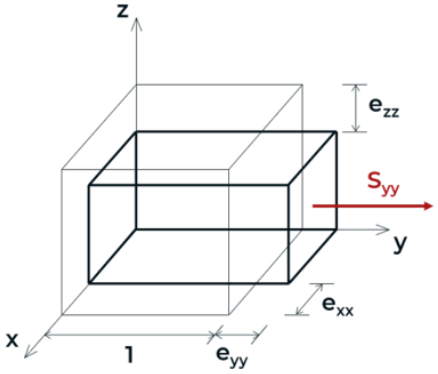
Notation

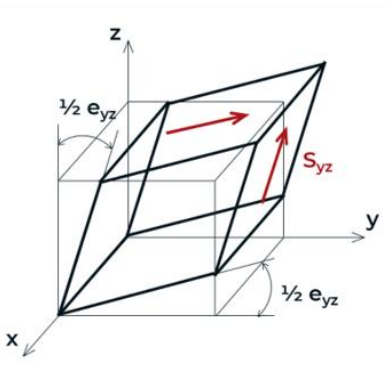
(NLE) – Nonlinear elastic material
(NLP) – Nonlinear plastic material
T – Total
E – Elastic
P – Plastic

6.1.21 Solid element strains

The strain results are available for linear and nonlinear analysis as well.

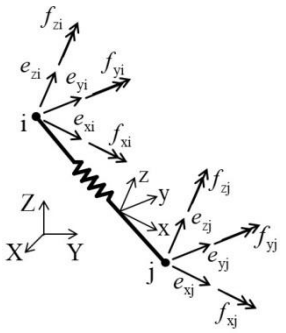
Strain components	
exx	Axial strain in local *x* direction
eyy	Axial strain in local *y* direction
ezz	Axial strain in local *z* direction
exy	Shear strain in local *xy* plane
exz	Shear strain in local *xz* plane
eyz	Shear strain in local *yz* plane
e1	Maximum principal strain
e2	Middle principal strain
e3	Minimum principal strain
ae1	Angle of the maximum principal strain direction
ae2	Angle of the middle principal strain direction
ae3	Angle of the minimum principal strain direction
ae	Angle of the principal strain direction





6.1.22 Spring element deformations

Deformation components	Deformations in local directions
ex	$e_x = e_{xj} - e_{xi}$
ez	$e_y = e_{yj} - e_{yi}$
ez	$e_z = e_{zj} - e_{zi}$
	Rotational deformations about local axes
fx	$f_x = f_{xj} - f_{xi}$
fy	$f_y = f_{yj} - f_{yi}$
fz	$f_z = f_{zj} - f_{zi}$



6.1.23 Truss/beam/rib stresses

The display modes for stress results are the same as for the internal forces. The table of the stress results are similar to those of internal forces.

Truss

The $S_x = N_x/A_x$ stress value is calculated for each truss element.
A positive value means tension.

Beams / Ribs

The following stress values are calculated in each stress point of each cross-section of the beam/rib element:

Normal stress from tension/compression and bending is calculated disregarding warping stress:

$$S_{x,i} = \frac{N_x}{A_x} + \frac{M_y I_z + M_z I_{yz}}{I_y I_z - I_{yz}^2} z_i - \frac{M_z I_y + M_y I_{yz}}{I_y I_z - I_{yz}^2} y_i$$

where y_i, z_i are the stress point coordinates. Positive stress value means tension in the cross-section.
Resultant shear stress is calculated from shear and twisting (Saint-Venant) disregarding warping shear stress. [16, 42]

For thick-walled cross-sections $V_i = \sqrt{V_{y,i}^2 + V_{z,i}^2}$,

where shear stress components are:

$$V_{y,i} = \frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial y}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial y}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial y}\right)_i - z_i\right]$$

$$V_{z,i} = \frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial z}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial z}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial z}\right)_i + y_i\right]$$

where Φ_y and Φ_z are the shear stress functions for shear in y and z direction, ω is the warping function.

For thin-walled cross-sections:

$$V_i = \left| \frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial s}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial s}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial s}\right)_i - m_i\right] \right| + \frac{M_x}{I_x} t_i$$

where the last two terms are the shear stress from twisting derived from shear flow in closed and open subsections. m_i is the distance of the center of gravity from the segment, t_i is the wall thickness of the segment. ω, Φ_y and Φ_z are centerline values.

Von Mises stress is defined as $S_{o,i} = \sqrt{S_{xi}^2 + 3V_i^2}$

If a cross-section contains two or more separate parts V_i and S_o is not calculated.

Mean shear stresses: $V_{y,mean} = \frac{V_y}{A_x}$, $V_{z,mean} = \frac{V_z}{A_x}$, if $A_y = A_z = 0$ then $A_y = A_y = A_x$

Stresses of composite cross-sections

The modulus-weighted [16, Chapter 1.2] stresses are calculated for each part of the composite cross-sections. The reference moduli E_{ref} and G_{ref} belong to the material of the outer part of the cross-section.

Normal stress from tension/compression and bending is calculated disregarding warping stress:

$$S_{x,i} = \frac{E_i}{E_{ref}}\left(\frac{N_x}{A_x} + \frac{M_y I_z + M_z I_{yz}}{I_y I_z - I_{yz}^2} z_i - \frac{M_z I_y + M_y I_{yz}}{I_y I_z - I_{yz}^2} y_i\right)$$

where y_i, z_i are the stress point coordinates and E_i is the Young's modulus of the part which the stress point belongs to.

Resultant shear stress is calculated from shear and twisting (*Saint-Venant*) disregarding warping shear stress. [16, 42]

For stress points belonging to thick-walled parts $V_i = \sqrt{V_{y,i}^2 + V_{z,i}^2}$,

where shear stress components are:

$$V_{y,i} = \frac{G_i}{G_{ref}}\left(\frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial y}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial y}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial y}\right)_i - z_i\right]\right)$$

$$V_{z,i} = \frac{G_i}{G_{ref}}\left(\frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial z}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial z}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial z}\right)_i + y_i\right]\right)$$

where Φ_y and Φ_z are the shear stress functions for shear in y and z direction, ω is the warping function and G_i is the shear modulus of the part which the stress point belongs to.

For stress points belonging to thin-walled parts:

$$V_i = \frac{G_i}{G_{ref}}\left\{\left|\frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial s}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial s}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial s}\right)_i - m_i\right]\right| + \frac{M_x}{I_x} t_i\right\}$$

where the last two terms are the shear stress from twisting derived from shear flow in closed and open subsections. m_i is the distance of the centre of gravity from the segment, t_i is the wall thickness of the segment. ω, Φ_y and Φ_z are centerline values.

Von Mises stress is defined as $S_{o,i} = \sqrt{S_{xi}^2 + 3V_i^2}$

☞ **Stress points on the boundary of the two component cross-sections always belong to the thin-walled cross-section.**

For composite cross-sections $V_{y,mean}$ and $V_{z,mean}$ is not calculated.

☞ **The modulus-weighted computation method is based on the assumption that there is no shear connection between the parts of the composite cross-section. However, it has no effect for the normal stress, because of the coincidence of the center of gravity of the parts. But the shear stress would be different if there was shear connection.**

7DOF beams

In case of 7DOF beams the warping stresses are also regarded [16, 42].

Normal stress from tension/compression, bending and restrained warping:

$$S_{x,i} = \frac{N_x}{A_x} + \frac{M_y I_z + M_z I_{yz}}{I_y I_z - I_{yz}^2} z_i - \frac{M_z I_y + M_y I_{yz}}{I_y I_z - I_{yz}^2} y_i + \frac{B}{I_\omega}\omega_i$$

where y_i, z_i are the stress point coordinates and ω_i is the measure of warping. Positive stress value means tension in the cross-section.

Resultant shear stress is calculated from shear, twisting (*Saint-Venant*) and restrained warping.

For thick-walled cross-sections $V_i = \sqrt{V_{y,i}^2 + V_{z,i}^2}$,

where shear stress components are:

$$V_{y,i} = \frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial y}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial y}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial y}\right)_i - z_i\right] + \frac{M_w}{I_\omega}\left(\frac{\partial \Phi_\omega}{\partial y}\right)_i$$

$$V_{z,i} = \frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial z}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial z}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial z}\right)_i + y_i\right] + \frac{M_w}{I_\omega}\left(\frac{\partial \Phi_\omega}{\partial z}\right)_i$$

where Φ_y and Φ_z are the shear stress functions for shear in y and z direction, Φ_ω is the shear stress function for warping and ω is the warping function.

For thin-walled cross-sections:

$$V_i = \left|\frac{V_y}{A_x}\left(\frac{\partial \Phi_y}{\partial s}\right)_i + \frac{V_z}{A_x}\left(\frac{\partial \Phi_z}{\partial s}\right)_i + \frac{M_x}{I_x}\left[\left(\frac{\partial \omega}{\partial s}\right)_i - m_i\right] + \frac{M_w}{I_\omega}\left(\frac{\partial \Phi_\omega}{\partial s}\right)_i\right| + \frac{M_x}{I_x} t_i$$

where the last three terms are the shear stress from twisting derived from shear flow in closed subsections, from warping and from twisting derived from shear flow in open subsections. m_i is the distance

of the center of gravity from the segment, t_i is the wall thickness of the segment. ω, Φ_y, Φ_z and Φ_ω are centerline values.

Von Mises stress is defined as $S_{o,i} = \sqrt{S_{xi}^2 + 3V_i^2}$

Further stress components for nonlinear material

NL elastic / Strain energy based material model

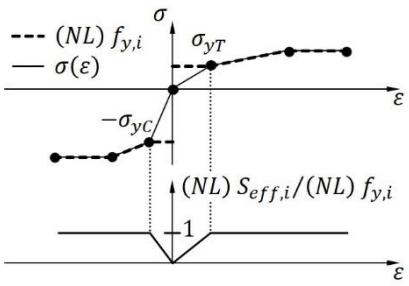
Effective stress.
Only the normal stress affects the materially nonlinear behaviour of truss, beam and rib elements. The shear stress has no effect (further information see at **References** [5])

$$(NL)\ S_{eff,i} = |S_{x.i}|$$

Actual yield strength
It is the actual limit of proportionality influenced by the sign and the magnitude of the normal stress

$$(NL)\ f_{y,i} = \begin{cases} \sigma(\varepsilon_{eff,i}) & if \quad S_{x,i} > \sigma_{yT} \\ \sigma_{yT} & if \quad 0 \le S_{x,i} \le \sigma_{yT} \\ \sigma_{yC} & if \quad -\sigma_{yC} \le S_{x,i} \le 0 \\ |\sigma(-\varepsilon_{eff,i})| & if \quad S_{x,i} < -\sigma_{yC} \end{cases}$$

where σ_{yT} and σ_{yC} are the closest points of the stress-strain diagram of the material (see **3.1.15 Material Library**) on the positive and on the negative region and $\varepsilon_{eff,i}$ is the effective strain [(NLE) *eeff*, see **6.1.18 Truss, beam and rib element strains at stress points**]



Utilization:

$$(NL)\ Utilization_i = \frac{(NL)\ S_{eff,i}}{(NL)\ f_{y,i}}$$

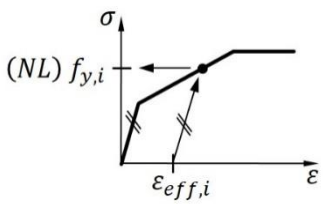
Plastic / von Mises material model

Effective stress.
Only the normal stress affects the materially nonlinear behaviour of truss, beam and rib elements. The shear stress has no effect [5]

$$(NL)\ S_{eff,i} = |S_{r,x.i}|$$

where $S_{r,x,i} = S_{x,i} - B_{x,i}$ is the relative stress with respect to the centre of the yield surface. See **3.1.15 Material Library**

Actual yield strength
It is the actual yield strength influenced by the hardening effect. It is computed on the basis of the stress-strain diagram of the material (see **3.1.15 Material Library**)



where $\varepsilon_{eff,i}$ is the effective plastic strain ((NLP) *eeff*, see **6.1.18 Truss, beam and rib element strains at stress points)**

Utilization:

$$(NL)\ Utilization_i = \frac{(NL)\ S_{eff,i}}{(NL)\ f_{y,i}}$$

Plastic / Bresler-Pister material model

Effective stress.

Only the normal stress affects the materially nonlinear behaviour of truss, beam and rib elements. The shear stress has no effect [5]

$$(NL)\ S_{eff,i} = |S_{x.i}|$$

Actual yield strength

It is the actual yield strength influenced by the sign of the normal stress:

$$(NL)\ f_{y,i} = \begin{cases} \sigma_{yT} & if \quad S_{x,i} \geq 0 \\ \sigma_{yC} & if \quad S_{x,i} < 0 \end{cases}$$

where σ_{yT} and σ_{yC} are tensile and compressive yield strength respectively (see **3.1.15 Material Library**)

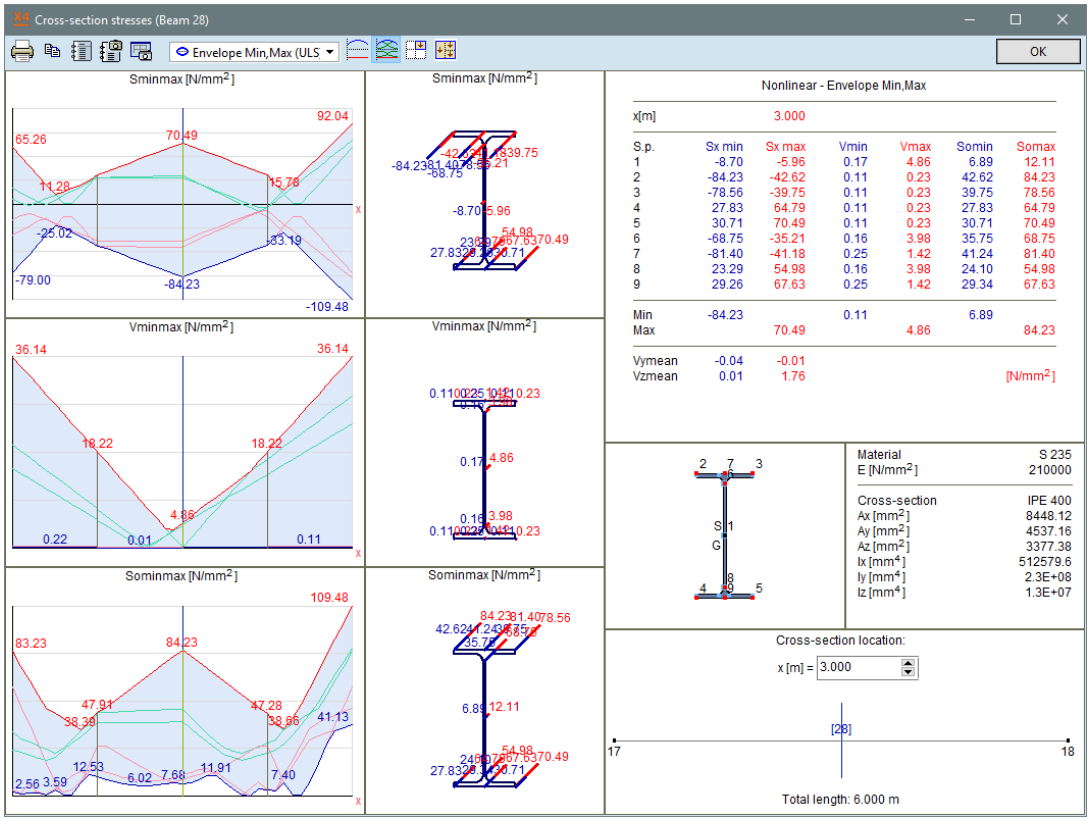
Utilization:

$$(NL)\ Utilization_i = \frac{(NL)\ S_{eff,i}}{(NL)\ f_{y,i}}$$

Beam stresses *Sminmax, Vminmax, Sominmax* are minimum / maximum values within the cross-section and displayed like internal forces.

You can click a beam/rib element to display stress diagrams. On the left the minimum/maximum values along the line are displayed. Dragging the blue line with the mouse the evaluation position can be changed. The axonometric diagrams in the middle and the tables on the right show the stress distribution within the section at the evaluation point.

Select more elements before clicking to display them in one diagram. Continuous beams/ribs can be displayed in one diagram if conditions described in section **6.1.10 Truss/beam internal forces** are satisfied.



You can display the diagrams corresponding to any load case or combination, as well as envelopes. You can turn on and off the display of envelope functions and set the position along the member where you want the results displayed.

Save diagrams to the Drawings Library

Associative diagrams can be saved to the Drawings Library. Drawings from this library can be inserted into reports. After changing and recalculating the model diagrams in the library and reports change accordingly.

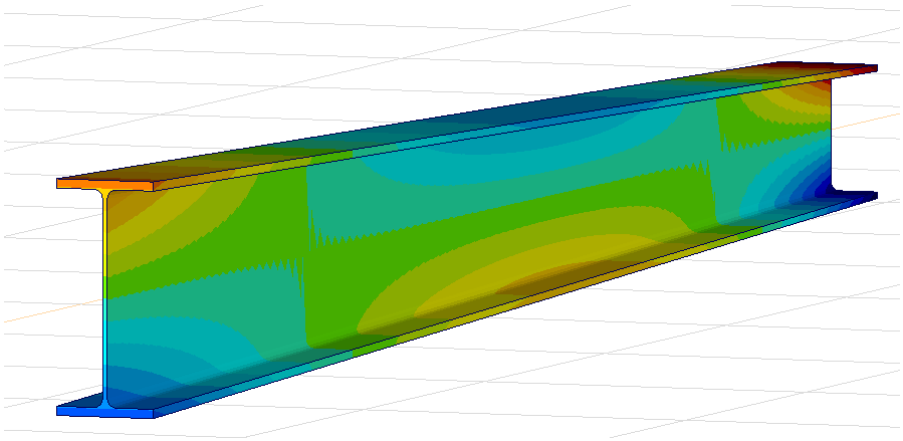
Selecting envelope or critical combinations only one of the min and max components will appear depending on the component. If extreme values are located in one cross-section only you will see values of the other components as well. Otherwise a * will appear and the cross-section location will be the first one.

Result Tables

See **6.1.6 Result tables**

Result display on a solid model

Distribution of normal stress in the beam cross-section can be displayed in the *Solid model* result display mode.



6.1.24 Surface element stresses

The following stress components are calculated at each node of the element in the top, center, and bottom fiber:

Component	Membrane	Plate	Shell
s_{xx}	$s_{xx}=\frac{n_x}{t}$	$s_{xx}=\pm\frac{6}{t^2}m_x$	$s_{xx}=\frac{n_x}{t}\pm\frac{6}{t^2}m_x$
s_{yy}	$s_{yy}=\frac{n_y}{t}$	$s_{yy}=\pm\frac{6}{t^2}m_y$	$s_{yy}=\frac{n_x}{t}\pm\frac{6}{t^2}m_y$
s_{xy}	$s_{xy}=\frac{n_{xy}}{t}$	$s_{xy}=\pm\frac{6}{t^2}m_{xy}$	$s_{xy}=\frac{n_{xy}}{t}\pm\frac{6}{t^2}m_{xy}$
s_{xz}		$s_{xz}=\frac{3v_{xz}}{2t}$	$s_{xz}=\frac{3v_{xz}}{2t}$
s_{yz}		$s_{yz}=\frac{3v_{yz}}{2t}$	$s_{yz}=\frac{3v_{yz}}{2t}$
s_1	$s_1=\frac{s_x+s_y}{2}+\sqrt{\left(\frac{s_x-s_y}{2}\right)^2+s_{xy}^2}$		
s_2	$s_2=\frac{s_x+s_y}{2}-\sqrt{\left(\frac{s_x-s_y}{2}\right)^2+s_{xy}^2}$		
α_s	$\mathrm{tg}(2\alpha_s)=\frac{2s_{xy}}{s_x-s_y}$		

☞ **In the case of plane strain membrane elements $s_{zz}\neq 0$, and is determined as $s_{zz}=v\cdot(s_{xx}+s_{yy})$**

Von Mises stress

The Von Mises stress is computed:

$$s_0=\sqrt{0.5\left[(s_{xx}-s_{yy})^2+(s_{yy}-s_{zz})^2+(s_{zz}-s_{xx})^2\right]+3(s_{xy}^2+s_{yz}^2+s_{zx}^2)}$$

Further stress components for nonlinear materials

NL elastic / Strain energy based material model

Effective stress

$$(NL)\ s_{eff}=\sqrt{s_{xx}^2+s_{yy}^2+s_{zz}^2-2v(s_{xx}s_{yy}+s_{xx}s_{zz}+s_{yy}s_{zz})+2(1+v)s_{xy}^2}$$

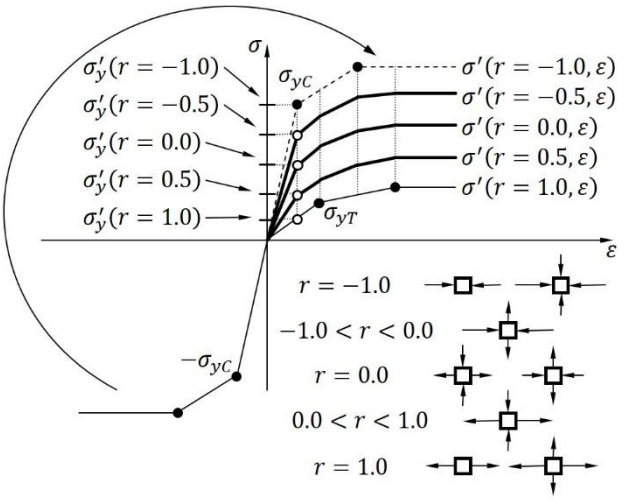
☞ **Only the normal stresses s_{xx}, s_{yy}, s_{zz} and the shear stress s_{xy} affect the materially nonlinear behaviour of surface elements. The shear stresses s_{xz} and s_{yz} have no effect. Further information see at References [5]**

Actual yield strength

It is the limit of proportionality influenced by the actual multiaxial stress state:

$$(NL)\ f_y=\begin{cases}\sigma_y'(r) & \text{if } 0\le s_{eff}\le\sigma_y'(r)\\ \sigma'(r,\varepsilon_{eff}) & \text{if } s_{eff}>\sigma_y'(r)\end{cases}$$

where $\sigma'(r,\varepsilon)$ is the interpolated stress-strain diagram relating to the actual multiaxial stress state, $\sigma_y'(r)$ is its closest point to the origin and ε_{eff} is the effective strain [(NLE) *eeff*, see 6.1.20 Surface element strains at stress points).



where r is the ratio of the tensile and compressive components of the multiaxial stress state:

$$r = \begin{cases} -1 & if \quad \frac{I_1}{\sigma_{VM}} < -1 \\ \frac{I_1}{\sigma_{VM}} & if \quad -1 \le \frac{I_1}{\sigma_{VM}} \le 1 \\ 1 & if \quad \frac{I_1}{\sigma_{VM}} > 1 \end{cases}$$

where I_1 is the first invariant of the stress tensor and σ_{VM} is the von Mises equivalent stress:

$$I_1 = s_{xx} + s_{yy} + s_{zz}$$

$$\sigma_{VM} = \sqrt{0.5\left[\left(s_{xx} - s_{yy}\right)^2 + \left(s_{yy} - s_{zz}\right)^2 + \left(s_{zz} - s_{xx}\right)^2\right] + 3s_{xy}^2}$$

Utilization:

$$(NL)\ Utilization = \frac{(NL)\ s_{eff}}{(NL)\ f_y}$$

Plastic / von Mises material model

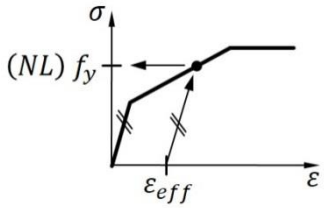
Effective stress is calculated according to the yield criterion:

$$(NL)\ s_{eff} = \sqrt{0.5\left[\left(s_{r,xx} - s_{r,yy}\right)^2 + \left(s_{r,yy} - s_{r,zz}\right)^2 + \left(s_{r,zz} - s_{r,xx}\right)^2\right] + 3s_{r,xy}^2}$$

where $s_{r,xx} = s_{xx} - b_{xx}$, $s_{r,yy} = s_{yy} - b_{yy}$, $s_{r,zz} = s_{zz} - b_{zz}$ and $s_{r,xy} = s_{xy} - b_{xy}$ are the relative stresses with respect to the centre of the yield surface (see **3.1.15 Material Library**)

☞ **Only the normal stresses s_{xx}, s_{yy}, s_{zz} and the shear stress s_{xy} affect the materially nonlinear behaviour of surface elements. The shear stresses s_{xz} and s_{yz} have no effect. [5]**

Actual yield strength
It is the yield strength influenced by hardening effect. It is computed on the basis of the stress-strain diagram of the material (see **3.1.15 Material Library**).



where ε_{eff} is the effective plastic strain *(NLP) eeff*, see **6.1.20 Surface element strains at stress points**

Utilization:

$$(NL)\ Utilization = \frac{(NL)\ s_{eff}}{(NL)\ f_y}$$

Plastic / Bresler-Pister material model

The *effective stress* is defined by the Euclidean distance of the point $[s_1, s_2, s_3]$ from the origin in the space of pricipal stresses:

$$(NL)\ s_{eff} = \sqrt{s_1^2 + s_2^2 + s_3^2}$$

where $[s_1, s_2, s_3]$ is the principal stress representation of the stress state $[s_{xx}, s_{yy}, s_{zz}, s_{xy}]$.

☞ **Only the normal stresses s_{xx}, s_{yy}, s_{zz} and the shear stress s_{xy} affect the materially nonlinear behaviour of surface elements. The shear stresses s_{xz} and s_{yz} have no effect. [5]**

Actual yield strength

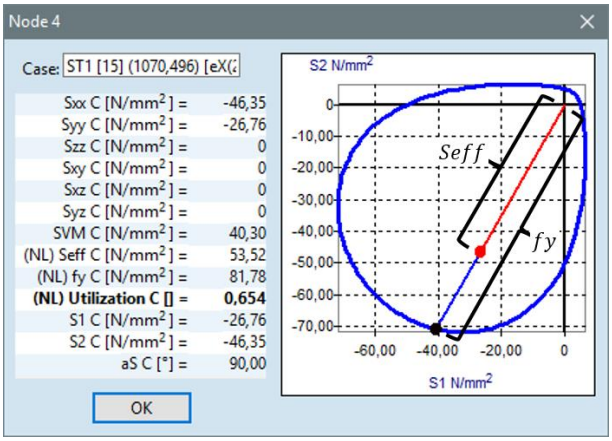
It is the yield strength influenced by the stress state

$$(NL)\ f_y = \sigma(s_1, s_2, s_3)$$

where $\sigma(s_1, s_2, s_3)$ is the point on the yield surface determined by the actual stress state (see the picture below and see 3.1.15 Material Library).

Utilization:

$$(NL)\ Utilization = \frac{(NL)\ s_{eff}}{(NL)\ f_y}$$



Stress values can be displayed as a diagram, section diagram, as isolines or isosurfaces.

Result Tables See 6.1.6 Result tables

Stress state for non-linear materials

It is a result component to identify the tensioned, compressed and sheared regions of the structure. Its numerical value is based on the value of the so called *stress triaxiality factor* that is the ratio of the hydrostatic and the deviatoric stress component.

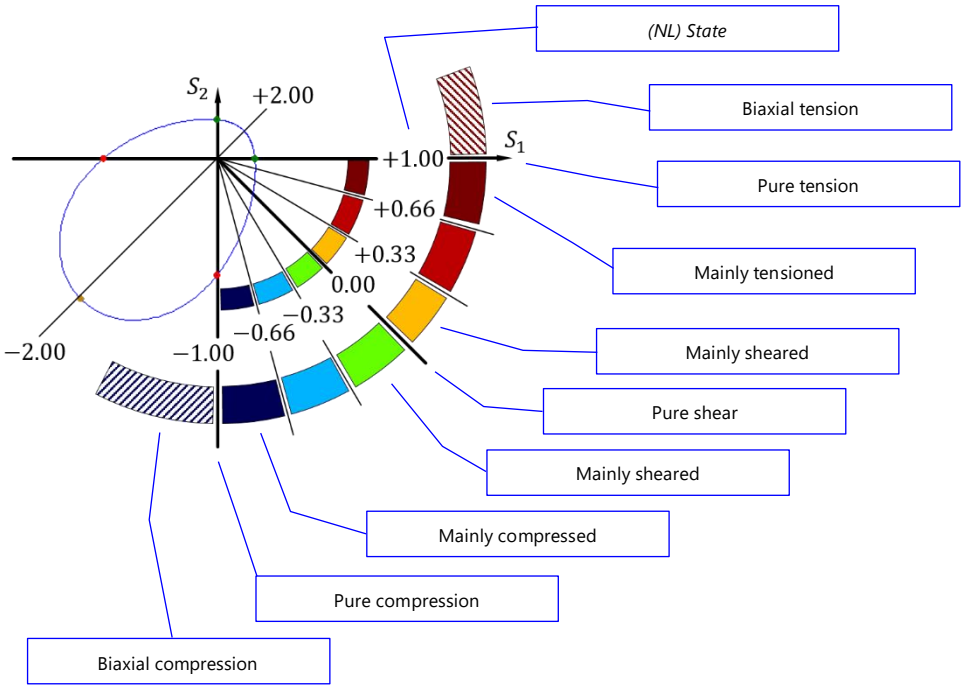
$$(NL)\ State = 3\frac{\sigma_m}{\sigma_{VM}}$$

where σ_m is the hydrostatic or mean stress and σ_{VM} is the von Mises equivalent stress characteristic for the deviatoric or shear stresses:

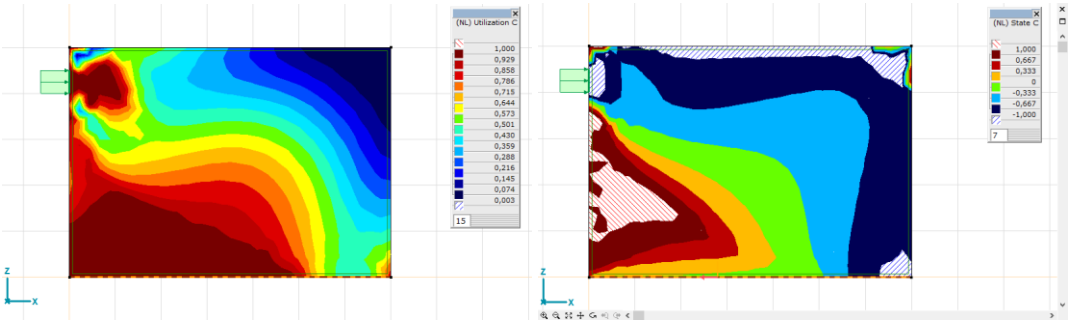
$$\sigma_m = \frac{s_{xx} + s_{yy} + s_{zz}}{3}$$

$$\sigma_{VM} = \sqrt{0.5\left[(s_{xx} - s_{yy})^2 + (s_{yy} - s_{zz})^2 + (s_{zz} - s_{xx})^2\right] + 3s_{xy}^2}$$

The following picture illustrates the stress state on the plane of the pricipal stresses S1-S2 with the belonging values of *(NL) State*.



Using *(NL) Utilization* together with *(NL) State* the critical regions and the failure mode of them can be determined.



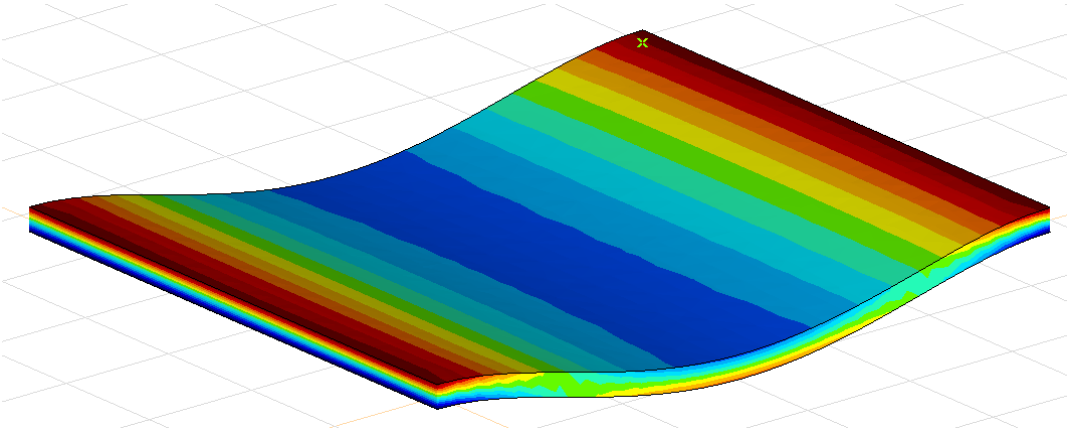
Stress values can be displayed as a diagram, section diagram, as isolines or isosurfaces.

Result Tables

See **6.1.6 Result tables**

Result display on a solid model

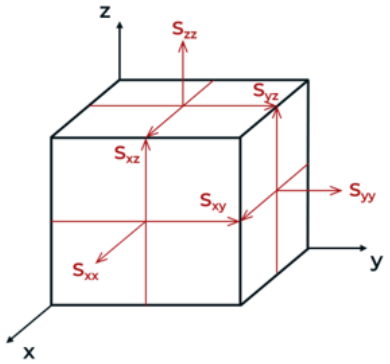
Top and bottom values of surface stresses can be displayed simultaneously in the *Solid model* result display mode.



6.1.25 Solid element stresses

The following stress components are calculated at each node of the solid element:

Strain components	
Sxx	Axial strain in local *x* direction
Syy	Axial strain in local *y* direction
Szz	Axial strain in local *z* direction
Sxy	Shear strain in local *xy* plane
Sxz	Shear strain in local *xz* plane
Syz	Shear strain in local *yz* plane
Seff	For linear material, this is the Von Mises stress: $S_{eff} = \sqrt{0.5\left[\left(S_{xx} - S_{yy}\right)^2 + \left(S_{yy} - S_{zz}\right)^2 + \left(S_{zz} - S_{xx}\right)^2\right] + 3\left(S_{xy}^2 + S_{xz}^2 + S_{yz}^2\right)}$
S1	Maximum principal stress
S2	Middle principal stress
S3	Minimum principal stress
aS1	Angle of the maximum principal stress direction
aS2	Angle of the middle principal stress direction
aS3	Angle of the minimum principal stress direction
aS	Angle of the principal stress direction



6.1.26 Spring element nonlinear results

Translational components	Description
e,x T	Deformation in local x direction
(NLP) e,x E	Elastic deformation in local x direction
(NLP) e,x P	Plastic deformation in local x direction
(NLP) eeff,x P	Effective plastic deformation in local x direction
(NLP) deeff,x P	Effective plastic deformation increment in local x direction (static) Effective plastic deformation rate in local x direction (dynamic)
R,x	Spring force in local x direction
(NL) Fy,x	Actual limit force in local x direction
(NL) Util.,x	Utilization in local x direction
...,y	... in local y direction
...,z	... in local z direction
Rotational components	**Description**
e,xx T	Rotational deformation about local x axis
(NLP) e,xx E	Elastic rotational deformation about local x axis
(NLP) e,xx P	Plastic rotational deformation about local x axis
(NLP) eeff,xx P	Effective plastic rotational deformation about local x axis
(NLP) deeff,xx P	Effective plastic rotational deformation increment about local x axis (static) Effective plastic rotational deformation rate about local x axis (dynamic)
R,xx	Moment about local x axis
(NL) My,xx	Actual limit moment about local x axis
(NL) Util.,xx	Rotational utilization about local x axis
...,yy	... about local y axis
...,zz	... about local z axis

NL elastic behaviour

Actual limit force, actual limit moment

It is the actual limit strength influenced by the sign of the load:

$$(NL)\ F_{y,x} = \begin{cases} F_{yT} & ha \quad R_x \geq 0 \\ F_{yC} & ha \quad R_x < 0 \end{cases}$$

where F_{yT} and F_{yC} are the limit forces defined by the user for tension and compression.

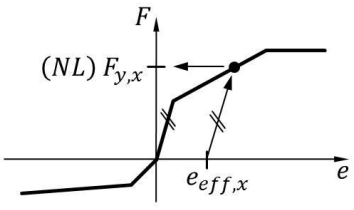
Utilization:

$$(NL)\ Utilization_x = \left| \frac{R_x}{(NL)\ F_{y,x}} \right|$$

Plastic behaviour

Actual limit force, actual limit moment

It is the actual limit strength influenced by the hardening effect. It is computed on the basis of the force-displacement diagram of the material:



where $e_{eff,x}$ is the effective plastic deformation ((NLP) *eeff,x P).*

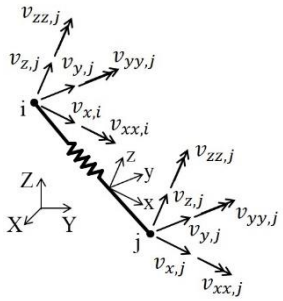
Utilization:

$$(NL)\ Utilization_x = \left| \frac{R_x}{(NL)\ F_{y,x}} \right|$$

6.1.27 Spring element dynamic results

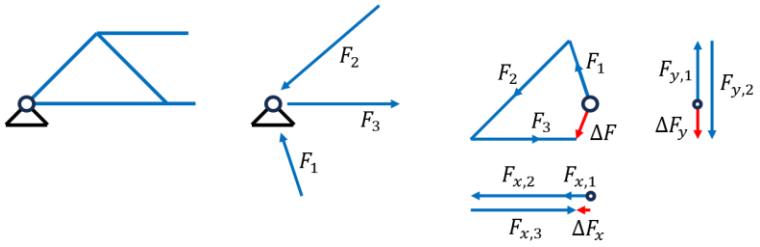
Translational components	Description
(D) v,x	Spring deformation rate in local x direction $v_x = v_{x,j} - v_{x,i}$
R,x	Spring force in local x direction
(D) R,x D	Damper force in local x direction
(D) R,x S	Total spring-damper force in local x direction $(\mathrm{D})\,\mathrm{R,x\ S} = \mathrm{R,x} + (\mathrm{D})\,\mathrm{R,x\ D}$
...,y	... in local y direction
...,z	... in local z direction

Rotational components	Description
(D) v,xx	Spring rotational deformation rate about local x axis $v_{xx} = v_{xx,j} - v_{xx,i}$
R,xx	Spring moment about local x axis
(D) R,xx D	Damper moment about local x axis
(D) R,xx S	Total spring-damper moment in local x direction $(\mathrm{D})\,\mathrm{R,xx\ S} = \mathrm{R,xx} + (\mathrm{D})\,\mathrm{R,xx\ D}$
...,yy	... about local y axis
...,zz	... about local z axis



6.1.28 Nodal equilibrium

In case of an ideally exact calculation the force equilibrium of $\sum \underline{F}_i = \underline{0}$ and the moment equilibrium of $\sum \underline{\mathrm{M}}_i = \underline{0}$ would hold at every node. In case of numerical solution, especially in case of nonlinear iteration, a residual vector is obtained instead of zero vector: $\sum \underline{\mathrm{F}}_{\mathrm{i}} = \Delta\underline{\mathrm{F}}$, $\sum \underline{\mathrm{M}}_{\mathrm{i}} = \Delta\underline{\mathrm{M}}$.



Using the following result components the absolute and relative measure of the closure error of the vector polygon can be obtained. The relative value is defined as the ratio of the error to the half of the sum of all forces or moments acting on the node. The denominator can be interpreted as the internal force of the node.

Components of the absolute difference from the force equilibrium:

$$\mathrm{DFx} = \left|\sum \mathrm{F_{x,i}}\right|,\ \mathrm{DFy} = \left|\sum \mathrm{F_{y,i}}\right|,\ \mathrm{DFz} = \left|\sum \mathrm{F_{z,i}}\right|$$

Resultant value of the absolute difference from the force equilibrium:

$$\mathrm{DFr} = \left|\sum \underline{\mathrm{F}}_{\mathrm{i}}\right| = \sqrt{\mathrm{DFx}^2 + \mathrm{DFy}^2 + \mathrm{DFz}^2}$$

Components of the relative difference:

$$dFx = \frac{\left|\sum \mathrm{F_{x,i}}\right|}{\frac{1}{2}\sum\left|\mathrm{F_{x,i}}\right|} = \frac{\mathrm{DFx}}{\frac{1}{2}\sum\left|\mathrm{F_{x,i}}\right|},\ dFy = \frac{\left|\sum \mathrm{F_{y,i}}\right|}{\frac{1}{2}\sum\left|\mathrm{F_{y,i}}\right|} = \frac{\mathrm{DFy}}{\frac{1}{2}\sum\left|\mathrm{F_{y,i}}\right|},\ dFz = \frac{\left|\sum \mathrm{F_{z,i}}\right|}{\frac{1}{2}\sum\left|\mathrm{F_{z,i}}\right|} = \frac{\mathrm{DFz}}{\frac{1}{2}\sum\left|\mathrm{F_{z,i}}\right|}$$

Resultant value of the relative difference:

$$dFr = \sqrt{\mathrm{dFx}^2 + \mathrm{dFy}^2 + \mathrm{dFz}^2}$$

Components of the absolute difference from the moment equilibrium:

$$\mathrm{DMx} = \left|\sum \mathrm{M_{x,i}}\right|,\ \mathrm{DMy} = \left|\sum \mathrm{M_{y,i}}\right|,\ \mathrm{DMz} = \left|\sum \mathrm{M_{z,i}}\right|$$

Resultant value of the absolute difference from the moment equilibrium:

$$\mathrm{DMr} = \left|\sum \underline{\mathrm{M}}_{\mathrm{i}}\right| = \sqrt{\mathrm{DMx}^2 + \mathrm{DMy}^2 + \mathrm{DMz}^2}$$

Components of the relative difference:

$$dMx = \frac{\left|\sum \mathrm{M}_{\mathrm{x,i}}\right|}{\frac{1}{2}\sum\left|\mathrm{M}_{\mathrm{x,i}}\right|} = \frac{\mathrm{DMx}}{\frac{1}{2}\sum\left|\mathrm{M}_{\mathrm{x,i}}\right|},\quad dMy = \frac{\left|\sum \mathrm{M}_{\mathrm{y,i}}\right|}{\frac{1}{2}\sum\left|\mathrm{M}_{\mathrm{y,i}}\right|} = \frac{\mathrm{DMy}}{\frac{1}{2}\sum\left|\mathrm{M}_{\mathrm{y,i}}\right|},\quad dMz = \frac{\left|\sum \mathrm{M}_{\mathrm{z,i}}\right|}{\frac{1}{2}\sum\left|\mathrm{M}_{\mathrm{z,i}}\right|} = \frac{\mathrm{DMz}}{\frac{1}{2}\sum\left|\mathrm{M}_{\mathrm{z,i}}\right|}$$

Resultant value of the relative difference:

$$dMr = \sqrt{\mathrm{DMx}^2 + \mathrm{DMy}^2 + \mathrm{DMz}^2}$$

The results are displayed as arrows similarly to the nodal support results.
DFxyz DMxyz, dFxyz, dMxyz result components refer to a special display mode where the individual components are displayed simultaneously as three arrows pointing in the respective direction.

In case of nonlinear analysis it is often happens that despite the converged calculation the force or moment equilibrium is not fulfilled at certain nodes. Usually at nodes to which elements of both low and high stiffness are connected, or at nodes of which individual DOFs have significantly different connecting stiffness. The occurrence of this phenomenon is even more probable in case of geometrically nonlinear analysis. However, the diffeerence from the equilibrium is difficult to detect by comparing the internal forces of the connecting elements if more than two spatially located elements are connected to the node. The result components introduced above display directly the difference from the equilibrium.
The relative components allow to evaluate the error of the equilibrium at different nodes and in different DOFs on a common scale. While the absolute components display the difference from the equilibrium in the units of the applied force and torque making it easier to assess the magnitude of the difference from an engineering viewpoint.
The individual errors of equilibrium can be reduced using the following options:

1. By turning on the force convergence criterion (see **5.1 Static analysis**)
2. By reducing the limit of the force convergence criterion from the default value of 10^{-3} to 10^{-4} to 10^{-5}. Further reduction is not recommended.

By reducing the large connecting stiffnesses. For example, In case of linear analysis the default nodal support having extreme high stiffness of 10^{10} kN/m or 10^{10} kNm/rad works well. But in case of nonlinear analysis it can cause an error in the nodal unequilibrium. Therefore, it should be decreased to as low as 10^{6} kN/m or 10^{5} kN/m. These reduced values may seem too low but in civil engineering practice these supports can still be regarded as infinitely rigid. In general it is necessary to find a compromise value that is low enough for equilibrium to be satisfied with sufficient accuracy but it is still high enough for the support to be regarded as infinitely rigid.

6.1.29 Influence lines

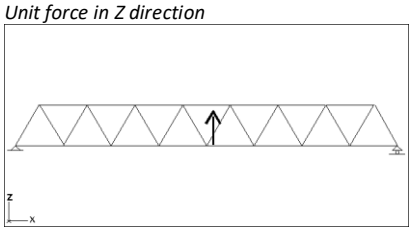
Displays the internal force influence lines corresponding to the unit applied forces P_X, P_Y, P_Z that act in the positive direction of the global coordinate axes. An ordinate of the influence line represents the value of the respective internal force that occurs in the respective cross-section caused by an applied unit force at the position of the ordinate.

Truss

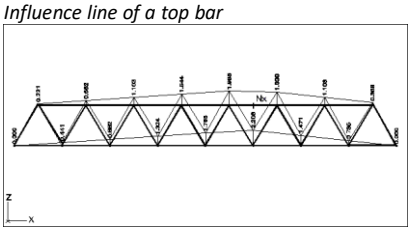
Clicking a truss shows the elements' absolute maximum ordinate value.

Displaying the axial force influence line diagrams of a truss girder:

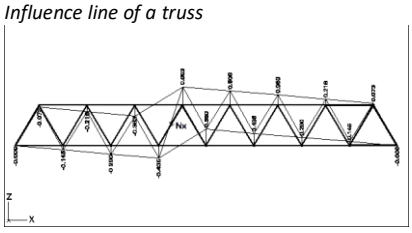
Unit force in Z direction



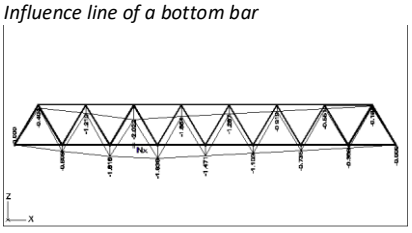
Influence line of a top bar



Influence line of a truss



Influence line of a bottom bar

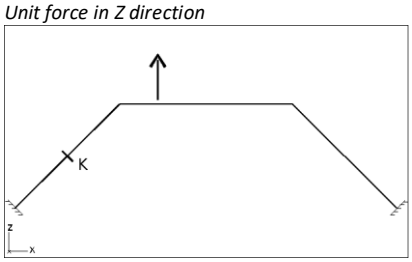


Beam

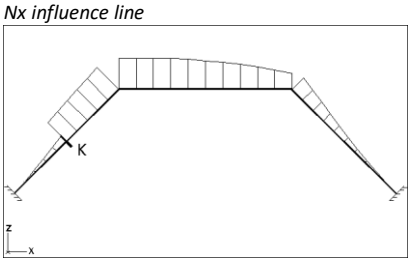
Clicking a beam shows the elements' absolute maximum ordinate value and its location.

Displaying the internal force influence line diagrams of a frame:

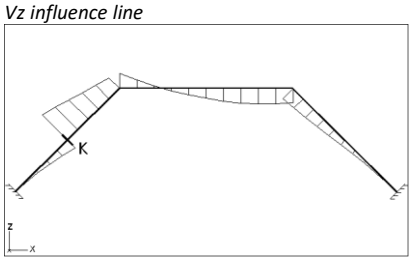
Unit force in Z direction



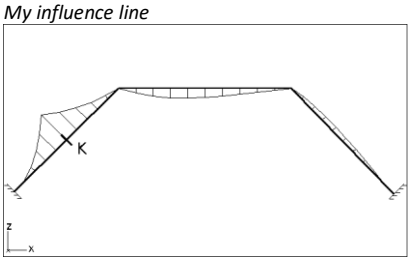
Nx influence line



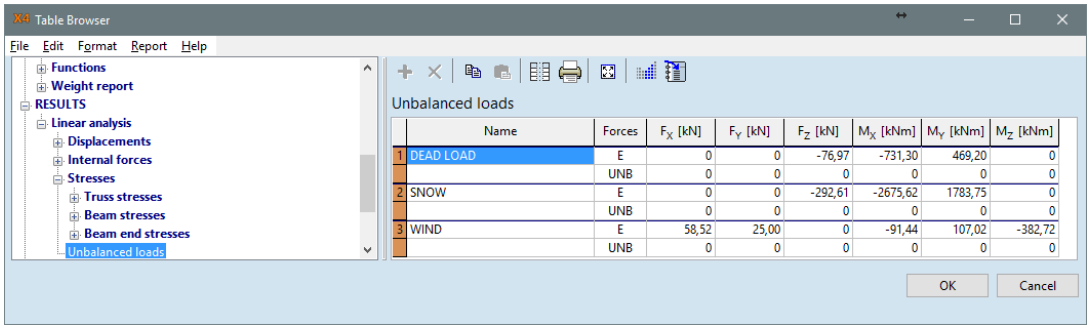
Vz influence line



My influence line



6.1.30 Unbalanced loads



The resultant of all external loads in the global coordinate system *(E)* is calculated for each load case. The unbalanced loads for each load case is also displayed *(UNB)*. The unbalanced loads does not appear on the supports, therefore, if there are non-zero unbalanced load components, it usually means that a part of the external loads are supported by constrained degrees of freedom and not the supports.

☞ **It is recommended to check the unbalanced loads after each analysis run.**

6.1.31 Results of construction stages

Static analyses are performed for the specified constructions stages and the so-called final state. Of course, the results of gradually applied permanent constructional loads are also present in final state, but these results are evluated by linear superposition from previous stages. Not permitted and non constructional loads are analysed only in the final state.

There are the same available result components in construction stage analysis then in the case of a conventional analysis.

Stages and final state are displayed as independent tree nodes in the load case tree. The results of construction stages and final state can be plotted and listed separately. This is also true for the window of result tables (see **6.1.6 Result tables**). In many cases, arrows help the easy switch between construction stages.

The informations related to the investigated stage is displayed in the info windows where such info windows are available.

Invalid results

In the case of any modification (activation of inactivation of elements, etc.) that makes the analysis results invalid, the results of the construction stage and subsequent stages are deleted. In this case, it is possible to run static analysis for *Construction stages not yet analysed* (see **5.1.1 Construction stage analysis – STG module**).

axe and stg files

Typically, the model data are stored in *axs*, while the analysis results are stored in *axe* files. In the case of construction stage analysis, the results of stages are stored in *stg* files and the results of the final state is still stored in *axe* files. The format of *stg* files is the following: *ModelName.stgX* where X represents the index of the construction stage.
It is possible to transfer the *axe* file only together with *axs* files in order to share the results in final state. It has to be noted that some results require stage results. If some of these results were not calculated, but they need to be calculated (e.g. the user creates a new envelope), it is not possible without the results stored in *stg* files.

Design modules

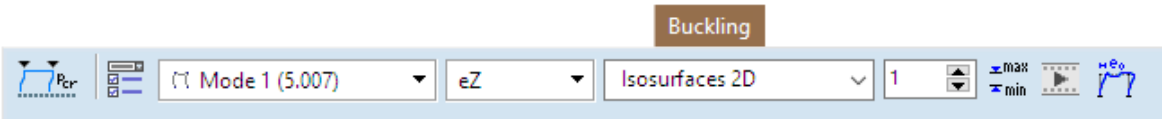
The following design modules in the following circumstances are functional for every construction stage:

RC1 Design of surface reinforcement for RC surfaces, crack width calculation and verification of longitudinal reinforcement in surfaces
RC2 Design and verification of RC columns and beams
RC3 Shear resistance and reinforcement calculation for plates and shells
RC6 Stress-strain analysis of RC surfaces
SD1 Design and verification of steel line elements
SD9 Steel cross-section optimization
TD1 Design and verification of timber line elements
TD9 Timber cross-section optimization
CLT Design of CLT domains

The following design modules in the following circumstances are functional for the last construction stage and the final state only (the structure is the same in both cases)

RC3 Punching analysis
RC4 Footing design
RC5 Design of RC cores and walls
RC6 Stress-strain analysis of RC line elements and virtual beams with actual reinforcement
MD1 Masonry wall design

6.2 Buckling

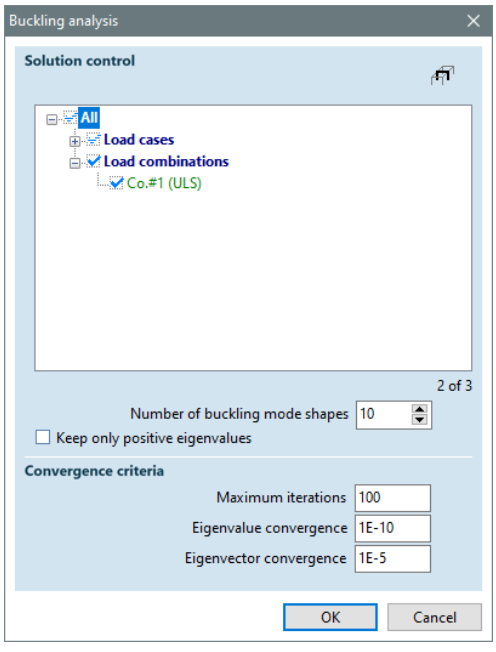


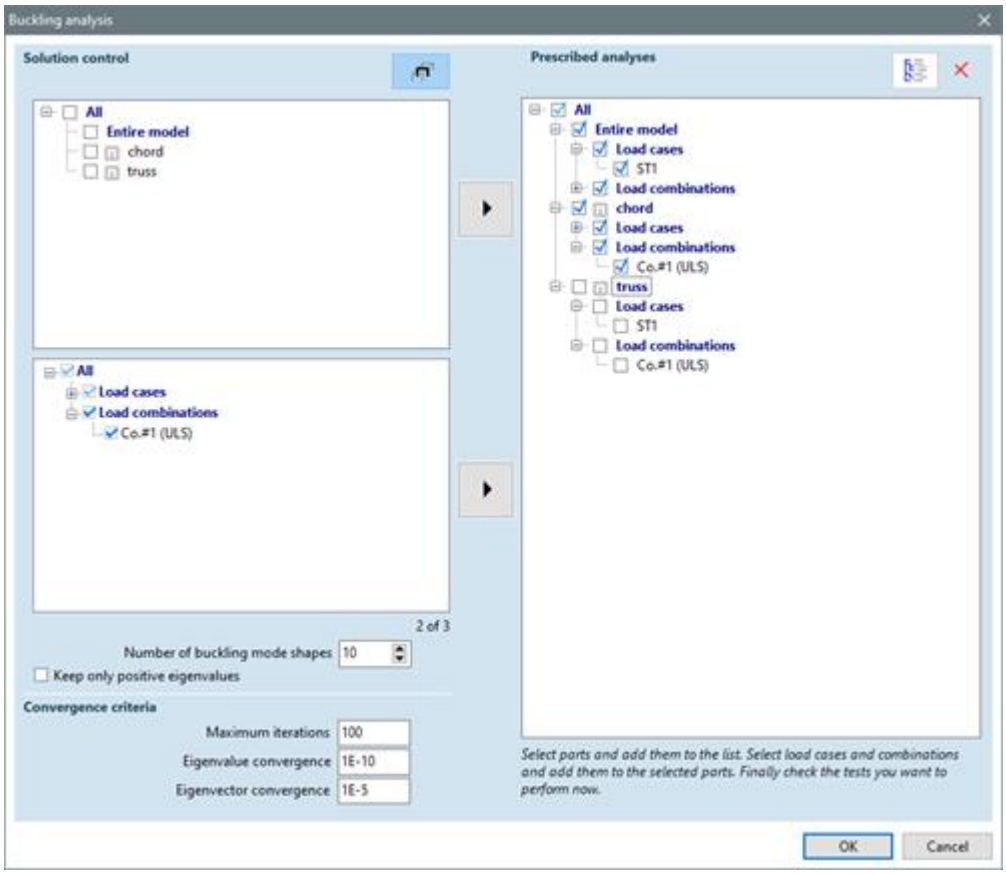
Starting the analysis

To start the buckling analysis, click on the first icon on the toolbar. Select the load cases and load combinations for which you want to run the eigenvalue analysis on the *Solution control* panel. Set the convergence criteria and the number of shapes to be determined.

Buckling analysis of parts

By default, the software takes second-order effects into account on the entire model. Clicking on the button in the top right corner, allows buckling analysis for just a part of the structure (see 2.16.17 Parts). This way the geometric stiffness matrix containing the second-order effects will only include elements within the selected part, second-order effects are ignored on other elements. The rest of the structure will provide elastic support for the selected part with its original stiffness.





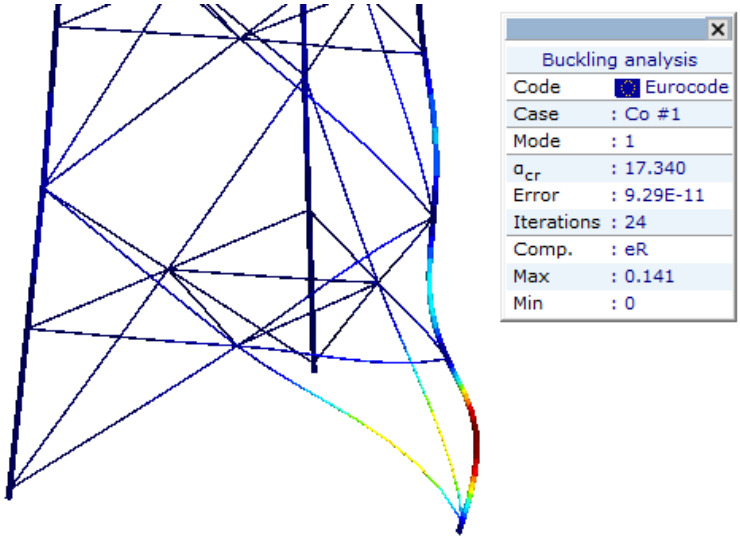
In the *Prescribed analyses* panel, you can set up the buckling analyses you want to run for the whole structure or for the selected parts. Select the parts from the tree on the left and add them to the tree on the right using the upper arrow button. Load cases and load combinations can be assigned to the selected parts on the right with the lower arrow button. After closing the dialog with OK, the program runs the eigenvalue analyses only for cases checked in the tree on the right.
Buckling analyses for structural parts can be effectively used to find the buckling shapes and critical load multipliers for elements in large and complex models.

☞ **The application of buckling analysis for a structural part only needs special attention and the elements included in the part need to be carefully selected.**
The analysis may significantly overestimate the critical load if the second-order effects are significant for supporting elements that are not included in the structural part.

Results of buckling analysis

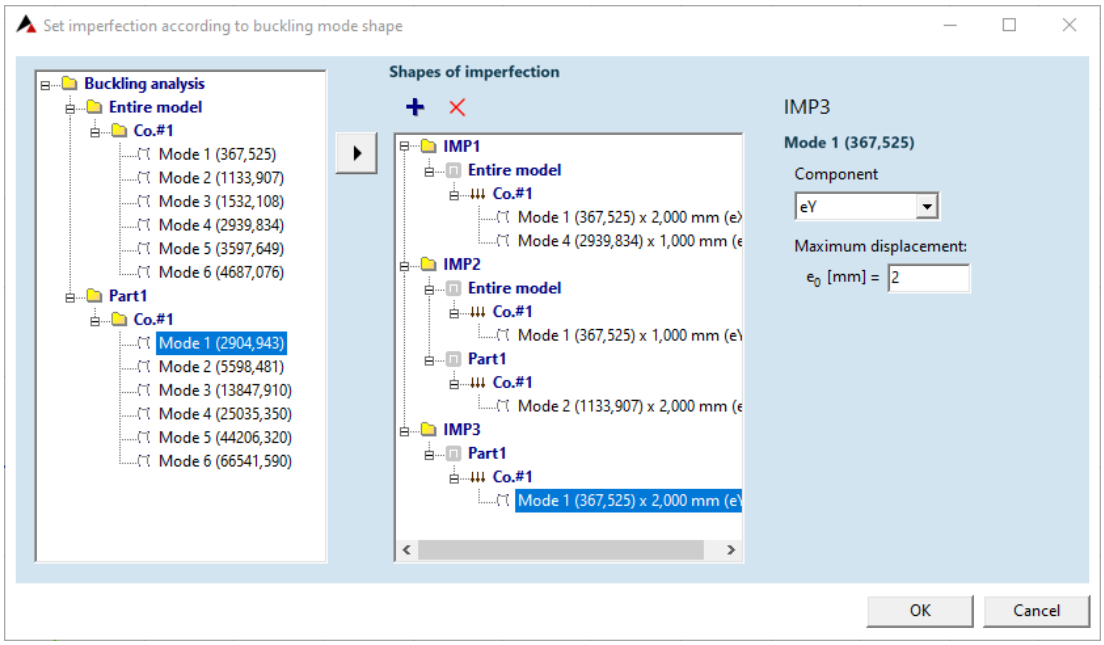
The buckling shape is displayed in the model window, results are available in the *Table Browser*.
The info window displays the following information:

α_{cr}	the critical load multiplier
Error	relative error of the eigenvalue
Iterations	the number of iterations performed until convergence was achieved



6.2.1 Creating imperfect geometry from buckling shapes – IMP module

The module allows imperfections to be taken into account by creating an imperfect geometry by scaling and superimposing buckling shapes. Second-order effects from imperfections can be considered via geometrically nonlinear analysis run with imperfect geometry.



The tree on the left lists the buckling mode shapes obtained from buckling analysis. The tree in the middle displays the shapes of imperfection constructed from the available buckling shapes. The parameters assigned to each buckling mode shape can be set on the right side of the window.

Creates a new imperfect shape

Delete imperfect shapes or constituents. In case of multiple selection, all selected items are deleted. Deleting an imperfect shape, part or load case deletes all associated subfolders and buckling shapes.

Use the *Add* button to add the buckling shapes selected in the left tree to the imperfect shapes selected in the middle. Multiple buckling shapes can be selected. If a buckling load case is selected, all associated shapes are copied. Multiple imperfections or their constituents can be selected in the middle tree. Selected buckling shapes are added to each imperfection affected by the selection. Added buckling shapes are automatically organized into load case folders, and it is not possible to group them or change their order. If a selected buckling shape is already present in the target imperfection, it is not added again.

☞ **Selected buckling shapes can also be added by dragging them over to an imperfect shape. Selected shapes will be added only to the target of the dragging operation.**

Component, Maximum displacement

To create imperfections, a displacement or rotation component or resultant must be selected. Scaling and superposition are performed according to the following formulas: A scaling displacement or rotation must be specified for each buckling shape. Buckling shapes are scaled so that the absolute value of the largest deformation in the direction of the selected component is to be equal to the absolute value of the scaling value. The sign of the scaling value can be used to control whether the direction of the deformations of the shape is the same as or opposite to the direction of the buckling shape.

$$IMP = \sum_{(i)} c_i \cdot \eta_{cr,i}$$

where IMP is the compiled imperfect shape, $\eta_{cr,i}$ is the ith buckling shape, c_i is the factor of the ith buckling shape:

$$c_i = \frac{e_{0,i}}{|\eta_{cr,i}|_{max}}$$

where $e_{0,i}$ is the scaling displacement or rotation, $|\eta_{cr,i}|_{max}$ is the absolute value of the maximum deformation in the direction of the selected component in the ith buckling shape.

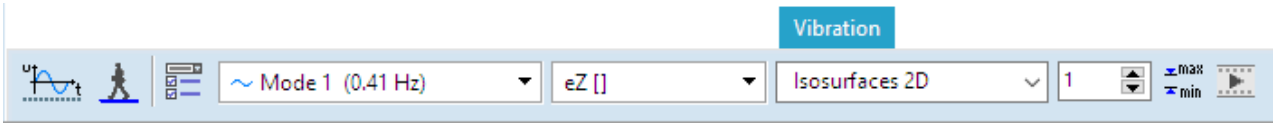
☞ **The entered component and scaling value are applied to all selected shapes.**

The constructed imperfect shapes are appended to the dropdown list of the buckling tab, so they can also be displayed in the main window.
Creating and deleting imperfect shapes automatically creates and deletes the associated load cases, but they can also be edited among the shape? load cases. See **4.10.28 Geometric imperfections based on buckling shapes**

If a buckling shape is deleted due to changes in the model imperfect shapes that use it as a constituent are not deleted only invalidated. After running the buckling analysis again these imperfect shapes are updated automatically from the new shapes.

☞ **The updated imperfect shapes need to be reviewed as the order of buckling modes might have been changed, existing shapes might have disappeared, or new ones might have formed.**

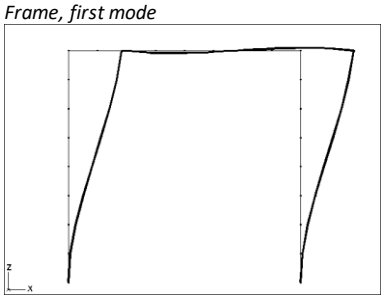
6.3 Vibration



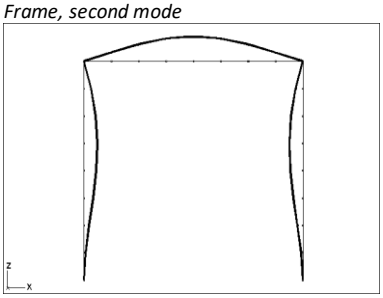
Displays the results of a vibration analysis (mode shapes and frequencies).
You must specify the mode shape number.
The mode shapes are normalized with respect to the mass.

Displaying mode shapes:

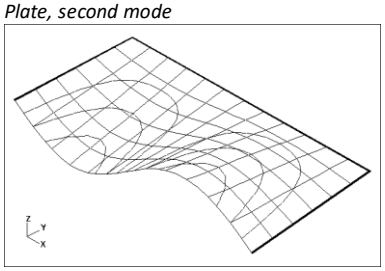
Frame, first mode



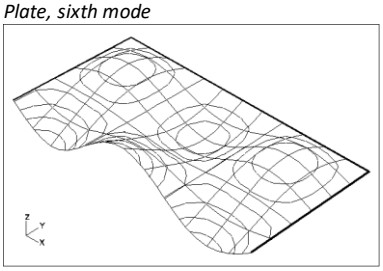
Frame, second mode



Plate, second mode



Plate, sixth mode



In the Info Window the following will appear:

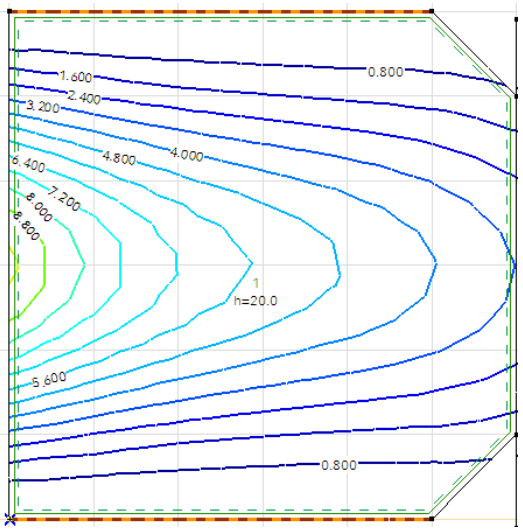
f	the frequency
ω	the circular frequency
T	the period
Ev	the eigenvalue
Error	the relative Error of the eigenvalue
Iteration	the number of iterations performed until convergence was achieved

☞ **AXISVM stores the vibration analysis results corresponding to each case.**

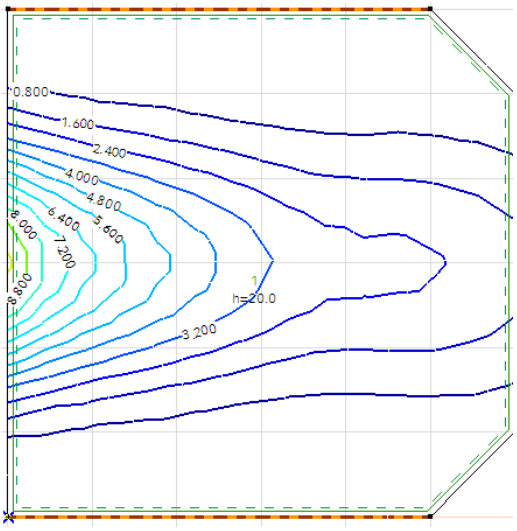
Displaying vibration response factor

Select *Vibration response factor* from the end of the list of vibration modes. Vibration response factors determined with different excitation methods *(R full, R extr., R self)* can be selected from the list of result components.

Full excitation of a slab *(R full)*



Self-excitation of a slab *(R self)*



Values displayed in the info window are:

Design method:	*CCIP-016* or *SCI P354*
ξ:	Damping [%]
N, L:	Number of steps [-] (*CCIP-016*), Length of the walking path [m] (*SCI P354*)
m:	Mass of the walker [kg]
$f_{p,min}$, $f_{p,max}$	Minimum and maximum of the pace frequency on slabs (*CCIP-016*, *SCI P354*), on stairs (*SCI P354*) [Hz]

Result table

See **6.1.6 Result tables**

Activated mass

The program calculates the activated mass (some article use the term 'modal mass') in each X, Y and Z directions, using the mode shapes ordinates and the masses.
For beams:

$$M_{mod} = \mu \int_l \delta^2(x) \cdot dl$$

For surfaces:

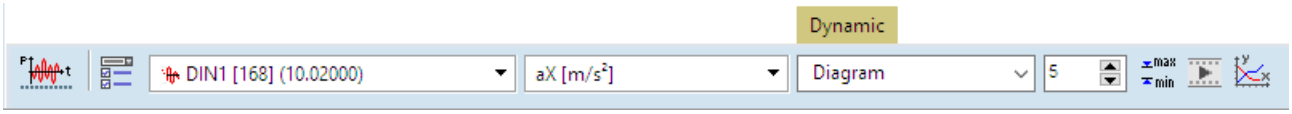
$$M_{mod} = \mu \int_A \delta^2(x,y) \cdot dA$$

where:
μ – the distribution of mass
δ – the normalized mode shape ordinates in the given direction

The calculated 'activated mass' values for each mode shapes and for each direction is displayed in the Frequency Tables/Activated masses table.
The activated masses and frequencies are useful to determine and verify floor designs for vibrations due to walking. In Eurocode there are few guidelines to the above design. Hence the EU sponsored research project worked out the OS-RMS90 guideline to vibration design of floors.
Further info can be found in the following PDF file:

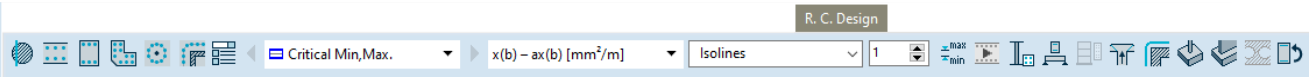
http://www.stahlbau.stb.rwth-aachen.de/projekte/2007/HIVOSS/docs/Guideline_Floors_EN02.pdf

6.4 Dynamic



Displays the results of a dynamic analysis.
Available settings and display modes are the same as for static results. See **6.1 Static.**

6.5 R.C. Design

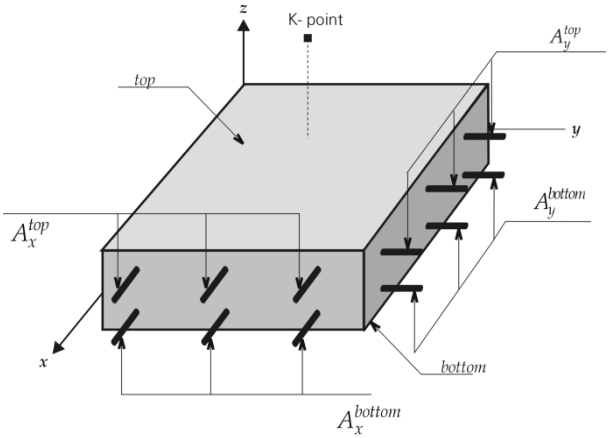


6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module

Design codes

The design codes and national annexes in the software for which this functionality is available are listed in 3.3.7 Design codes.

The calculation of the reinforcement of membrane, plate, and shell elements is based on the 3rd stress condition. Reinforcement directions are the same as the local x and y directions. The nominal moment and corresponding axial strengths are determined based on the restricted direction optimal design.



Result components

mxD, myD, nxD, nyD: design forces
ax(b): calculated reinforcement area at the bottom in x direction
ay(b): calculated reinforcement area at the bottom in y direction
ax(t): calculated reinforcement area at the top in x direction
ay(t): calculated reinforcement area at the top in y direction
sx(b): calculated reinforcement spacing at the bottom in x direction
sy(b): calculated reinforcement spacing at the bottom in y direction
sx(t): calculated reinforcement spacing in at the top x direction
sy(t): calculated reinforcement spacing in at the top y direction
x(b): actual (applied) reinforcement at the bottom in x direction
y(b): actual (applied) reinforcement at the bottom in y direction
x(t): actual (applied) reinforcement at the top in x direction
y(t): actual (applied) reinforcement at the top in y direction
x(b)–ax(b): reinforcement difference at the bottom in x direction
y(b)–ay(b): reinforcement difference at the bottom in y direction
x(t)–ax(t): reinforcement difference at the top in x direction
y(t)–ay(t): reinforcement difference at the top in y direction
vEd: resultant specific shear force
avEd: principle direction of the resultant specific shear force relative to the x axis
vRd,c: shear resistance of concrete cross-section
vRd,max: maximum shear resistance of the cross-section with shear reinforcement
vEd/vRd,max: shear utilization of the concrete cross-section
vSz–vRd,c: difference between the resultant shear force perpendicular to the surface and the shear resistance
wk(b): crack opening in the axis of bottom reinforcement
wk(t): crack opening in the axis of top reinforcement
wk2(b): crack opening at the bottom of the plate
wk2(t): crack opening at the top of the plate
wR(b): crack direction at the bottom of the plate
wR(t): crack direction at the top of the plate

Result display on a solid model

Top and bottom values can be displayed simultaneously in the *Solid model* result display mode.

Reinforcement parameters

In the surface reinforcement design, the following parameters must be assigned to the finite elements:



Materials

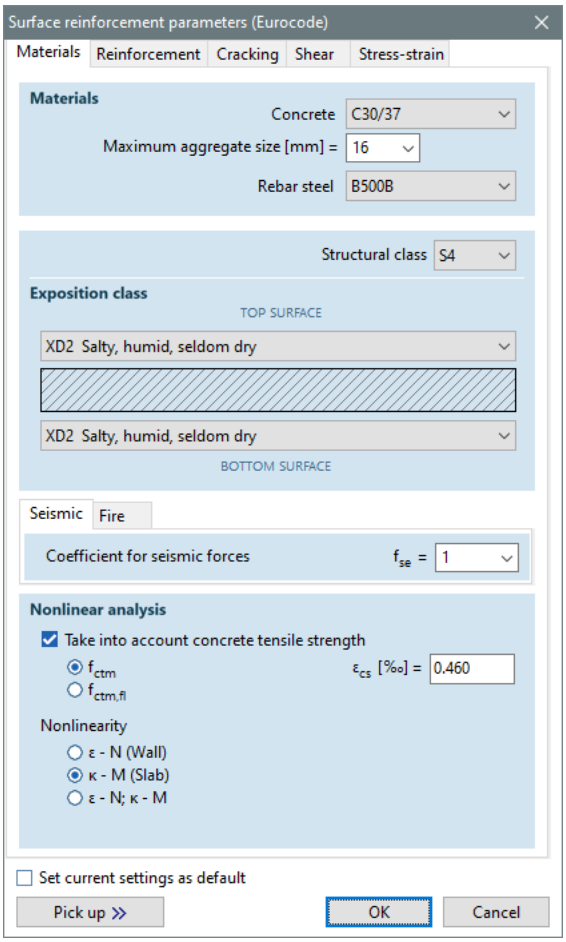
Concrete and rebar materials must be specified. The required minimum of concrete cover is determined from the structural class and the environment at the top and bottom of the surface.

Calculating the minimum rebar spacing all design code takes into account the maximum aggregate size. The Swiss code (SIA) also checks this value for the mimum cover.

You can see coefficient of seimic forces at **4.10.25 Seismic loads – SE1 module**.

The details of fire design of reinforced concrete surfaces are summarized in a following section (**6.5.14.2 Fire design of reinforced concrete surfaces – RC8-S module**).

For those exposure classes for which the Eurocode standard does not provide a recommended value for $c_{min,dur}$, the value is taken from national standard recommendations.



Nonlinear analysis

The value of some parameters considered during nonlinear analysis of surface elements (see **6.5.6 Nonlinear analysis of RC surfaces**) can be set here

- taking into account the tensile or flexural tensile strength of concrete;
- final concrete strain due to shrinkage;
- type of dominant nonlinear behaviour (wall, slab or shell)

This feature is available for the following domain types (see **4.9.6 Domain**):

- normal domains,
- composite ribbed domains,
- parametric ribbed domains,
- composite trapezoidal plates (if the *Trapezoidal plate used only as formwork* option is checked).

☞ **In case of walls the flexural moments, in case of slabs the membrane forces are calculated elastically.**

☞ **The purpose of the consideration of nonlinear behavior in nonlinear static analysis is the more accurate deflection calculation for surface concrete elements by verification in serviceability limit states considering nonlinear (material and structural) behavior. The verification of reinforced concrete elements in ultimate limit states using this fiber integration model is not recommended. It is also important to note that the present model cannot be used in pushover analysis in order to take into account nonlinear behaviour of reinforced concrete elements instead of plastic hinges (see 4.10.25 Seismic loads – SE1 module) because the presented material models do not consider cyclic degradation in concrete and steel strength / stiffness, buckling of steel reinforcement bars, etc.**

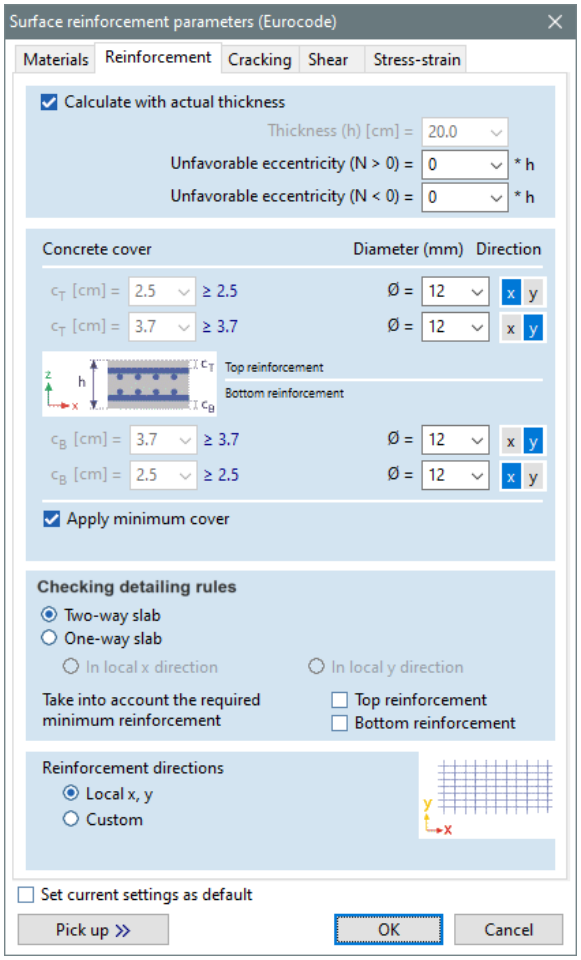
Reinforcement

Calculate with actual thickness:
If this option is activated the program calculates reinforcement using the actual (constant or variable) thickness.
h is the total thickness used in the calculation, which may differ from the actual plate thickness.

In case of Eurocode 2, the *Unfavorable eccentricity* will always be added to the actual value (calculated from normal forces and moments) to increase the absolute value of the eccentricity.

Apply minimum cover can be checked or c_T and c_B top and bottom concrete covers can be entered (i.e., the least distance between the surface of embedded reinforcement and the outer surface of the concrete) for the outer and inner reinforcement layers separately.

The top two rows under *Concrete cover, Diameter* and *Direction* represent the top rebars (the 1st row is the outer one, the 2nd row is the inner one), the bottom two rows represent the bottom rebars (the 3rd row is the inner one, the 4th row is the outer one). The actual scheme in local *x-z* view is displayed accordingly.



Apply minimum cover

The program determines the minimum top and bottom concrete cover from the environment class according to the current design code.

Take into account the required minimum reinforcement

The program determines the required minimum top and bottom reinforcement according to the current design code. If the calculated amount of reinforcement is smaller than these values the required minimum is used.

Detailing rules

In this panel you can take into account the load transfer mode when calculating minimum and maximum rebar spacing and minimum steel percentage according to the design code and the respective national annex. Rebar spacing is adjusted according to design rules only if *Follow rebar layout rules for spacing* is checked in *Result display parameters / Miscellaneous settings* (**6.1 Static**).

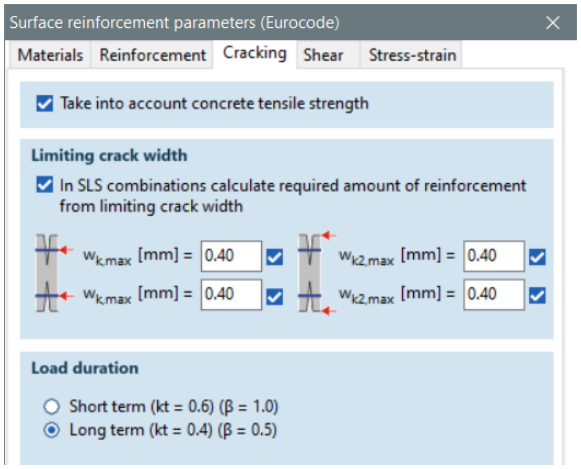
If the calculated amount of reinforcement would result in rebar spacings above the maximum the maximum is used. If it would be under the minimum spacing the slab cannot be reinforced.

Cracking

On this tab the load duration can be set.
This parameter is used in cracking analysis.

Limiting crack width. If checked, the required reinforcement is calculated based on the specified limiting crack width in SLS combinations. Limiting crack width can be specified at the perimeter of the concrete ($w_{k2,max}$) and at the centre of reinforcement ($w_{k,max}$).

The limitations can be switched on or off individually.



Take concrete tensile strength into account

If this option is activated the software does not calculate crack width where the tensile stress in concrete remains below the mean value of axial tensile strength.

Reinforcement calculation based on limiting cracking width

In SLS load combinations, the required reinforcement is calculated based on user specified limiting cracking width. If the actual reinforcement parameters is adjusted to the reinforcement paramters of the domain (identical diameters and concrete cover, as defined in this window), the calculated crack width will not exceed the limiting value.

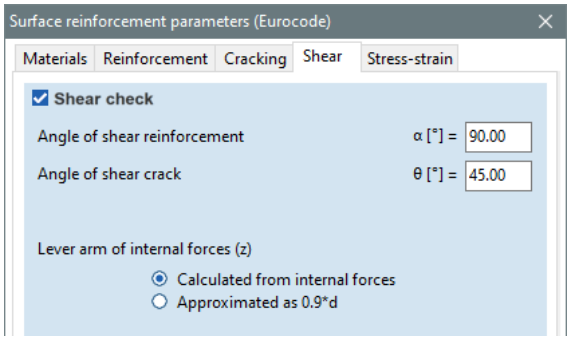
In case of a simple load case or an user defined load combination (not categorized as ULS or SLS), the required reinforcement is calculated considering both ULS and SLS requirements and the greater value is selected. In case of Auto critical combination formula (see **6.1 Static**) required reinforcement is calculated considering both ULS (in ULS combinations only) and SLS (the type of the SLS combination applied depends on the standard) requirements and the greater value is shown.

Stress-strain

Stress-strain analysis parameters are described in **6.5.18.1.1 Stress-strain analysis parameters**.

Shear

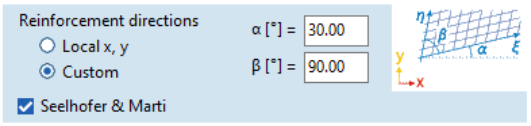
The angle of shear reinforcement and shear crack can be set for the calculation of $V_{Rd,max}$ shear resistance (see **6.5.8 Shear resistance**). The shear check can be disabled by un-checking the checkbox in the top left corner of the panel.



Reinforcement directions

The required reinforcement can be calculated with the consideration of local x/y or custom reinforcement directions. If custom reinforcement directions are selected, ξ/η symbols are used instead of x/y to identify the reinforcement directions. α is the angle between local x axis of the surface and ξ reinforcement direction, while β refers to the angle between the two (ξ and η) reinforcement directions.
Orthogonal reinforcement mesh with custom α angle (-360°<α<360°) and skew reinforcement (β≠90°) can be also defined. Lower and upper bounds for angle β are 45° and 135°.

If Seelhofer & Marti checkbox is checked, the software takes into account an additional condition from the technical article of Hans Seelhofer and Peter Marti *(H. Seelhofer & P. Marti: Dimensioning of skewly reinforced concrete plates)* by the check of compressed concrete strut (see **6.5.1.3 Calculation of skew reinforcement according to Eurocode 2 and SIA 262**). The software always considers the guidelines of Annex F of EN 1992-1-1 by the check of concrete strut in compression.



☞ **If at least one skewly reinforced surface exists in the model, the labels are replaced on many drawings and tables. In these cases *ξ/η* symbols are used instead of *x/y*. For example, *ax(b)* is replaced with *aξ(b)* (if every surface is skewly reinforced) or with *a(x/ξ)(b)* (if there are skewly and orthogonally reinforced surfaces).**

☞ **In case of Airdeck and hollow core surface elements, required skew reinforcement cannot be calculated. If Airdeck or hollow core surface element is included in the selection, Reinforcement directions panel is not visible.**

Required amount of reinforcement in domains

The software creates a summary of the required amounts of reinforcement as *Required amount of reinforcement in domains* (see in the Table Browser) for load cases, load combinations, envelopes and critical combinations. The content of the table can be sorted according to the types of structural elements (slab, wall or other structure), concrete, rebar steel grades, parameters of the domains and the calculated specific and total amount of reinforcement. The *Comment* column displays a warning if the calculation of required reinforcement failed (e.g., the reinforcement cannot be calculated) or the thickness of the domain is different from the thickness found in the reinforcement parameters.
The summary gives information about the minimum and maximum required amount of reinforcement.

The table lists only the results of the domains that are in the active parts or the selected ones (if some domains are selected).

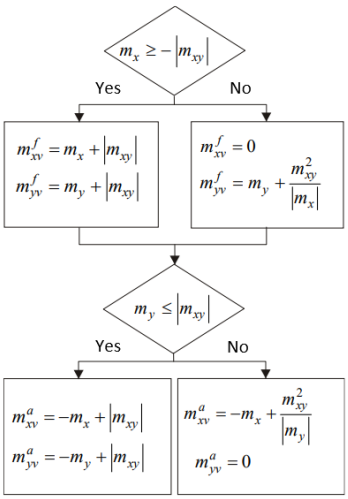
☞ **The calculated amount of reinforcement does not contain the additional reinforcement that comes from the overlapping of rebars, needed anchor length of rebars, erecting rebars and space bars.**

☞ **In the interpretation of the total and specific amount of reinforcement, the following shall be taken into account:**
In the case of Ribbed or Composite ribbed slabs (with concrete ribs), the calculated amount of reinforcement includes only the reinforcement of the slabs, not that of the ribs.
The concrete volume of domains includes the volume of the concrete ribs if they are logical and not actual ribs. In case of actual ribs, only the volume of the slab is taken into account.
In case of a trapezoidal steel deck, using actual domain thickness, the presented concrete volume includes the volume of the concrete ribs, even if the trapezoidal plate is used only as a formwork.
Setting individual thickness for the domain when defining reinforcement parameters overrides the actual thickness. The concrete volume and the amount of reinforcement is calculated with that value as a normal, solid concrete slab. The trapezoidal slab is an exception: the individual thickness means the total height of the concrete in the trapezoidal slab, the concrete volume is calculated according to this.

6.5.1.1 Calculation of orthogonal x/y reinforcement according to Eurocode 2

Plate

If m_x, m_y, m_{xy} are the internal forces at a point, then the nominal moment strengths are as follows:
The moment optimum is: $\begin{matrix}\Delta m_2=0\\ \Delta m_1=\min!\end{matrix}$ $m_x \geq m_y$



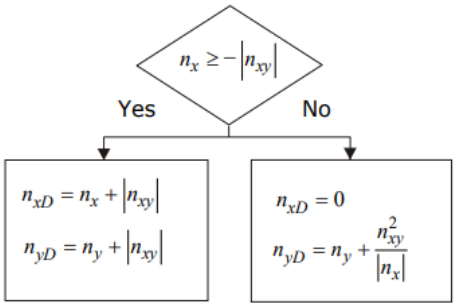
Results

AXISVM calculates the tension and/or compression reinforcements (for doubly reinforced sections).

☞ **The error message 'The section cannot be reinforced' appears if the total cross sectional area of top and bottom reinforcement in x or y reinforcement directions is greater than the limit value for maximum reinforcement area. This limit can be set in the *Design codes* window under the *Reinforced concrete* tab (see 3.3.7 Design codes).**

Membrane

Only plane stress membranes can be reinforced.
If n_x, n_y, n_{xy} are the internal forces at a point, then the nominal axial strengths are as follows:
The axial force optimum is: $\begin{matrix}\Delta n_2=0\\ \Delta n_1=\min!\end{matrix}$ $n_y \geq n_x$



Results

AXISVM calculates the tension or compression reinforcements. Compression reinforcement is calculated only in the points at which the axial compression resistance of the section without reinforcement is lower than the compressive design axial force.

Shell

If $n_x, n_y, n_{xy}, m_x, m_y, m_{xy}$ are the internal forces in a point, the design axial forces and moments are established based on the reserve axial force optimum and reserve moment optimum criterias that were emphasized, at the membrane reinforcement and plate reinforcement description.

The program calculates the necessary tensile and compressive reinforcement.

Results

The following values are provided as results: *ax(b), ax(t), ay(b), ay(t).*
These represent the calculated *t*op and *b*ottom reinforcement in *x* and *y* directions.

☞ **The error message 'The section cannot be reinforced' appears if the total cross sectional area of top and bottom reinforcement in x or y reinforcement directions is greater than the limit value for maximum reinforcement area. This limit can be set in the *Design codes* window under the *Reinforced concrete* tab (see 3.3.7 Design codes).**

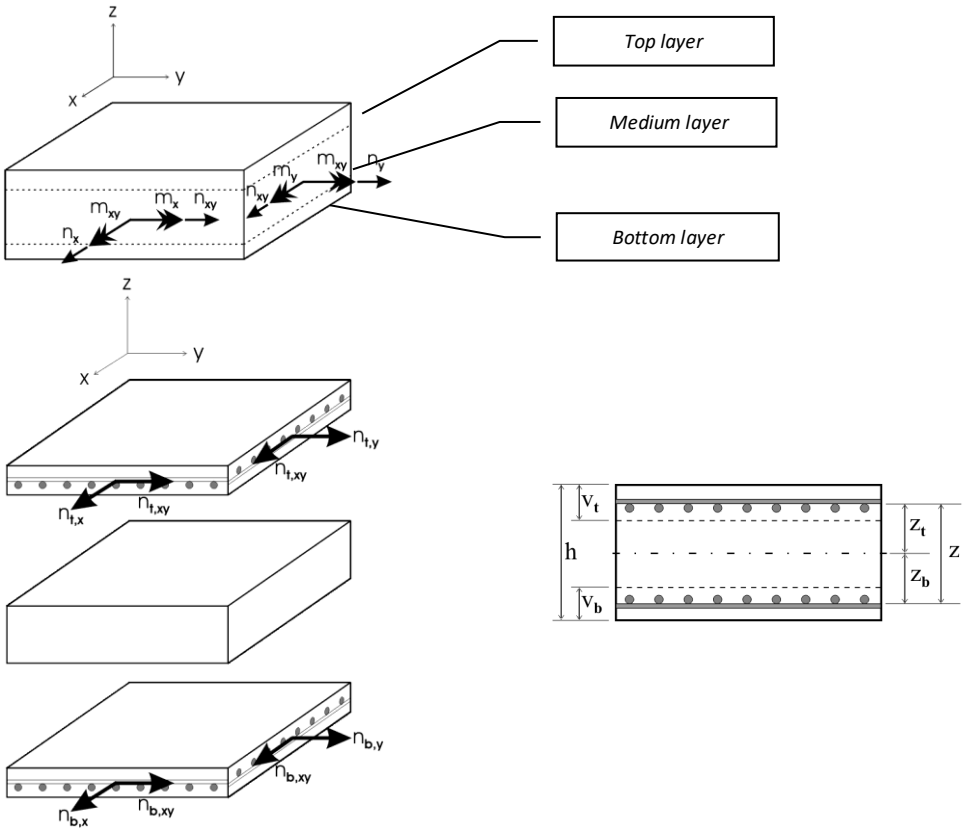
Tables

The following symbols are used in tables:
(-) compression reinforcement bar
??? the section cannot be reinforced in the corresponding direction

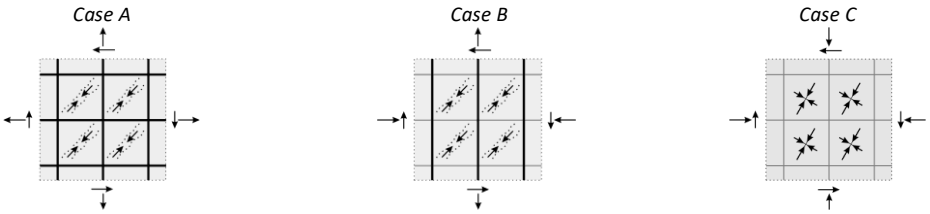
6.5.1.2 Calculation of orthogonal x/y reinforcement according to DIN EN and SIA 262

Plate, Membrane, Shell

Reinforcement of membranes, plates and shells are calculated according to the three-layer method.
The internal forces $(n_x, n_y, n_{xy}, m_x, m_y, m_{xy})$ are calculated in the perpendicular directions of the reinforcement.
The surface is divided into three layers. Membrane forces for the top and bottom layers are calculated then design forces and the required amount of reinforcement is determined.



Besides calculating the required reinforcement zones of concrete are checked for shear and compression according to *A, B* and *C* cases. The program considers reduced compressive strength if tensile stress arises in the concrete and the principal tensile stress exceeds the design tensile strength of concrete *(f_{ctd})*.



Error message

The error message 'The section cannot be reinforced' appears.

If the compressed zone of the concrete fails due shear forces.

If the compression principal stress is higher than f_{cd}.

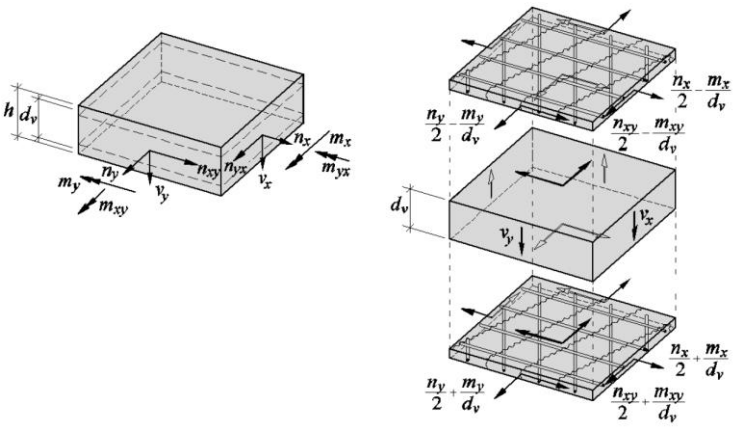
$A_x > 0.04A_c$, or $A_y > 0.04A_c$, where A_c is the concrete cross-section area.

Tables

The following symbols are used in tables:
(-) compression reinforcement bar
??? the section cannot be reinforced in the corresponding direction
No symbol appears when tension reinforcement is required.

6.5.1.3 Calculation of skew reinforcement according to Eurocode 2 and SIA 262

Orthogonal reinforcement meshes with reinforcement directions differing from local x and y axes of the surface and reinforcement meshes with arbitrary reinforcement directions are called skew reinforcement and calculated as described in this section. *ξ/η* symbols are used instead of *x/y* to identify the reinforcement directions. α is the angle between local *x* axis of the surface and *ξ* reinforcement direction, while *β* refers to the angle between the two (*ξ* and *η*) reinforcement directions.
During the calculation of required reinforcements, a sandwich model with three layers is used. Calculation methodology described in the paper of Peter Marti and Hans Seelhofer is invoked (H. Seelhofer & P. Marti: Dimensioning of skewly reinforced concrete plates). The calculation methodology is identical for membranes, plates and shells.



Reference: H. Seelhofer & P. Marti: *Dimensioning of skewly reinforced concrete plates*

Calculation

Both top and bottom layers of the sandwich model are responsible to resist the bending moments and membrane forces. The inner layer resists shear forces.
Membrane forces and bending moments are decomposed into equivalent membrane forces in case of the top and the bottom layer, respectively. Before the decomposition, the internal forces are transferred from *x-y* coordinate system into an orthogonal *x'-y'* coordinate system where *x'* direction is identical with *ξ* reinforcement direction.

Bottom layer:

$$n^b_{x'} = \frac{n_{x'}}{2} - \frac{m_{x'}}{d_v} \qquad n^b_{y'} = \frac{n_{y'}}{2} - \frac{m_{y'}}{d_v} \qquad n^b_{x'y'} = \frac{n_{x'y'}}{2} - \frac{m_{x'y'}}{d_v}$$

Top layer:

$$n^t_{x'} = \frac{n_{x'}}{2} + \frac{m_{x'}}{d_v} \qquad n^t_{y'} = \frac{n_{y'}}{2} + \frac{m_{y'}}{d_v} \qquad n^t_{x'y'} = \frac{n_{x'y'}}{2} + \frac{m_{x'y'}}{d_v}$$

In order to achieve optimal reinforcement, the thickness of the layers is iteratively changed. The lower bound for layer thickness is 5 mm, while the upper limit is set to *H*/2 (in case of membrane-like loading conditions, where *H* is the thickness of the surface) or $\xi_{c0}d$ (in case of plate-like loading conditions, where *d* is the effective depth of the reinforcement mesh). The d_v level arm is calculated on the following way:

$$d_v = min\left(d_b - \frac{t_t}{2}, d_t - \frac{t_b}{2}\right)$$

Where t_b, t_t and d_v are the thickness of bottom layer, the thickness of top layer and the level arm, respectively.

The d_b and d_t are the effective depths related to the bottom and top reinforcement meshes, respectively. The effective depth is calculated based on the average reinforcement positions of ξ and η reinforcements.

The stresses in the top and bottom layers are calculated as follows:

- bottom layer:

$$\sigma_{x'}^b = \frac{n_{x'}^b}{t_b}; \quad \sigma_{y'}^b = \frac{n_{y'}^b}{t_b}; \quad \tau_{x'y'}^b = \frac{n_{x'y'}^b}{t_b}$$

- top layer:

$$\sigma_{x'}^t = \frac{n_{x'}^t}{t_t}; \quad \sigma_{y'}^t = \frac{n_{y'}^t}{t_t}; \quad \tau_{x'y'}^t = \frac{n_{x'y'}^t}{t_t}$$

These stresses are transformed into ξ-η coordinate system (σ_ξ, σ_η, $\tau_{\xi\eta}$) on the following way (*H. Seelhofer & P. Marti: Dimensioning of skewly reinforced concrete plates*):

$$\sigma_\xi = \sigma_{x'} \sin\beta + \sigma_{y'} \cos\beta \cot\beta - 2\tau_{x'y'} \cos\beta$$
$$\sigma_\eta = \frac{\sigma_{y'}}{\sin\beta}$$
$$\tau_{\xi\eta} = \tau_{\eta\xi} = \tau_{x'y'} - \sigma_{y'} \cot\beta$$

Required *ξ/η* reinforcement is calculated as follows (*k*=1):

$$\rho_\xi = \frac{\frac{\sigma_\xi}{\sin\beta} + k\left|\frac{\tau_{\xi\eta}}{\sin\beta}\right|}{f_y} \qquad A_{s,\xi} = \rho_\xi tB$$

$$\rho_\eta = \frac{\frac{\sigma_\eta}{\sin\beta} + \frac{1}{k}\left|\frac{\tau_{\xi\eta}}{\sin\beta}\right|}{f_y} \qquad A_{s,\eta} = \rho_\eta tB$$

The concrete strut in compression is checked according to the guidelines of Annex F of EN 1992-1-1 and according to the instructions in the paper of *Seelhofer and Marti*:

$$(\rho_\eta + \rho_\xi) f_y \le \nu f_{cd} + \sigma_{x'} + \sigma_{y'}$$

The program considers reduced compressive strength only if tensile stress arises in the concrete and the principal tensile stress exceeds the design tensile strength of concrete (f_{ctd}).

If the resistance of the concrete strut is not adequate, the layer thickness is increased. Compression reinforcement is designed in the surface if the upper limit for layer thickness is reached and the utilization ratio of the concrete is still greater than 1.0. If the stresses in the layer and the utilization of concrete strut are low, the layer thickness is decreased.

Error message

The error message 'The section cannot be reinforced' appears in the following cases.

- If the resistance of the concrete strut is not adequate with maximum layer thickness notwithstanding that compression reinforcement is applied.
- The total cross sectional area of top and bottom reinforcement whether in ξ or η reinforcement directions is greater than the limit value for maximum reinforcement area. This limit can be set in the Design codes window under the Reinforced concrete tab (see **3.3.7 Design codes**)

Results

The following values are provided as results: *aξa, aξf, aηa, aηf*

Tables

The following symbols are used in tables:
(-) compression reinforcement bar
??? the section cannot be reinforced in the corresponding direction

6.5.1.4 Calculation of reinforcement of composite slabs with trapezoidal sheeting according to EC and SIA standards

The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

Composite slabs with trapezoidal sheeting can be created as written in the previous sections of the manual (see **4.9.6.5 Trapezoidal steel deck**). The geometry of the composite slab needs to satisfy the detailing rules of the selected design code. The software checks the most important detailing rules (according to section 4.9.2. of Eurocode 4 and section 7.2. of SIA 264) and it warns the user if the geometry has the be modified.

☞ **The presented calculation method can be used only if the longitudinal shear behaviour is ductile.**

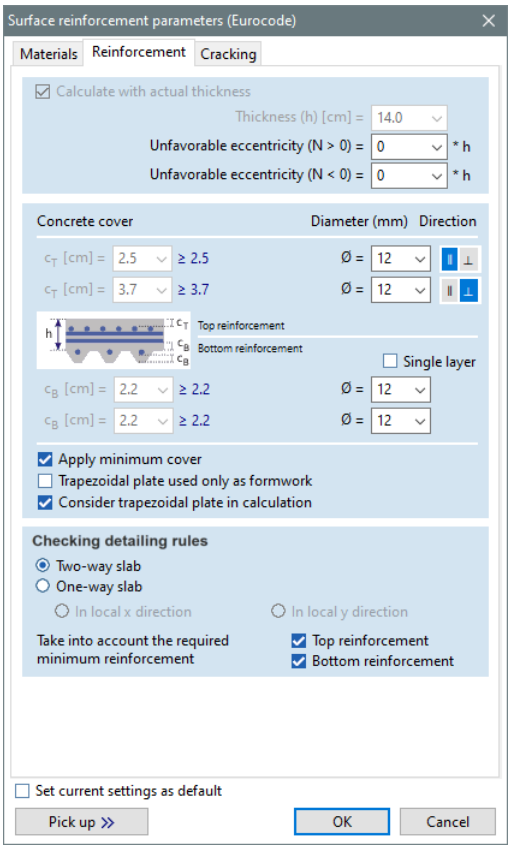
☞ **The presented method is developed for the calculation of reinforcement in composite slabs with trapezoidal sheeting. The verification of trapezoidal sheeting without concrete infill is not part of the module.**

Longitudinal reinforcement can be calculated with the presented method. Shear reinforcement and shear resistance are not calculated. Furthermore, if a significant concentrated load is present, the punching shear resistance of the slab needs to be verified. This module does not cover the punching shear verification. RC3 module can be used for verification against punching failure if *Trapezoidal plate used only as formwork* option is checked.

☞ **This calculation does not cover the verification in construction stage. The weight of wet concrete and temporary supports need to be considered.**

Surface reinforcement parameters

In case of a composite slab with trapezoidal sheeting, the *Surface reinforcement parameters window* becomes slightly different comparing to the presented window in previous sections. The main difference is that parameters need to be defined according to reinforcement directions parallel with the ribs and perpendicular to the ribs of the slab, respectively, not parallel with *x* and *y* axes. The main direction of the reinforcement can be specified only for top reinforcement. The concrete cover of the bottom reinforcement perpendicular to the ribs needs to be specified from the lower edge of concrete plate and not from the lower edge of the rib. The small figure in the window helps the understanding of reinforcement directions and concrete covers.



Due to the fact that the parameters are defined according to reinforcement directions parallel with the ribs and perpendicular to the ribs of the slab, slabs with different rib directions cannot be selected at the same time. Similarly, surface reinforcement parameters cannot be defined for a set of domains with different type simultaneously is composite slabs with trapezoidal sheeting are also selected.

The following features are supported only if the *Trapezoidal plate used only as formwork* option is checked:

- calculation of shear resistance and reinforcement
- calculation of skew reinforcement
- nonlinear analysis with the consideration of cracks and reinforcement

Single layer reinforcement

If *Single layer* checkbox is checked, only one layer reinforcement can be given in the concrete plate and bottom reinforcement perpendicular to the ribs cannot be defined.

Consider trapezoidal plate in calculation

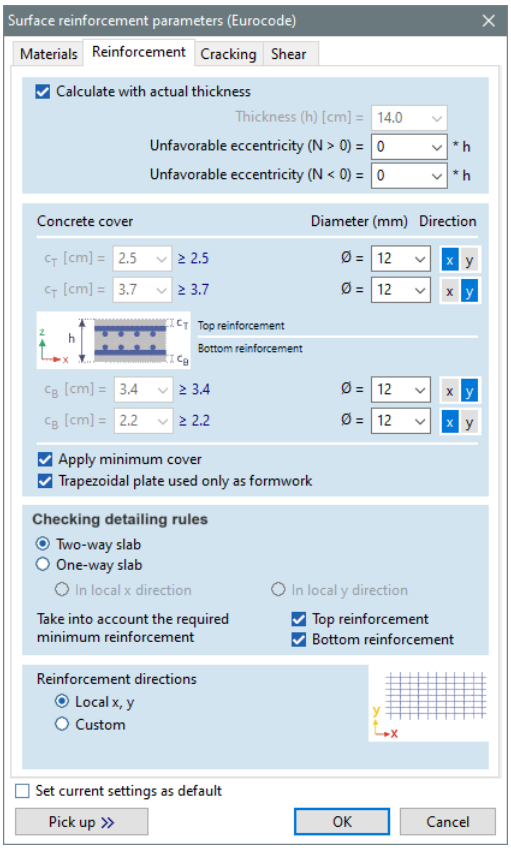
If *Consider trapezoidal plate in calculation* checkbox is not checked, the required longitudinal reinforcement is calculated using the same concrete section but without the consideration of the trapezoidal sheeting in equilibrium equations.

Trapezoidal plate used only as formwork

If *Trapezoidal plate used only as formwork* checkbox is checked, the software takes into account only the upper concrete plate in the equations and it neglects the ribs and the trapezoidal sheet. In this case, reinforcement parameters and concrete covers can be defined regarding x/y reinforcement directions.

The following features are available for a composite slab with trapezoidal sheet only if the *Trapezoidal plate used only as formwork* checkbox is checked (only the upper concrete plate is taken into account):

- punching check (see **6.5.15 Punching analysis – RC3 module**)
- calculation of shear resistance and shear reinforcement (see **6.5.8 Shear resistance and reinforcement calculation for plates and shells**),
- calculation of skew reinforcement (see **6.5.1.3 Calculation of skew reinforcement according to Eurocode 2 and SIA 262**),
- nonlinear analysis with the consideration of cracking and reinforcement (see **6.5.6 Non-linear analysis of RC surfaces).**



Calculation of crack width

In case of composite slabs with trapezoidal sheeting, limiting crack width cannot be given on *Cracking* panel. Independently of the state of *Trapezoidal plate used only as formwork* checkbox, crack width is calculated only for the upper edge of the concrete plate. The details of the calculation of crack width can be found in the following section (see **6.5.7 Cracking**).
Crack width values are calculated parallel with the ribs and perpendicular to the ribs. If *Trapezoidal plate used only as formwork* checkbox is checked, the software calculated the crack width according to principal stress directions as for a conventional slab.

Calculation of longitudinal reinforcement

Composite slabs with trapezoidal sheeting are designed mostly as one-way slabs without the presence of membrane forces. In the course of calculation required longitudinal reinforcement, the software does not consider the membrane forces in the slab.
Membrane forces are typically caused by modelling issues, and special attention needs to be paid for the definition of boundary conditions.

Perpendicular to the ribs

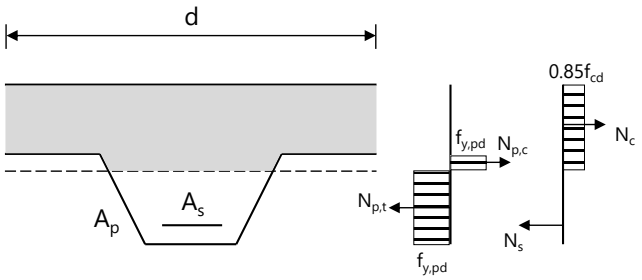
The calculation of required longitudinal reinforcement perpendicular to the ribs is identical with the calculation of reinforcement for regular slabs (see **6.5.1.1 Calculation of orthogonal x/y reinforcement according to Eurocode 2 and 6.5.1.2 Calculation of orthogonal x/y reinforcement according to DIN EN and SIA 262.** The reinforcement design bending moments are calculated on the same way from m_x, m_y and m_{xy} bending moments.
Positive and negative design bending moment (m_{xD}, m_{yD}) may be calculated for the same position due to the presence of considerable torsional moment (m_{xy}). If *Single layer* checkbox is checked, the longitudinal reinforcement is calculated for the single layer from both positive and negative bending moments, respectively.

Parallel with the ribs (lower edge in tension)

The following model and equations are used for the calculation of longitudinal reinforcement parallel with the ribs. The width of the reference cross-section is *d*.

The stresses in the concrete, in the reinforcement and in the trapezoidal sheeting are calculated according to the position of plastic neutral axis. The parts of steel trapezoidal sheeting can be in compression or in tension based on the position of plastic neutral axis. The effect of local buckling of compressed parts of the sheeting is taken into account by using effective widths.

If *Consider trapezoidal plate in calculation* checkbox is not checked, the trapezoidal steel sheeting is neglected in the equations.



The equation of equilibrium and the bending resistance are formulated as follows:

$$0.85 f_{cd} A_c = A_s f_{y,d} + (A_{p,t} - A_{p,c}) f_{yp,d}$$

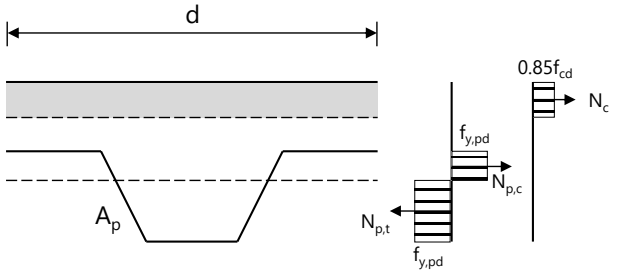
$$M_{pl,Rd} = A_s f_{y,d} \left(z_s - \frac{x_c}{2}\right) + A_{p,t} f_{yp,d} \left(z_{p,t} - \frac{x_c}{2}\right) - A_{p,c} f_{yp,d} \left(z_{p,c} - \frac{x_c}{2}\right)$$

If compression longitudinal reinforcement is required, components representing the forces in the compression reinforcement are added to the equations.

Steps of the calculation of required reinforcement:

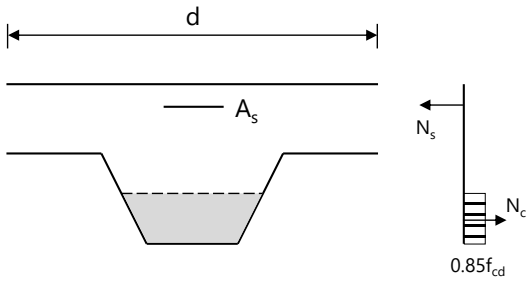
- The plastic bending resistance of the composite slab is calculated without any additional reinforcement. If the resistance of the slab is adequate after comparison with design bending moment, no additional reinforcement is required and the algorithm quits.
- In cases where $M_{pl,Rd} < M_{Ed}$, additional reinforcement is required. First of all, the position of plastic neutral axis is compared with x_{c0}:
    - $x_c < x_{c0}$: Amount of longitudinal reinforcement and plastic bending resistance are calculated using x_{c0} height of compression zone (this is the upper limit of compression zone where the tension reinforcement is still in plastic state).
        - $M_{Ed} < M_{pl,0,Rd}$: No compression reinforcement is required. The exact amount of required tension reinforcement is found with iteration.
        - $M_{Ed} > M_{pl,0,Rd}$: Additional compression reinforcement is required that is calculated from the difference of bending moments $\Delta M = M_{Ed} - M_{pl,0,Rd}$. The tension reinforcement is calculated from equilibrium equations $x_c = x_{c0}$.
    - $x_c > x_{c0}$: Additional compression reinforcement is applied iteratively while one of the following conditions is met $M_{pl,Rd} > M_{Ed}$ or $x_c = x_{c0}$. In the first case, no additional reinforcement is required, only the calculated compression reinforcement. In the latter case, required tension reinforcement is calculated from the difference of bending moments $\Delta M = M_{Ed} - M_{pl,0,Rd}$. The exact value of compression reinforcement is calculated from equilibrium equations $x_c = x_{c0}$

The effectiveness of shear connection needs to be considered by the calculation of the position of the plastic neutral axis if η < 1. In this case, we have two neutral axes, one for the concrete and longitudinal reinforcement and one for the trapezoidal steel sheeting. The equilibrium equation is written twice in order to obtain the position of these axes. The contribution of trapezoidal sheeting is multiplied with η in the first case and the contribution of concrete and reinforcement are multiplied with η in the second case.
After that, the stresses in concrete and in the reinforcement are calculated using the first neutral axis, but the stresses in the trapezoidal sheeting are calculated using the second neutral axis.



Parallel with the ribs (upper edge in tension)

In this case, the contribution of trapezoidal sheeting is neglected.



The equation of equilibrium and the bending resistance are formulated as follows:

$$0.85 f_{cd} A_c = A_s f_{y,d}$$

$$M_{pl,Rd} = A_s f_{y,d} \left(z_s - \frac{x_c}{2} \right)$$

If compression longitudinal reinforcement is required, components representing the forces in the compression reinforcement are added to the equations.

Steps of the calculation of required reinforcement:

- x_{c0} compression zone height and bending resistance $M_{pl,0,Rd}$ are calculated first.
  - $M_{Ed} < M_{pl,0,Rd}$: No compression reinforcement is required. The exact height of compression zone and the amount of required tension reinforcement is found with iteration.
  - $M_{Ed} > M_{pl,0,Rd}$: Additional compression reinforcement is required that is calculated from the difference of bending moments $\Delta M = M_{Ed} - M_{pl,0,Rd}$. The tension reinforcement is calculated from equilibrium equations $x_c = x_{c0}$.

Note

☞ **The software converts the calculated required reinforcement for 1 meter reference-width, thus the unit of presented reinforcement values is mm²/m, etc.**

☞ **The software calculates and shows the required reinforcement in the ribs for the element under the mouse's position if the calculated bottom reinforcement result component, that is parallel to the ribs, is selected.**

6.5.2 Actual reinforcement

Actual Reinforcement



Lets you apply an actual reinforcement to domains or surface elements.
Using the actual reinforcement you can perform a non-linear plate deflection analysis.

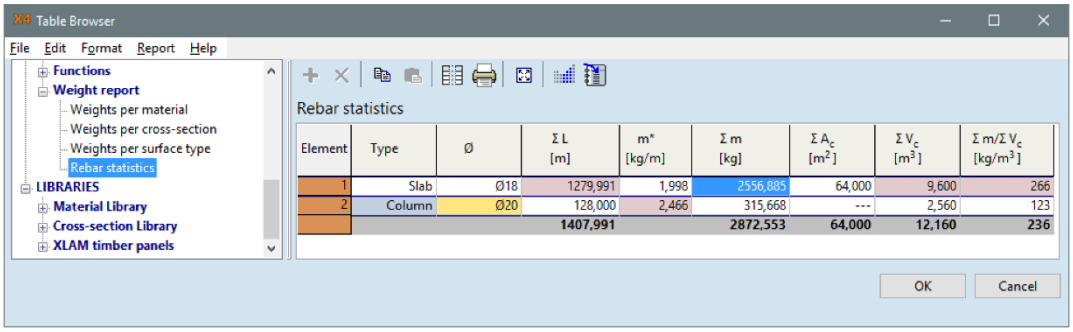
There are two ways to define actual reinforcement:

1.) select surface elements / domains then click the button on the toolbar to specify reinforcement

2.) click the button with no selection, specify reinforcement then draw mesh-independent reinforcement domains or assign the reinforcement to domains.

Rebar statistics

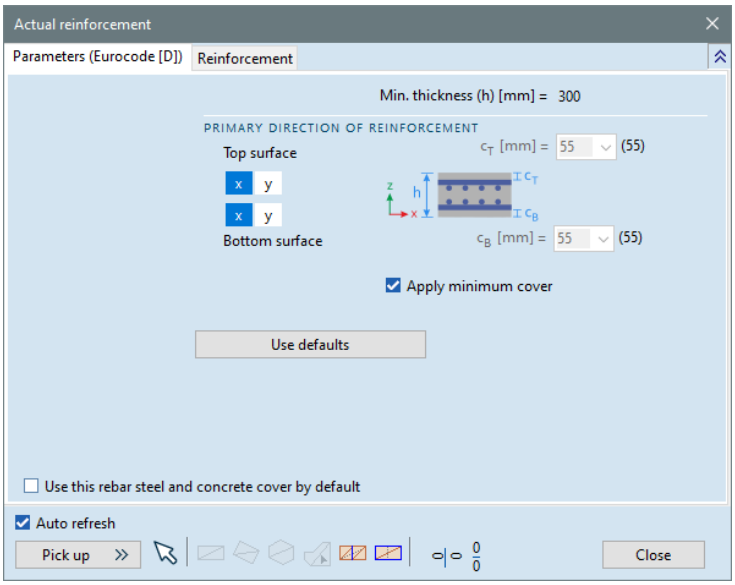
The actual surface, beam and column reinforcements can be checked by displaying *Rebar statistics* in the *Weight report* section of the *Table Browser*. This table lists total length and mass of rebars and the total reinforced concrete surface and volume per rebar diameter.



6.5.2.1 Reinforcement for surface elements and domains

Parameters

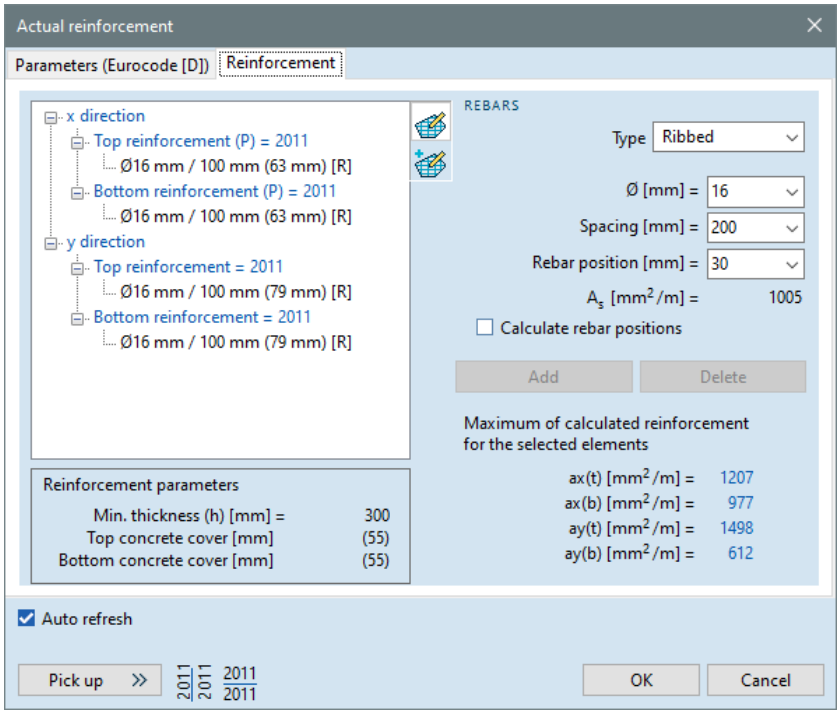
The first tab displays the parameters required by the design code for cracking calculation.
The actual concrete cover and primary directions can be different from those used to determine the required amount of reinforcement (see **6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module**).
Changing these values does not overwrite surface reinforcement parameters.



Min. Thickness

Min. Thickness displays the minimum thickness entered as surface reinforcement parameter for the selected elements, and not the minimum thickness of the elements.

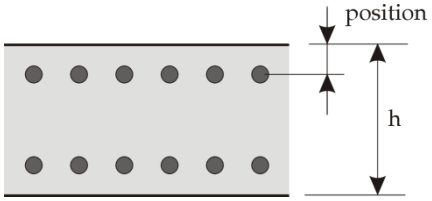
Reinforcement



Add and Delete The applied reinforcement is shown in a tree view on the left. By selecting a reinforcement you can change its parameters in the right side. Changing the values updates the tree. By selecting a location (e.g., x Direction / Top Reinforcement) you can set a new reinforcement on the right side and add it.

Use the *Delete* button (or **[Del]** key) to delete reinforcement or the *Add* button (or **[Ins]** key) to add reinforcement to a group. If you select a node of the tree view the *Delete* button (or **[Del]** key) will delete all the reinforcements under that node. The *Add* button (or **[Ins]** key) will add reinforcement to the corresponding group.
Calculate rebar positions sets the rebar positions according to the actual concrete cover and primary directions.

☞ **Rebar position is defined as the distance between the side of the concrete and the axis of the rebar.**



When modifying an existing reinforcement domain, two methods are available:



Overwrite New reinforcement overwrites the existing one.



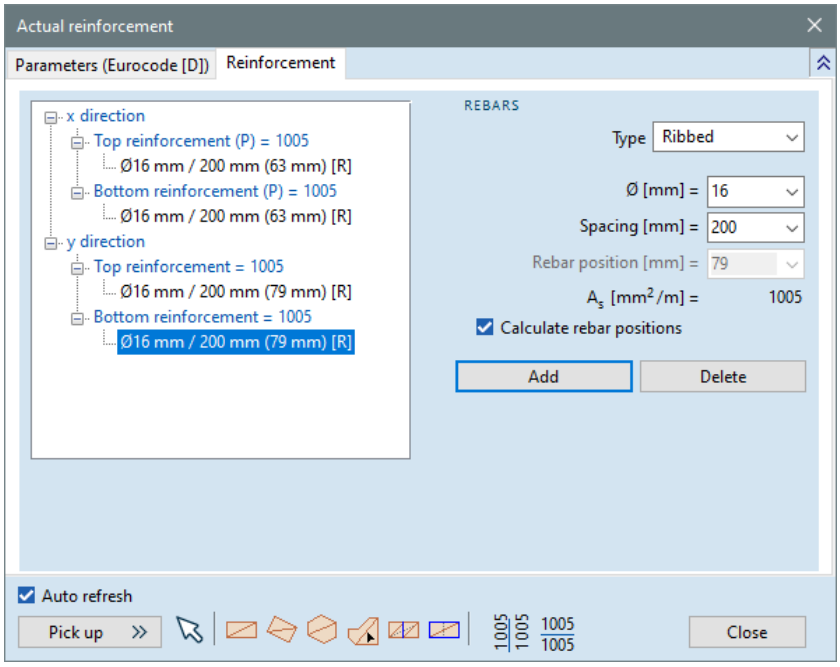
Add New reinforcement is added to the existing one.

Auto refresh If this option is checked changing reinforcement updates the drawing immediately.

☞ **Composite slabs with trapezoidal sheeting cannot be selected together with normal or other types of slabs. If composite slabs with trapezoidal sheeting and other type of slabs are present in the model, the type of domain has to be selected before the window opens. Reference domain needs to be selected if more than one composite slabs with trapezoidal sheeting are selected, but the main geometrical parameters differ. In the case of composite slabs with trapezoidal sheeting, the actual reinforcements need to be defined according to reinforcement directions parallel with the ribs and perpendicular to the ribs of the slab.**

6.5.2.2 Mesh-independent reinforcement

To define mesh-independent reinforcement set the parameters and the reinforcement scheme first then draw rectangular or polygonal reinforcement domains or apply the reinforcement to entire domains.
If no surfaces or domains are selected clicking the button on the toolbar displays this dialog. Reinforcement can be edited the same way as above.



The bottom toolbar can be used to control the way the actual reinforcement is placed on the structure. Mesh-independent reinforcement can also be defined.



Displays the selection toolbar to select existing domains or reinforcement domains. The current reinforcement is applied when the selection is completed.



Option to draw rectangular reinforcement domains.



Option to draw skewed rectangular reinforcement domains.



Option to draw polygonal reinforcement domains.

Reinforcement is applied only where reinforcement domains fall on surface elements or domains.



Option to apply reinforcement to domains just by clicking them.

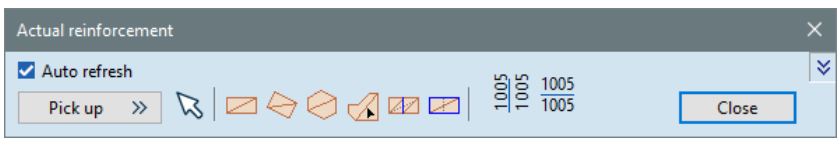


Division of reinforcement domains
Select the reinforcement zones to be divided, then click on two points of the dividing line. The selected reinforcement zones will be replaced by the zones resulting from their division.



Union of reinforcement domains
Select the reinforcement zones to be merged. The shape resulting from the union will be the new reinforcement domain.

The dialog can be reduced to a toolbar. Clicking the icon at the top right corner shrinks or opens up the dialog. The reinforcement amounts specified are displayed as symbols. The amounts of top and bottom *y* reinforcement are written along a vertical line. The amounts of top and bottom *x* reinforcement are written along a horizontal line.

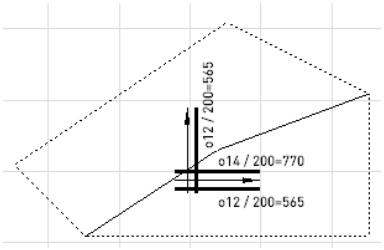


Toolbar icons remain the same.



Contours of reinforcement domains are identified by the cursor. Clicking reinforcement domains allow making changes in the reinforcement. **[Shift]** + clicking selects multiple reinforcement domains. Clicking on one of the selected domains allow making changes in multiple reinforcement domains. This is the same method used for elements or mesh-independent loads.

Mesh-independent reinforcement domains are displayed as contours made of dashed brown lines. A symbol showing top and bottom reinforcement amounts in *x(ξ)* and *y(η)* directions appear at the center. Center-point is connected to two vertices of the domain polygon by continuous brown lines.



Auto refresh Updates the drawing immediately when the reinforcement changes.

6.5.3 Verification of actual reinforcement of RC surfaces

AXISVM can check the utilization of the actual surface reinforcement. While the *Reinforcement difference* result components show only the difference between the calculated and actual reinforcement for each layer, the utilization component is determined by the capacity curve of the RC cross-section. The function is available for the Eurocode (except the German standard) and the NTC standards.
Domains can be membrane, plate or shell.
Linear, non-linear or dynamic results can be taken into account, and the combinations can contain fire load cases.

Limitations

The following types of RC slabs can be verified:

- normal
- hollow core
- ribbed
- composite ribbed
- trapezoidal steel deck slabs
- and slabs with custom stiffness matrix

☞ **Only orthogonal reinforcement can be checked (custom reinforcement directions are not supported).**

Parameters

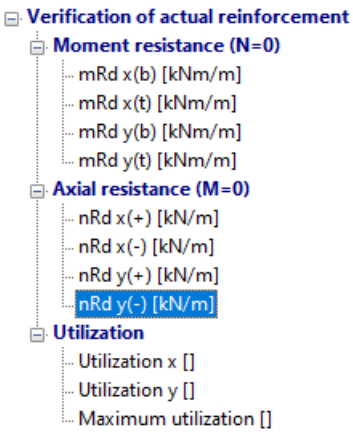
The results are based on the parameters specified for the actual and calculated reinforcement (e. g. concrete cover, rebar diameters, layer directions). Therefore, it only makes sense to compare the results for the actual and calculated reinforcement with identical or nearly identical parameters.

☞ **Reinforcement with various diameters/positions placed in a given layer is considered with an average reinforcement position.**

Result components

The component group *Verification of actual reinforcement* only appears if the model contains at least one domain with actual reinforcement that meets the above criteria.

The displayed result components can be divided into three groups:



Moment resistance (N = 0):
The moment resistance of the cross-section in local *x* and *y* directions with zero axial force considering moments with opposite signs (bending at the top 't' or the bottom 'b' of the cross-section under tension).

Axial resistance (M = 0):
Axial resistance ('+' = tension, '–' = compression) of the cross-section with zero bending in local *x* and *y* directions.

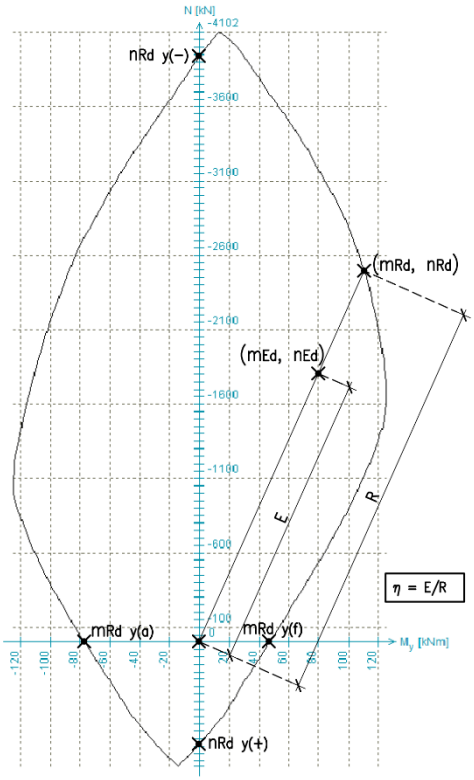
Utilization:
The program calculates the utilization in the local *x* and *y* directions. *Maximum utilization* is the higher value of the two.

Calculation of the result components

Results are based on a linearized capacity curve for each direction (*x* and *y*). (Only specific points of the capacity curve are determined, and a linear interpolation is applied between them.)

In the calculation, an elastic, perfectly plastic stress-strain diagram is assumed for the reinforcement, and a rectangular stress distribution for the concrete. This model is the same as that used in calculating the required reinforcement.

Moment and axial resistance results, interpretation of the utilization components



The pure bending resistance (*mRd (N=0)*) and axial resistance (*nRd (M=0)*) are the limit points of the capacity curve.

The point of maximum capacity *(mRd, nRd)* is obtained by fitting a half-line to the origin and the design stress point *(mEd, nEd)* and finding its intersection with the capacity curve (assuming constant eccentricity).

The utilization *(η = E / R)* is given by the ratio of the length of the sections starting from the origin to the point of design internal force *(mEd, nEd)* and to the point of maximum capacity *(mRd, nRd)*.

☞ **The calculation takes into account the type of combination (SLS, ULS, ...). The safety factors depend on the type of the combination.**
The interaction of the internal forces related to the x and y directions cannot be considered.

Result display and tables

The results can be displayed in any display mode that can be applied to surfaces.
The summary of the results is available in the Table Browser.

6.5.4 Actual reinforcement against biaxial bending (column)

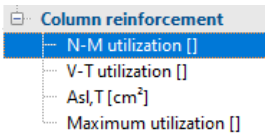
Parameters for reinforcement against biaxial bending can be specified in the dialog below.

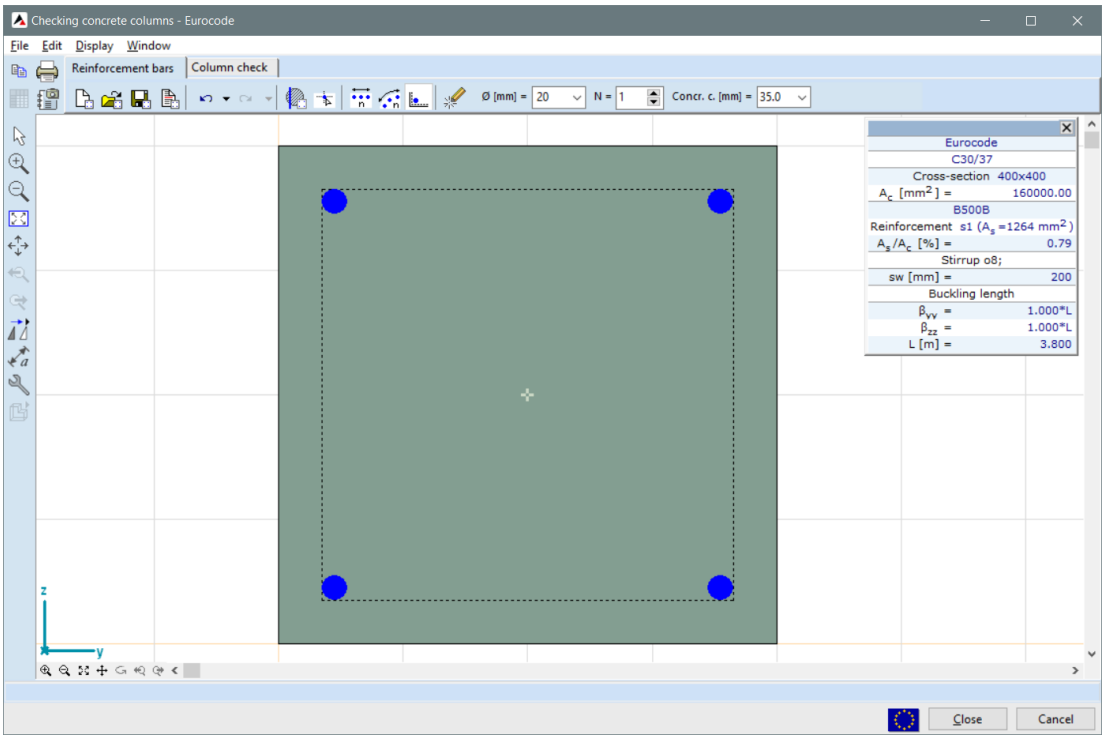
Each finite element can have different reinforcement parameters. (Turn on *Settings / Preferences / Editing / Enable selection of finite elements on lines*). Column check based on actual reinforcement can be performed later (see **6.5.10 Column reinforcement – RC2 module**)

☞ **This type of reinforcement is used for reinforced concrete elements subjected to biaxial bending. It can be assigned to horizontal elements as well, though in these cases the use of beam reinforcement parameters (see 6.5.12 Beam reinforcement parameters (uniaxial bending)) may be more practical.**

Design of multiple columns

Reinforcement parameters can be assigned to multiple columns at the same time. If you select a *Column reinforcement* result component from the drop-down menu, all elements with actual column reinforcement will be checked.





Saves the current drawing to the Drawings Library.

Defines a new reinforcement

Opens an existing cross-section or reinforcement.

☞ **Only thick walled cross-sections with graphics data can be opened.**

Saves the reinforcement under a name for further use.

List of existing column reinforcements. You can sort them and delete the marked rows.

Defining reinforcement

The following icons are available on the Defining reinforcement menu:

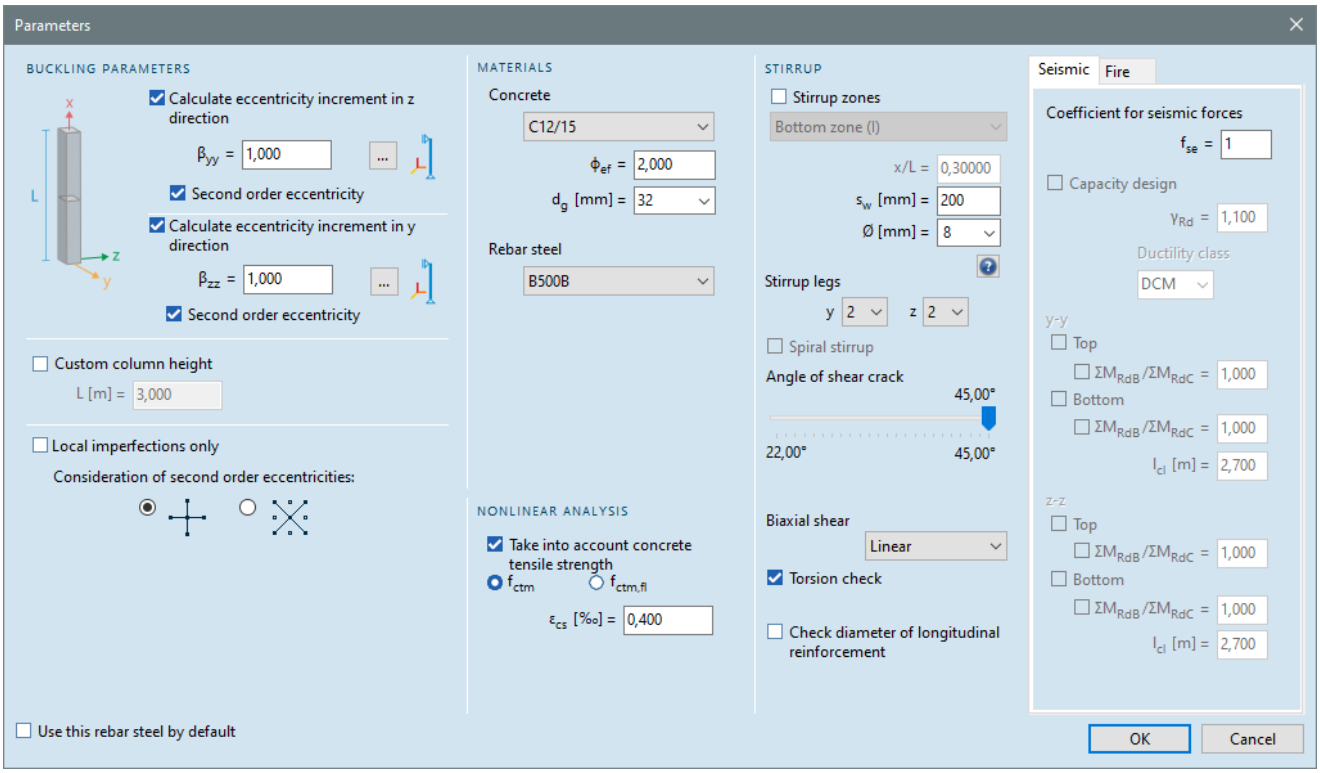
Parameters

Lets you specify the parameters for verification of reinforced concrete columns. The verification covers bending check with or without axial force, the check against failure caused by shear forces and torsional effects.
The unfavorable eccentricity increments determined based on the buckling parameters are displayed in the internal force check table. It can be controlled if eccentricity increments prescribed by the design code are applied in a certain direction or not. Furthermore, the consideration of second order eccentricities can also be controlled in each direction. Second order eccentricities can be considered at the same time or independently in each directions. In checked directions β_{yy} and β_{zz} buckling length factors can be specified. (β_{yy} in *x-z* plane and β_{zz} in *x-y* plane). It is also possible to change the column height used in buckling length calculations and select the buckling shape.
You can see coefficient of seimic forces at **4.10.25 Seismic loads – SE1 module**.
The considered concrete class in the calculation can be altered compared to concrete class assigned to the structural element. It allows the user to evaluate structural safety of the column with different concrete class without new static analysis.
ϕ_{ef} is the so-called effective creep ratio according to Section 5.8.4 of EN 1992-1-1 standard.
Related to non-linear analysis of the column (see **6.5.5 Nonlinear analysis of reinforced concrete beam and column elements**) the consideration of tensile strength and shrinkage strain can be specified. Constant or variable stirrup spacing (three different spacing with different diameters) can be defined to the column. x/L refers to a relative distance from the bottom of the column related to boundary of zones with different stirrup parameters. The number of stirupp legs is uniform along the element, however, it can be different parallel to y and z axes, respectively.

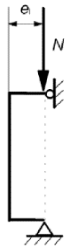
The considered shear crack angle may be also defined by the user. In case of columns with circular cross section, spiral stirrup can be used.
The following options are available for the check of *Biaxial shear:*

- *None* (the check of biaxial shear is disabled),
- *Linear* (linear expression is used by the sum of shear utilizations) and
- *Quadratic* (quadratic expression is used by the sum of shear utilizations).

Check *Torsion check* to consider the torsion moment when designing the column.
In case of dissipative structures with DCM or DCH ductility class (EN 1998-1-1), the design values of shear forces are determined in accordance with the capacity design rule (if seismic load case is included in the selected load combination), on the basis of the equilibrium of the column under end moments $M_{i,d}$ (with i=1,2 denoting the end sections of the column), corresponding to plastic hinge formation for positive and negative directions of seismic loading (see **6.5.10 Column reinforcement – RC2 module**).
The following parameters can be specified by the user: the safety factor, the ductility class, the clear length of the column (clear length: the distance between end sections where plastic hinges can be formed) and the ratio between the sum of moment capacity of beams and columns connected to the same joint, respectively.
Detailed information according to fire design of reinforced concrete columns can be found in **6.5.14.1 Fire design of reinforced concrete columns and beams – RC8-B module**.

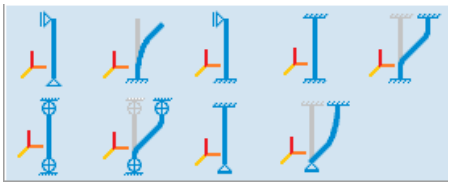


If global imperfections are taken into account by extra loads acting on the structure, they must be ignored in column reinforcement calculation. To achieve this, check *Local imperfections only*. If a sway buckling shape is selected only a constant local geometric imperfection is considered. (see **6.5.10 Column reinforcement – RC2 module**).



Buckling shapes

Second order eccentricities are calculated based on the selected buckling shape (see **6.5.10 Column reinforcement – RC2 module**). Buckling length factors are changed accordingly, but can be overridden



☞ **The given buckling shapes and buckling length factors are associated with the local y and z axes of the section. Accordingly, during the calculation of local geometric imperfections and second order eccentricities flexural inertias related to y and z axes are taken into account (**see 6.5.10 Column reinforcement – RC2 module**).**

Reinforcement bars

To position / by cover



Generates a reinforcement bar with a specified diameter to the location of the cursor.
If the cursor is on a corner or on the contour line the reinforcement will be generated according to the concrete cover.

By spacing



Inserts evenly *N+1* new rebars between two selected points.

Along a circular arc



Inserts evenly *N+1* new rebars between a selected starting point and an end point of a circular arch.
If an entire circle is drawn inserts N rebars evenly around the circle.

Placing rebars at the corners



Places rebars in positive and negative corners (> 5°) if there is no rebar in the corner.

Incorrect rebar spacing

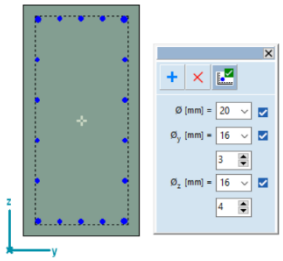
When placing rebars in the cross-section, the program displays the conflicting rebars in red and does not allow them to be placed.
If the gap between the rebars does not meet the standard requirements, the rebars are displayed in orange *during editing*. This latter feature is only a warning and does not prevent the placement of rebars that are too close to each other.

Apply reinforcement scheme

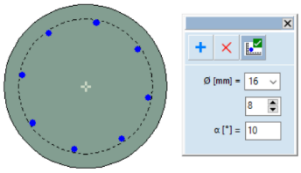


Reinforcement scheme wizard allows the user to create simple rebar arrangement for rectangular and circular cross section.

In the case of rectangular cross section, the diameter of the corner and side rebars can be defined differently. On each side the applied number of rebars are distributed with the same spacing, equally dividing the distance between the corner bars.



In the case of circular cross section the diameter of the rebars, the number of rebars (at least 6 pieces) and the start angle of the distribution can be set.





The *Add reinforcement* tool works as an additive, by keeping the existing reinforcement the user can add new reinforcement scheme based on the set parameters.

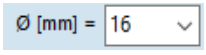


Delete all the reinforcement in the cross-section.



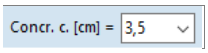
The rebar positions are automatically checked and the program gives a warning message if the positions do not satisfy the rules. The check can be switched off with the „Check rebar position” tool button, but no rebar collisions are allowed.

Diameter

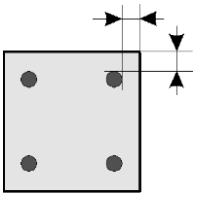


Lets you define or modify the diameter of a rebar.
To modify, select the rebars and than the enter the diameter or select a value from the list.

Cover



Lets you define or modify the concrete cover.



☞ **In this case the concrete cover is the distance from the extreme fiber to the rebar.**

Modifying the geometry of the rebars:

1. Move the cursor over the centroid of the rebar.
2. Use the left button (keep depressed) to move the rebar to its new location, or, enter its new coordinates numerically in the coordinate window.



The division number which defines the number of rebars as *N+1*.



Creates new rebars by copying existing ones by translation.



Creates new rebars by copying existing ones by roatation.



Creates new rebars by mirroring existing ones.

Display of actual reinforcement

After actual reinforcement and its prameters have been assigned to the element and the display of cross-section shapes is turned on *(Display options / Symbols / Graphics symbols / Cross-section shape)*, see **2.16.21 Display options**, actual reinforcement can also be seen in the model view. See also **6.5.12 Beam reinforcement parameters (uniaxial bending)**

Nonlinear analysis

The value of some parameters considered during nonlinear analysis (see **6.5.5 Nonlinear analysis of reinforced concrete beam and column elements**) can be set by the user:

- consideration of the tensile or flexural tensile strength of concrete
- final concrete strain due to shrinkage

ormalised axial force in seismic design situation

The deformation capacity of reinforced concrete elements in seismic design situation depends, among other things, on the sign and magnitude of axial force in the element. For this reason, the axial force in the elements need to be limited. In order to help the design process and the check the axial force in the elements, normalised (normalised to $A_c f_{cd}$) axial force as result component can be calculated and graphically shown for concrete line elements.

$$v_d = \frac{N_{Ed}}{A_c f_{cd}}$$

It is important to note that in this calculation the compression force is positive. In the case of tension force, a negative value is displayed, which is of course only indicative, as the concrete cannot withstand high tensile forces. The software warns the user in the case of the presence of tension force by a warning message.
This result component is calculated only for seismic load cases, seismic load combinations (with or without actual seismic load case in the combination, the type of the combination matters), seismic envelopes and seismic critical combinations.

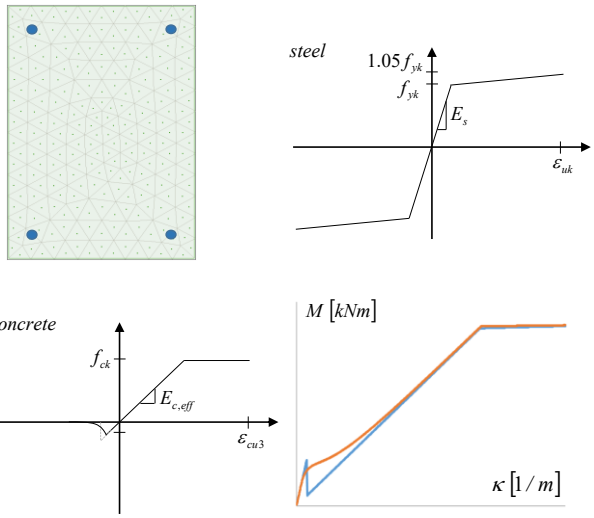
☞ **The normalised axial force is calculated for every concrete line element (including truss, beam and rib elements) considering the specified concrete material and cross-section by the definition of the line element. If actual beam reinforcement is assigned to the element, the concrete material and cross-section of actual reinforcement will be considered instead of original parameters.**

Fire design

Detailed information according to fire design of reinforced concrete columns can be found in a following section (*see* **6.5.14 Fire design of reinforced concrete elements – RC8 module**).

6.5.5 Nonlinear analysis of reinforced concrete beam and column elements

In case of nonlinear analysis actual reinforcements, concrete parameters and steel/concrete nonlinear behaviour can be taken into account (see **5.1 Static analysis**). The design codes and national annexes in the software for which this functionality is available are listed **3.3.7 Design codes**. Internal forces compatible with strains are calculated through numerical integration of fiber stresses at Gauss integration points based on ε normal strains, κ_y and κ_z curvatures. Concrete cross section is divided into a number of triangle fibers; for steel reinforcements independent fibers are assigned with circle shape. Fiber stresses are calculated based on the strain at the center of the fiber and based on concrete/steel nonlinear material model. The difference between the material models built in for EC and SIA standards is only values of strengths, ultimate strains; the shape of the material model is identical.



In order to avoid convergence problems, concrete material model has been changed on tension side. Based on the results of verification tests, the calculated displacements are close to displacements calculated with standardized formulae. The error was lower than 5% in every investigated case.
Tensile stress in the concrete is calculated with the following formula that is based on an approximate method from section 7.4.3 of EN 1992-1-1:

$$\sigma = E\varepsilon(1-\zeta)$$

$$\zeta = \min\left[\max\left[1-\alpha\beta\left(\frac{f_t}{E\varepsilon}\right)^{\gamma};0\right];1\right]$$

α and γ factors are selected (as a function of ratio of concrete's and steel's Young's modulus and as a function of the reinforcement ratio) according to the results of a parametric study in order calculate deformations in good agreement with section 7.4.3. of EN 1992-1-1.

☞ **β is a coefficient taking account of the influence of the duration of the loading. According to EN 1992-1-1, β may be selected 0.5 and 1.0 in case of sustained loads and short-term loads, respectively. If creep is considered in the nonlinear analysis, β is selected 0.5. If creep is not considered in the nonlinear analysis, β=0.5 or β=1.0 can also be used. The value for these cases can be specified in the Design codes window (see 3.3.7 Design codes).**

Creep and shrinkage

The consideration of creep and shrinkage in nonlinear static analysis can be set by the initialization of the analysis (see **5.1 Static analysis**). The creep is taken into account by the modification of the Young's modulus of the concrete. In case of shrinkage, two additional κ_y and κ_z curvatures (calculated from the arrangement and the amount of reinforcement) are added to the calculated strains according to the given shrinkage strain (see **6.5.12 Beam reinforcement parameters (uniaxial bending)** and **6.5.4 Actual reinforcement against biaxial bending (column)**).

☞ **The purpose of consideration of actual reinforcement in nonlinear static analysis is the more accurate deflection calculation for line reinforced concrete elements by the verification in serviceability limit states considering nonlinear material and structural behavior. Using this fiber integration model for modeling material nonlinearity is not recommended by the verification of reinforced concrete elements in ultimate limit states. It is also important to note that the present model cannot be used in pushover analysis in order to take into account nonlinear behaviour of reinforced concrete elements instead of plastic hinges (see 4.10.26 Pushover loads – SE2 module) because the presented material models do not consider cyclic degradation in concrete and steel strength / stiffness, buckling of steel reinforcement bars, etc.**

☞ **In case of composite sections, nonlinear behavior is not considered. Elements with composite sections are treated as linear elements. To consider any creep effect in case of a composite section, concrete material with reduced Young's modulus needs to be assigned.**

☞ **The analysis does not take into account fire effects assigned to the elements. Namely it does not consider the temperature distribution within the cross-sections, the spalling of damaged concrete and the degradation of stiffness and strength of materials.**

After completing nonlinear analysis with actual reinforcement, the following error messages may appear:

1) Increment X, Beam Y: Normal force exceeds the tension/compression resistance
Normal force exceeded the tension/compression design resistance for centrally loaded beams

2) Increment X, Beam Y: Bending moment exceeds the bending resistance
The bending moment at Increment X exceeded the bending resistance in certain cross-sections of the beam

3) Increment X, Beam Y: Longitudinal tension reinforcement is insufficient

Longitudinal reinforcement on the tension side is less than the recommended minimum value according to Eurocode 2, therefore a brittle behaviour is expected due to the sudden and considerable cracking that may lead to convergence problems

4) Increment X, Beam Y: Actual strain exceeds the limiting strain

The strain at the axis of the reinforcing steel or in the extreme compression fiber exceeded the limiting value

These messages appear to help detecting and correcting errors committed when defining the actual reinforcement.

6.5.6 Nonlinear analysis of RC surfaces

In case of the linear static analysis the plate deflection is calculated according to the elastic theory. In fact the behaviour of RC plates is non-linear due to two opposite effects. The actual reinforcement increases the bending strength but cracking decreases it.
The non-linear RC plate deflection analysis follows up these two effects taking into account the actual or calculated reinforcement.

☞ **The below presented method, for consideration of material nonlinearity of RC surfaces in nonlinear analysis, can be used for calculation of deflections by verification in serviceability limit states. Using this fiber integration model for modelling material nonlinearity is not recommended by verification in ultimate limit states. It is also important to note that the present model cannot be used in pushover analysis in order to take into account nonlinear behaviour of reinforced concrete elements instead of plastic hinges (see 4.10.26 Pushover loads – SE2 module) because the presented material models do not consider cyclic degradation in concrete and steel strength / stiffness, buckling of steel reinforcement bars, etc.**

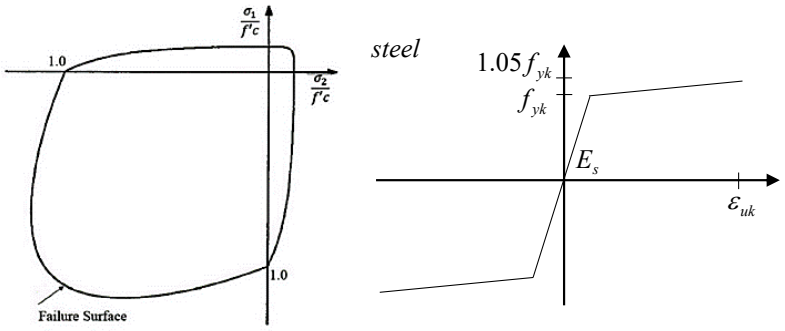
Design codes

The design codes and national annexes in the software for which this functionality is available are listed in 3.3.7 Design codes.

Calculation

Nonlinearity ε-N; κ -M

Internal forces compatible with strains are calculated through numerical integration of fiber stresses at Gauss integration points based on ε_x, ε_y and ε_{xy} normal strains, κ_x, κ_y and κ_{xy} curvatures. The concrete cross section is divided into a number of fibers (namely layers in this case); for steel reinforcements independent fibers are assigned with circle shape. Fiber stresses are calculated based on the strain at the center of the fiber and based on concrete/steel nonlinear material model. In order to avoid convergence problems, concrete material model has been slightly changed on tension side similarly to beams and columns.



Nonlinearity ε-N (Wall)

In this case, nonlinear behavior is considered only in the connection of normal strains and membrane forces. Flexural moments are calculated elastically without consideration of membrane forces.

Nonlinearity κ –M (Slab)

In this case, nonlinear behavior is considered only in the connection of curvatures and flexural moments. Flexural moments are integrated independently in *x* and *y* directions, the effect of membrane forces on the bending behavior is considered. Membrane forces and *Mxy* are calculated elastically.

☞ **In case of slabs with considerable torsion a nonlinear analysis based only on κx-Mx and κy-My bending behaviour may underestimate the deflection, so it is recommended to use the first ε-N; κ-M complex model. That can follow nonlinear behaviour related to the six strain and internal force components.**
In case of κ-M nonlinearity model, in order to calculate more realistic deformations, torsion stiffness in the stiffness matrix is considered on a reduced value if the surface is cracked at a specific point in both x and y main directions. If the height of cracked zone is considerable (>0.5H), torsion stiffness is considered on 15% of the original elastic value. Linearly interpolated value is used between the two states.

☞ **The nonlinear analysis of skewly reinforced concrete surfaces is not allowed. These surfaces are treated as materially linear elements without the consideration of cracking and the effect of reinforcement. The Young's modulus is calculated with the consideration of the defined creep factor.**

☞ **The analysis does not take into account fire effects assigned to the elements. Namely it does not consider the temperature distribution within the cross-sections, the spalling of damaged concrete and the degradation of stiffness and strength of materials.**

Amount of reinforcement

Actual or calculated reinforcement can be taken into account in nonlinear static analysis by choosing Use reinforcement in calculation in the Nonlinear static analysis dialog (see **5.1 Static analysis**). The convergence of such nonlinear analyses is sensitive to the amount of reinforcement. In order to avoid convergence problems, AXISVM applies the minimum reinforcement prescribed by the design code where (top or bottom) actual reinforcement is defined only in one direction. The convergence is usually better in case of actual reinforcement. It is recommended to use the Take into account the minimum required reinforcement option while defining reinforcement parameters to surfaces (see **6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module**)

Elements within the contour of columns

If *Adjust mesh to column heads* option is selected by domain meshing, finite elements within the contour of RC columns can be identified. In order to avoid convergence problems, nonlinear behavior of these elements is neglected in the calculation.

Creep and shrinkage

The consideration of creep and shrinkage in nonlinear static analysis can be set by the initialization of the analysis (see **5.1 Static analysis**). The creep is taken into account by the modification of the concrete's Young's modulus. The shrinkage is considered only in flexural deformations.

Nonlinearity ε-N; κ -M

Two calculated (based on the arrangement and the amount of reinforcement and on the state of RC section) shrinkage induced curvatures (namely κx and κy) are added to the calculated strains according to the given shrinkage strain (see **6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module)**.

Nonlinearity ε-N (Wall)

The reinforcement in walls is typically symmetrical, thus shrinkage curvatures are not taken into account in this case.

Nonlinearity κ –M (Slab)

Two calculated (based on the arrangement and the amount of reinforcement and on the state of RC section) shrinkage induced are added to the calculated strains according to the given shrinkage strain.

Warning messages

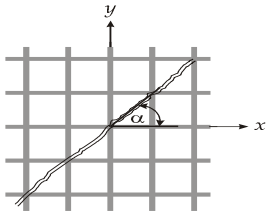
After completing nonlinear analysis with actual reinforcement, the following warning messages may appear:

- "Normal force exceeds the resistance."
- "Actual strain exceeds the limiting strain."
  (Note: In case of the reinforcement or concrete in compression)

6.5.7 Cracking

Design Codes

The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

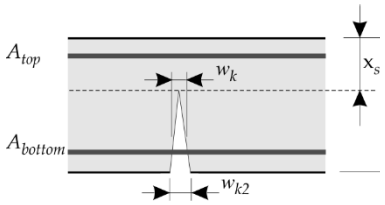


After the assignment of the actual reinforcement the program calculates the crack width and crack directions in the membrane, plate and shell elements. The calculation can be perfromed with reinforcement directions relative to the surface element's local *x* and *y* axes or with arbitrary ξ and η reinforcement directions. The program displays the crack openings in a color coded mode, can draw the crack map and the crack angles.

The set of the parameters can be seen in the previous section.

Results The calculation of crack width can be performed for required reinforcement (marked with *, e.g., *wk** and *wk2**) not only for actual reinforcement. The crack widths calculated for requires reinforcement can be considered only as approximate values because the final actual reinforcement will be different from the required reinforcement. In the table of results the following information can be found:

Aa(x/ξ), Aa(y/η)	actual reinforcement in ***x*** and ***y*** (or in ***ξ*** and ***η*** in case of skew reinforcement) directions
wk	crack width at the axis of the rebar
wk2	crack width at the edge of the slab
xs2	position of the neutral axis relative to the edge on the compressed side
σs2	rebar stress
wR	angle of cracking relative to the local x direction
nx, ny, nxy, mx, my, mxy	surface forces and moments



☞ **A warning message will appear if the calculated rebar stress is higher than the characteristic yield strength.**

☞ **The software does not calculate crack width in fire design situation.**

6.5.7.1 Cracking calculation according to Eurocode 2

$w_k = s_{r,max}(\epsilon_{sm} - \epsilon_{cm})$, where $s_{r,max}$ is the maximum cracking,

ϵ_{sm} is the strain of the rebar, ϵ_{cm} is the strain of the concrete between cracks

$$\epsilon_{sm} - \epsilon_{cm} = \frac{\sigma_{s2} - \frac{k_t f_{ctm}}{\rho_{\rho,eff}}\left(1 + \frac{E_s}{E_{cm}}\rho_{\rho,eff}\right)}{E_s} \geq 0.6 \cdot \frac{\sigma_{s2}}{E_s}$$

$s_{r,max} = \mathrm{k}_3 \cdot c + \mathrm{k}_4 \cdot k_1 k_2 \frac{\overline{\Phi}}{\rho_{\rho,eff}}$ ($s_{r,max} = \frac{\sigma_{s2}\phi}{3.6 f_{ct,eff}}$ in case of DIN EN 1992-1-1)

where

$\overline{\Phi}$	is the average rebar diameter,
c	is the concrete cover,
k_1	is a factor depending on rebar surface (ribbed or plain),
k_2	is a factor depending on the character of the eccentric tension,
k_3	is a factor may be equal to 3.4
k_4	is a factor may be equal to 0.425

k3 and k4 factors can be set in the Design codes window under the Reinforced concrete tab (see 3.3.7 Design codes)

k_t	is a load duration factor,	
	for short-term loads	$k_t = 0.6$
	for long-term (permanent) loads	$k_t = 0.4$

$\rho_{\rho eff} = A_s / A_{c,eff}$ is the effective reinforcement ratio

If plain rebars are used or the spacing of ribbed rebars exceeds $5 \cdot (c + \frac{\overline{\Phi}}{2})$, then $s_{rmax} = 1.3 \cdot (h - x_2)$.

The program takes account of the fact that cracking is not perpendicular to any of the reinforcement directions and calculates its angle relative to the x axis.

6.5.8 Shear resistance and reinforcement calculation for plates and shells – RC3 module

Design Codes

The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

AXISVM calculates the shear resistance of the reinforced plate or shell without shear reinforcement, shear reinforcement (*asw*) if required and the maximum shear resistance of plate and shell structures with shear reinforcement according to the failure of concrete in compression. Furthermore, the resultant design shear force, the principal direction (ϕ), the difference between the design shear force and the shear resistance without shear reinforcement and the ratio of the design shear force and maximum shear resistance with shear reinforcement are also calculated.

This feature is available for the following domain types (see **4.9.6 Domain**):

- normal domains
- composite ribbed domains
- parametric ribbed domains
- domains with custom stiffness matrix
- AIRDECK slabs
- composite slabs with trapezoidal sheeting (if the Trapezoidal plate used only as formwork option is checked)

☞ **Due to the differences in the regulations related to shear and punching, this calculation and a verification based on the presented result components cannot substitute punching verification!**

$v_{Ed} = \sqrt{v_{xz}^2 + v_{yz}^2}$ is the resultant shear force, where v_{xz}, and v_{yz} are the shear force components in planes with normals in the local x and y direction.

$\phi = \arctan \frac{v_{yz}}{v_{xz}}$ is the angle of the normal of the plane, in which resultant shear force of q_{Rz} acts (this angle can be displayed as a result component).

$d = (d_x + d_y)/2$ is the average effective height.

Orthogonal x/y reinforcement:

$\rho_l = \rho_x \cos^2 \phi + \rho_y \sin^2 \phi$ is the reinforcement ratio of the longitudinal reinforcement.

ρ_x and ρ_y are rebar ratios calculated from tension reinforcement in *x* and *y* directions of the reinforcement.

Skew reinforcement:

$\rho_l = \rho_\xi \cos^2(\phi - \alpha) + \rho_\eta \cos^2[\beta - (\phi - \alpha)]$ is the reinforcement ratio of the longitudinal reinforcement.

ρ_ξ and ρ_η are rebar ratios calculated from tension reinforcement in ξ and η directions, respectively.

☞ **The calculation of the shear resistance is based on the actual reinforecement assigned to the surfaces.**

☞ **The calculation of the shear resistance and shear reinforcement does not take into account fire design rules, elevated temperatures in the cross-section, spalling of concrete and degradation in stiffness and strength.**

6.5.8.1 Calculation according to Eurocode 2

Shear resistance is

$$V_{Rd,c} = \left[C_{Rd,c} k (100 \cdot \rho_l f_{ck})^{1/3} + k_1 \sigma_{cp}\right] \cdot d \geq \left(v_{min} + k_1 \sigma_{cp}\right) \cdot d$$

where

$$C_{Rd,c} = \frac{0.18}{\gamma_c}, \qquad k = 1 + \sqrt{200/d} \leq 2.0, \qquad k_1 = 0.15$$

$$\sigma_{cp} = \frac{N_{Ed}}{A_c} \leq 0.2\, f_{cd}, \qquad v_{min} = 0.035 \cdot k^{3/2} \cdot f_{ck}^{1/2}$$

N_{Ed} is the normal force in the shell perpendicular to the plane of q_{Rz}.

N_{Ed} is positive in compression.

The reinforcement ratio is $\rho_l \leq 0.02$

$$V_{Rd,max} = \alpha_{cw} z \nu_1 f_{cd} \frac{\cot\theta + \cot\alpha}{1 + \cot^2\theta}$$

where

$$\nu_1 = \nu = 0.6\left[1 - \frac{f_{ck}}{250}\right]$$

$$\alpha_{cw} = 1.0 \leftarrow \sigma_{cp} = 0$$

$$\alpha_{cw} = \left(1 + \frac{\sigma_{cp}}{f_{cd}}\right) \leftarrow 0 < \sigma_{cp} \leq 0.25 f_{cd}$$

$$\alpha_{cw} = 1.25 \leftarrow 0.25 f_{cd} < \sigma_{cp} \leq 0.5 f_{cd}$$

$$\alpha_{cw} = 2.5\left(1 - \frac{\sigma_{cp}}{f_{cd}}\right) \leftarrow 0.5 f_{cd} < \sigma_{cp} \leq f_{cd}$$

z is the level arm of internal forces

Calculation of level arm of internal forces

The level arm of internal forces is calculated according to one of the two following ways based on the settings specified by the user (See **6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module**):

- calculation from internal forces based on the equilibrium of a three-layered sandwich model;
- approximately: $z = 0.9d$

Calculation of required shear reinforcement

If v_{Ed} is greater than $V_{Rd,c}$, shear reinforcement is required to resist shear forces. The required reinforcement (e.g., [mm²/m²]) is calculated with the help of the following formula. If v_{Ed} is greater than $V_{Rd,max}$, the section cannot be reinforced against shear force. In these cases, the software warns the user with a warning message.

$$a_{sw} = \frac{v_{Ed}}{z f_{ywd} 1m[(\cot\theta + \cot\alpha)\sin\alpha]}$$

$V_{Rd,c}$, $V_{Rd,max}$ shear resistances, furthermore the difference between design shear force and the shear resistance $V_{Ed} - V_{Rd,c}$ and $V_{Ed}/V_{Rd,max}$ can also be displayed with isolines and isosurfaces.

6.5.8.2 Calculation according to SIA 262

Shear resistance is

$$V_{Rdc} = k_d \tau_{cd} d$$

where $\tau_{cd} = 0.3\sqrt{f_{ck}}/\gamma_c$; $k_d = 1/(1 + \varepsilon_v d k_g)$; $k_g = \frac{48}{16 + D_{max} \cdot min\left\{1; \left(\frac{60}{f_{ck}}\right)^2\right\}} \geq 1{,}2$

According to the settings, the ε_v parameter is calculated as follows:

$$\varepsilon_v = 1{,}5\frac{f_{yd}}{E_s} \text{ or } \varepsilon_v = \frac{m_d}{m_{Rd}}\frac{f_{yd}}{E_s}.$$

If direction of V_{Ed} resultant design share force is not parallel to either x or y reinforcement, ε_v is increased by $1/(cos^4\phi + sin^4\phi)$.

$$V_{Rd,max} = z k_c f_{cd} (\cos\alpha + \cot\beta \sin\alpha)\sin\alpha$$

where

$$k_c = \frac{1}{1.2 + 55\varepsilon_1} \leq 0.65 \qquad \varepsilon_1 = \varepsilon_x + (\varepsilon_x + 0.002)\,cot^2\,\alpha$$

z is the level arm of internal forces

Calculation of required shear reinforcement

If v_{Ed} is greater than $V_{Rd,c}$, shear reinforcement is required to resist shear forces. The required reinforcement (e.g., [mm²/m²]) is calculated with the help of the following formula. If v_{Ed} is greater than $V_{Rd,max}$, the section cannot be reinforced against shear force. In these cases, the software warns the user with a warning message.

$$a_{sw} = \frac{v_{Ed}}{z f_{ywd} 1m[(\cot\theta + \cot\alpha)\sin\alpha]}$$

$V_{Rd,c}$, $V_{Rd,max}$ shear resistances, furthermore the difference between design shear force and the shear resistance $V_{Ed} - V_{Rd,c}$ and $V_{Ed}/V_{Rd,max}$ can also be displayed with isolines and isosurfaces.

6.5.9 Punching reinforcement parameters for multiple elements



Punching reinforcement parameters for multiple columns
6.5.15.1.1 Punching analysis of columns according to Eurocode
6.5.15.2.1 Punching analysis of columns according to SIA 262



Punching reinforcement parameters for multiple wall ends and corners
6.5.15.1.2 Punching analysis of wall ends and corners according to Eurocode
6.5.15.2.2 Punching analysis of wall ends and wall corners according to SIA 262

Punching reinforcement parameters may be assigned to the selected columns, node supports (if their stiffness was calculated from column parameters), or to the selected nodes at wall ends and wall corners. If a punching analysis result component is selected from the drop-down menu (See **6.5.15.3 Punching result components**), all elements with parameters assigned will be checked for punching.

6.5.10 Column reinforcement – RC2 module



The reinforced column check can be performed based on the following design codes:

Design Codes

The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

Reinforcement bars

Actual reinforcement can be entered the same way as described in **6.5.4 Actual reinforcement against biaxial bending (column)**

Column Check

The program performs the verification against bending with or without axial force, shear forces and torsional moments of the column based on the cross-section properties and reinforcement parameters and determines the eccentricity increments for the forces in the selected columns (or any $N_x, M_{yb}, M_{zb}, M_{yt}, M_{zt}$ values) based on the given buckling parameters and according to the requirements of the current design code.
Calculates $N_{x,Ed}, M_{y,Ed}, M_{z,Ed}$ design forces using the eccentricity increments and checks if these points are within the interaction diagram.
The display of the diagram can be set in the Display Parameters window.
In case of shear verification, the software determines the shear resistance of the concrete and the stirrups and the upper bound of shear resistance related to the failure of concrete strut in compression.
In case of verification against torsional effects, the software determines the torsion resistance of the concrete and the upper bound of torsion resistance related to the failure of concrete strut in compression. During the calculation of utilization, the combined effect of shear and torsion is also accounted.

Display

Allows setting the display modes for the interaction diagram.

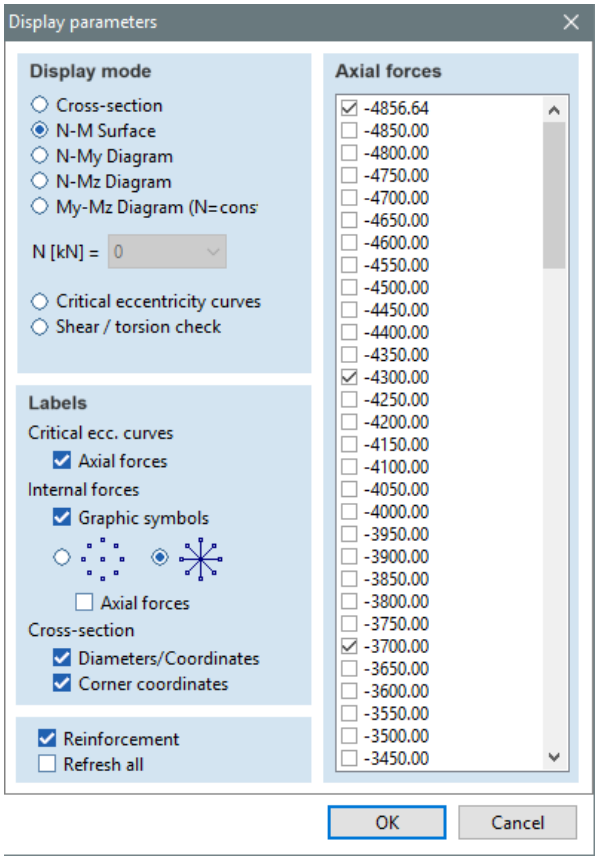
Select display mode by clicking a radio button in the Display Mode group box. It has the same effect as selecting it from the dropdown list.

Select axial force values to use when drawing the 3D interaction diagram (N-M Surface) from the check list.

In the Labels group you can turn on and off axial force labeling, the display of graphic symbols for internal forces of selected columns in the N-My-Mz space and display options for the cross-section display mode.

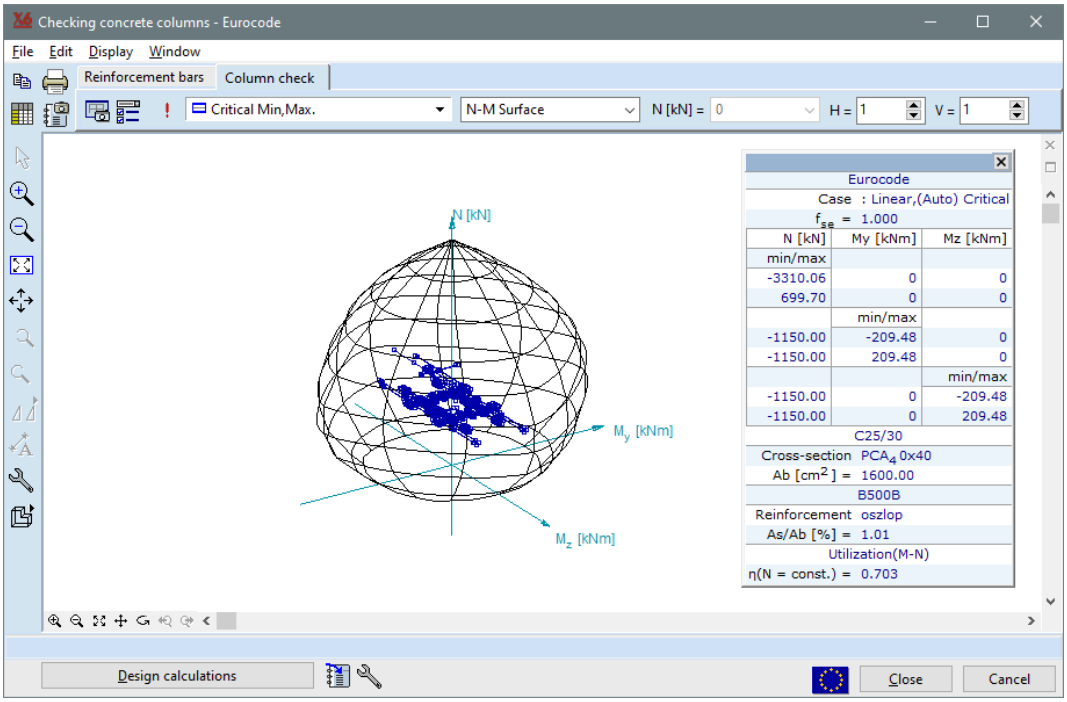
Blue color shows that the $N_{x,Ed}, M_{y,Ed}, M_{z,Ed}$ values are within the interaction diagram.

Red color shows that $N_{x,Ed}, M_{y,Ed}, M_{z,Ed}$ values are ouside the interaction diagram. Normal forces for these points are always displayed.



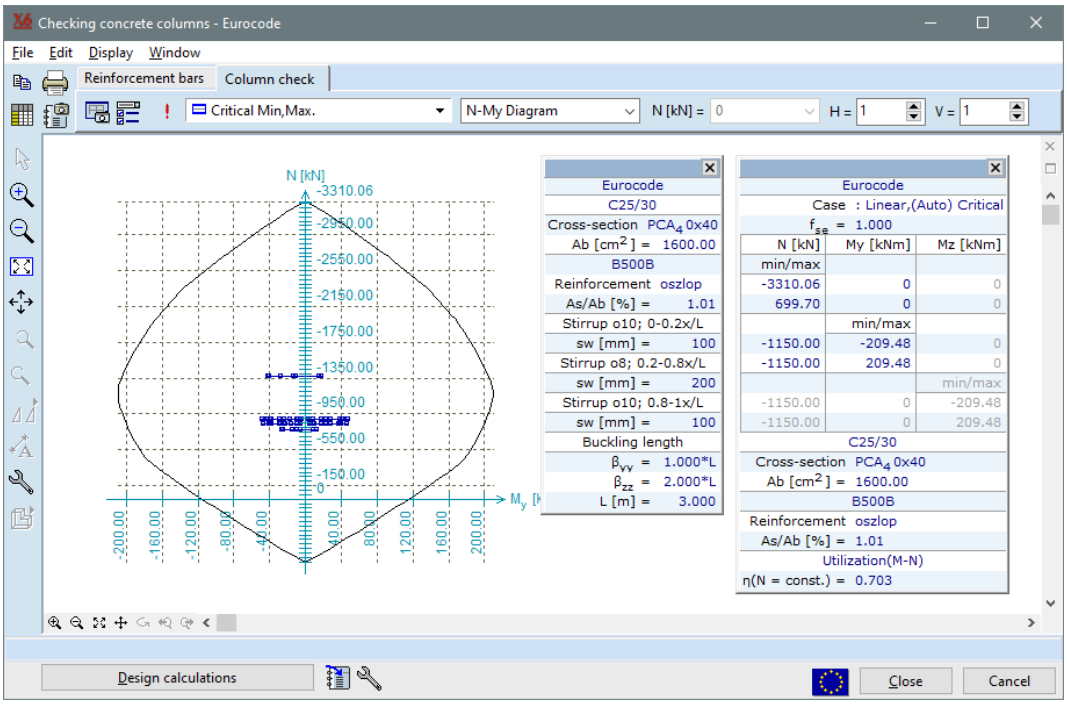
N-M surface

Displays the N_x-M_y-M_z strength interaction 3D diagram.



N-M diagram

Displays the N_x-M_y, or N_x-M_z load-moment strength interaction diagram.

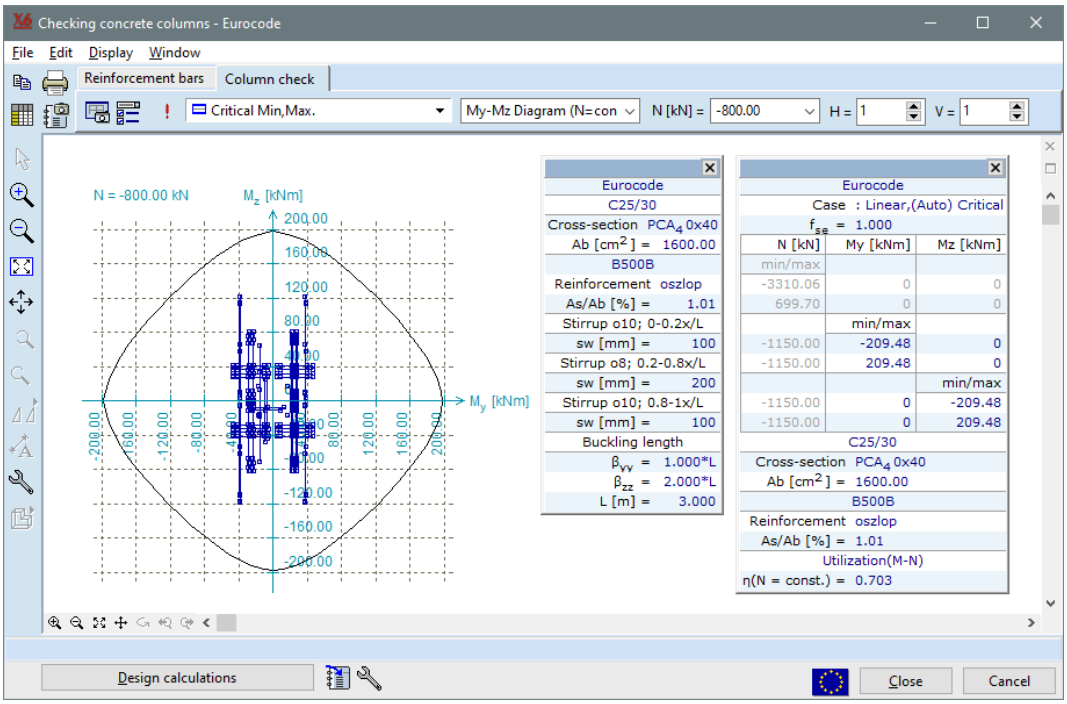


This display mode can be used with cross-sections that are symmetric. You can display the design values of the internal forces, by enabling the *Write values to* check-box.

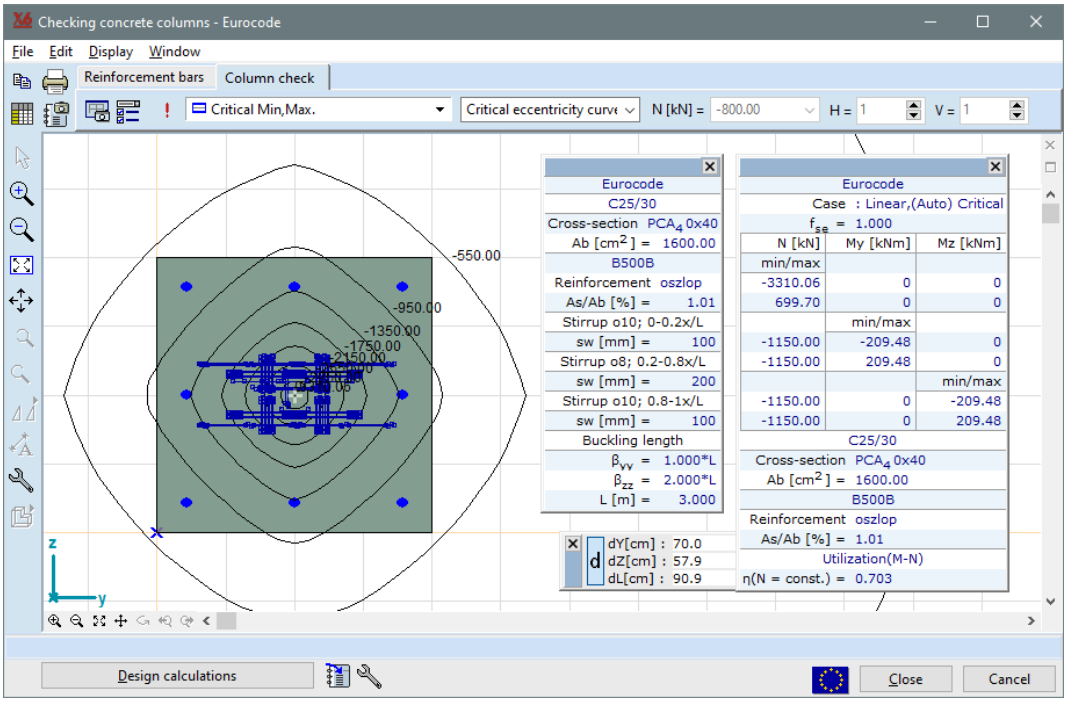
The design values of the internal forces are displayed as follows:

My-Mz diagram

Displays the M_y-M_z interaction diagram at a given N value.

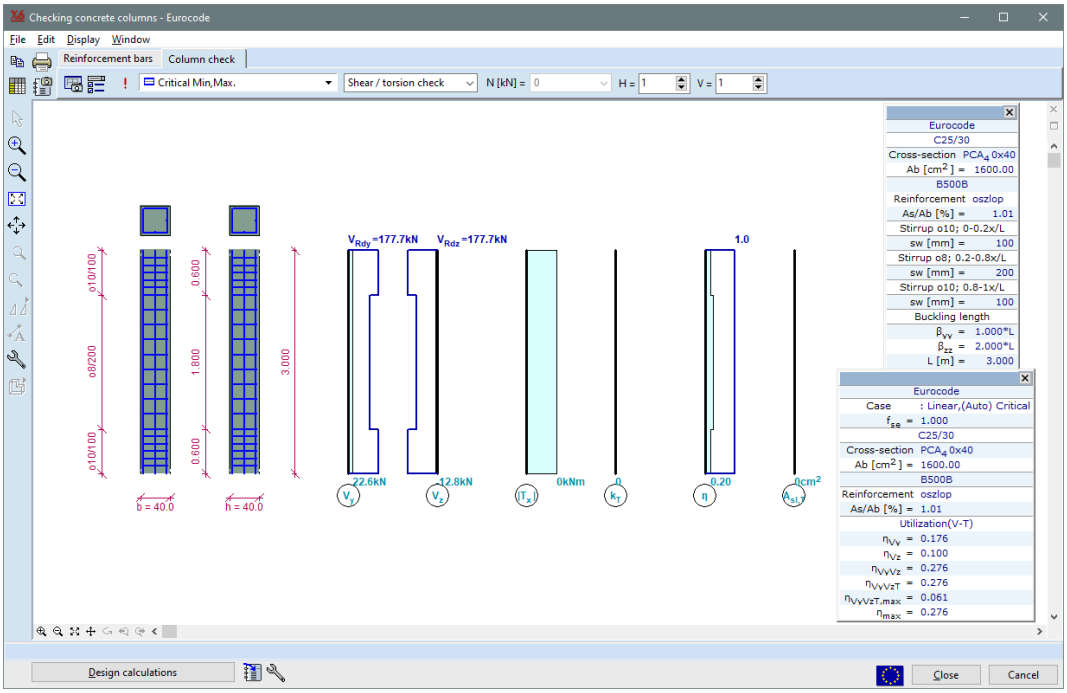


Load eccentricity limit curves



Displays the load eccentricity limit curves based on $\frac{M_{y,Rd,i}}{N_i}$ or $\frac{M_{z,Rd,i}}{N_i}$.

Shear/torsion check



The following figures and diagrams are shown in the window:

a) reinforcement drawing of the column,
b) V_y and V_z shear forces together with shear resistance (blue line – without torsion),
c) T_x torsional moment with torsion resistance (blue line),
d) k_T factor (utilization of stirrups for torsion) along the column,
e) η maximal utilization along the column,
f) $A_{sl,T}$ required additional longitudinal reinforcement for torsion along the column.

Shear and torsion check can be performed only for rectengular and circular sections.

In case of Critical or Envelope results, the results for only one load combination (that results the maximum utilization) is shown. Detailed results for every load combinations and cases are available in the table.

In order to increase the performance of the calculation, the column is divided into several sections. New section boundary is created where any of internal forces reaches min./max. value, where stirrup's parameters change and where V_y, V_z or T_x internal force diagram changes.
k_T, η and $A_{sl,T}$ result components are calculated at the bottom and at the top of every sections. Maximum values are shown only for every section in the window.
See **6.5.10.4 Shear and torsion check of reinforced concrete columns**

☞ **Detailed verification results are available for each section, if the cursor is moved over the black lines representing the column axis.**

Special window layouts

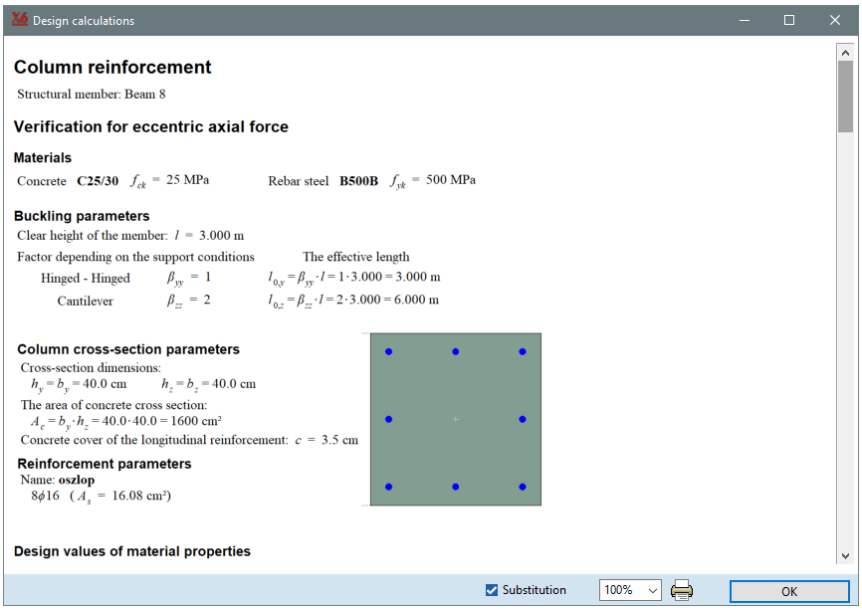
The *Window / Special window layouts* menu item offers different predefined result window layouts that can be customized by the user later.

Design calculations

The details of calculations according to the current design code are displayed as a multi-page document. References to sections and formulas of the design code appear in blue. It is only available if column check is performed and column design results are available.



Clicking on the *Settings* icon beside the *Design calculations* button allows setting the basic units for force and length used in the design calculations. More detailed design calculation can be selected as well.
A short summary of design calculations is available, which can be printed on its own or together with the detailed calculations.





Add design calculation to the current report
The current report can be selected in the Table Browser menu. See *Report / Current report*. See **2.9 Table Browser**

Detailing rules

The software checks the most critical detailing rules according to the selected design code. If there are not fulfilled detailing rules, the software warns the user in warning messages and in the design calculations.

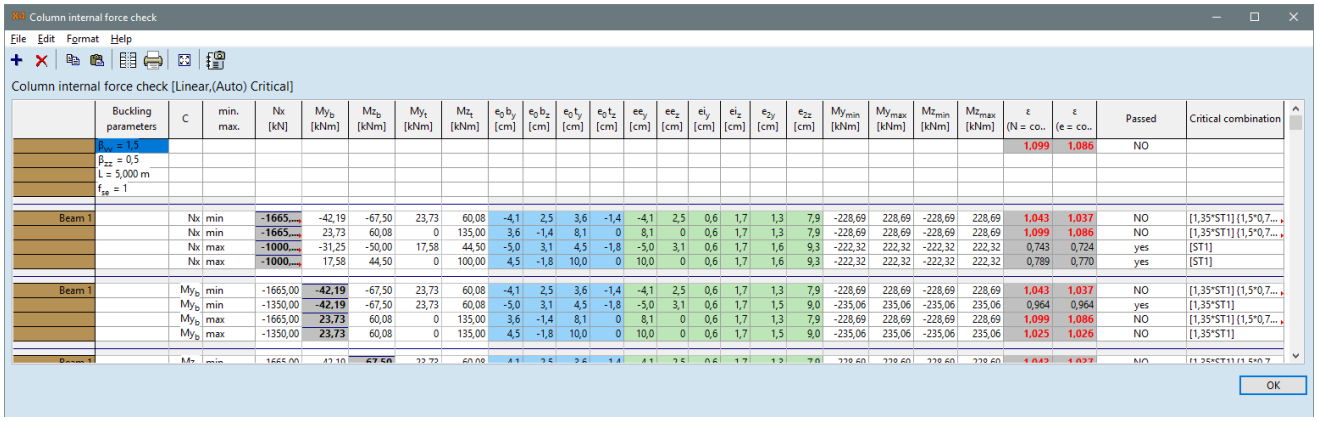
Utilization values in table

The calculated utilizations are available in table format. If *N-M* surface, *N-My* diagram, *N-Mz* diagram, *My-Mz* diagram or *Critical eccentricity curves* display mode is active, the presented utilizations are related to the column verification against bending with or without axial force. If Shear/torsion check display mode is active, the results of shear and torsion verification are shown.
The *Column internal force check* table contains the maximum normal forces and moments at the top and bottom end of the selected columns and different eccentricity values. Additional columns displaying $My_{Rd,min}$, $My_{Rd,max}$, $Mz_{Rd,min}$, $Mz_{Rd,max}$ moment resistance maximum values at the given *Nx* are also available.

Utilization – bending with or without axial force

The program calculates two types of utilization. The first one is *ε (N = const.),* the moment utilization: this is defined on the *My-Mz* diagram as the ratio of the distance of design force point from the origin to the distance of the intersection point of the curve and the half line drawn from the origin through the same point from the origin. The second one is *ε (e = const.)* the utilization for constant eccentricity.

It is defined in the *N-My-Mz* space as the ratio of the distance of design force point from the origin to the distance of the intersection point of the N-M surface and the half line drawn from the origin through the same point from the origin.



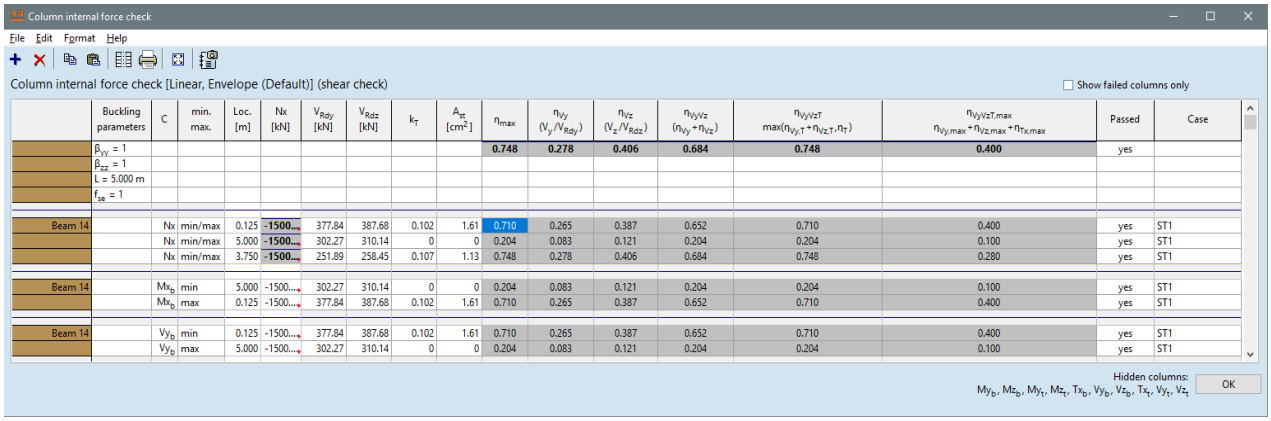
Utilization – shear/torsion

The program shows the following utilizations in the table:

- η_{Vy}: Shear utilization in local y direction without torsion.
- η_{Vz}: Shear utilization in local z direction without torsion.
- η_{VyVz}: Overall shear utilization without torsion.
- η_{VyVzT}: Overall utilization with shear and torsion interaction.
- $\eta_{VyVzT,max}$: Utilization of concrete compression strut.
- η_{max}: Maximum utilization (max. value of the above listed utilizations).

Besides the utilizations, the program lists the design shear resistances related to local y and z axes, the shear resistant reduction factor of stirrups due to torsion and the additional longitudinal reinforcement required for torsion.

See **6.5.10.4 Shear and torsion check of reinforced concrete columns**



Show failed columns only

Check this option to display the failed columns only.

Custom internal forces in result table

Adds new custom internal forces to the table: By clicking this button, new custom internal forces can be assigned to the column. The software saves these internal forces and it takes into account these values in column design check together with internal forces coming from the analysis.

Deletes selected rows from the table.

Pasting custom internal forces from the clipboard

Custom internal forces can be pasted in the table from the clipboard as well. If *Paste lines at the end* option is selected, the table previously copied to the clipboard needs to consist of exactly 11 columns according to the number and order of internal force columns in the result table *(N_x, M_{ya}, M_{za}, M_{yf}, M_{zf}, T_{xa}, V_{ya}, V_{za}, T_{xf}, V_{yf}, V_{zf})*. The same number of columns and rows need to be selected in the result table as the copied table has if *Overwrite table beginning from current cell* option is selected.

Result components

If there are reinforced concrete columns with actual column reinforcement in the model, new results components become available under *R. C. Design* tab. These new result components show the calculated utilization values and required additional longitudinal reinforcement due to torsion:

- *N-M utilization*,
- *V-T utilization*,
- $A_{sl,T}$ (required additional longitudinal reinforcement due to torsion),
- *Maximum utilization* (greater of *N-M utilization* and *V-T utilization)*.

Results in the Table Browser

Design results are also available in the Table Browser (**see 2.9 Table Browser**), where the results of every reinforced concrete column are available in summary tables.

6.5.10.1 Eccentricity calculation

Eccentricity calculation in AXISVM is based on a method assuming that the value of bending moments along the column are the sum of first order bending moments, bending moments due to geometrical imperfections and second order bending moments (EN 1992-1-1 5.8.8.2). Any of the sections along the column may be critical.

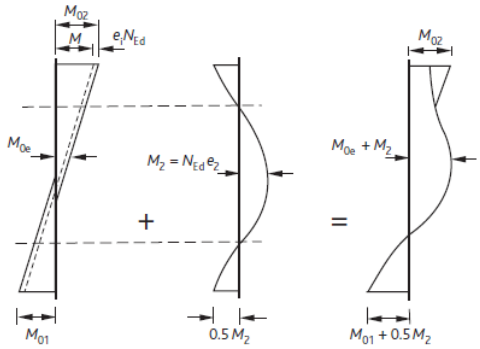
$$M_{Ed} = M_0 + M_i + M_2$$

where

M_0 first order bending moment

M_i bending moment due to geometrical imperfections

M_2 second order bending moment



(*A. J. Bond et al., How to Design Concrete Structures using Eurocode 2. UK, 2006*)

According to this method the verification of a column with arbitrary boundary conditions can be performed if the bending moments are available in every section.

First order bending moments

AXISVM analyses the first order bending moment diagram along the column and it divides the column based on the number of moment extremes. This way the effect of intermediate forces on the column can be taken into account. Bending moments between extreme values are determined with linear interpolation.

Bending moments due to imperfections

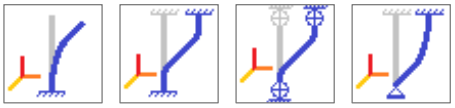
In case of braced elements (non-sway buckling shapes) bending moments due to geometrical imperfections are taken to be constant along the column because 1) the axis of columns on different storeys are not necessarily collinear, 2) if the column has rigid connections, a local curvature causes the same bending moment *(N·ei)* at the ends than at the middle but of opposite sign.

Buckling shapes of braced elements:



In case of sway buckling shapes, linear bending moment diagram is assumed along the beam if it is specified in the code.

Buckling shapes of sway elements:



If global imperfections are taken into account by extra loads acting on the structure, they must be ignored in column reinforcement calculation. To achieve this, check *Local imperfections only*. If a sway buckling shape is selected only a constant local geometric imperfection is considered (calculated with a $\beta \leq 1.0$ buckling factor).

Second order moments

Based on the selected design code, second order moments are calculated from the first order bending moments (extended with bending moments due to imperfections) or based on the buckling shape. The first calculation method is called nominal stiffness method, while the second one is the nominal curvature method.

Nominal curvature method

Second order moments are calculated with trigonometric functions assuming that the bending moment at the maximum amplitude is equal to $N \cdot e_2$. The maximum amplitude is obtained according to the selected buckling shape (see **6.5.4 Actual reinforcement against biaxial bending (column)**)



1) $e_2 = \sin\left(\frac{\pi x}{L}\right) e_{2,0}$



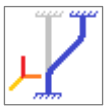
2) $e_2 = \cos\left(\frac{\pi x}{2L}\right) e_{2,0}$



3) $e_2 = \cos\left(\frac{3\pi x}{2L}\right) e_{2,0}$



4) $e_2 = \cos\left(\frac{2\pi x}{L}\right) e_{2,0}$



5) $e_2 = \cos\left(\frac{\pi x}{L}\right) e_{2,0}$



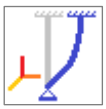
6) $e_2 = \left[1 - \frac{\beta - 0.5}{0.5}\right] \cos\left(\frac{2\pi x}{L}\right) e_{2,0} - \left[1 - \frac{1-\beta}{0.5}\right] \sin\left(\frac{\pi x}{L}\right) e_{2,0}$



7) $e_2 = \cos\left(\frac{\pi x}{L}\right) e_{2,0}$



8) $e_2 = \cos\left(\frac{3\pi x}{2L} + \frac{\pi}{2}\right) e_{2,0}$



9) $e_2 = \cos\left(\frac{\pi x}{2L} + \frac{\pi}{2}\right) e_{2,0}$

$e_{2,0}$ is the maximum amplitude of second order eccentricity.

Nominal stiffness method

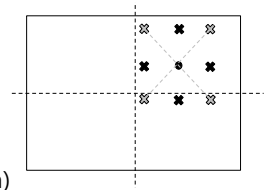
This method ignores the buckling shape. The sum of first order bending moments and bending moments due to imperfections are multiplied based on the ratio of design normal force and the buckling force. For further details see EN 1992-1-1. Dutch and Danish NA-s require application of this method.

Minimal eccentricity

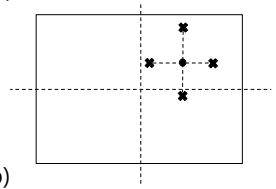
Design codes usually require the consideration of a minimum eccentricity if the calculated eccentricity is lower than this limit. In AXISVM, if the sum of first order eccentricity, eccentricity due to imperfections and second order eccentricity does not reach the minimum, eccentricity due to imperfections is increased in order to ensure that the sum of eccentricities is not less than the minimum

Biaxial bending

A reinforced concrete column is generally subjected to biaxial bending because geometric imperfections and second order eccentricities may be considered in both principal directions. The center of the cross represents the sum of first order eccentricity and eccentricity from geometric imperfections. Second order eccentricities are considered with both +/- sign. Second order eccentricities can be considered in both y and z directions simultaneously (a) or independently (b) (A. W. Beeby, R. S. Narayanan (2009): *Designer's guide to Eurocode 2: Design of concrete structures*). (see **6.5.4 Actual reinforcement against biaxial bending (column)**)



a)

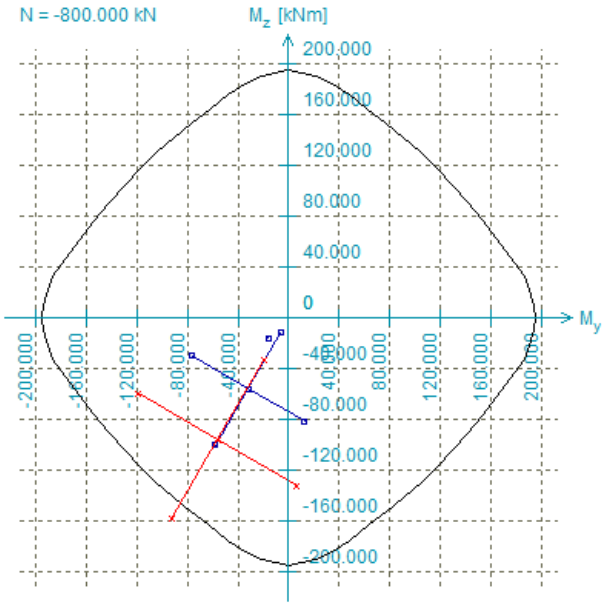


b)

☞ **Note: Figures and illustrations in this section have been saved considering case (b). If the second order eccentricities were considered simultaneously (case(a)), the figures and illustrations would show different representation of eccentricities.**

Double symmetric columns

Double symmetric, circle or square shaped RC columns are very common in practice. If $I_y = I_z$ both for the cross section and the reinforcement and the boundary conditions are also identical in *y* and *z* directions, second order eccentricities are applied neither in *y* nor in *z* direction but according to the resultant first order bending moments at the critical cross section for buckling.



6.5.10.2 Check of reinforced columns according to Eurocode 2 (bending with axial force)

Design moments in bending directions are $M_d = N_d \cdot e_d$
where Nd is the normal force in the column and $e_d = e_e + e_i + e_2$ is the critical eccentricity in the given bending direction. Minimal eccentricity: max (20 mm, h/30).

e_e is the **initial eccentricity** calculated from the first order force and moment.
Initial eccentricities at the ends of the investigated section:

$$e_e = M_I / N_d$$

Initial eccentricities at intermediate cross-section of the investigated section (equivalent eccentricity):

$$e_e = \max \begin{Bmatrix} 0.6 \cdot e_a + 0.4 \cdot e_b \\ 0.4 \cdot e_a \end{Bmatrix} \text{ and } |e_a| \geq |e_b|,$$

where e_a and e_b are the initial eccentricities at the ends of the investigated section.

e_i is the **eccentricity from geometric imperfections.**
$e_i = \alpha_h \Theta_0 \frac{l_0}{2}$, where l_0 is the buckling length, $\alpha_h = 2/\sqrt{l}$ and $\frac{2}{3} \leq \alpha_h \leq 1$, where l is the column height.
Θ_0 – according to NA of the selected code.

e_2 is the **second order eccentricity.**
Second order eccentricity is considered if $\lambda \geq \lambda_{lim} = 20 \frac{ABC}{\sqrt{n}}$ where

$$n = \frac{N_{Ed}}{A_c f_{cd}} \qquad A = \frac{1}{1+0.2\varphi_{ef}}, \qquad B = \sqrt{1+2\omega}, \qquad \omega = \frac{A_s f_{yd}}{A_c f_{cd}},$$

C is a constant that can be changed in *Design codes* window.

☞ **The calculation of λlim may be different according to NA of the selected code!**

In case of DIN EN 1992-1-1: $n < 0.41 \rightarrow \lambda_{lim} = \frac{16}{\sqrt{n}}; n > 0.41 \rightarrow \lambda_{lim} = 25.0$

In case of Italian Code: $\lambda_{lim} = \frac{25}{\sqrt{n}}$

In case of NS EN 1992-1-1 : $\lambda_n = \sqrt{\frac{n}{1+k_a \omega}} \geq \lambda_{n,lim} = 13 A_\varphi$

The prescribed calculation method of e_2 can be nominal curvature method or nominal stiffness method according to NA of the selected code. In some NAs, the application of both methods is allowed. In these cases, nominal curvature method is applied.

Nominal curvature method

$$e_2 = \frac{1}{r}\frac{l_0^2}{c}$$

where $\frac{1}{r} = K_r K_\varphi \frac{f_{yd}}{E_s \cdot 0.45 \cdot d'}$

$8 \le c \le \pi^2$ based on the first order bending moment diagram

$K_r = \min\left\{\frac{N_u' - N_{Ed}}{N_u' - N_{bal}}; 1.0\right\}$, $K_\varphi = \max\{1 + \beta\varphi_{ef}; 1.0\}$,

$\beta = 0.35 + \frac{f_{ck}}{200} - \frac{\lambda}{150}$, where f_{ck} is in N/mm2 unit

$d' = \frac{h}{2} + i_s$, where i_s is the radius of inertia of the longitudinal reinforcement

Nominal stiffness method

$$e_2 = (e_e + e_i)\left[1 + \frac{\beta}{N_B/N_d - 1}\right],$$

where $\beta = \pi^2/c_0$

N_B is the buckling load based on nominal stiffness

$EI = K_c E_{cd} I_c + K_s E_s I_s$ is the nominal stiffness

where

E_{cd} is the design value of the modulus of elasticity of concrete, see 5.8.6 (3)

I_c is the moment of inertia of concrete cross-section

E_s is the design value of the modulus of elasticity of reinforcement, 5.8.6 (3)

I_s is the second moment of area of reinforcement, about the centre of area of the concrete

K_c is a factor for effects of cracking, creep etc, see 5.8.7.2. (2), (3)

K_s is a factor for contribution of reinforcement, see 5.8.7.2. (2), (3)

Eccentricities are determined in both bending planes. The program checks the following design situations:

$\lambda_y/\lambda_z \le 2$ and $\lambda_z/\lambda_y \le 2$,

$\frac{e_{dy}/b_{eq}}{e_{dz}/h_{eq}} \le 0{,}2$ or $\frac{e_{dz}/h_{eq}}{e_{dy}/b_{eq}} \le 0{,}2$ *

if $e_{dy}/b_{eq} > e_{dz}/h_{eq}$

$M_{dy,1} = 0$

$M_{dz,1} = -N_d\left(e_{ey} + e_{iy} \pm e_{2y}\right)$

otherwise

$M_{dy,2} = N_d(e_{ez} + e_{iz} \pm e_{2z})$

$M_{dz,2} = 0$

otherwise

$M_{dy,1} = N_d(e_{ez} + e_{iz})$

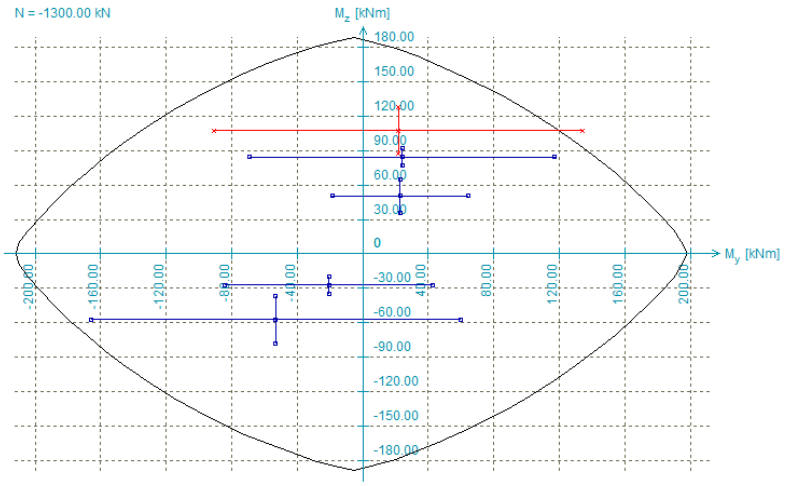
$M_{dz,1} = -N_d\left(e_{ey} + e_{iy} \pm e_{2y}\right)$

$M_{dy,2} = N_d(e_{ez} + e_{iz} \pm e_{2z})$

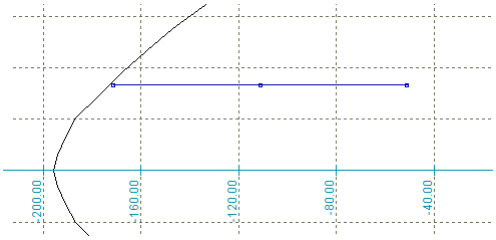
$M_{dz,2} = -N_d\left(e_{ey} + e_{iy}\right)$

* except DIN EN 1992-1-1 NA

AXISVM checks whether the calculated design loads *(M_{dy}, M_{dz}, N_d)* are inside the N-M strength interaction diagram. If it is not satisfied in any of the design situations, the column with the given cross-section and reinforcement fails.

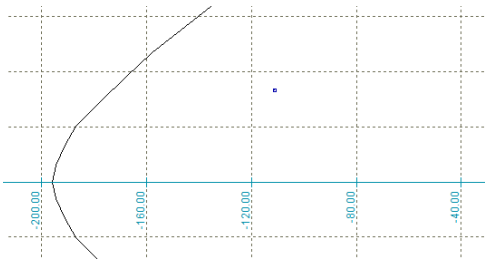


The following situations may occur:



Possible reasons:

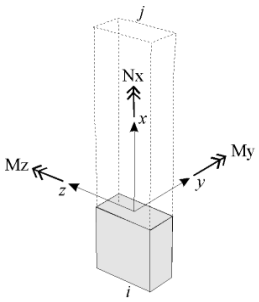
- the slenderness related to the buckling about y or z axis is lower than the limit value λ_{lim};
- at free ends or at ends with hinged supports the second order moment is zero;
- in case of biaxial bending, if the criteria met, eccentricites are shown only in the relevant direction;
- the consideration of second order eccentricity is turned off about one axis.



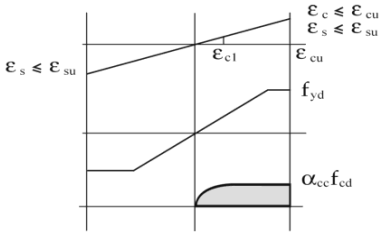
Possible reasons:

- the slenderness related to the buckling about both y and z axis is lower than the limit value λ_{lim};
- at free ends or at ends with hinged supports the second order moment is zero;
- the consideration of second order eccentricity is turned off for both axes.

On *N-MR* strength interaction diagrams and on load eccentricity limit curves points represent these design loads.Custom force and moment values can also be entered into the table. These points will be displayed in the *N-MR* strength interaction diagrams and in the load eccentricity limit curves.Signs of the forces and moments are determined according to the picture.



The calculation takes the following assumptions:



σ,ε diagrams:

☞ **Longitudinal reinforcement bars thinner than 1/15 of the stirrup distance will be ignored for compression.**

6.5.10.3 Check of reinforced columns according to SIA 262 (bending with axial force)

Design moments in bending directions are $M_d = N_d \cdot e_d$
where Nd is the normal force in the column and $e_d = e_{1d} + e_{0d} + e_{2d}$ is the critical eccentricity in the given bending direction.

e_{0d}: **increment due to geometric imperfections**

$e_{0d} = \max\left\{\alpha_i \frac{l_{cr}}{2}; \frac{d}{30}\right\}$

where $\frac{1}{200} \geq \alpha_i = \frac{0.01}{\sqrt{l}} \geq \frac{1}{300}$

l_{cr} is the buckling length, l is the actual length, *d* is the effective height of the cross-section.

e_{1d} : is the **initial eccentricity** calculated from the first order force and moment.
Initial eccentricities at the ends of the investigated section:

$$e_{1d} = M_I / N_d$$

Initial eccentricities at intermediate cross-section of the investigated section (equivalent eccentricity):

$$e_{1d} = \max\begin{Bmatrix} 0.6 \cdot e_a + 0.4 \cdot e_b \\ 0.4 \cdot e_a \end{Bmatrix} \; and \; |e_a| \geq |e_b|,$$

where e_a and e_b are the initial eccentricities at the ends of the investigated section.

e_2: **second order increment of the eccentricity.**

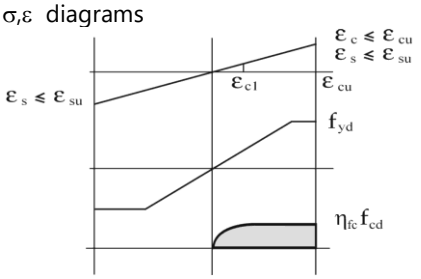
$e_{2d} = \chi_d \frac{l_{cr}^2}{\pi^2}$,

where $\chi_d = \frac{2f_{yd}}{E_s(d-d')}$ if approximate curvature option is checked, otherwise $\chi_d = \frac{\varepsilon_{sd}-\varepsilon'_{sd}}{d-d'} + \frac{|\varepsilon_{c,\infty}|}{d}$.
Eccentricities are determined in both bending planes. The program checks the following design situations:

$$M_{dy,1} = N_d(e_{1z} + e_{0z})$$
$$M_{dz,1} = -N_d(e_{1y} + e_{0y} \pm e_{2y})$$
$$M_{dy,2} = N_d(e_{1z} + e_{0z} \pm e_{2z})$$
$$M_{dz,2} = -N_d(e_{1y} + e_{0y})$$

AXISVM checks whether the calculated design loads *(M_{dy}, M_{dz}, N_d)* are inside the N-M strength interaction diagram. If it is not satisfied in any of the design situations, the column with the given cross-section and reinforcement fails.

The calculation takes the following assumptions: σ,ε diagrams



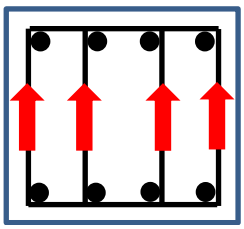
☞ **Longitudinal rebars will not be taken into account for compression if any of the following criteria is met (s is the stirrup distance):**

∅ < 8, s > 15 ∅, s > a_{min}, s > 300 mm

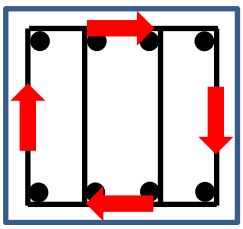
6.5.10.4 Shear and torsion check of reinforced concrete columns

Shear resistant reduction factor of stirrups due to torsion (k_T)

According to design codes and design manuals, the transverse reinforcement cannot be fully utilized for shear and torsion, respectively. For this reason, the program calculates the utilization of the stirrups considered only torsional effects (k_T) and it calculates a reduced shear resistance for stirrups against shear forces, the considered reduction factor is 1-k_T (only used for the external stirrups). This reduced shear resistance is used by the calculation of the overall utilization with shear and torsion interaction (η_{VyVzT}). The equilibrium forces in the stirrups against shear and torsional effects, respectively:



V_z shear force



T_x torsional moment

If the utilization of stirrups considered only torsion (k_T) exceeds 1.0, the resistance of stirrups cannot be considered against shear forces. In these cases, overall utilization with shear and torsion interaction is η_{VyVzT} cannot be calculated that is marked with ***???*** symbol.

Torsion is considered by the verification only if torsional moment considerably influences the shear resistance (k_T > 0.01), the torsion moment exceeds 1% of torsional cracking moment, or the following condition is satisfied:

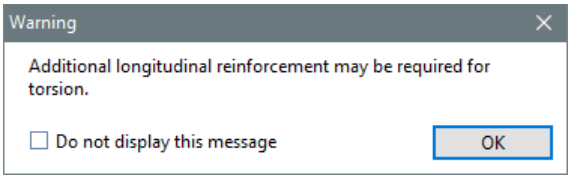
$$\frac{V_{y,E}}{V_{y,R,c}} + \frac{V_{z,E}}{V_{z,R,c}} + \frac{T_{x,E}}{T_{x,R,c}} > 1$$

where $V_{y,R,c}$, $V_{z,R,c}$ and $T_{x,R,c}$ are the shear resistance of the concrete related to local *y* and *z* axes and the torsional cracking moment, respectively. Otherwise, the shear resistant reduction of stirrups due to torsion is not considered *(k_T = 0)*.

Additional longitudinal reinforcement required for torsion ($A_{sl,T}$)

In case of a reinforced concrete beam/column, longitudinal reinforcement is also required against torsional moment if condition above is satisfied. This longitudinal reinforcement should be distributed over the length of side of the section. Bending reinforcement may not be considered against torsion. Torsional effects are typically not considerable in case of columns, torsional longitudinal reinforcement cannot be defined. The program calculates an additional longitudinal reinforcement may be required due to torsion.

☞ **If additional longitudinal reinforcement is necessary a warning message appears. If *Do not display this message* option is checked, the message is not displayed while the Checking concrete columns window is open.**



Overall shear utilization without torsion (η_{VyVz})

Rectangular column section:

$$\eta_{VyVz} = \frac{V_{y,E}}{V_{y,R}} + \frac{V_{z,E}}{V_{z,R}} \leq 1 \text{ (linear sum)}$$

$$\eta_{VyVz} = \sqrt{\left(\frac{V_{y,E}}{V_{y,R}}\right)^2 + \left(\frac{V_{z,E}}{V_{z,R}}\right)^2} \leq 1 \text{ (quadratic sum)}$$

where $V_{y,E}$ and $V_{z,E}$ are calculated design shear forces related to local y and z axes, while $V_{y,R}$ and $V_{z,R}$ are the design shear resistances related to local *y* and *z* axes, respectively.
In case of circular sections, the program calculates the resultant shear force from $V_{y,E}$ and $V_{z,E}$. The resultant shear force is compared to shear resistance of the section calculated in resultant's direction:

$$\eta_{VyVz} = \frac{\sqrt{V_{y,E}^2 + V_{z,E}^2}}{V_R} \leq 1$$

Overall utilization with shear and torsion interaction (η_{VyVzT})

η_{VyVzT} utilization is evaluated similarly to η_{VyVz}, however, in this case the resistance of external stirrup legs is reduced with a factor *(1 – k_T)* based on the torsional utilization.
Torsion is considered only by the verification if torsional moment considerably influences the shear resistance *(k_T > 0.01)*, the torsion moment exceeds 1% of torsional cracking moment, or the following condition is satisfied:

$$\frac{V_{y,E}}{V_{y,R,c}} + \frac{V_{z,E}}{V_{z,R,c}} + \frac{T_{x,E}}{T_{x,R,c}} > 1$$

Utilization of the concrete compression strut ($\eta_{VyVzT,max}$)

In case of both shear and torsion verification, the resistance cannot exceed a shear force and torsional moment value, respectively, related to the failure of concrete compression strut. If shear forces and torsional moments are both present, the effect of shear and torsion interaction need to be considered by the calculation of the utilization of concrete compression strut.

Rectangular column section:

$$\eta_{VyVzT,max} = \frac{V_{y,E}}{V_{y,R,max}} + \frac{V_{z,E}}{V_{z,R,max}} + \frac{T_{x,E}}{T_{x,R,max}} \leq 1$$

In case of circular sections, the program calculates the resultant shear force from $V_{y,E}$ and $V_{z,E}$. The resultant shear force is compared to shear resistance of the section calculated in resultant's direction. The utilization is calculated as follows:

$$\eta_{VyVzT,max} = \frac{\sqrt{V_{y,E}^2 + V_{z,E}^2}}{V_{R,max}} + \frac{T_{x,E}}{T_{x,R,max}} \leq 1$$

If $\eta_{VyVzT,max}$ exceeds 1.0, the resistance of the section can be increase only by changing concrete class or increasing sections dimensions.

Shear resistance of circular cross section

The calculation shear resistance of reinforced concrete columns with circular cross section is mainly identical with the calculation performed in case of rectangular cross sections, there are only few modifications. For further information, we recommend the following research: John Orr - *Shear Capacity of Circular Concrete Sections.*

Main differences considered in the calculation:

a) By the calculation of shear resistance of the concrete *($V_{y/z,R,c}$)* and the maxmin design shear resistance *($V_{y/z,R,max}$)*, 0.8D is used as the width of cross section, where D is the diameter.

b) In case of circular stirrups, a reduction factor *(λ_1)* is used by the calculation of shear resistance due to the fact that the stirrups are not parallel to the shear force along the height of the section. λ_1 factor is calculated on the following way [Turmo et. al (2008): *Shear truss analogy for concrete members of solid and hollow circular cross sections*] [J. Orr (2009): *Shear Capacity of Circular Concrete Sections*], where z_0 is the distance between the center of tension reinforcement and the center of gravity of the section, z is the level arm and r_s is the distance between the center stirrup reinforcement and the center of gravity of the section:

$$\lambda_1 = \int_0^1 \sqrt{1-((z_0 - zX)/r_s)^2}dX$$

c) In case of spiral stirrups, another reduction factor is also used to consider that the vertical component of resisting force is not parallel to shear force. λ_2 factor is calculated on the following way [I. Feltham (2004): *Shear in reinforced concrete piles and circular columns]*, where p is the pitch of the spiral stirrup:

$$\lambda_2 = \left(\left(\frac{p}{2\pi r_s}\right)^2 + 1\right)^{-0.5}$$

Level arm

During the calculation of shear resistance of a reinforced concrete section, the level arm (z) of internal forces plays an important role. The level arm is interpreted as the distance between the center of tension reinforcement and the center of compression zone. In case of beams, z is approximately selected 0.9d, where d is the effective depth. However, if axial force is present, this approximation cannot be used. The program automatically calculates the level arm of internal forces with the consideration of the axial force in the section.

☞ **The program checks the detailing rules in the standard regarding minimum/maximum longitudinal reinforcement ratios, maximum distance of stirrups and minimum diameter of stirrups. Warning message calls the attention of the user in the information window and instantly after the design calculations if any of these detailing rules is not fulfilled. During the definition of the parameters, the user is informed of the fulfilment of detailing rules regarding maximum distance of stirrups and minimum diameter of stirrups by clicking the button.**

See 6.5.4 Actual reinforcement against biaxial bending (column)

6.5.10.5 Shear and torsion check according to Eurocode 2

Design shear resistance

$$V_{Rd} = Min\left[Max\left[V_{Rd,c}, V_{Rd,s}\right], V_{Rd,max}\right]$$

where $V_{Rd,c}$ is the shear resistance of concrete, $V_{Rd,s}$ is the shear resistance of stirrups, and $V_{Rd,max}$ is the maximal shear resistance of the section according to the resistance of concrete compression strut.

Shear resistance of concrete

$$V_{Rd,c} = \left[C_{Rd,c}k(100\rho_l f_{ck})^{1/3} + k_1\sigma_{cp}\right]b_w d \geq \left(v_{min} + k_1\sigma_{cp}\right)b_w d$$

where:

- ρ_{sl} ratio of tensile reinforcement,
- b_w is the smallest width of the cross-section in the tensile area (circular section 0.8D),
- d is the effective depth,
- σ_{cp} the axial stress in the cross-section due to loading or prestressing (compression is positive) $<0.2f_{cd}$,
- $C_{Rd,c}$, v_{min} and k_1 parameters that are regulated in the National Annex. The values of these factors can be set in the Design codes window under the Reinforced concrete tab (see 3.3.7 Design codes).

Shear resistance of stirrups

Rectangular cross section:

$$V_{Rd,s} = \frac{A_{sw}}{s} z f_{ywd} \cot\theta$$

Circular cross section:

$$V_{Rd,s} = \lambda_1 \lambda_2 \frac{A_{sw}}{s} z f_{ywd} \cot\theta$$

where:

- A_{sw} is the cross-sectional area of the shear reinforcement,
- s is the spacing of stirrups or the pitch of spiral stirrup,
- z is the level arm,
- f_{ywd} is the design yield strength of the shear reinforcement,
- θ is the angle of shear crack,
- λ_1 is the reduction factor for circular stirrups,
- λ_2 is the reduction factor for spiral stirrups.

Maximum design shear resistance

$$V_{Rd,max} = \alpha_{cw} b_w z \nu_1 f_{cd} \frac{\cot\theta + \cot\alpha}{1 + \cot^2\theta}$$

where:

- α_{cw} is a coefficient taking account of the state of the stress in the compression chord,
- ν_1 is a strength reduction factor for concrete cracked in shear,
- α is the angle between shear reinforcement and the beam axis perpendicular to the shear force (in case of stirrups α=90°).

Torsional cracking moment – torsional resistance of concrete

$$T_{Rd,c} = 2 A_k f_{ctd} t_{ef}$$

where:

- A_k is the area enclosed by the centre-lines of the connecting walls,
- t_{ef} is the effective wall thickness.

Maximum design torsional moment

The maximum torsional moment related to the resistance of concrete compression strut is calculated as follows:

$$T_{Rd,max} = 2\nu\alpha_{cw} f_{cd} A_k t_{ef} \sin\theta \cos\theta$$

Calculation of k_T

$$k_T = \frac{T_{Ed} s}{2 A_{sw} A_k f_{ywd} \cot\theta}$$

where:

- T_{Ed} is the design torsional moment,
- A_{sw} cross sectional area of a stirrup leg.

Calculation of $A_{sl,T}$

$$A_{sl,T} = \frac{T_{Ed} u_k \cot\theta}{2 A_k f_{ywd}}$$

where:

- u_k is the perimeter of the area A_k.

6.5.10.6 Shear and torsion check according to SIA 262

Shear resistance of stirrups

Rectangular cross section:

$$V_{Rd,s} = \frac{A_{sw}}{s} z f_{yd} \cot\alpha$$

Circular cross section:

$$V_{Rd,s} = \lambda_1 \lambda_2 \frac{A_{sw}}{s} z f_{yd} \cot\alpha$$

where:

- A_{sw} is the cross-sectional area of the shear reinforcement,
- s is the spacing of stirrups or the pitch of spiral stirrup,
- z is the level arm,
- f_{yd} is the design yield strength of the shear reinforcement,
- α is the angle of shear crack,
- λ_1 is the reduction factor for circular stirrups,
- λ_2 is the reduction factor for spiral stirrups.

Maximum design shear resistance

$$V_{Rd,c} = b_w z k_c f_{cd} \sin\alpha \cos\alpha$$

$$\varepsilon_1 = \varepsilon_x + (\varepsilon_1 + 0.001)\cot^2\alpha$$

where:

- k_c is a strength reduction factor for concrete cracked in shear.

Maximum design torsional moment

The maximum torsional moment related to the resistance of concrete compression strut is calculated as follows:

$$T_{Rd,max} = 2k_c f_{cd} A_k t_k \sin\theta \cos\theta$$

Calculation of k_T

$$k_T = \frac{T_{Ed} s}{2A_{sw} A_k f_{yd} \cot\theta}$$

where:

- T_{Ed} is the design torsional moment,
- A_{sw} cross sectional area of a stirrup leg.

Calculation of $A_{sl,T}$

$$A_{sl,T} = \frac{T_{Ed} u_k \cot\theta}{2A_k f_{yd}}$$

where:

- u_k is the perimeter of the area A_k.

6.5.10.7 Capacity design: calculation the design value of shear force according to Eurocode and SIA

In case of dissipative structures with DCM or DCH ductility class, in order to avoid shear failure in columns, the design values of shear forces are determined in accordance with the capacity design rule (if seismic load case is included in the selected load combination), on the basis of the equilibrium of the column under end moments $M_{i,d}$.

Calculation of $M_{i,d}$

$$M_{i,d} = \gamma_{Rd} M_{Rc,i} min\left\{1; \frac{\sum M_{RB}}{\sum M_{RC}}\right\}$$

where:

- i=1,2 denotes the end sections of the column,
- γ_{Rd} is the factor accounting for overstrength due to steel strain hardening and confinement of the concrete of the compression zone of the section,
- $M_{Rc,i}$ is the design value of moment capacity at end i in the sense of the seismic bending moment under the considered sense of the seismic action;
- ΣM_{RB} and ΣM_{RC} the sum of moment capacity of beams and columns connected to the same joint, respectively.

Calculation of V_{Ed}

$$V_{Ed} = V_{Ed,0} + V_{Ed,EQ}$$

where:

- $V_{Ed,0}$ denotes the shear force from non-seismic actions (from dead loads, etc.),
- $V_{Ed,EQ}$ is the design value of seismic shear force.

Plastic hinge at the top and at the bottom

$$V_{Ed,EQ} = \frac{M_{1,d} + M_{2,d}}{l_{cl}}$$

where:

- l_{cl} is the clear length of the column (set 90% of the column length as default).

Plastic hinge at the top

If plastic hinge may be formed only at the top of the column (e.g., due to hinged support, etc.), it can be set in the parameter window that plastic hinge is expected only at the top. In this case, the design value of seismic shear force is calculated on the following way:

$$V_{Ed,EQ} = \frac{\left(M_{1,Ed,0} + x \cdot M_{1,Ed,EQ}\right) + M_{2,d}}{l_{cl}}$$

where:

- $M_{1,Ed,0}$ denotes the bending moment from non-seismic actions (from dead loads, etc.) at the bottom,
- $M_{1,Ed,EQ}$ is the design value of seismic bending moment at the bottom from seismic analysis,
- x is a multiplier for seismic bending moment to reach the bending moment capacity of plastic hinge at the top.

Plastic hinge at the bottom

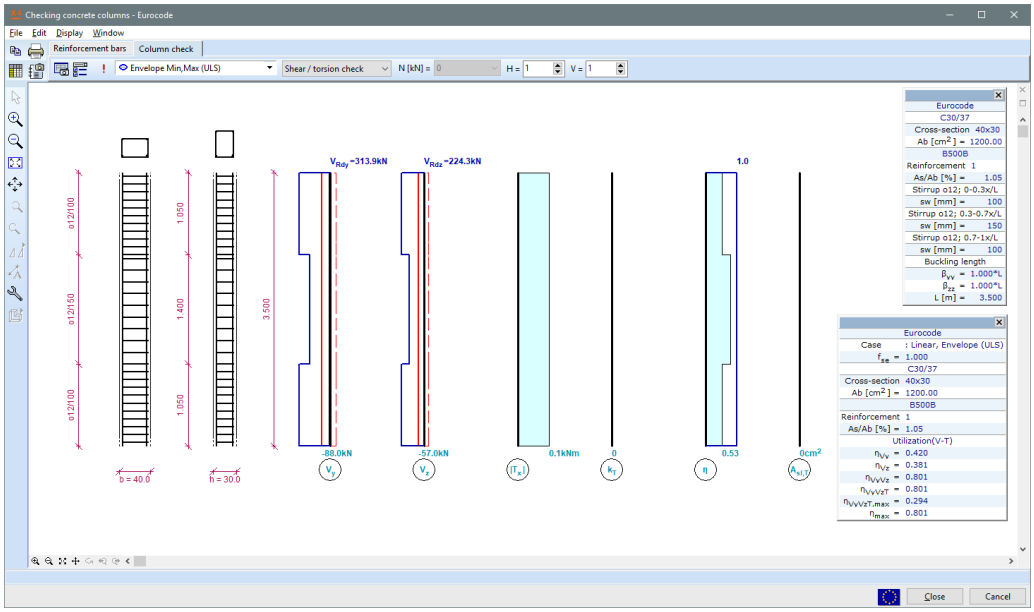
$$V_{Ed,EQ} = \frac{M_{1,d} + \left(M_{2,Ed,0} + x \cdot M_{2,Ed,EQ}\right)}{l_{cl}}$$

where:

- $M_{2,Ed,0}$ denotes the bending moment from non-seismic actions (from dead loads, etc.) at the top,
- $M_{2,Ed,EQ}$ is the design value of seismic bending moment at the top from seismic analysis,
- x is a multiplier for seismic bending moment to reach the bending moment capacity of plastic hinge at the bottom.

☞ **If seismic shear force from seismic analysis is greater than the shear force calculated based on capacity design rules, shear force from seismic analysis is used as VEd,EQ.**

Results



The above-described calculation of design seismic shear force is performed only if there is seismic load case in the selected load combination. A seismic shear force calculated in accordance with capacity design rules is added to non-seismic shear forces in *y* and *z* directions respectively. The design value of shear force including seismic shear force coming from capacity design is represented by dashed and solid red lines. The seismic shear force needs to be considered with both positive and negative sign considering positive and negative directions of seismic loading. Solid red line represents greater absolute values.

☞ **The program checks the detailing rules in the standard regarding minimum/maximum longitudinal reinforcement ratio, maximum distance of stirrups and minimum diameter of stirrups. Warning message calls the attention of the user in the information window and instantly after the design calculations if any of these detailing rules is not fulfilled. During the definition of the parameters, the user is informed of the fulfilment of detailing rules regarding maximum distance of stirrups and minimum diameter of stirrups by clicking button, see 6.5.4 Actual reinforcement against biaxial bending (column). Checking the fulfilment of further standard specific detailing rules is the responsibility of the user.**

6.5.11 Composite column design – RC2 module

The design codes and national annexes in the software for which the design check of composite columns subjected to compression/tension force and biaxial bending can be performed are listed in **3.3.7 Design codes.**

The chapter **3.1.16.1 Cross-Section Editor** shows how to create a composite section in the software. Columns fulfil the following criteria can be designed only in RC2 module according to the design codes:

- steel grade needs to between S235 to S460;
- strength class of concrete need to be between C20/25 to C50/60;

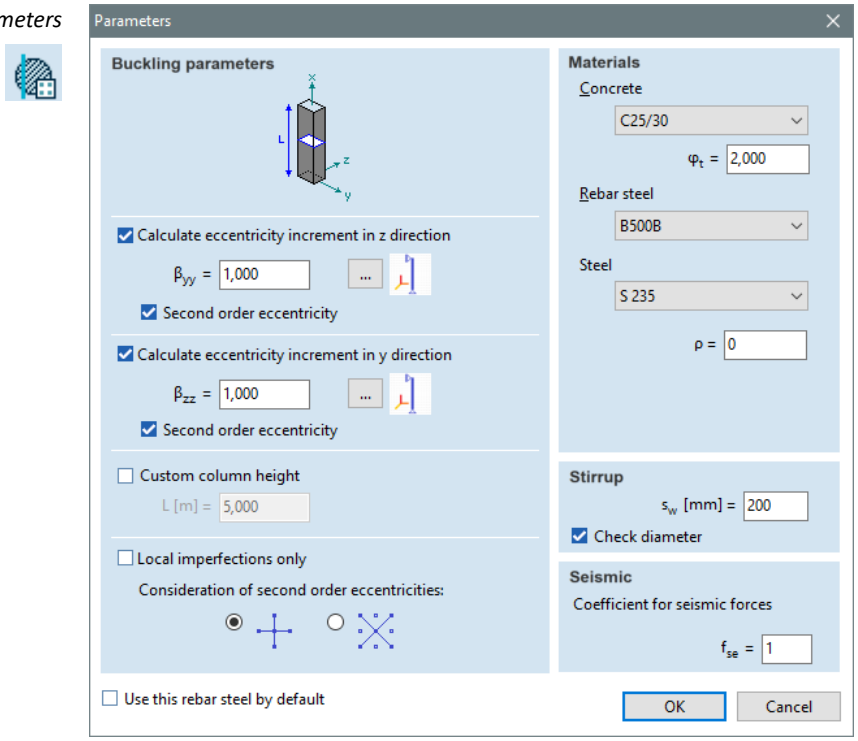
- column is isolated and part of a framed structure where the other structural members are either composite or steel members;
- the column section is doubly symmetrical.

It has to be noted that both Eurocode and SIA standard prescribes further criteria that need to be checked by the user. The program shows warning messages in the following cases:

- if the steel grade or the strength class of concrete is out of the acceptable range;
- if the effect of local buckling cannot be neglected (the design check performed by the software does not take into account the effect of local buckling);
- if the steel contribution ratio (δ) is out of the acceptable range.

The steps of design check of composite columns are very similar or identical with the steps need to be done in course of the design of conventional reinforced concrete columns (i.e., how to define reinforcement, which button need to be clicked). For this reason, the steps described in **6.5.4 Actual reinforcement against biaxial bending (column)** and **6.5.10 Column reinforcement – RC2 module** sections are not duplicated, however, the differences are underlined.

Parameters

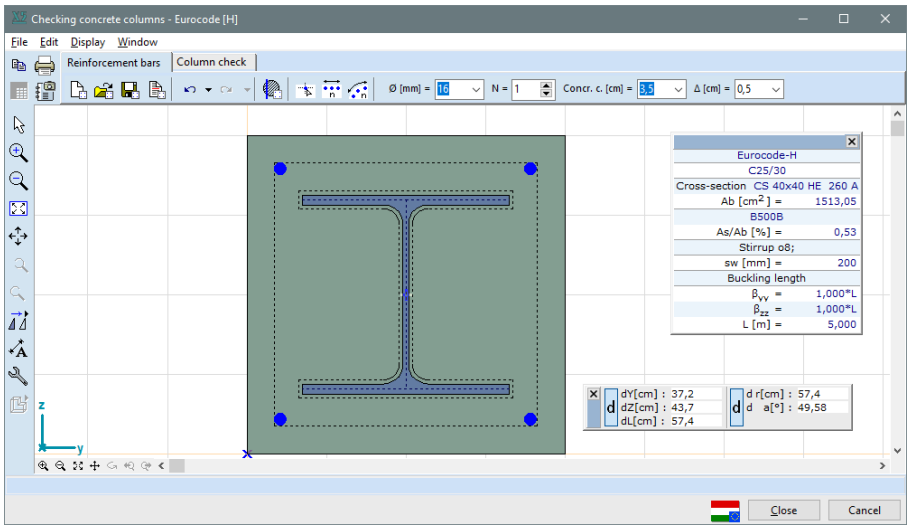


This window is used to define the design parameters similarly to conventional reinforced concrete columns. Most of the parameters are described in previous section see **6.5.4 Actual reinforcement against biaxial bending (column).** Additionally, the steel grade of steel sections and reduction factor (ρ) considering the effect of shear force on bending and compression resistance can be specified. ρ factor is used if the shear force $V_{a,Ed}$ on the steel section exceeds 50% of the design shear resistance $V_{pl,a,Rd}$ of the steel section. This factor is used to reduce the considered yield strength of the steel material int the web of the steel section:

$$(1-\rho)\cdot f_{yd}$$

Definition of reinforcement

The reinforcement of the section can be defined in the following window. It is identical compared to definition of reinforcement in case of conventional reinforced concrete column see **6.5.4 Actual reinforcement against biaxial bending (column).** In case of composite columns, the concrete cover on the steel section (Δ) can be also specified.



Column check

The program performs the verification against bending with or without axial force of the column based on the cross-section and given design properties and reinforcement parameters. The details of evaluation of the bending utilization are explained in the next sections. The graphical interpretation of the results and the result tables are discussed in the aforementioned section *see* **6.5.10 Column reinforcement – RC2 module.**

☞ **Fire design of composite columns is not supported in RC8-B module.**

6.5.11.1 Design check according to Eurocode and SIA standards

The utilization of the column subjected to uniaxial y-y and z-z bending moments are evaluated on the following way:

$$\eta_y = \frac{M_{y,Ed}}{\alpha_{M,y} M_{y,N,Rd}} \leq 1.0$$

$$\eta_z = \frac{M_{z,Ed}}{\alpha_{M,z} M_{z,N,Rd}} \leq 1.0$$

where

- $M_{y,N,Rd}$ and $M_{z,N,Rd}$ are the design value of the bending resistance of composite section taking into account the compressive normal force,
- $\alpha_{M,y}$ and $\alpha_{M,z}$ are coefficients related to bending of a composite column about the y-y axis and the z-z axis respectively (S235 – S355 α_M=0.9; S420 – S440 α_M=0.8).

Interaction curves for combined compression and uniaxial bending related y-y and z-z axes are calculated separately. The utilization of the column subjected to biaxial bending is calculated on the following way:

$$\eta = \frac{M_{y,Ed}}{M_{y,N,Rd}} + \frac{M_{z,Ed}}{M_{z,N,Rd}} \leq 1.0$$

☞ **It may happen that in the info window or in the result table ??? symbol can be seen instead of utilization of the column. This symbol means that the utilization of the column cannot be evaluated. In most of the cases, the reason of this error is that the design axial force is higher than the compression/tension resistance of the section, thus, the bending moment-based utilization of the column cannot be calculated.**

Design bending moment

$M_{y,Ed}$ and $M_{z,Ed}$ bending moments contain first order bending moment, the bending moment coming from the imperfection and second order effects. The second order effects are taken into account with the following multiplier coefficient (k) that is used to multiply the sum of first order bending moments and bending moments from the imperfection:

$$k = \frac{\beta}{1 - \frac{N_{Ed}}{N_{cr,eff}}} \geq 1.0$$

where

- β is an equivalent moment factor,
- $N_{cr,eff}$ is the critical normal force for the relevant axis and corresponding to the effective flexural stiffness calculated as follows.

$$N_{cr,eff} = \frac{\pi^2 (EI)_{eff,II}}{L_{cr}^2}$$

$$(EI)_{eff,II} = 0.9 \cdot \left(E_a I_a + E_s I_s + 0.5 \cdot E_{cm} \frac{1}{1 + \varphi_t} I_c \right)$$

where

- $E_a I_a$ is the bending stiffness of steel section,
- $E_s I_s$ is the bending stiffness of reinforcement,
- E_{cm}, φ and I_c are the secant modulus of elasticity of concrete, the creep coefficient and second moment of area of the un-cracked concrete section.

6.5.12 Beam reinforcement parameters (uniaxial bending)



Beam reinforcement parameters and actual beam reinforcement can be assigned to concrete rib and beam elements without performing design and check of beam reinforcement. Beam reinforcement design parameters can be different on each finite element. To select only a part of a structural member check the following option: *Settings / Preferences / Editing / Enable selection of finite elements on lines.*

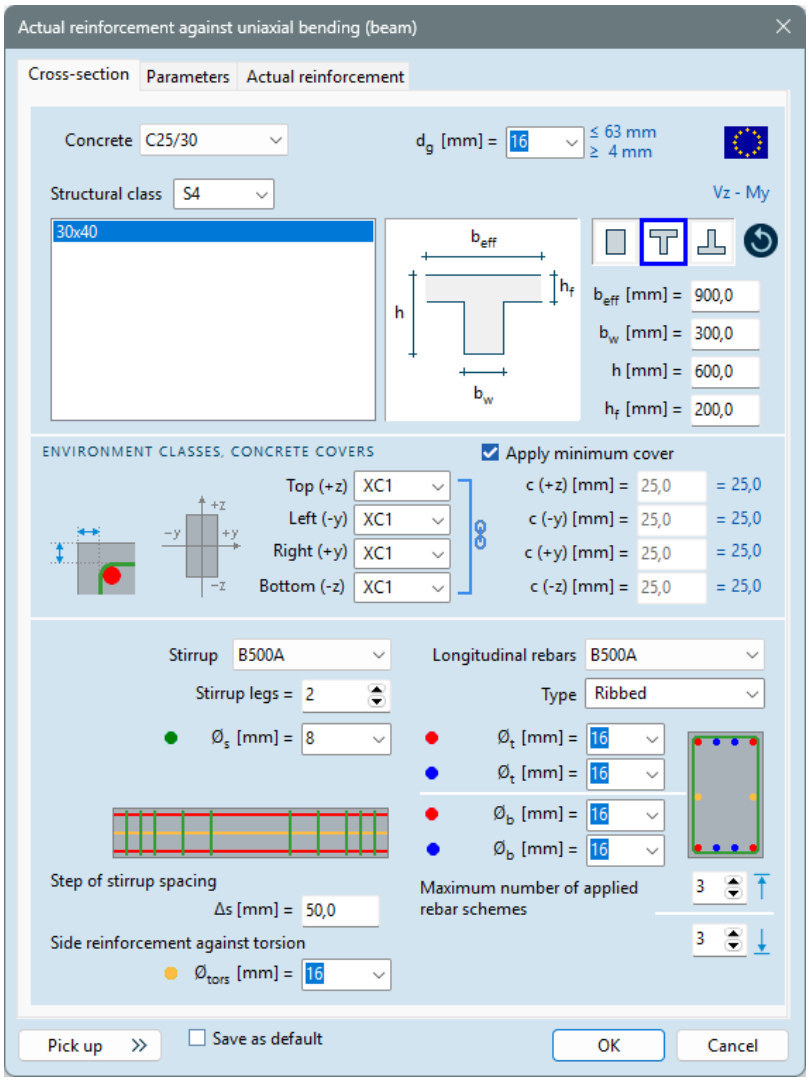
☞ **Beam reinforcement parameters dialog is suitable to define the reinforcement parameters against uniaxial bending. This can be done for vertical elements as well.**

Design of beam reinforcement (determining the required amount of reinforcement and checking it, see **6.5.13 Beam reinforcement design – RC2 module)** uses these parameters.

Designing multiple beams

Beam reinforcement parameters can be assigned to multiple selected beams simultaneously. If you select a result component from the *Calculated beam reinforcement* or *Applied beam reinforcement* category (See **6.5.13.6 Result components of beam reinforcement**), the design calculations for the beams will automatically be performed for all beams where this is required.

Cross-section

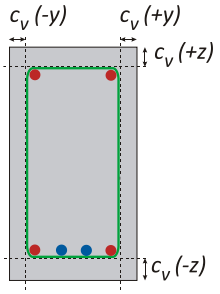


The list contains all beam design cross-sections found in the selection. The geometry parameters of the selected list item will be displayed and can be edited. Changes won't lead to recalculation of forces so cross-section dimensions should not be changed unless it is really necessary.
Concrete grade can also be overridden. Structural class and maximum aggregate size *(D_{max})* must be entered.

☞ **After changing cross-section dimensions, it is recommended to change the actual cross-section of the element and recalculate the model.**

Environment classes, concrete cover

Environment classes and concrete covers must be specified on all four sides of the beam *(+z, -y, +y, -z)*.
Concrete cover is the least distance between the stirrups and the outer surface of the concrete. The minimum required cover calculated according to the design code from environment classes and other parameters is displayed in blue at the end of each row. Checking *Apply minimum cover* will set the c_v edit fields to the calculated value. Clicking on the link symbol right to the environment classes will set the same environment class on all sides.



Reinforcement parameters

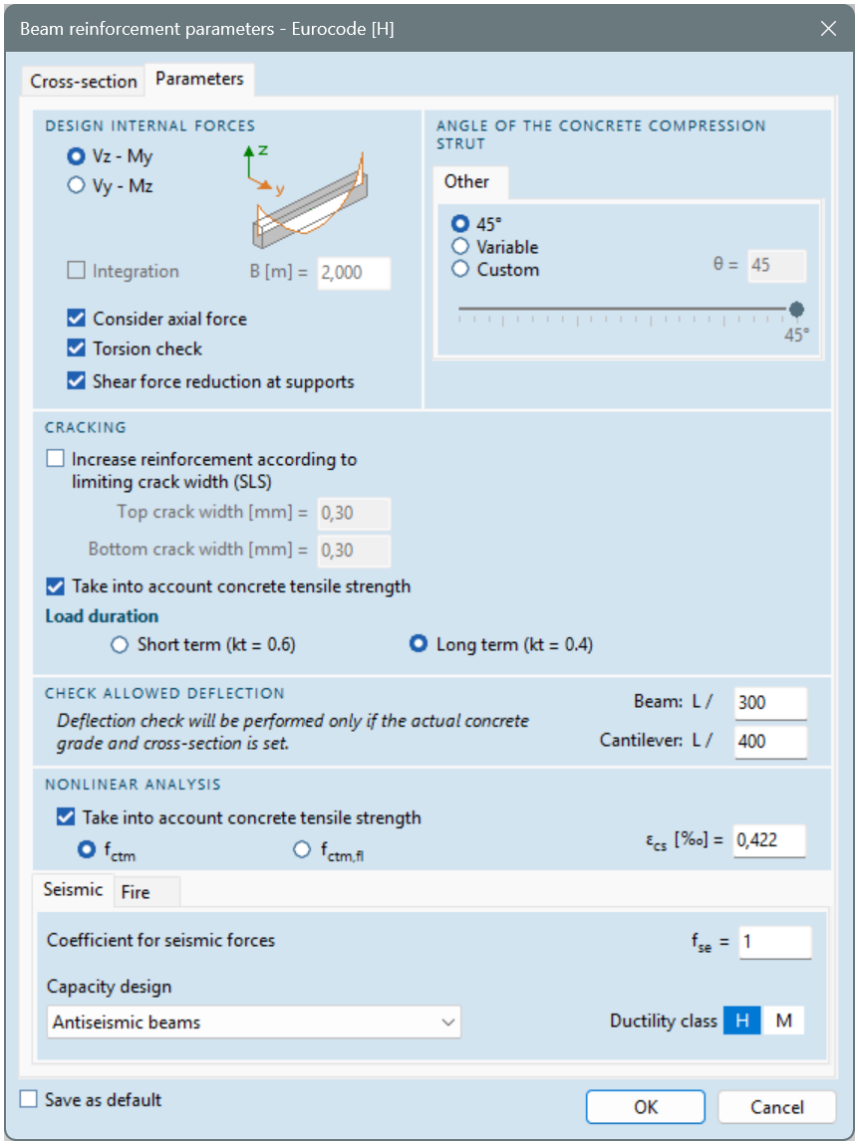
Parameters (i.e., material, diameter, etc.) of stirrups (green) and torsion rebars (yellow) can be entered on the left side, parameters of corner rebars (red) and other longitudinal rebars (blue) on the right. $Ø_t$ is the diameter of the top rebars, $Ø_b$ is the same for bottom rebars, $Ø_t$ is the diameter of side reinforcement against torsion. Checking *Use this rebar and stirrup steel by default* sets the default value of the rebar and stirrup steel grade to the value set in the dialog.

☞ **The following parameters are used only in beam reinforcement design. The program distributes stirrups and rebars according to these parameters before checking the calculated reinforcement.**

Step of stirrup spacing: Calculated stirrup spacing values will be integer multiples of the step size entered here.

Maximum number of applied rebar schemes limits the number of different rebar distribution schemes applied along the beam (number of top and bottom distributions can be set separately).

Parameters



These parameters are required to design and check beam reinforcement.

Design internal forces

Beam reinforcement can be calculated for bending in one direction. So, the beam can be designed either for Vz-My or for Vy-Mz forces.
Enable the *Consider axial force* option to include axial force in beam reinforcement design.
Enable *Torsion check* to consider the torsion moment as well when designing the beam.
Enable *Shear force reduction at supports* to apply shear force reduction methods according to the current design code.
In the case of ribs connected to domains, the *Integration* switch allows you to control whether the software reads the internal forces of the reinforced concrete beam as the results of rib elements, or integrates them in the background using a virtual strip object. The advantage of the latter is that the virtual strip also integrates the internal forces of the connected domain within the width *B* specified by the user (the total integration width is *B*, i.e., *B*/2 to the right and left of the beam axis). In the case of integration, the software can also account for the axial force arising in the beam. Otherwise, the axial force arising in the rib elements cannot be used for design, as it does not include the effect of normal stresses arising in the domain.

Angle of the concrete compression strut

Eurocode 2 allows specification of the θ angle of the concrete compression strut. According to 6.2.3 (2) $1 \leq \operatorname{ctg}\Theta \leq 2.5$. In case of a flat strut (steep cracking angle) cracks intersect only few stirrups, so concrete gets more shear stress. In case of a steep strut (flat cracking angle) cracks intersect many stirrups so shear reinforcement gets more shear stress.
In the variable truss angle method (second option) the strut angle is optimized for minimum shear reinforcement. If tension or torsional moments are not negligible the standard method must be selected (first option) where the fixed strut angle is 45°.

Cracking

Checking *Increase reinforcement according to limiting crack width* the maximum allowed crack width values can be entered. In this case the program increases the top / bottom reinforcement (maintaining the relation $A_s \leq A_{s,max}$. Maximum reinforcement can be set in the Design codes window under the Reinforced concrete tab (see **3.3.7 Design codes**)) to reduce the crack width under the specified value. To perform cracking analysis the load duration must be specified. See **6.5.7.1. Cracking calculation according to Eurocode 2.**
If the option *Take concrete tensile strength into account* is selected no cracking calculations will be performed in points where the tensile stress is below the concrete tensile strength.

Load duration

k_t is a factor depending on load duration.
For short term loads $k_t = 0.6$, for long term loads $k_t = 0.4$

Check allowed deflection

The program checks the allowed deflection according to the criteria set for beams and cantilevers. L represents the beam length. This check will be performed only if the actual concrete grade and cross-section is set.

For coefficient of seimic forces see **4.10.25 Seismic loads – SE1 module**.

Capacity design

It can be set for the beams whether they are primary (anti-seismic beams) or secondary seismic elements (non-anti-seismic beams). Plastic hinges (see **6.5.13.1 Steps of beam reinforcement design)** can be defined only in case of anti-seismic beams. The ductility class can be selected (M – DCM, H – DCH) that influences the maximum allowable distance between stirrups in order to provide the expected ductility. During the calculation of shear resistance, θ=45° is considered for the angle of concrete compression strut within the critical regions if ductility class H is selected (EN 1998-1-1).

Normalised axial force in seismic design situation

The deformation capacity of reinforced concrete elements in seismic design situation depends, among other things, on the sign and magnitude of axial force in the element. For this reason, the axial force in the elements need to be limited. In order to help the design process and the check the axial force in the elements, a normalised axial force result component can be calculated and displayed for concrete line elements.

$$\upsilon_d = \frac{N_{Ed}}{A_c f_{cd}}$$

It is important to note that in this calculation the compression force is positive. In the case of tension force, a negative value is displayed, which is of course only indicative, as the concrete cannot withstand high tensile forces. If tension force is detected, a warning message is displayed.
This result component is calculated only for seismic load cases, seismic load combinations (with or without actual seismic load case in the combination, the type of the combination matters), seismic envelopes and seismic critical combinations.

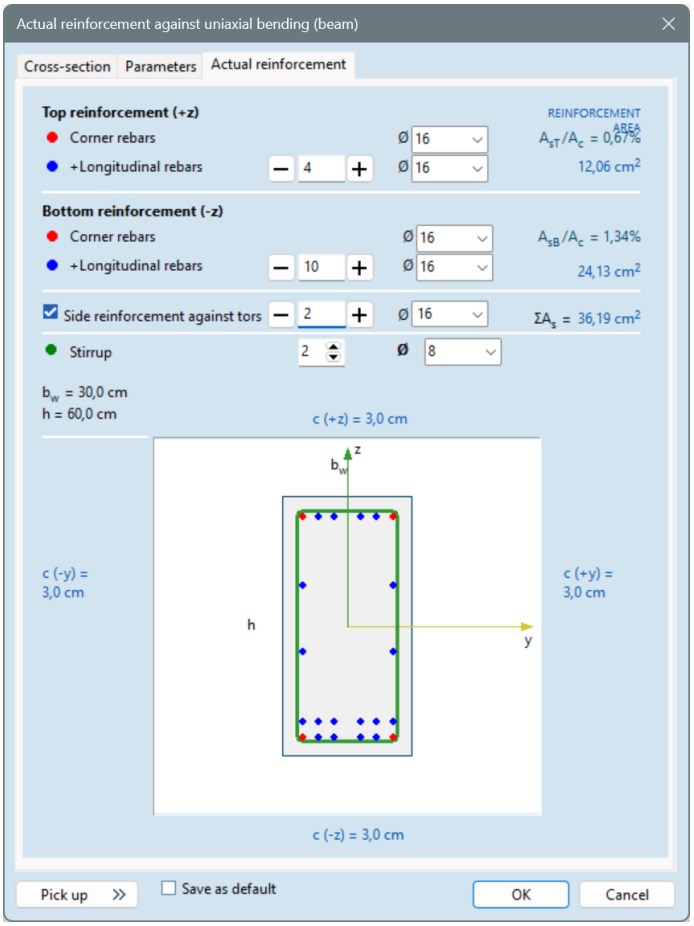
☞ **The normalised axial force is calculated for every concrete line element (including truss, beam and rib elements) considering the specified concrete material and cross-section by the definition of the line element. If actual beam reinforcement is assigned to the element, the concrete material and cross-section of actual reinforcement will be considered instead of original parameters.**

Fire design

Detailed information according to fire design of reinforced concrete beams can be found in a following section (see **6.5.14 Fire design of reinforced concrete elements – RC8 modules**).

Actual reinforcement against uniaxial bending (beam)

An actual reinforcement can be assigned to the selected finite elements. Nonlinear analysis can take into account this constant reinforcement when calculating deflection.
The necessary number of the corner rebars at the top and bottom of the beam are always there, their diameter can be changed. Number and diameter of additional top / bottom and side rebars can be specified. Rebars are distributed automatically.

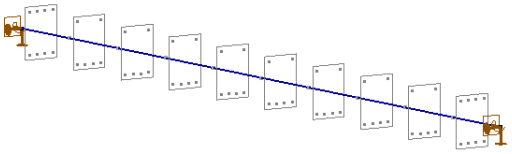


Pick up

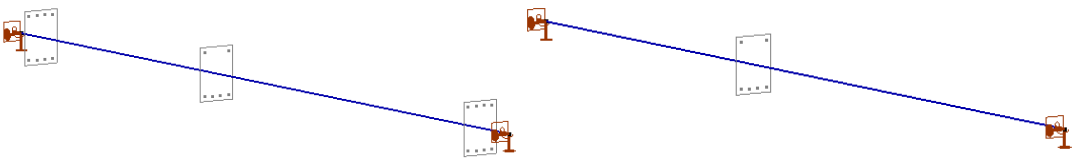
Clicking *Pick up* allows picking up beam reinforcement design parameters defined for another beam.

Display of actual reinforcement

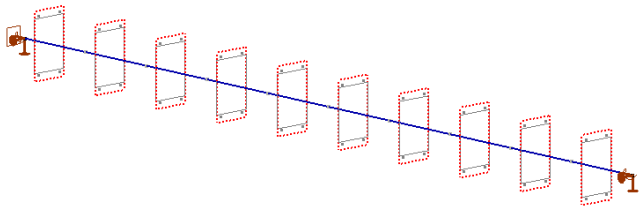
After actual reinforcement and its prameters have been assigned to the element and the display of cross-section shapes is turned on *(Display options / Symbols / Graphics symbols / Cross-section shape,* See **2.16.21 Display options***)* actual reinforcement can also be seen in the model view.



If a mesh is assigned to the line element (see **4.12.1.1 Meshing of line elements**), each finite element can have different actual reinforcement (Turn on *Settings / Preferences / Editing / Enable selection of finite elements on lines*). If line mesh display is disabled, three cross-sections are shown, one at the beginning, one at the midpoint and one at the end of the element. In case of constant actual reinforcement only one cross section is shown.



During the definition of beam reinforcement parameters, it is possible to change the cross section. If the cross section assigned to the line element is not identical to the cross section specified in the beam reinforcement parameters dialog, the modified cross section is displayed red dashed line in the model view.



Nonlinear analysis

The value of some parameters considered during nonlinear analysis (see **6.5.5 Nonlinear analysis of reinforced concrete beam and column elements**) can be set by the user:

- consideration of the tensile or flexural tensile strength of concrete;
- final concrete strain due to shrinkage;

6.5.13 Beam reinforcement design – RC2 module

The beam reinforcement design module can be applied to beam structural elements modeled by beam or rib finite elements, that have the same material and constant or variable rectangular or T cross sections, assuming that the load is applied in the symmetry plane of the cross section.
The computed longitudinal top and bottom reinforcement are of the same steel grade, while the stirrups could have steel grade different from the longitudinal ones.

Design codes

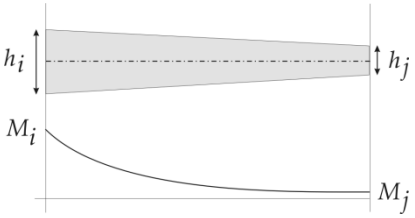
The design codes and national annexes in the software for which the design check of reinforced concrete beams can be performed are listed in **3.3.7 Design codes**.

☞ **The beams are structural elements with one dimension (the length) significantly greater than the dimensions of the cross-section, subjected to shear and to bending with or without axial force of large eccentricity. Axial force is considered in reinforcement design only if the *Consider axial force* option is selected (see above).**

Beam reinforcement parameters

Beam design (calculating the required amount of reinforcement, placing stirrups and rebars, and checking the reinforced beam) uses parameters entered in the Beam reinforcement parameters dialog (see **6.5.12 Beam reinforcement parameters**).

Variable cross-section



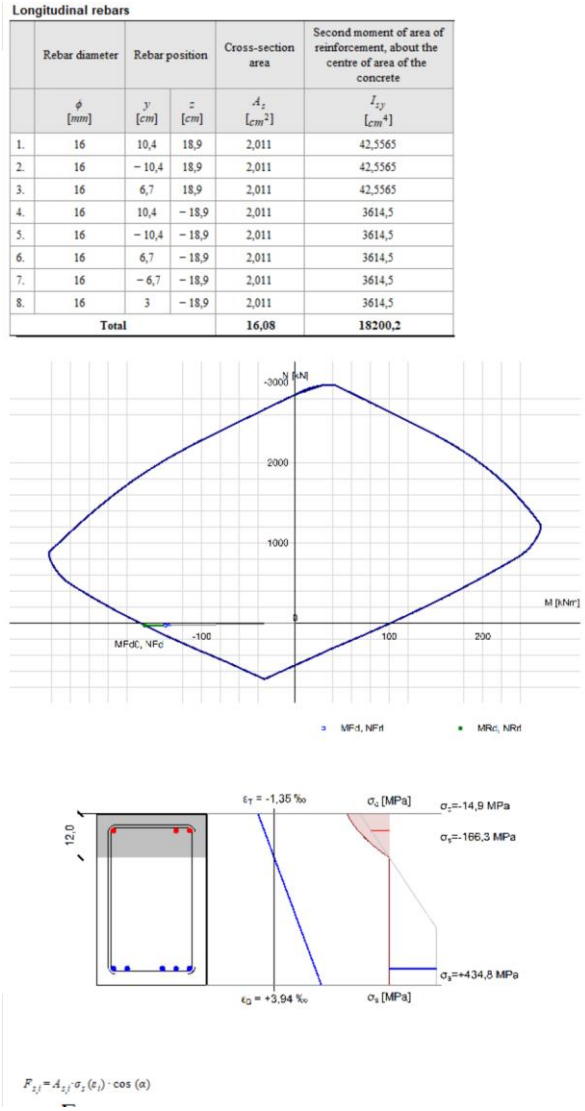
The change in shear force due to variable cross-section is taken into account.
Where sign of the moment does not change a simple rule can be applied: if section height changes the same way as the moment along the line shear capacity increases otherwise it decreases.
If $|h_i - h_j| \cdot |M_i - M_j|$ is positive then the shear force decreases.

The shear force is modified by the transverse components of the forces in the inclined tension and compression chords. These forces are determined from the cross-sectional equilibrium under the acting bending moment and axial force, taking the reinforcement into account. A factor of 0.8 is applied to a favourable correction and 1.2 to an unfavourable correction.

☞ **The program performs design calculations described below. Every other analysis, if prescribed by the design code, have to be done by the user.**
The present version of the module does not deal with out of plane bending, the effect of complex internal force states, lateral-torsional buckling or the effect of peak stresses perpendicular to the axis due to the action of concentrated forces and is not suitable for the reinforcement design of short cantilevers.

Design calculations

Click the *Design calculation* button on the *Beam reiforcement* dialog to display the details of calculations according to the current design code. References to sections and formulas of the design code appear in blue. If the *Calculated beam reinforcement* tab is active, the design calculations are based on the calculated rebar layout. If the *Actual beam reinforcement* tab is active, the short report is based on actual rebar layout.



Longitudinal rebars

	Rebar diameter	Rebar position		Cross-section area	Second moment of area of reinforcement, about the centre of area of the concrete
	ϕ [mm]	y [cm]	z [cm]	A_s [cm^2]	I_{zy} [cm^4]
1.	16	10,4	18,9	2,011	42,5565
2.	16	− 10,4	18,9	2,011	42,5565
3.	16	6,7	18,9	2,011	42,5565
4.	16	10,4	− 18,9	2,011	3614,5
5.	16	− 10,4	− 18,9	2,011	3614,5
6.	16	6,7	− 18,9	2,011	3614,5
7.	16	− 6,7	− 18,9	2,011	3614,5
8.	16	3	− 18,9	2,011	3614,5
Total				16,08	18200,2

Clicking on the *Settings* icon beside the *Design calculations* button allows to set the level of elaboration and the basic units for force and length used in the design calculations. Important results also appear converted to standard AXISVM units (*see* **3.3.8 Units and Formats**).
A short summary of design calculations is available, which can be printed on its own or together with the detailed calculations.

For further description about *Design calculations* window *see* **6.6 Steel design** and **6.6.1.3 Diagrams and design calculations**

6.5.13.1 Steps of beam reinforcement design



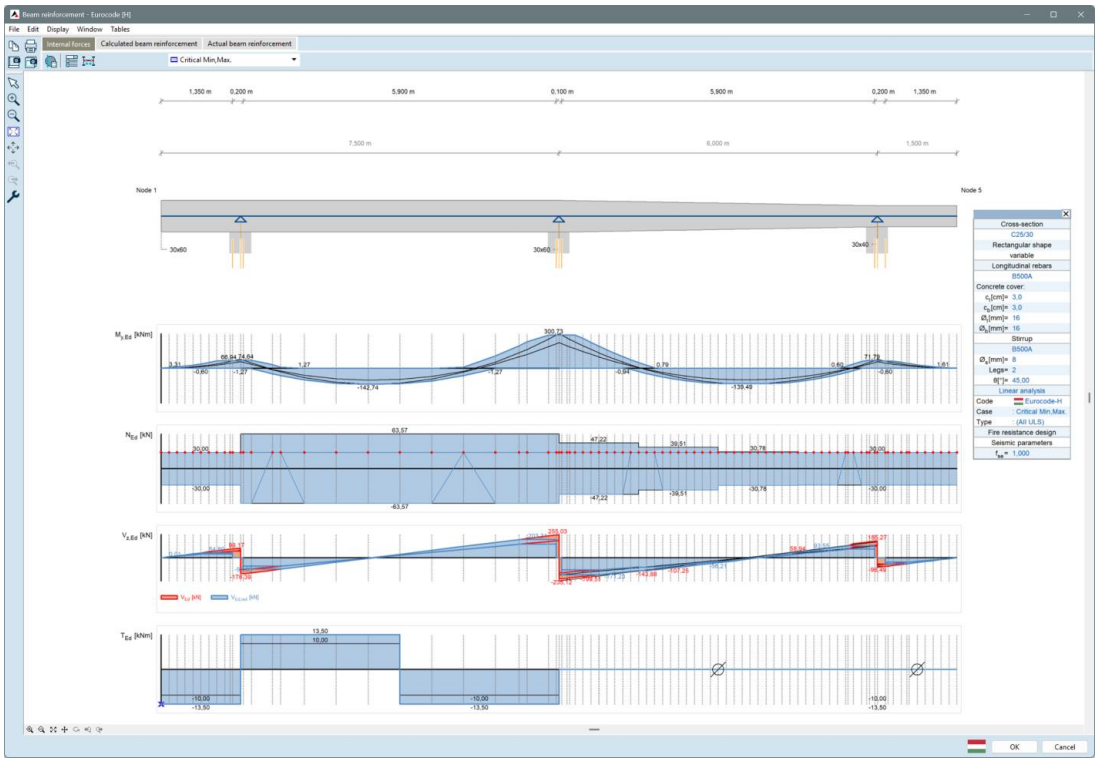
Beam reinforcement design

The design is performed in two steps:

1. Design of longitudinal reinforcement for moments about *y* or *z* axis *(My,* or *Mz)*, including axial force only if the *Consider axial force* option is selected.
2. Determination of spacing of vertical stirrups considering shear forces about y or z axis (*Vy* or *Vz*) and the twisting moment *(Tx)*.

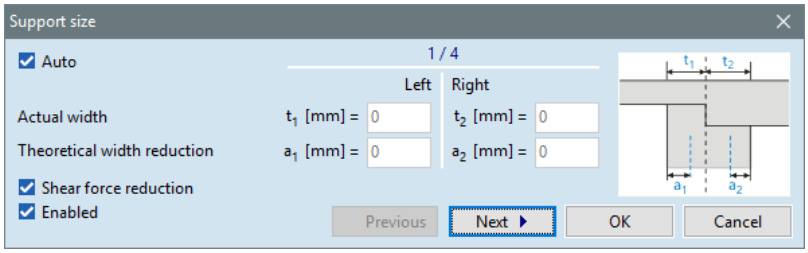
Axial force can be considered together with bending in beam reinforcement design. For members subjected to axial force with small eccentricity, use of the column design module is recommended.
The increase in the tension in the longitudinal rebars due to the shear cracks are taken into account by shifting the moment.

The following diagrams can be displayed on the *Design* tab: Design moment *(My_{Ed})*, calculated top/bottom reinforcement *(A_s)*, design shear force *(Vz)*, stirrup spacing *(s_w)*, design torsion moment *(Tx)*, torsion reinforcement on sides *(A_{SL})*.
The *Internal forces* tab displays the internal force diagrams for the selected load case, load combination or envelope.



Setting support size

AXISVM identifies all types of possible supports (columns, walls, beams, nodal supports) automatically but their width can be modified. Click on a support to get to the dialog below



Checking *Auto* sets the detected width values. Uncheck it to set the *Actual width* and the *Theoretical width reduction* manually. It lets you specify the a_1 and a_2 segments on the side of the support that will be ignored in the calculations. The internal forces are linearly interpolated within the segments.

Checking *Shear force reduction* activates a method described in the design code to reduce shear force above the supports. Uncheck *Enabled* to ignore the support in the design calculation. The shear force obtained from capacity design is not reduced by this option.

Beam parameters

See **6.5.12 Beam reinforcement parameters (uniaxial bending)**

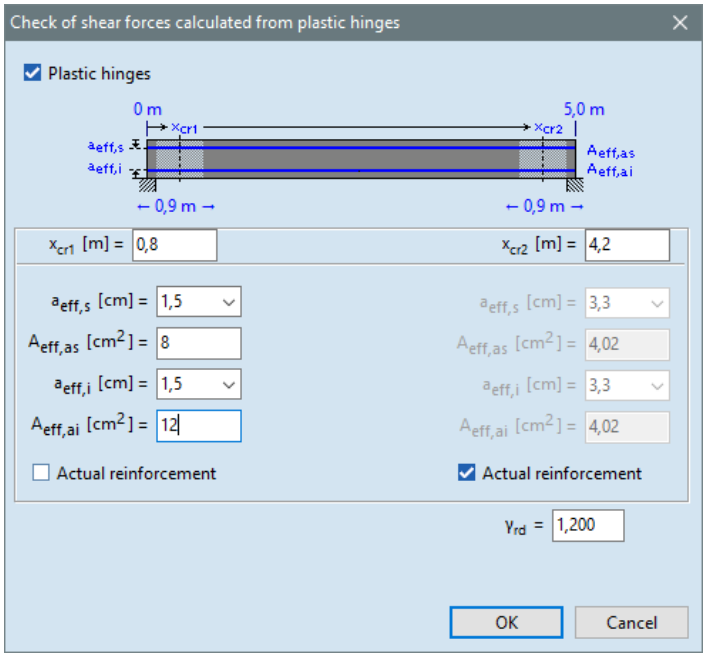
Display results

The display of diagrams and labels on each tab can be customized.

Plastic hinges

In case of dissipative structures with DCM or DCH ductility class (EN 1998-1-1), in order to avoid shear failure of beams, the design values of shear forces are determined in accordance with the capacity design rule (if seismic load case is included in the selected load combination), on the basis of the equilibrium of the beam under: a) the transverse load acting on it in the seismic design situation and b) end moments

$M_{i,d}$ (with i=1,2 denoting the end sections of the beam), corresponding to plastic hinge formation for positive and negative directions of seismic loading.
The following parameters can be given by the user: the safety factor, the distance between plastic hinges and the reinforcement of plastic hinges. If *Actual reinforcement* checkbox is checked, the reinforcement parameters related to plastic hinges are calculated automatically based on the defined actual beam reinforcement (*see* **6.5.13.3 Checking actual beam reinforcement**). The length below the plastic hinges in the window is the length of critical region where the program considers stricter requirements regarding detailing rules of shear reinforcement in order achieve the expected ductility.



Considering capacity design rules, the design shear force is calculated on the following way (if seismic load case is included in the selected combination):

$$V_{Ed} = V_{Ed,0} + V_{Ed,EQ}$$

where:

- $V_{Ed,0}$ denotes the shear force from non-seismic actions (from dead loads, etc.),
- $V_{Ed,EQ}$ is the design value of seismic shear force.

$$V_{Ed,EQ} = \gamma_{Rd} \frac{M_{1,d} + M_{2,d}}{l}$$

where:

- l is the distance between plastic hinges, $l = x_{cr2} - x_{cr1}$

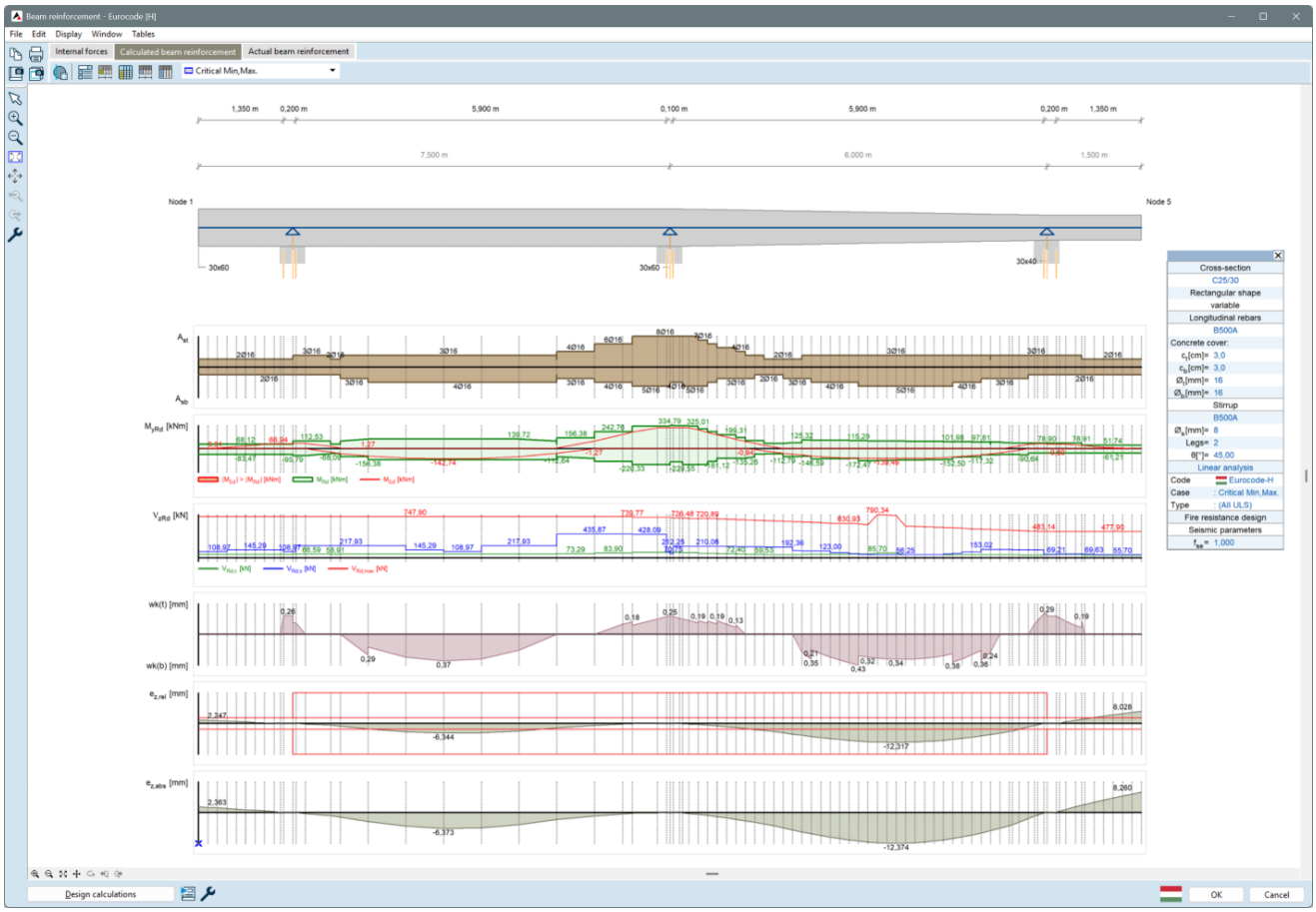
The seismic shear force is considered with both positive and negative sign according to the positive and negative direction of seismic loads.

☞ **If seismic shear force from seismic analysis is greater than the shear force calculated based on capacity design rules, seismic shear force from seismic analysis is used as $V_{Ed,EQ}$.**

☞ **The program considers detailing rules regarding max. distance of stirrups by the calculation of stirrup arrangement. Checking the fulfilment of further standard specific detailing rules of beams with plastic hinges is the responsibility of the user.**

6.5.13.2 Checking calculated beam reinforcement

After clicking on the *Calculated beam reinforcement* tab, the following diagrams can be displayed: Envelope of reinforcement (A_s), side reinforcement against torsion (A_{SL}), cracking (w_k), stirrup spacing (s_w), moment resistance (My_{Rd}), maximum shear force (Vz_{Rd}), torsion resistance (Tz_{Rd}), bending moment utilization (My_{Ed}/My_{Rd}), shear utilization (Vz_{Ed}/Vz_{Rd}), torsion utilization (Tz_{Ed}/Tz_{Rd}), relative deflection ($e_{z,rel}$), absolute deflection ($e_{z,abs}$).



Ultimate Limit State (ULS)

Bending and axial force, including the effect of torsion

The bending moment resistance M_{Rd} is determined from the calculated reinforcement, considering the acting axial force if the Consider axial force option is selected. For members subjected to axial force with small eccentricity, use of the column design module is recommended. Bending utilization is also displayed as a diagram.

Shear and torsion

The program calculates the shear and torsion resistance and utilization ratios:
- without shear/torsion reinforcement ($V_{Rd,c}$, $T_{Rd,c}$, $\eta_{Rd,c}$), (only in Eurocode)
- the maximum shear resistance / torsion moment limited by the concrete compression strut $V_{Rd,max}$, $T_{Rd,max}$, $\eta_{Rd,max}$).

Service Limit State (SLS)

Cracking analysis

After clicking on the *Calculated beam reinforcement* tab AXISVM determines the top and bottom rebar scheme from the top and bottom reinforcement amounts calculated from the selected load case or combination and according to the minimum rebar distance specified in the code.
If the required rebars do not fit in one row multiple rows are introduced. A row consists of two rebars minimum.
In case of T shapes rebars will be placed within the flange only if rebars do not fit into the web. Flange rebars can form only one row and their number cannot exceed the half of the total number of rebars.
If envelope is selected cracking is calculated from all SLS combinations included in the envelope. If no SLS combinations are included in the envelope all ULS combinations are used.
Cross-section properties and cracking is calculated with the rebars distributed according to the above scheme. If an SLS combination is chosen the cracking is calculated with the rebar scheme determined from the required reinforcement of the SLS combination. This cracking value can be higher than cracking calculated with rebars from the critical of envelope results.

Cracking

If *Parameters / Increase reinforcement according to limiting crack width* was checked when defining beam reinforcement parameters AXISVM will increase the number of rebars on the tension side until the calculated crack width falls below the limit, provided the total area of reinforcement does not exceed limit value (4-9% of the concrete cross-section) prescribed by the design code ($A_s \leq A_{s,max}$)

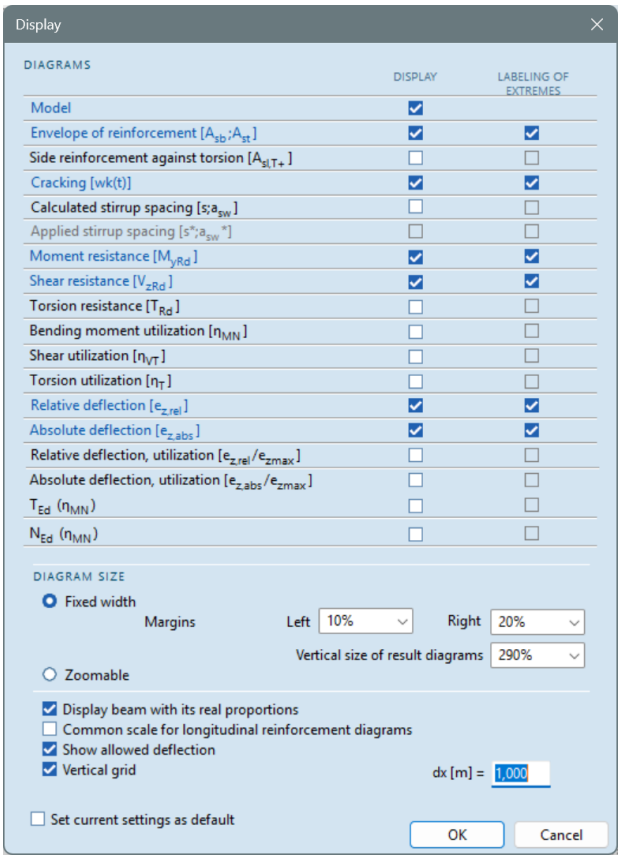
Deflection

The deflection is calculated using an approximation remaining on the safe side.
The program calculates the ζ distribution factors at the moment field maximum locations and at the theoretical support edges and assumes that this factor is constant 1) between the support edges and the zero moment point and 2) between zero moment points in the field.
The absolute deflection determined by the linear analysis is corrected using the support displacement values.
The approximated deflection at a certain point of the beam is $e = e_I \cdot (1 - \zeta) + e_{II} \cdot \zeta$, where
ζ is the distribution factor, $\zeta = 1 - \frac{1}{2} \cdot \left(\frac{M_{cr}}{M}\right)^2$.
e_I is the approximated deflection of the non-cracked reinforced beam: $e_I = e_0 \cdot I_I / I_b$
e_{II} is the approximated deflection of the cracked reinforced beam: $e_{II} = e_0 \cdot I_{II} / I_b$
e_0 is the corrected deflection taking into account the effective modulus of elasticity of the concrete and the support displacements $e_0 = e_{lin} \cdot E_{cm} / E_{c,eff}$

Result display



Click on this icon to set diagrams and diagram labels to display.



Check table (ULS)



Two rows of data are displayed for each section, one for the top reinforcement and one for the bottom. Each row contains ULS internal forces and values of different intermediate result components.
A shorter summary is also available displaying values only at certain important sections.

Check table (SLS)



Similar tables are generated for SLS internal forces, crack width and deflection values.
Top cracking is calculated from the (max) moment causing tension within the upper part. Bottom cracking is calculated from the (min) moment causing tension within the lower part. If no tension appears on a side (max is negative or min is positive), calculations are performed with zero moment. In this case the table shows zero and shows the actual moment in brackets.

☞ **Internal forces that appear in these two tables are different only if the reinforcement was calculated for an envelope or critical combination. If a load case or an individual load combination was selected the internal forces will be the same.**

6.5.13.3 Checking actual beam reinforcement

Changing actual beam reinforcement

On the *Actual beam reinforcement* tab, it is possible to change the actual longitudinal reinforcement or stirrup spacing assigned to finite elements.
After changing the reinforcement or stirrup spacing, the results are recalculated automatically, if *Display / Recalculate automatically* is checked. If this feature slows down the definition (e.g., the beam has to be checked for a huge number of critical combinations) deactivate this option. In this case click on the *Calculate* button at the bottom to recalculate the results.



Select all (Ctrl+A)
Selects all finite elements of the beam.



Apply calculated reinforcement / stirrup spacing
Sets the actual reinforcement or stirrup spacing according to the values calculated for the current load case or combination over the entire beam.

Click on one of the two exclusive buttons to select the definition mode

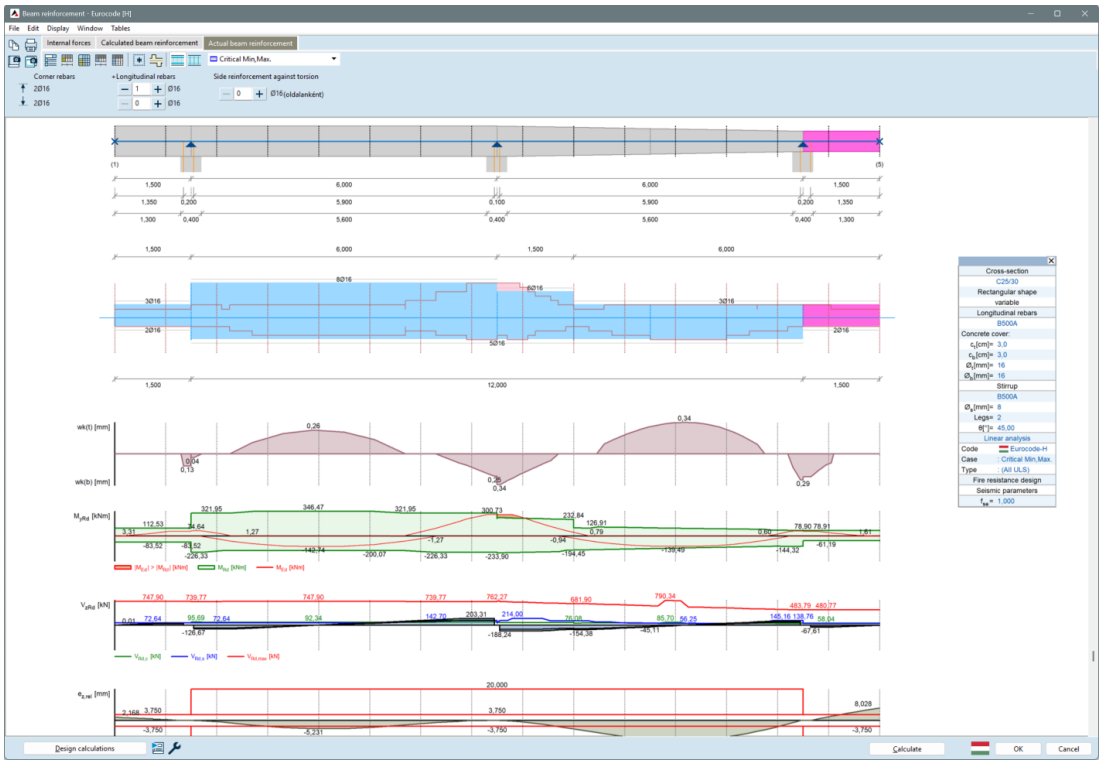
 *Longitudinal reinforcement*  *Stirrup spacing*



Longitudinal reinforcement
If this button is down the longitudinal reinforcement can be defined.
Select finite elements or remove the selection by clicking or dragging the mouse. To add non-adjacent elements to the selection press Shift during the operation. Diameter of corner rebars and other longitudinal rebars are set among the beam reinforcement parameters. Edit boxes allow changing the number of top, bottom and side rebars in the selected elements. The – and + buttons decrease/increase these values.
The diagram of the actual reinforcement is filled with sky blue, the A_s reinforcement calculated for the current load case, combination or envelope can be seen behind it as a red diagram filled with pink.

Checking the actual reinforcement

The program displays the selected results the same way as on the previous tab but for the actual reinforcement. The tables are the same but display results for the actual reinforcement.



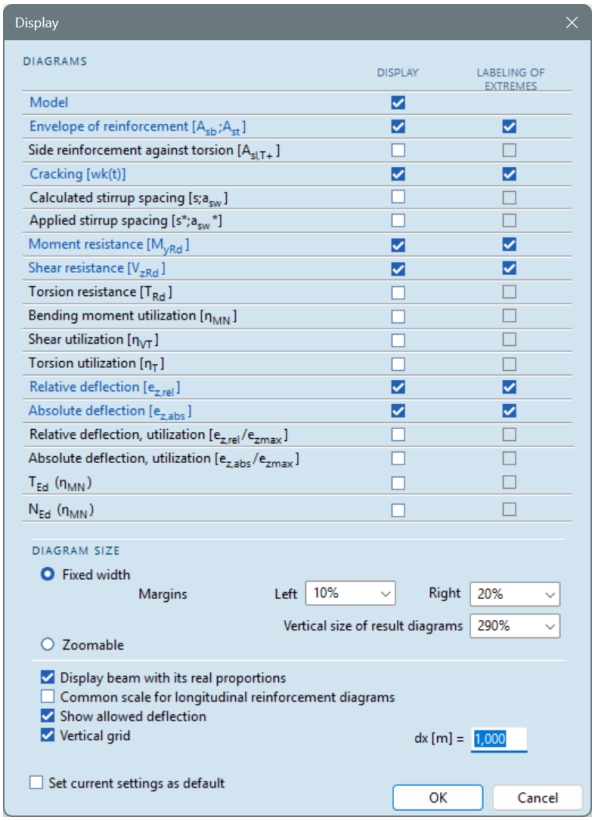
Result display



Click on this icon to set diagrams and diagram labels to display.
Special controls for the *Actual beam reinforcement* tab:

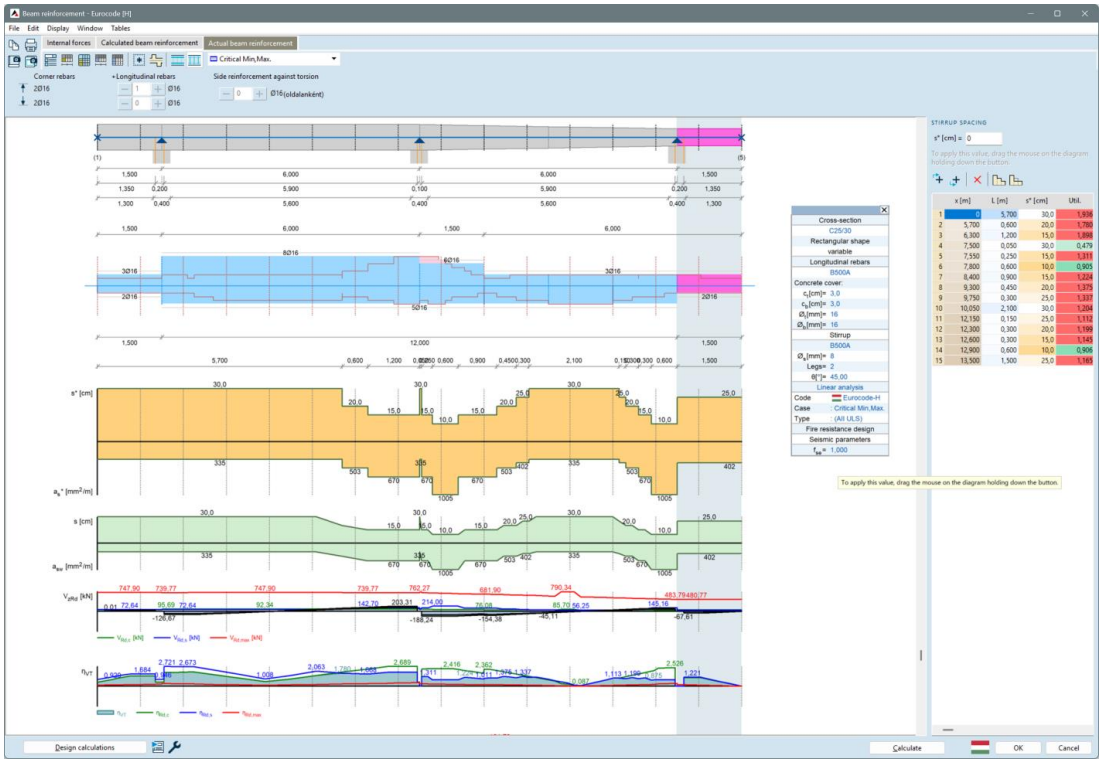
Margins. Set the left and right margin of the diagrams as a percentage of the total width. This way you can leave space for the info window.

Vertical size of result diagrams. Sets the vertical size of diagrams between 100% and 300%. This value can also be changed in the diagram window by rotating the mouse wheel.





Stirrup spacing
If this button is down the actual stirrup spacing can be set.
A diagram of applied stirrup spacing *(s*)* and the specific area of reinforcement *(a_s*)* appears at the top. A grid appears on the right displaying the stirrup spacing as a step function constant in continuous sections along the beam. The *x* coordinate of the start point of section, the L length of the section, and the s* spacing over that section can be edited. If the results have been calculated the utilization of each section is also displayed (if this value is greater than 1 the background is red otherwise it is green).



Editing stirrup spacing

After clocking on the *Apply calculated reinforcement / stirrup spacing* button the table shows the calculated values. Use the buttons below to edit the table describing the step function



Insert a new row before the current one
Creates a new section by dividing the row before the current one into two equal parts. *s** remains the same but can be edited separately.



Insert a new row after the current one
Creates a new section by dividing the current section into two equal parts. *s** remains the same but can be edited separately.



Delete selected rows
After deleting the selected rows, the previous section will be extended.



Apply calculated stirrup spacing
Imports the calculated spacing.



Apply calculated minimum/maximum stirrup spacing
Imports the calculated spacing but reduces the number of different spacings to two, the minimum and the maximum.

Editing cells

The *x* start-point of a section can be changed to any value between the previous and the next *x* values. Changing x will also adjust the length of the current and the previous section accordingly. *x* values outside this range will be rejected.

The L length of a section can be changed but the end point of the section must fall to the left of the end of the next section. The length of the next section will be adjusted accordingly.

6.5.13.4 Beam reinforcement according to Eurocode

EN 1992-1-1

Symbols, material properties, partial factors

f_{cd}	design value of the compressive strength of the concrete
f_{ctd}	design value of the tensile strength of concrete
α_{cc}	= 0.80-1.00 (prescribed by national annexes); is the coefficient taking account of long term effects on the compressive strength and of unfavourable effects resulting from the way the load is applied.
α_{ct}	is a coefficient taking account of long-term effects on the tensile strength and of unfavourable effects, resulting from the way the load is applied.
γ_c	is the partial safety factor for concrete (2.4.2.4(1))
f_{yd}	design value of flow limit of rebar steel
ε_{su}	limiting strain of rebar steel
E_s	(=200 kN/mm^2); Young modulus of rebar steel
γ_s	= 1.15; is the partial factor for reinforcing (2.4.2.4(1))

Shear & torsion reinforcement design of stirrups

In reinforcement design (*Calculated beam reinforcement* tab), the maximum stirrup spacing permitted by the detailing rules is rounded down to a multiple of the specified spacing increment. The external legs of the closed stirrups are then checked for the combined effects of shear and torsion using this spacing. If the check is not satisfied, the spacing is reduced in the specified increments until the check is satisfied or the minimum permitted spacing is reached.
When the check is performed on the *Actual beam reinforcement* tab, the actual stirrup spacings are used without modification.
The design is based on the following values of design shear resistance:

$V_{Rd,c}$	Design shear resistance of the cross-section without shear reinforcement.
$V_{Rd,max}$	Maximum shear force that can be transmitted without the failure of the inclined compression bars.
$V_{Rd,s}$	Design shear resistance of the cross-section with shear reinforcement.
$T_{Rd,c}$	Design torsional resistance of the cross-section without shear reinforcement.
$T_{Rd,max}$	Maximum torsional moment that can be transmitted without the failure of the inclined compression bars.

AXISVM calculates the shear and torsion reinforcement using the specified inclination angle of the concrete compression struts, subject to the restrictions of the selected design code
The relation between the capacity of inclined compression concrete bars and the design values is checked.

$$\frac{V_{Ed}}{V_{Rd,max}} + \frac{T_{Ed}}{T_{Rd,max}} \leq 1,$$

where

$$V_{Rd,max} = \frac{\alpha_{cw} b_w z v_1 f_{cd}}{\cot\theta + \tan\theta}$$

and

$$T_{Rd,max} = 2\nu\alpha_{cw} f_{ctd} t_{efi} A_k \sin\theta \cos\theta$$

If the cross-section does not fail it is checked if shear & torsion reinforcement is required according to the formula

$$\frac{V_{Ed}}{V_{Rd,c}} + \frac{T_{Ed}}{T_{Rd,c}} \leq 1,$$

where $V_{Rd,c} = \left[C_{Rd,c} k (100\varrho_l f_{ck})^{1/3} + k_1 \sigma_{cp}\right] b_w d$ and $T_{Rd,c} = 2 f_{ctd} t_{efi} A_k$

If shear & torsion reinforcement is required,

$$\frac{\sum A_{sl} f_{yd}}{u_k} = \frac{T_{Ed}}{2A_k} \cot\theta$$

therefore

$$A_{sl} = \frac{T_{Ed} u_k}{2A_k f_{yd} \tan\theta}$$

Stirrup spacing is determined by checking the external legs of closed stirrups for the combined effects of shear and torsion, taking the number of stirrup legs into account. The selected spacing must also satisfy the detailing rules of the design code.

Using the variable angle truss method, significant saving of shear reinforcement can be achieved if the compressed concrete beams have extra resistance, i.e:

$$\frac{V_{Ed}}{V_{Rd,max}} + \frac{T_{Ed}}{T_{Rd,max}} \ll 1$$

By changing the shear crack inclination angle the compressed concrete beams gets more load while shear reinforcement gets less. The actual saving depends on the design rules.
The inclination angle of the concrete compression struts affects both the required shear reinforcement and the resistance of the concrete struts. The calculation must satisfy the shear–torsion interaction checks and the angle limits specified by the selected design code. $\frac{V_{Ed}}{V_{Rd,max}} + \frac{T_{Ed}}{T_{Rd,max}} \leq 1$

☞ **The cross-section fails if critical shear force is higher than the shear resistance of the compressed concrete beams, i.e.:**

$$\frac{V_{Ed}}{V_{Rd,max}} + \frac{T_{Ed}}{T_{Rd,max}} > 1$$

Design rules applied in calculation:
On the basis of equation 9.2.2 (9.5N) $\rho_{w;min} = 0.08\sqrt{f_{ck}}/f_{yk}$ and of equation 9.2.2 (9.4) $\rho_w = A_{sw}/sb_w$, so the ratio of shear reinforcement is $s_{max,1} = A_{sw}/\rho_{wmin} b_w$ 9.2.2 (9.6N) states that: $s_{max,2} = 0.75\, d$.

Longitudinal beam reinforcement

Longitudinal reinforcement is designed using the moment–axial force resistance curves of the reinforced cross-section. These curves account for strain compatibility and the material stress–strain relationships. Yielding of all tension reinforcement is not assumed. The acting axial force is included if the Axial force option is selected.

The required top and bottom reinforcement along the beam and the moment diagram shift is calculated for each load case.

Due to inclined cracks tension reinforcement is designed for a force greater than calculated from *M/z*. This is taken into account by different design codes by shifting the moment diagram.

Cracking is calculated according to **6.5.7.1 Cracking calculation according to Eurocode 2**

Construction rules considered in the program

The detailing rules are checked according to the design codes and national annexes listed in **3.3.7 Design codes**.

6.5.13.5 Beam reinforcement according to SIA 262

SIA 262

Symbols, material properties, partial factors

f_{cd}	design value of the compressive strength of the concrete
f_{ct}	design value of the tensile strength of the concrete
γ_c	= 1.5; partial factor of the concrete
f_{yd}	design value of flow limit of rebar steel
ε_{su}	limiting strain of rebar steel
E_s	(= 200 kN/mm^2); Young modulus of rebar steel
γ_s	= 1.15; partial factor of the steel
k_c	= reduction factor for compressive strength of the concrete in a cracked zone

Shear & torsion reinforcement design of stirrups

Stirrup spacing is determined using the procedure described in the Eurocode section.
The program checks the concrete compression struts and the stirrups using the following relationships based on SIA 262:

Design value of the resistance of the concrete compression strut: $V_{Rd,c} = b_w z k_c f_{cd} \sin\alpha \cos\alpha$

Design value of the torsional resistance of the concrete compression struts:

$$T_{Rd,c} = 2A_k t_k k_c f_{cd} \sin\alpha \cos\alpha$$

The shear/torsion utilization ratio of the concrete compression struts: $\eta_{Rdc} = \frac{|V_d|}{V_{Rd,c}} + \frac{|T_{Ed}|}{T_{Rd,c}}$

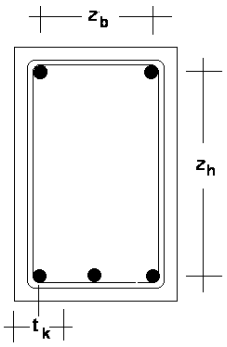
The resistance of shear reinforcement: $V_{Rd,s} = \frac{A_{sw}}{s} z f_{ywd} \cot\alpha$

Equivalent panel shear force due to torsion: $V_{d,i} = \frac{T_d}{2A_k} z_i$

Longitudinal reinforcement from torsion:

$$A_{slT} = \frac{\sum V_{d,i} \cot\alpha}{f_{yd}} = \frac{\frac{T_d}{z_h z_b}(z_h + z_b)\cot\alpha}{f_{yd}}$$

which should be placed evenly along the cross-section contour.



Longitudinal beam reinforcement based on SIA 262

Longitudinal reinforcement is designed using the method described in the Eurocode section, with material parameters according to SIA 262.

Construction rules considered in the program

The detailing rules are checked according to the design codes and national annexes listed in **3.3.7 Design codes**.

☞ **AXISVM displays warning messages in the following cases:**

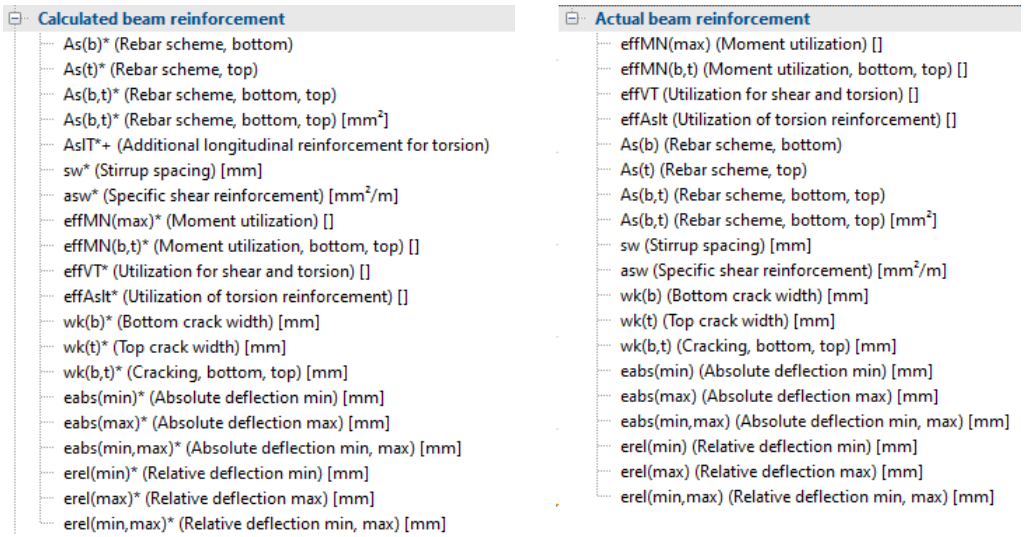
Message **The cross section is not acceptable for shear/torsion**

Event The utilization of concrete compression struts is greater than 1.

Solution Increase the cross section of the concrete, or/and the concrete grade.

6.5.13.6 Result components of beam reinforcement

The following global result components based on the beam design calculation results are displayed:



6.5.14 Fire design of reinforced concrete elements – RC8 modules

The fire design of reinforced concrete columns and beams can be performed in the software according to the guidelines of Eurocode 2 and SIA 262 design codes with the RC8-B (for columns and beams) and RC8-S (for membranes, plates and shells) modules. The fire design is based on the conventional design, **so RC2 module is a prerequisite to use RC8-B module and RC1 module is a prerequisite to use RC8-S module**. In this description only those parts, rules and procedures are described that are different or not used in the course of conventional design.
Fire design of reinforced concrete beams and columns can be performed in AXISVM if the selected load combination includes a fire load case (**4.10.1 Load cases, load groups**) and a fire effect has been assigned to any of the selected elements (**4.10.34 Fire effect on concrete line elements – RC8-B module, 4.10.35 Fire effect on concrete domains – RC8-S module**) in that fire load case. If different fire effects have been assigned to the selected elements, an error message appears and fire design cannot be performed.
Fire design has to be activated in the design parameters dialog (see below).

Design codes

The calculation can be performed according to Eurocode 2 and SIA 262. For a list of the supported Eurocode national annexes, see the previous chapters on the fire load definition (*see* **4.10.34 Fire effect on concrete line elements – RC8-B module, 4.10.35 Fire effect on concrete domains – RC8-S module**).

Member analysis

EN 1992-1-2 discusses different methods: member analysis, analysis of a part of the structure and analysis of the entire structure. Member analysis is the most widespread verification method due to its simplicity. Analysis of a part or whole of the structure usually requires complex nonlinear numerical models and nonlinear statical analysis.

Guiding principles of member analysis according to EN1992-1-2:

- Design member internal forces can be calculated with linear static analysis;
- „Only the effects of thermal deformations resulting from thermal gradients across the cross-section need be considered. The effects of axial or in-plane thermal expansions may be neglected." (EN1992-1-2, 2.4.2 (4));
- „The boundary conditions at supports and ends of member, applicable at time t = 0, are assumed to remain unchanged throughout the fire exposure." (EN1992-1-2, 2.4.2 (5)).

RC8-B performs member analysis. It has to be checked if the current structure/structural part can be verified with member analysis according to the guidelines of the design code.
In certain cases, analysis of a part of the structure can be also executed in AXISVM if

- boundary conditions are set carefully,
- fire effects are taken into account as thermal load and
- we consider the guidelines in section 2.4.3 of EN1992-1-2.

Basis of fire design

The basis of fire design in AXISVM is the simplified calculation method from EN 1992-1-2. The prescriptive methods combined with member analysis allow fire design of structural elements exposed to standard or parametric fire.
The simplified calculation uses the so-called 500°C isotherm method, whereby concrete sections with temperatures above 500°C are considered as damaged zones and are ignored in the calculation of the cross-sectional resistance. The concrete core below 500°C is considered with its strength and stiffness at normal temperature. The stiffness and strength of reinforcing steel are reduced based on their temperature.

6.5.14.1 Fire design of reinforced concrete columns and beams – RC8-B module

Possible error and warning messages

Fire design of reinforced concrete beams and columns can be performed in AXISVM if

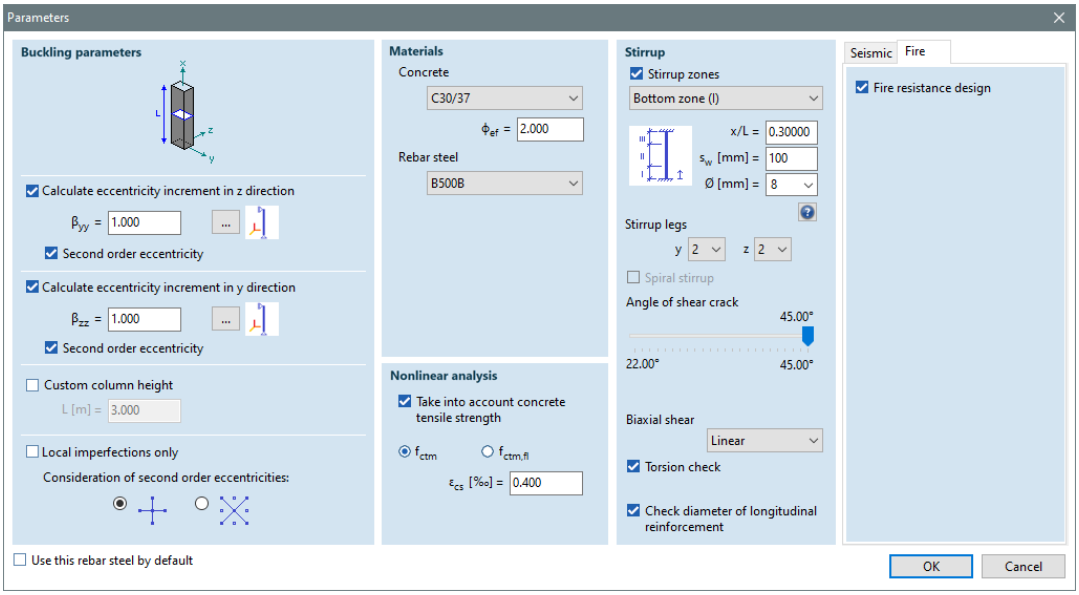
1) the selected load combination includes a fire load case and
2) a fire effect has been assigned to any of the selected elements in that fire load case.

Furthermore, if different fire effects have been assigned to the elements, fire design cannot be performed. In any of these criteria is not met, an error message appears.
In the case of reinforced concrete columns, the fire design can be performed for the original section of the member. It also leads to an error if the selected combinations (envelope or critical) come from different design situations (e.g., accidental, persistent, seismic) because their partial factors are different and N-M interaction surface cannot be properly generated.
In the case of reinforced concrete beams, there are additional cases when fire design cannot be performed as follows:

- inverted T-beams: fire load cannot be assigned to this type of cross-sections,
- T-beams: if not a T shaped exposition was selected among fire load parameters,
- T-beams and inverted T-beams: if design for Vy-Mz internal forces was selected in *Beam reinforcement parameters* window.

If the fire design cannot be performed, the software deletes accidental combinations with fire load from the list and it performs the design of the column/beam for the remaining conventional combinations in the envelope or in the critical set.
Like in case of steel and timber elements, fire design needs to be enabled in the design parameters dialog. The software sends a warning if fire design is not enabled, although fire load has been assigned to the element.
There are NAs that do not allow the use of Annex D of EN 1992-1-2. Shear and torsional design check is not available for these National Annexes because shear and torsion design is based on recommendations and guidelines of this Annex. The software displays a warning message if that is the case. It is important to note that it does not mean that the effect of shear and torsion is negligible.

Reinforced concrete columns



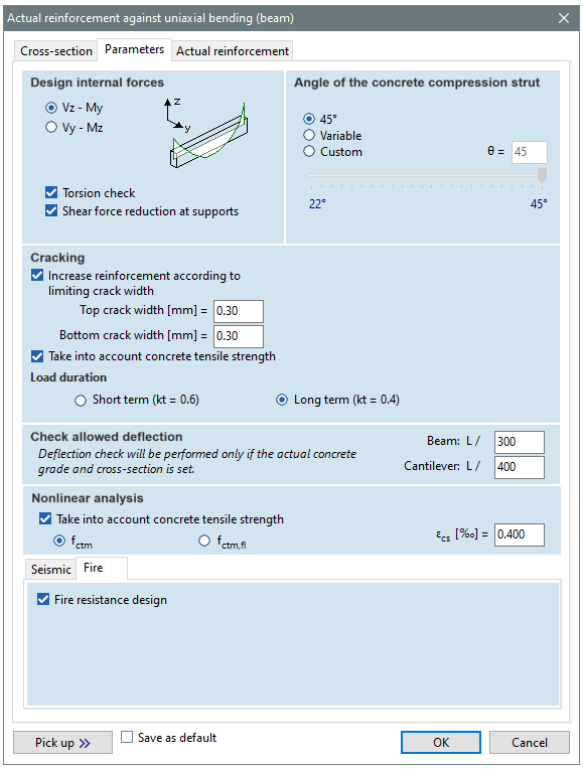
Verification against bending with or without axial force

- The recommendations and guidelines of Annex B of EN 1992-1-2 are taken into consideration by the calculation of N_x-M_y-M_z interaction surface.
- The accidental eccentricities are calculated as described in EN 1992-1-1 and SIA 262, however, with reduced sectional resistance and stiffnes due the damaged outer concrete layers and the change in the properties of concrete and steel on elevated temperature.

Shear and torsion check

- Shear and torsional check on elevated temperature is based on Annex D of EN 1992-1-2
- The elevated temperature of the stirrups is taken into consideration.
- Shear and torsional resistance without shear reinforcement are neglected in the calculation (EN 1992-1-1: $V_{Rd,c}$ = 0 and $T_{Rd,c}$ = 0; SIA262: V_{Rd} = 0 and T_{Rd} = 0).

Reinforced concrete beams



Verification against bending

- The recommendations and guidelines of Annex B of EN 1992-1-2 are taken into consideration during the verification or design of longitudinal reinforcement.
- The design is based on equilibrium equations presented in previous sections. These equations are formulated with reduced cross-sectional area. Damaged concrete with temperature above 500°C are neglected in the calculation. The modified strength of each rebar is taken into account based on the temperature of the reinforcement.

Shear and torsion check

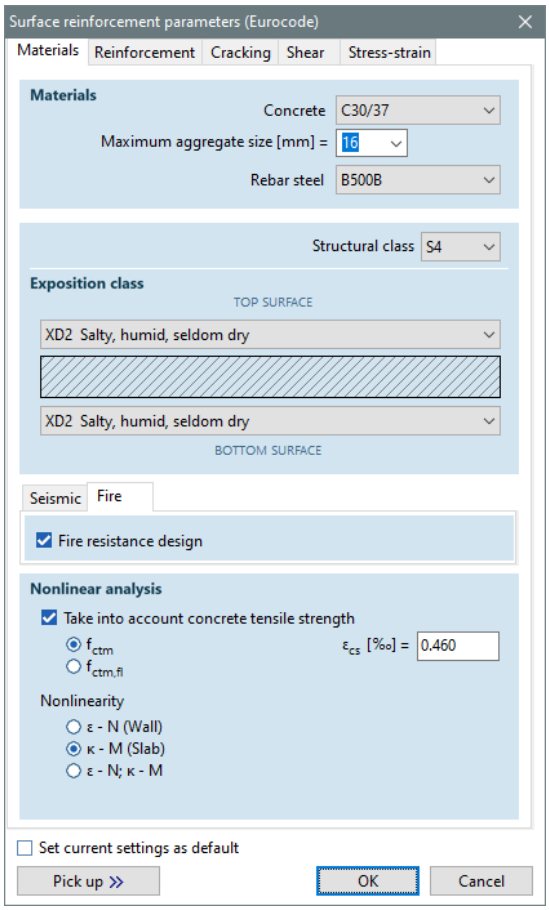
- Shear and torsional check on elevated temperature is based on Annex D of EN 1992-1-2.
- The elevated temperature of the stirrups is taken into consideration.
- Shear and torsional resistance without shear reinforcement are neglected in the calculation (EN 1992-1-1: $V_{Rd,c}$ = 0 and $T_{Rd,c}$ = 0; SIA262: V_{Rd} = 0 and T_{Rd} = 0).

6.5.14.2 Fire design of reinforced concrete surfaces – RC8-S module

Possible error and warning messages

The fire design has to be activated among design parameters similarly to columns and beams. The software sends warning message if the fire design is not activated or multiple fire load cases are included in the combination and multiple fire loads have been assigned to the element.

Surface reinforcement parameters



The required reinforcement can be calculated for reinforced concrete surfaces with RC8-S module in fire design situation. The reduced cross-section due to spalling and modified strength of each rebar based on the steel temperature are considered in the analysis. Simplified methods are implemented in the module according to Annex B of EN 1992-1-2. In stress-strain analysis (**6.5.18 Stress-strain analysis of reinforced concrete structures – RC6 module**) and in the verification of actual reinforcement (**6.5.3 Verification of actual reinforcement of RC surfaces**) the software considers the fire design rules together with the degradation of stiffness and strength of steel reinforcement and spalling of damaged concrete similarly to the calculation required reinforcement.
In the following cases, the software does not take into account the fire design rules in the calculations:

- The software does not calculate crack width considering fire conditions and it does not consider fire design rules by the calculation of shear resistance and shear reinforcement
- Nonlinear static analysis of reinforced concrete surfaces does not take into account the effect of fire
- The punching analysis and the design of reinforced cores and walls are always performed following the rules of normal temperature design and they do not consider the fire design rules and the effect of fire

6.5.15 Punching analysis – RC3 module

Standards

The design codes and national annexes in the software for which punching analysis can be performed are listed in **3.3.7 Design codes**.



The following design checks are included:

- column and reinforced concrete slab (including base slabs)
- wall end/corner and reinforced concrete slab (including base slabs)

As a special case, the program can analyze the situation when one or two beams are connected to the column. It is recommended to use the wall end/corner analysis, as that method can take into account the rib contour lines when finding the control perimeter. Neglecting the ribs and performing a simple column punching analysis is unsatisfactory and can lead to underdesign. RC ribs must be checked for shear separately.

If a wall and/or a beam is connected to a column (or multiple walls or beams meet forming a corner), and these elements can be taken into account, it is recommended to use the wall end/corner analysis.
See **6.5.15.1.2 Punching analysis of wall ends and corners according to Eurocode** or **6.5.15.2.2 Punching analysis of wall ends and wall corners according to SIA 262**
To analyse punching of a wall end or corner, select the node. To analyse punching of columns, select the column. If no additional elements are selected, all connecting domains will be analysed. The range of analysis can be narrowed down by selecting additional elements (domains, walls or ribs). Only the selected elements will be taken into account.

The check of resistance of compression struts and the calculation of required shear reinforcement are incorporated.
This feature is available for the following domain types (see **4.9.6 Domain**):

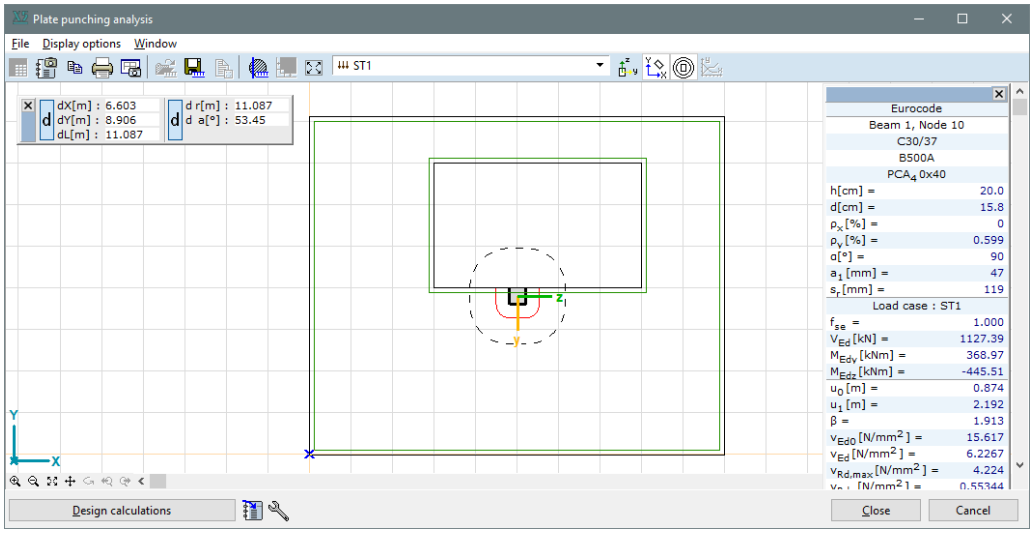
- normal domains
- composite ribbed domains
- parametric ribbed domains
- domains with custom stiffness matrix
- AIRDECK slabs
- composite slabs with trapezoidal sheeting (if the Trapezoidal plate used only as formwork option is checked)

☞ **The punching analysis does not take into account fire design rules, elevated temperatures in the cross-section, spalling of concrete and degradation in stiffness and strength.**

Openings and slab contour

The RC3 module automatically recognizes domains being closer than *6d* to the selected column/wall end. When finding the punching perimeter, it is important to decide if an opening must be considered as an opening or its size make it necessary to handle it as an edge of the slab. Edges of openings larger than 2 m^2 at the edge of the slab and edges of inner openings larger than 5 m^2 is considered to be an edge of the slab.

If multiple domains are connected to the column or multiple connecting domains are selected the one with the smallest thickness will be analysed. If there is only one selected domain that one will be chosen for punching analysis.



Reinforcement ratio and reinforcement directions

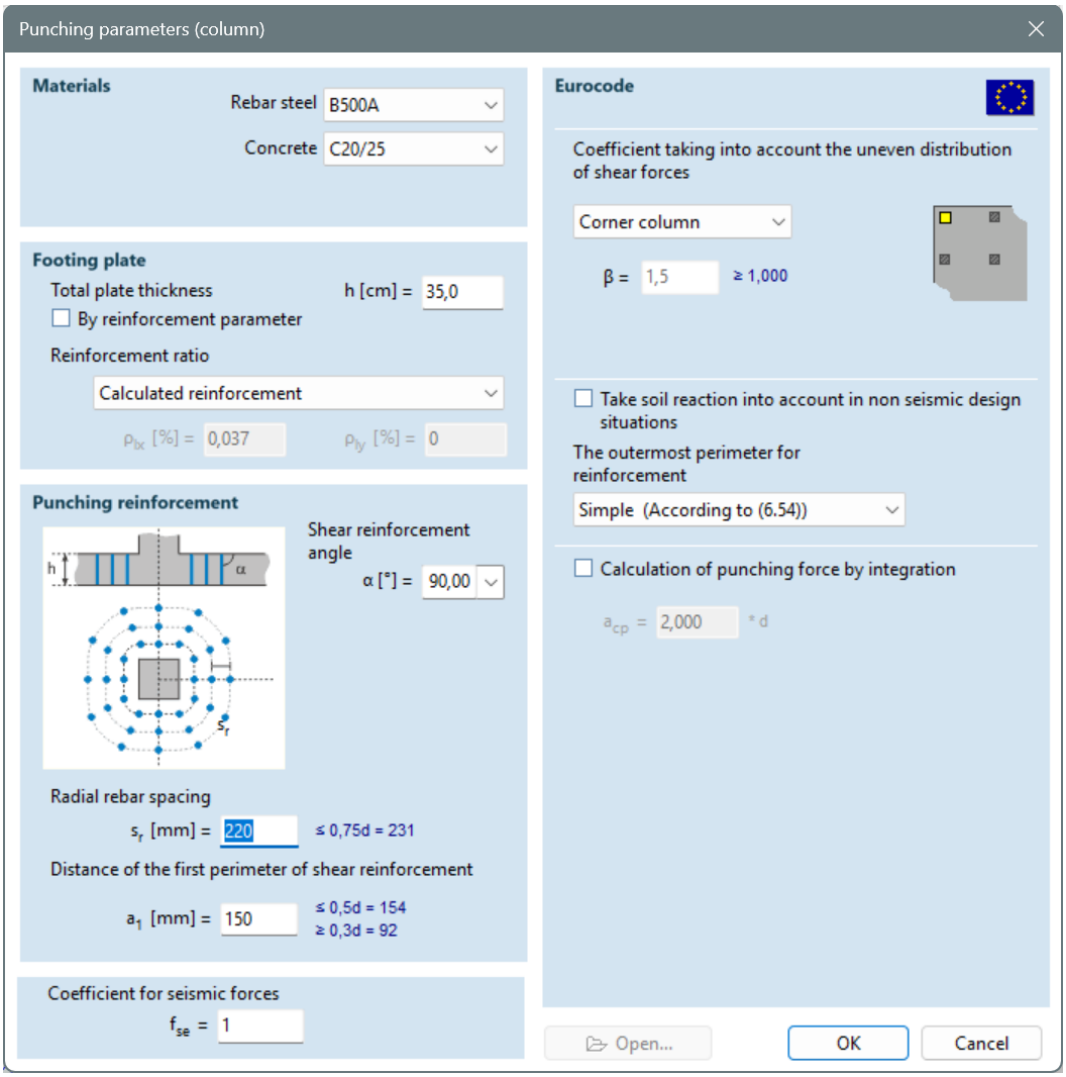
Reinforcement ratios (ρ_x, ρ_y) can be taken from the actual or calculated reinforcement or set *Custom ratio* and specify manually.
The software shows a warning message, if there are multiple domains with different local systems or different reinforcement directions within *6d* distance from the investigated column or wall end/corner. The different local systems and reinforcement directions are problematic due to the calculation of reinforcement ratios and membrane stresses. In case of different reinforcement directions, reinforcement ratios need to be given by the user as custom values.

6.5.15.1 Punching analysis according to Eurocode

6.5.15.1.1 Punching analysis of columns according to Eurocode



Punching shear control perimeters are determined based on the column cross-section and the effective plate thickness. Plate edges and holes are taken into account if they are closer to the column than six times the effective plate thickness. If column cross-section is concave, a convex section is used instead. Punching analysis is enabled if reinforcement parameters and actual reinforcement are defined and assigned to the reinforced concrete slab.
After clicking on the tool button, please select a column or a support with stiffnesses calculated from column parameters for analysis (if a rib element is connected to the column within the plane of the plate, analysis cannot be performed).



Materials

Concrete, rebar steel grade — Concrete and steel grade used in calculation. First, these parameters are automatically selected based on model settings, but they can be changed in the window.

Slab

Plate thickness (h) — Plate thickness is automatically selected but it can be changed in this window if *By reinforcement parameter* is turned off. In the info window the minimum mushroom head thickness is displayed as H1. The minimum mushroom head without punching shear reinforcement is displayed as H2.

Reinforcement ratio — ρ_{lx} and ρ_{ly} refers to the bonded tension steel in x- and y- directions respectively (longitudinal reinforcement). These should be calculated as mean values taking into account a slab width equal to the column width plus 3d each side. (EN 1992-1-1 6.4.4 (1)
You can select the applied or calculated reinforcement, or enter a custom value.

Punching reinforcement

Shear reinforcement angle

Angle between the plate and the punching shear rebars (45°-90°).

Radial rebar spacing

Radial rebar spacing is the difference between the radii of two neighbouring perimeters of shear reinforcement.
The **OK** button is not available until basic design criterion is met: $s_r \leq 0.75 \cdot d$. (section 9.4.3. (1))

Distance of the first perimeter of shear reinforcement

Distance of the first punching perimeter of shear reinforcement from the convex edge of the column.

Coefficient of seimic force

You can see coefficient of seimic forces at **4.10.25 Seismic loads – SE1 module**.

Coefficient taking into account the uneven distribution of shear forces (β factor)

Calculated (6.4.3. (3) (6.39))	$1 + k\frac{M_{Ed}}{V_{Ed}} \cdot \frac{u_1}{W_1}$	
Approximate value by column position (6.4.3. (6))*	*Internal column*	*1,15*
	Edge column	*1,4*
	Corner column	*1,5*
Custom	*user-specified value*	

*For structures where the lateral stability does not depend on frame action between the slabs and the columns, and where the adjacent spans do not differ in length by more than 25%.

Take soil reaction into account

If this option is checked, soil reaction within the rebar circle is considered when calculating the punching force. This effect increases with the radius and can reduce the size of the necessary reinforcement area. Its values per rebar circles are listed in the Punching analysis results dialog.
The program verifies the punching shear resistance according to EN 1992-1-1 6.4.4. (2) along control perimeters within 2d distance from the column edge.

The outermost perimeter for reinforcement

The u_{out} othermost perimeter at which shear reinforcement is not required can be determined according to formula (6.54) in EN 1992-1-1 6.4.5 (4) or can be calculated using two additional methods:

By summation of finite element stress
The punching force for u_{out} is calculated by summation of shear stress along the perimeter.

Shear force reduction by slab self-weight
Reduction of the support punching force by the self-weight of the slab inside the control perimeter.

☞ **The above methods are recommended for situations with a special geometry (slab openings and plate edges), where the length of control perimeters does not increase significantly with the distance from the support. In this case u_{out} cannot be determined using (6.54) or it provides unrealistic results.**

Calculation of punching force

By default the punching force is calculated from the difference in axial forces of connected columns. Check *Calculation of punching force by integration* to find the punching force by integrating shear forces of the slab. The line of integration is at *2d* from the convex outline of the column by default. This value can be set by the user.

☞ **In case of base plates, the punching force is integrated at 0.5d distance from the convex contour of the column ($V_{Ed,0}$). This value is used to check the resistance of concrete compression struts (compared to the design value of the maximum punching shear resistance).**

Open... Loads a saved parameter set.

After entering all parameters control perimeters will appear and the required number of punching rebars is displayed in the info window.

AXISVM calculates the effective parts of the control perimeter based on plate edges and holes. Continuous lines show that reinforcement is needed. AXISVM displays the required amount of reinforcement for each line. The info window shows the amount of critical punching reinforcement. When calculating the length of the critical perimeter it is assumed that rebar spacing on the perimeter is not above *2d* but the fulfilment of this requirement is not checked. If this requirement is not met, the user should choose a smaller diameter or place additional rebars.

Calculation process

The required punching reinforcement is calculated based on the following principles:
The column-slab connection does not fail if the shear stress is less than or equal to the design value of the maximum punching shear resistance along the control section and the design value of the punching shear resistance of the plate with punching shear reinforcement:

$$v_{Ed} \le v_{Rd,max} \text{ and } v_{Ed} \le v_{Rd,cs}$$

v_{Ed} design value of the shear stress
$v_{Rd,max}$ the design value of the maximum punching shear resistance along the control section
$v_{Rd,cs}$ the design value of the punching shear resistance of the plate with punching shear reinforcement

$$v_{Ed} = \beta \frac{V_{Ed}}{u_i \cdot d}$$

where u_i is the length of the control perimeter, d is the mean effective thickness of the plate.
β is a factor expressing additional stress due to eccentric forces:

$$\beta = 1 + k \frac{M_{Ed}}{V_{Ed}} \cdot \frac{u_i}{W_1}$$

Eurocode assumes that the critical section is at a distance of *2d* from the edge of the cross-section. The length of the critical perimeter and the static moment is calculated considering plate edges and holes of the actual geometry.
Design value of the punching resistance of the connection without punching shear reinforcement is:

$$v_{Rd,c} = C_{Rd,c} k (100 \rho_1 f_{ck})^{\frac{1}{3}} + k_1 \sigma_{cp} \ge v_{min} + k_1 \sigma_{cp}$$

where $\rho_1 = \sqrt{\rho_{lx} \rho_{ly}}$ ($\rho_1 = \sqrt{\rho_\xi \rho_\eta}$ in case of skew reinforcement)
If $v_{Ed} > v_{Rd,c}$ then the required punching reinforcement is determined along the critical perimeter

$$v_{Rd,cs} = 0.75 \cdot v_{Rd,c} + 1.5 \cdot \frac{d}{s_r} \cdot \frac{A_{sw} f_{ywd,ef}}{u_1 d} \sin \alpha \le k_{max} \cdot v_{Rd,c};$$
$$v_{Ed} \le v_{Rd,cs}$$

If $v_{Ed} > k_{max} \cdot v_{Rd,c}$, then the column-plate joint is not safe for punching
The program determines the distance of the outermost perimeter of shear reinforcement from the edge of the support.

Critical combination or load case

If the design value of the maximum punching shear resistance along the control perimeter is not enough ($v_{Ed} > v_{Rd,max}$), then the critical case is the case of maximum $v_{Ed}/v_{Rd,max}$. Otherwise, if punching shear reinforcement is not required ($v_{Ed} \le v_{Rd,c}$), then the critical case is the case of maximum $v_{Ed}/v_{Rd,c}$. If the punching shear reinforcement cannot be verified ($v_{Ed} > v_{Rd,cs}$), then the critical case is the case of maximum $v_{Ed}/v_{Rd,cs}$,. Otherwise, the critical case is the case of maximum shear reinforcement (A_{sw}).
The info window displays the critical result component in **bold.**

Results for the critical perimeter are calculated first (these are displayed in the *Punching analysis results* dialog). Then the required amount of reinforcement is determined for reinforcement circles defined in the parameters dialog. The critical perimeter is red, reinforcement circles are black. Dashed line shows the perimeter where the distance of points from the column is six times the effective plate thickness.
A thin blue line shows the perimeter where no punching reinforcement is needed.
This is also the outline of the mushroom head which can be designed with thickness $h_{min,2}$ and without punching reinforcement.

A thick blue line shows the perimeter where the critical punching force exceeds the compressing strength of the concrete so the plate with the original thickness cannot be properly reinforced. This is the outline of the mushroom head which can be designed with thickness $h_{min,1}$ and with punching reinforcement. Punching capacity can be increased by setting the plate thicker, using a better concrete grade or columns with bigger cross-section area.


Saves the drawing to the Drawing Library.


Loads a saved punching parameter set.

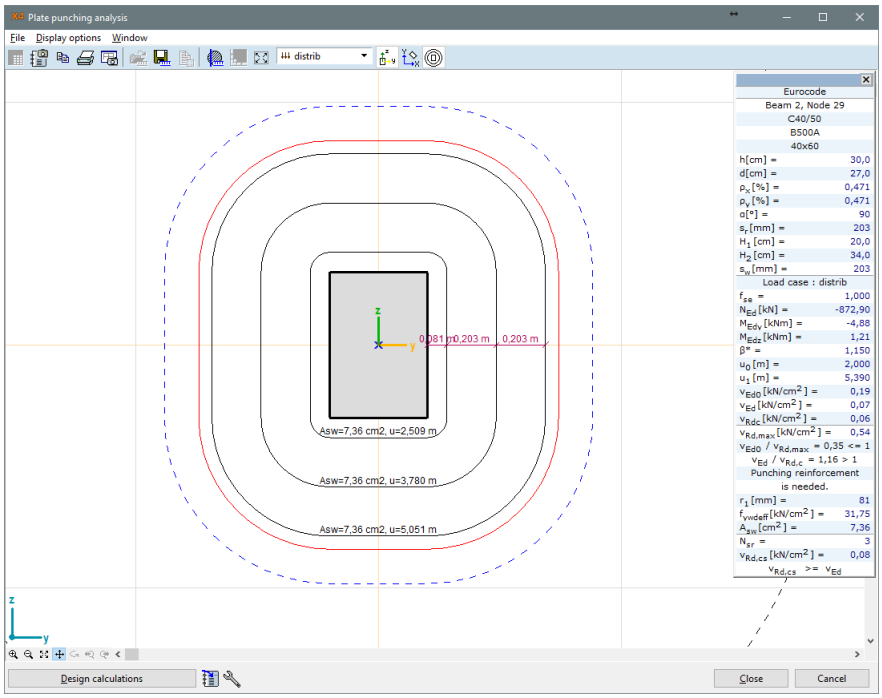

Saves the current punching parameters under a name. You can load back the saved parameters with the button *Loading*... on *Punching parameters* dialog.


Punching parameters dialog.

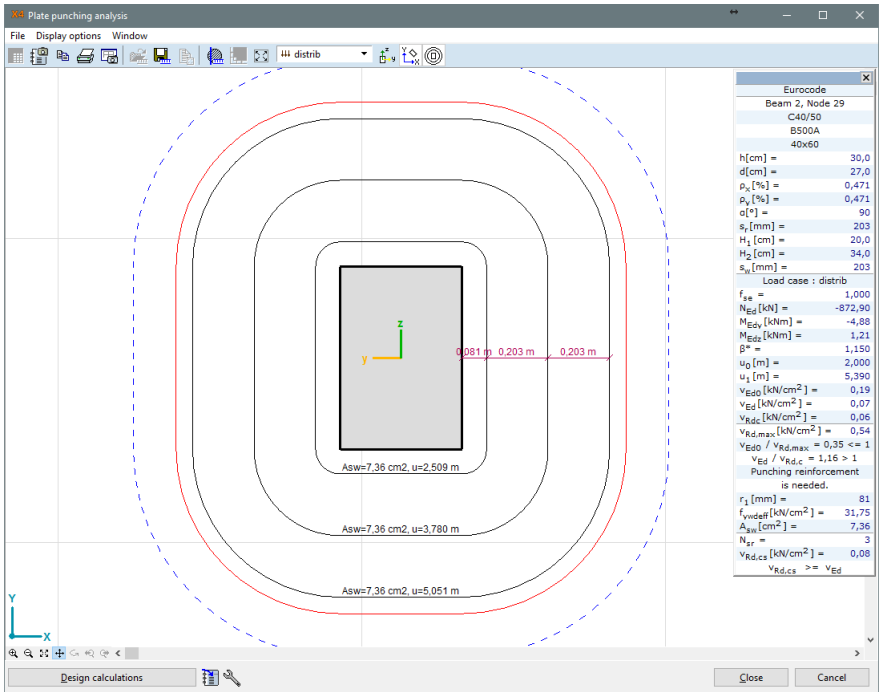

Inflates the plate boundary so that the entire column cross section is within the boundary.

Fits the diagram to the window.

Column local coordinates are used.



Global coordinates are used.



Turns on and off the display of rebar circles.

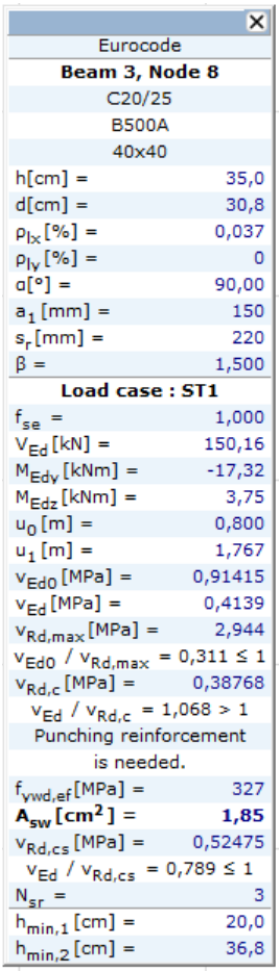
To follow design calculations in details, click on the *Design calculations* button at the bottom left. See also **6.6 Steel design** and **6.6.1.3 Diagrams and design calculations**

Clicking on this icon adds the design calculation to the current report.

Clicking on the *Settings* icon beside the *Design calculations* button allows setting the units for force and length used in the design calculations.

A short summary of design calculations is available, which can be printed on its own or together with the detailed calculations.

Info window Under the design code, element identifier and materials the following parameters are displayed.



h plate thickness

d effective plate thickness

ρ_{lx}, ρ_{ly} ratios of longitudinal reinforcement in x and y directions

α angle between the plate and the punching reinforcement

a_1 distance of the first perimeter of shear reinforcement from the convex edge of the column

s_r distance between perimeters of shear reinforcement

f_{se} factor scales only the internal forces from seismic load

V_{Ed} design value of punching force

$V_{Ed,0}$ design value of punching force integrated at 0.5d distance from the convex contour of the column (only for base plates)

M_{Edy}, M_{Edz} design value of bending moments

u_0 control perimeter at the column perimeter

u_1 critical control perimeter at *2d*

β^* calculated eccentricity factor

$v_{Ed,0}$ shear stress along the u_0 perimeter

v_{Ed} shear stress along the u_1 perimeter

$v_{Rd,max}$ the design value of the maximum punching shear resistance

$v_{Rd,c}$ the design value of the punching shear resistance of the slab without punching shear reinforcement

$v_{Ed,0}/v_{Rd,max}$ utilization on the u_0 perimeter

$v_{Ed}/v_{Rd,c}$ utilization (tension in concrete)

$f_{ywd,ef}$ tension in the punching reinforcement

A_{sw} punching reinforcement area on the critical control perimeter

$v_{Rd,cs}$ the design value of the punching shear resistance of the slab with punching shear reinforcement

$v_{Ed}/v_{Rd,cs}$ utilization of the plate with punching reinforcement

N_{sr} number of reinforcement circles

$h_{min,1}$ minimum plate thickness required with punching reinforcement

$h_{min,2}$ minimum plate thickness required without punching reinforcement

6.5.15.1.2 Punching analysis of wall ends and corners according to Eurocode

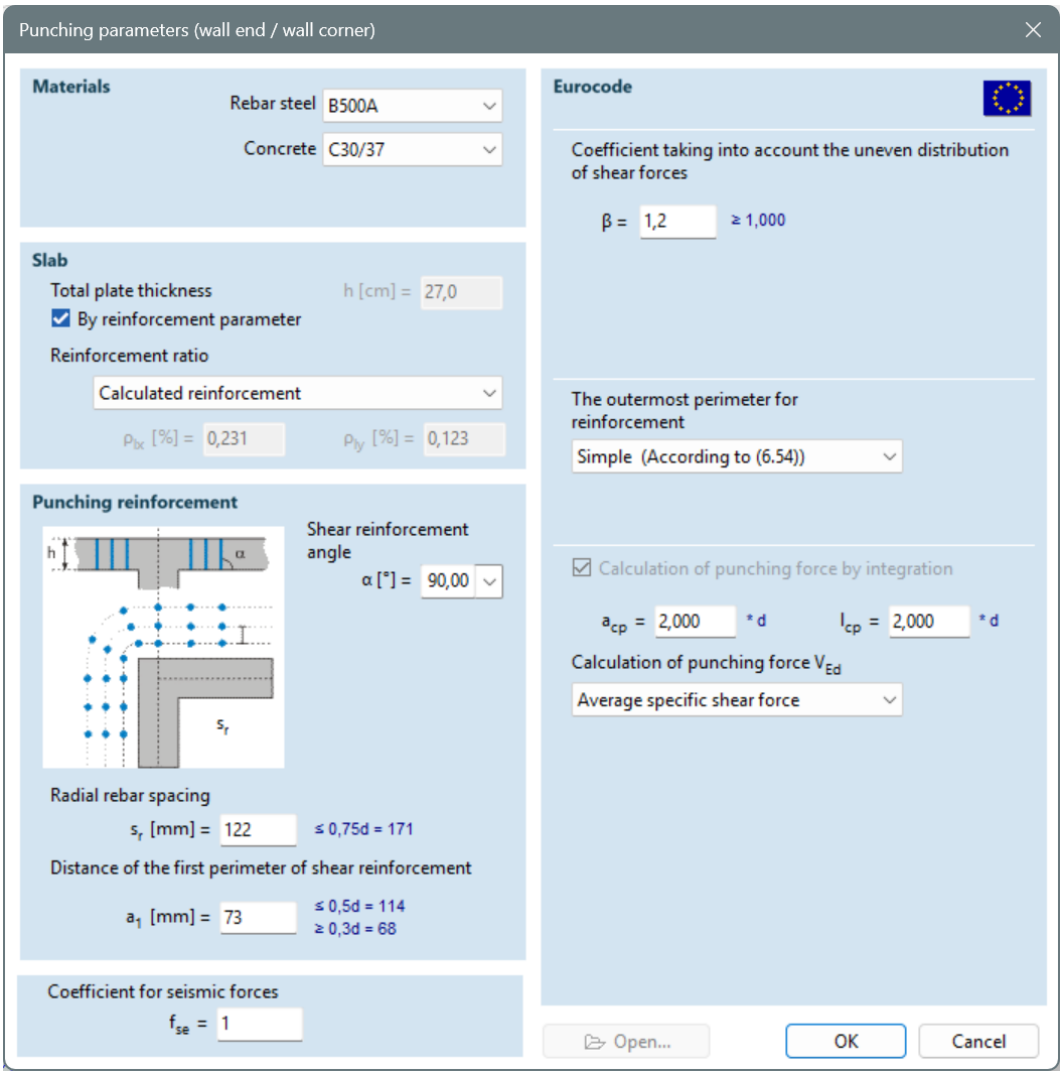


Punching analysis of wall ends and wall corners can be performed using RC3 module including the calculation of required reinforcement if necessary. The software generates the punching control perimeters according to the relative position of the walls. Plate edges and holes are taken into account if they are closer than six times the effective plate thickness.

After clicking on the tool button shown above, a node needs to be selected that identifies the wall end or the wall corner. The program identifies automatically the walls connecting to the node. Walls can be represented by vertical domains or line supports with section (with stiffnesses calculated from wall parameters), see **4.9.15 Line support**. The software is able to identify both elements.

The punching parameters can be defined in the following window:

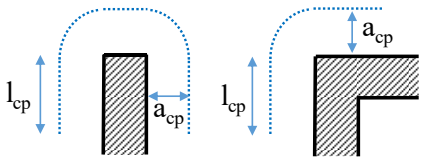
Punching parameters



Most of the parameters can be set in the dialog has been already presented in the previous sections (see **6.5.15.1.1 Punching analysis of columns according to Eurocode**).

The punching force is generally calculated according to one of the following ways in case of wall ends and corners:

- by integration of section force along the wall(s)
- by integration of shear forces along a segment around the wall end/corner



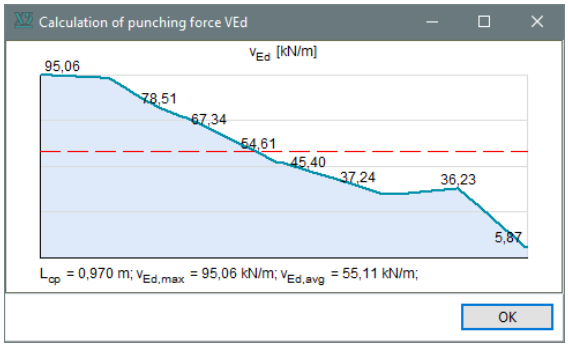
The second method is implemented in the software. a_{cp} and l_{cp} parameters (used to generate section line) can be specified by the user. The punching force is calculated from the shear forces along the line on the following way presented below.

The a_{cp} and l_{cp} parameters can be given in terms of d (effective depth) in the *Punching parameters* window. In the case of a_{cp}=2 and l_{cp}=2, the position of the integration line will be at 2d distance from the edge of the wall and the ends of the integration line will be at also 2d distance from the wall end/corner.

☞ **In case of base plates, the punching force is integrated at 0.5d distance from the contour of the wall end/corner ($V_{Ed,0}$). This value is used to check the resistance of concrete compressed strut (compared to the design value of the maximum punching shear resistance).**



The distribution of shear forces along the integration line is shown by clicking on this tool button:



☞ **It has to be noted that the line given by a_{cp} and l_{cp} parameters is a tool used to calculate the punching force and it is not considered as punching perimeter. The punching perimeter is selected according to the design code (see 6.5.15.1.1 Punching analysis of columns according to Eurocode and 6.5.15.2.1 Punching analysis of columns according to SIA 262).**

Calculation of VEd punching force

Calculated from maximum shear force:

$$V_{Ed} = v_{Ed,max} \cdot L$$

where

- $v_{Ed,max}$ is the maximum value of shear force along the integration line;
- L is the length of the integration line.

Calculated from average shear force:

$$V_{Ed} = \beta \cdot v_{Ed,avg} \cdot L$$

where

- β is coefficient that takes into account the effect of bending moment;
- $v_{Ed,avg}$ is the average value of shear force along the integration line;

v_{Ed} is calculated on the following way:

$$v_{Ed} = \sqrt{v_{xz}^2 + v_{yz}^2}$$

Soil reaction

In case of wall end/wall corner punching analysis the soil reaction force is ignored as the punching perimeters are not closed polygons and the punching force is typically integrated at *2d* from the contour of the wall thus the most of the soil interaction is already neglected.

Graphical interface and results

The graphical interface and buttons are described and presented in detail in the previous section (see 6.5.15.1.1 Punching analysis of columns according to Eurocode). Design results are listed in an info window and shown graphically in Plate punching analysis window. If punching reinforcement is required, calculated reinforcement is written on punching perimeters. After clicking on the Design calculations button, a report of the calculation details is displayed. Design calculations and drawings can be saved into the documentation.

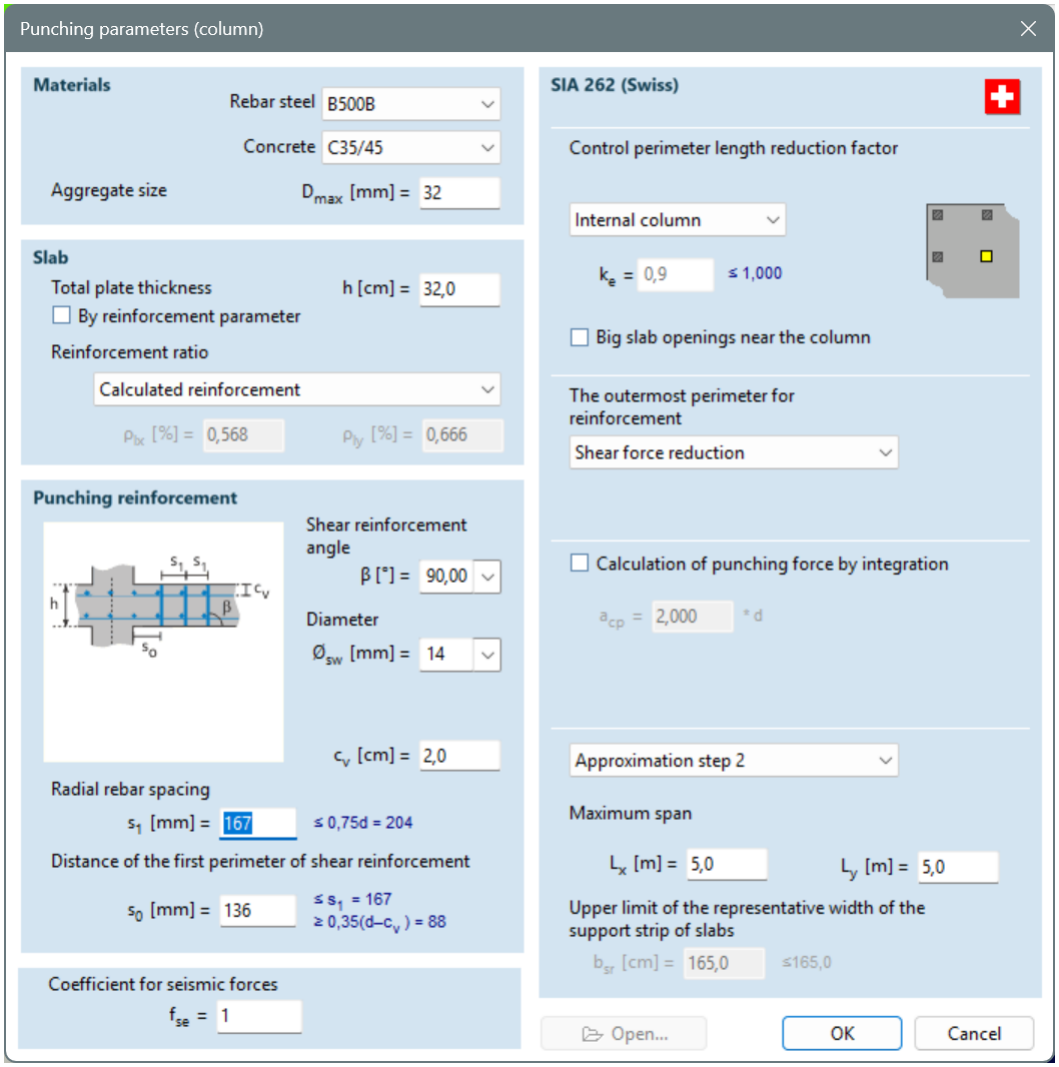
6.5.15.2 Punching analysis according to SIA 262

6.5.15.2.1 Punching analysis of columns according to SIA 262



Punching analysis is enabled if reinforcement parameters and actual reinforcement are defined and assigned to the reinforced concrete slab.
After clicking the tool button, please select a column or a support with stiffnesses calculated from column parameters for analysis.
Some details of design process are discussed in 6.5.15.1.1 Punching analysis of columns according to Eurocode section and not repeated here.

Parameters related to punching design check can be specified in the following window similarly to Eurocode conforming design.



Materials

Concrete, rebar steel grade — Concrete and steel grade used in calculation. First, these parameters are automatically selected based on model settings, but they can be changed in the window.

Aggregate size — Maximum grain size of aggregate, D_{max} for coefficient k_g in expression (37).

Slab

Plate thickness (h) — Plate thickness is automatically selected but it can be changed in this window if *By reinforcement parameter* is turned off.

Reinforcement ratio — ρ_{lx} and ρ_{ly} refers to the bonded tension steel in *x*- and *y*- directions respectively (longitudinal reinforcement). The bending resistance of the slab (m_{Rd}) is determined from the reinforcement ratio.

Punching reinforcement

β Angle between the plate and the punching shear rebars (45°-90°).

Φ_{sw} Diameter of punching shear rebars.

c_v Concrete cover of punching shear reinforcement on compression side.

Radial rebar spacing, s_1 Radial rebar spacing is the difference between the radii of two neighbouring perimeters of shear reinforcement.

Distance of the first perimeter of shear reinforcement, s_0 Distance of the first punching perimeter of shear reinforcement from the convex edge of the column.

Control perimeter length reduction factor k_e You can choose to have the program take into account the factor k_e with the value calculated according to the formula (57) in paragraph 4.3.6.2.4 or, depending on the position of the column, with the approximate value according to paragraph 4.3.6.2.5.

The outermost perimeter for reinforcement The u_{out} outhermost perimeter at which shear reinforcement is not required can be determined base of formula (58) in 4.3.6.3.1 by rearranging the formula: $u_{out} = \frac{V_d}{k_r \tau_{cd} d_v}$

or can be calculated using two additional methods:

By summation of finite element stress
The punching force for u_{out} is calculated by summation of shear stress along the perimeter.

Shear force reduction by slab self-weight
Reduction of the support punching force by the self-weight of the slab inside the control perimeter.

☞ **The last two methods are recommended for situations with a special geometry (slab openings and plate edges), where the length of control perimeters does not increase significantly with the distance from the support. In this case u_{out} cannot be determined base (57) or it provides unrealistic results.**

Calculation of punching force The punching force can be calculated from the difference in axial forces of connected columns or by the integration of shear forces of the slab (see **6.5.15.1.1 Punching analysis of columns according to Eurocode**).

Approximation step Approximation level can be selected according to SIA 262 (section 4.3.6.4.2). Approximation steps 1 and 2 can be selected.

l_x, l_y span in x-, y-directions

b_{sr} Limit value for width of column strips (for edge or corner columns) for the determination of bending resistances.

Calculation of punching force The punching force can be calculated from the difference in axial forces of connected columns or by the integration of shear forces of the slab (see **6.5.15.1.1 Punching analysis of columns according to Eurocode**).

Design principles The software determines the punching perimeter at $0{,}5d_v$ distance (section 4.3.6.2.2) from the convex edge of the column where punching resistance need to be checked. The resistance of compressive struts is calculated according to section 4.3.6.5.8:

$$V_{Rd,max} = 2 \cdot k_r \cdot \tau_{cd} \cdot d_v \cdot u$$

If $V_d > V_{Rd,max}$, the resistance of compressive struts is not adequate. In this case, the plate thickness or concrete strength need to be changed.
The punching shear resistance without shear reinforcement is calculated on the following way (section 4.3.6.3.1):

$$V_{Rd,c} = k_r \cdot \tau_{cd} \cdot d_v \cdot u$$

If $V_d \leq V_{Rd,c}$, punching shear reinforcement is not necessary, otherwise the required shear reinforcement is calculated as follows (sections 4.3.6.5.2 (65) and 4.3.6.5.3 (66)):

$$\Sigma A_{sw} = \frac{V_d}{k_e \cdot \sigma_{sd} \cdot sin\beta},$$

where

$$\sigma_{sd} = \frac{E_s \cdot \psi}{6} \cdot \left(1 + \frac{f_{ctm}}{f_{yd}} \cdot \frac{d}{\phi_{sw}}\right) \leq f_{yd}$$

$$\psi = 1{,}5 \cdot \frac{r_s}{d} \cdot \frac{f_{yd}}{E_s} \cdot \left(\frac{m_{sd}}{m_{Rd}}\right)^{3/2}$$

This required reinforcement needs to be applied between $0{,}35 \cdot d_v$ and d_v according to the code. Based on the given design parameters, the number perimeters of shear reinforcement are calculated can be placed between $0{,}35 \cdot d_v$ and d_v. Required reinforcement for a perimeter is calculated by dividing ΣA_{sw} with the number of perimeters. This A_{sw} required reinforcement is placed (outside d_v distance) on each perimeter until it is needed.

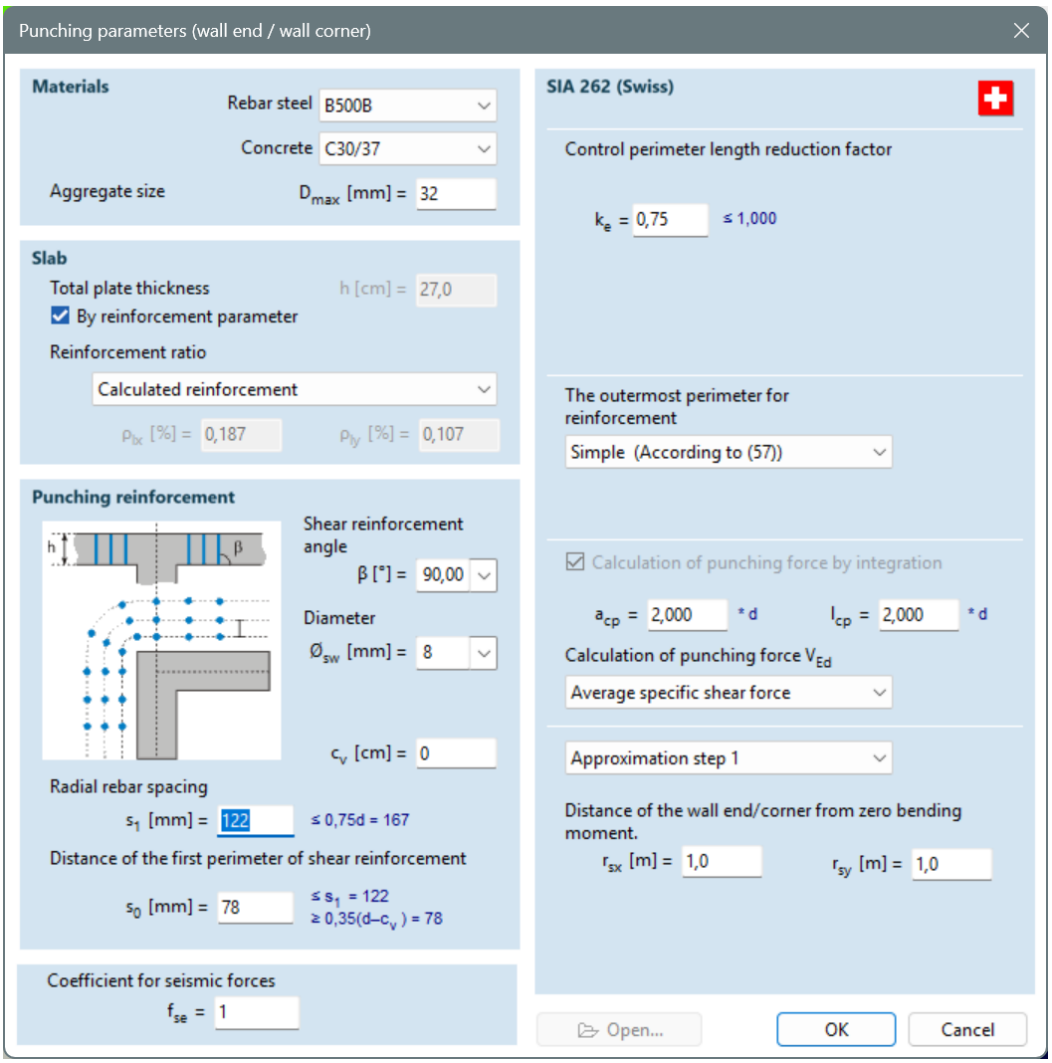
Info window Under the design code, element identifier and materials the following parameters are displayed.

h	plate thickness
d	effective plate thickness
ρ_{lx}, ρ_{ly}	ratios of longitudinal reinforcement in x and y directions
β	angle between the plate and the punching reinforcement
s_0	distance of the first perimeter of shear reinforcement from the convex edge of the column
s_1	distance between perimeters of shear reinforcement
$\varnothing_{sw}$	diameter of shear reinforcement
d_v	effective plate thickness
l_x, l_y	span in x-, y-directions
f_{se}	factor scales only the internal forces from seismic load
V_d	design value of punching force
$V_{d,0}$	design value of punching force integrated at 0.5d distance from the convex contour of the column (only for base plates)
M_{dy},M_{dz}	design value of bending moments
k_e	reduction factor for punching shear perimeter
u	length of punching shear perimeter
u'	$u' = k_{e\,u} \cdot u$
$V_{Rd,max}$	the design value of the maximum punching shear resistance
$V_{Rd,c}$	the design value of the punching shear resistance of the slab without punching shear reinforcement
$V_{Ed}/V_{Rd,max}$	utilization for $V_{Rd,max}$
$V_{Ed}/V_{Rd,c}$	utilization for $V_{Rd,c}$
$V_{d,s}$	punching force for calculation of shear reinforcement
σ_{sd}	design value of tensile stress in shear reinforcement
ΣA_{sw}	required shear reinforcement
A_{sw}	required shear reinforcement for one perimeter
$V_{Rd,s}$	design value of punching shear force for calculation of reinforcement
N_{sr}	number of perimeters of shear reinforcement

SIA 26x (Swiss)	
Beam 1, Node 6	
C35/45	
B500B	
40x40	
h[cm] =	32,0
d[cm] =	27,2
ρ_{lx}[%] =	0,568
ρ_{ly}[%] =	0,666
β[°] =	90,00
s_0[mm] =	136
s_1[mm] =	167
Ø[mm] =	14
d_v[cm] =	25,2
l_x[m] =	5,0
l_y[m] =	5,0
Load case : ST1	
f_{se} =	1,000
V_d[kN] =	1213,93
M_y[kNm] =	-20,84
M_z[kNm] =	-60,05
u[m] =	2,391
k_e =	0,900
u'=k_e·u[m] =	2,152
$V_{Rd,c}$[kN] =	648,73
$V_{Rd,max}$[kN] =	1297,45
V_d / $V_{Rd,max}$ = 0,936 ≤ 1	
V_d / $V_{Rd,c}$ = 1,871 > 1	
Punching reinforcement	
is needed.	
$V_{d,s}$[kN] =	606,96
σ_{sd}[MPa] =	426,59
Σ A_{sw}[cm²] =	**15,81**
A_{sw}[cm²] =	15,81
$V_{Rd,s}$[kN] =	606,99
N_{sr} =	2
$h_{min,1}$[cm] =	31,5
$h_{min,2}$[cm] =	38,7

6.5.15.2.2 Punching analysis of wall ends and wall corners according to SIA 262

The software determines the punching shear force as it is described in **6.5.15.1.2 Punching analysis of wall ends and corners according to Eurocode**. In case of SIA conforming design, β factor is not used. Reduction factor for perimeter length (k_e) can be specified by the user.
After clicking on tool button shown above, a node needs to be selected that identifies the wall end or the wall corner. The program identifies automatically the walls connecting to the node. Walls can be represented by vertical domains or line supports with section (with stiffnesses calculated from wall parameters) (see **4.9.15 Line support**). The software is able to identify both elements.
Some details of design process are discussed in **6.5.15.1.2 Punching analysis of wall ends and corners according to Eurocode** section and not repeated here.



Details of design principles and equations are discussed in the previous section. see **Punching analysis of columns according to SIA 262.** In case of wall ends and corners, the r_s distance can be specified by the user.

6.5.15.3 Punching result components

Result components

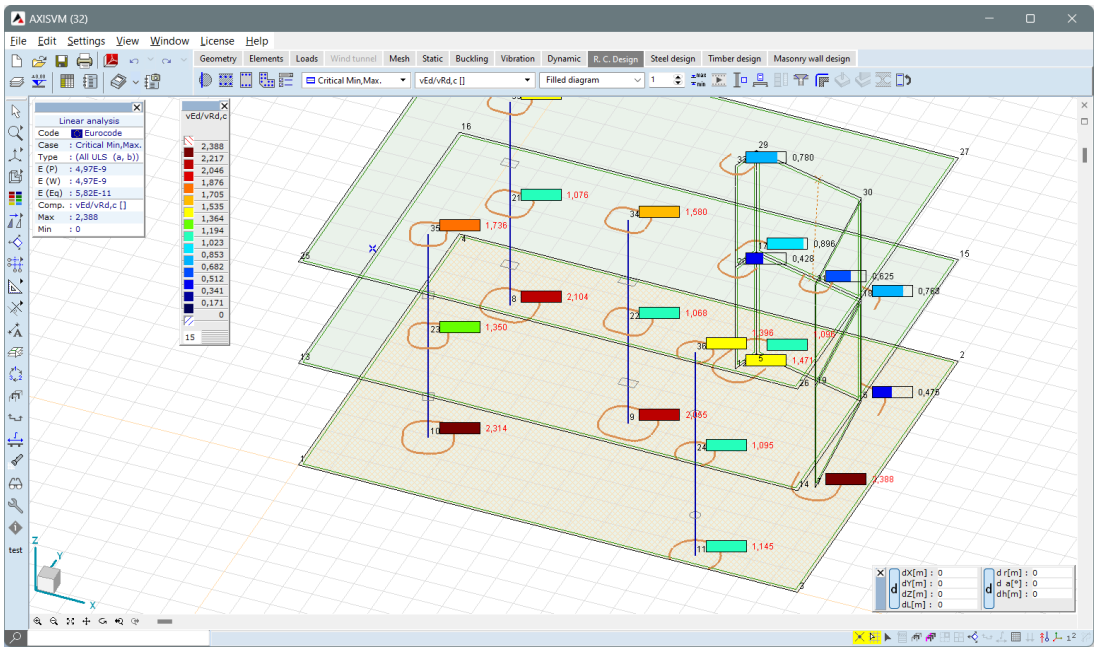
If columns, supports, wall ends or wall corners with punching design parameters exist in the model and static (linear/non-linear) or dynamic results are available, new result components become available on R. C. Design tab related to punching check results. With the help of these new result components, the punching results can be easily displayed and documented for every element and node.

Eurocode

- v_{Ed} – design value of punching force
- u_1 – critical control perimeter at *2d*
- v_{Ed} – shear stress along the u_1 perimeter
- $v_{Rd,c}$ – the design value of the punching shear resistance of the slab without punching shear reinforcement
- $v_{Ed,0}$ – shear stress along the u_0 perimeter
- $v_{Rd,max}$ – the design value of the maximum punching shear resistance
- $v_{Ed}/v_{Rd,c}$ – utilization for $v_{Rd,c}$
- $v_{Ed,0}/v_{Rd,max}$ – utilization for $v_{Rd,max}$ along the u_0 perimeter
- A_{sw} – punching reinforcement area on the critical control perimeter
- $v_{Rd,cs}$ – the design value of the punching shear resistance of the slab with punching shear reinforcement
- $v_{Ed}/v_{Rd,cs}$ – utilization for $v_{Rd,cs}$

SIA

- V_d – design value of punching force
- u – length of punching shear perimeter
- k_e – reduction factor for punching shear perimeter
- u' – effective length of punching shear perimeter
- $V_{Rd,c}$ – the design value of the punching shear resistance of the slab without punching shear reinforcement
- $V_{Rd,max}$ – the design value of the maximum punching shear resistance
- $V_d/V_{Rd,c}$ – utilization for $V_{Rd,c}$
- $V_d/V_{Rd,max}$ – utilization for $V_{Rd,max}$
- A_{sw} – required shear reinforcement for one perimeter
- $V_{Rd,s}$ – design value of punching shear force for calculation of reinforcement
- $V_d/V_{Rd,s}$ – utilization for $V_{Rd,s}$



Results in the Table Browser

Design results are also available in the Table Browser (see **2.9 Table Browser**), where the results of punching analyses of columns, supports, wall ends and corners are available in summary tables.

6.5.16 Footing design – RC4 module

6.5.16.1 Pad footing design



AXISVM can determine the necessary size and reinforcement of rectangular or circular pad foundations (with or without pedestal), and can check the footing against sliding and punching according to Eurocode7. It determines the settlement of the foundation as well.

Footing size

The size of the foundation can be entered or let AXISVM calculate it. If AXISVM calculates the size a maximum value must be specified.
Using the soil profile and the internal forces this module determines the necessary size of the foundation in an iterative process. Then it calculates the effective area of the foundation for load cases and combinations, the design forces, moments and resistances, determines the settlement (for load cases and Service Limit State [SLS] combinations), efficiencies and the shear reinforcement if necessary. The module also checks the stability of the footing.
Step sides must not be bigger than the respective side of the foundation.

☞ **The coordinate system used in footing calculations is the coordinate system of the support. However, eccentricities are calculated relative to the center of the footing.**

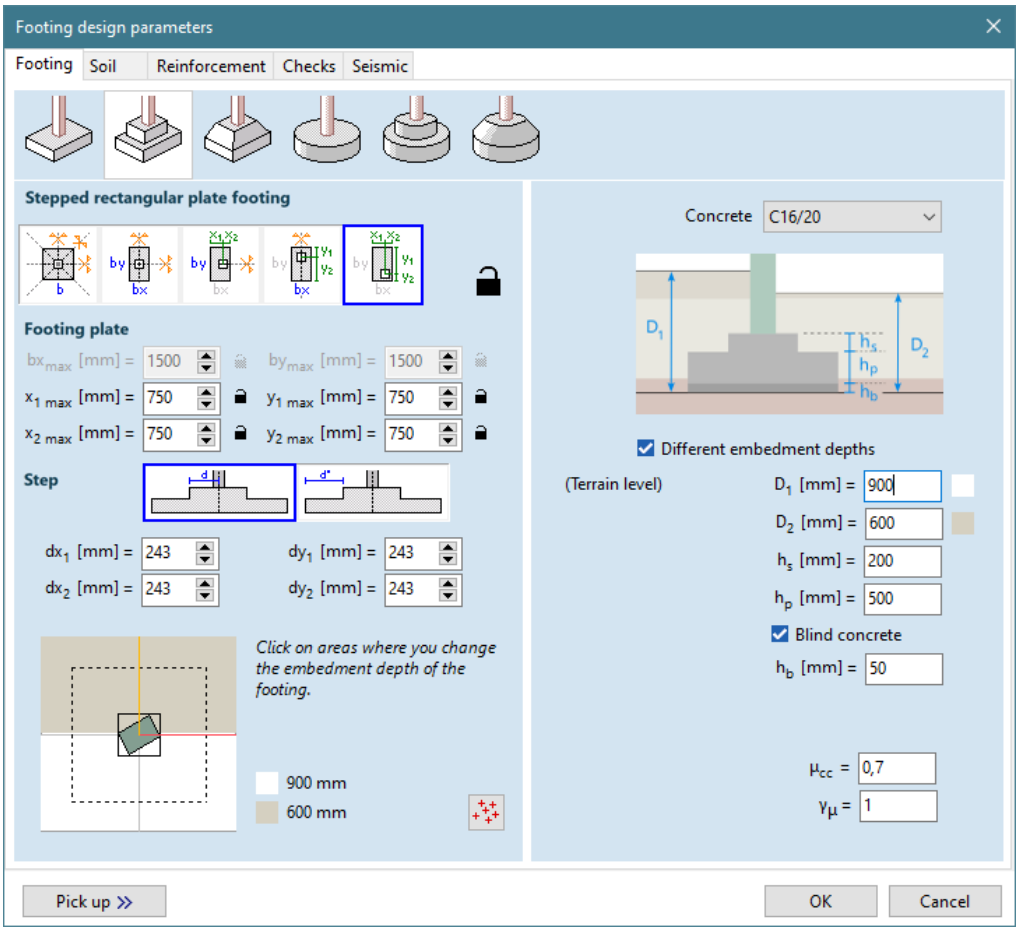
Footing design parameters



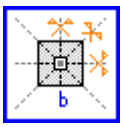
Click the *Footing design* icon and select one or more nodal supports with a vertical or slanted column. (If supports have been already selected, the dialog is displayed at the first click).
Footing design parameters have to be specified in a dialog.

Footing

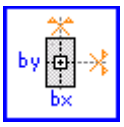
At the *Footing* tab select the footing type (simple plate / stepped / sloped, rectangular or circular) and set the geometry parameters and the friction coefficient between the footing and the blind concrete and the partial factor for the friction coefficient.



Symmetry constraints and geometric parameters for a rectangular footing:



Square footing
b is the side length,
the column is concentric,
value or upper limit of *b* must be entered



Rectangular footing
bx and *by* are the sides,
the column is concentric,
value or upper limit of *bx* and *by* must be entered



Single eccentric rectangular footing
the column is eccentric in *x* direction, concentric in *y* direction
x_1 and x_2 are the distance of the column axis from the edges of the footing
value or upper limit of x_1, x_2 and *by* must be entered

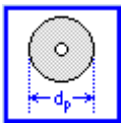


Single eccentric rectangular footing
the column is eccentric in *y* direction, concentric in *x* direction
y_1 and y_2 are the distance of the column axis from the edges of the footing
value or upper limit of y_1, y_2 and *bx* must be entered



Double eccentric rectangular footing
the column is eccentric in both directions
x_1 and x_2 are the distance of the column axis from the edges of the footing in *x* direction
y_1 and y_2 are the distance of the column axis from the edges of the footing in *y* direction
value or upper limit of x_1, x_2, y_1, y_2 must be entered

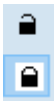
Parameters for a circular footing:



Circular footing
the column is concentric
d_p is the diameter of the footing
d_p or its upper limit ($d_{p,max}$) must be entered



If the lock button beside an edit field is down (closed), the entered value is fixed (the value is checked). If the lock icon is up (open) the entered value is the upper limit (the value will be calculated by the program). After pressing down the large lock button *(Verification of a footing with given dimensions)*, all values become locked and can be released only after opening the large lock.

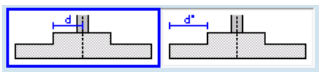


Design:

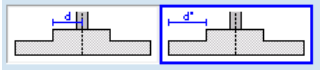
If the footing is safe for all checks and the large lock button is open, the program runs an iteration process to optimize the area (sizes) of the foundation performing the tests for verification. The bearing resistance and eccentricity verification are always performed, but design for excentricity is optional.

Stepped and sloped footings

There are two options to define the step size.



dx_1 and dx_2 are the distance of the edges of the step or the upper base of the frustum **from the column axis** in *x* direction. dy_1 and dy_2 are the same in *y* direction. These are always given values.
For circular footings: d_s is the diameter of the concentric cylindrical step or the upper base of the truncated cone. Value of the $d_s / 2$ radius must be entered.



dx^*_1 and dx^*_2 are the distance of the edges of the step or the upper base of the frustum **from the perimeter of the footing** in *x* direction. dy_1 and dy_2 are the same in *y* direction.
For circular footings: d_s is the diameter of the concentric cylindrical step or the upper base of the truncated cone. Value of $d_s / 2' = (d_p - d_s) / 2$ must be entered.

In the first case the step size is always fixed. In the second case step size will grow or shrink with the footing size if the size is not locked.

Other parameters:

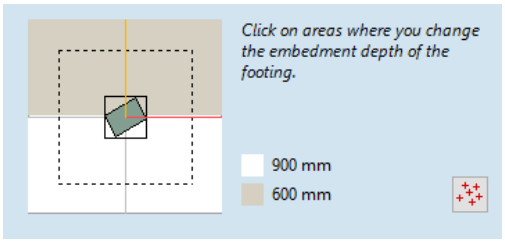
Concrete material of the footing

Different embedment depths

The embedment depth of footings can have two different values (D_1, D_2). These can be assigned to quarters (for pad footings) or sides (for strip footings) by clicking on the footing diagram.
By convention, D_1 is taken as the value measured from the ground level, so the position of soil layers will be determined accordingly. If $D_2 > D_1$, the topmost layer will be extended upwards.
The bearing capacity is calculated from the smaller embedment depth, D = min (D_1, D_2)
The volume of backfill is calculated from the embedment depth, so different values make the self-weight of a symmetric footing eccentric.

D or D_1	Embedment depth (distance between the bottom of the base plate and the top of the soil profile)
D_2	Embedment depth in selected quarters or sides
h_s	Step height (height of the step or the frustum)
h_p	Base plate thickness
h_b	Blind concrete thickness
μ_{cc}	Friction coefficient between the footing and the blind concrete
γ_μ	Partial factor of the friction coefficient

You can see coefficient of seimic forces at **4.10.25 Seismic loads – SE1 module.**



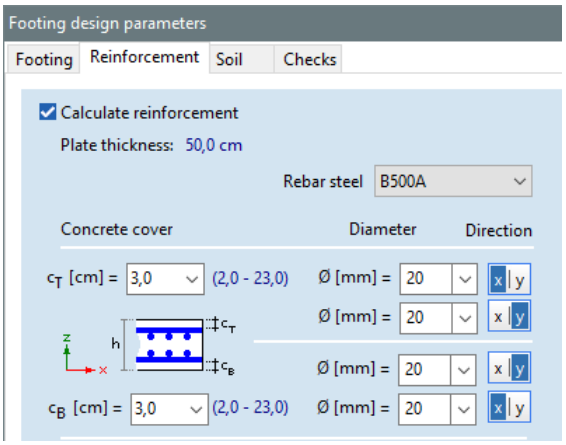
Under the edit fields on the left side the footing and the column is displayed in top view. Given sizes are drawn as continuous lines, upper limits as dashed lines. The forces appear as red crosses placed according to their eccentricities. This diagram is for orientation purposes only because the actual eccentricities are calculated taking into account the self -weight of the footing and the backfill reducing the eccentricity.
If the option *Different embedment depths* is enabled, the area around the column is divided into quarters and clicking into quarters changes the embedment depth in that quarter. This way a different embedment depth can be set to the upper/lower half or on the left/right side. This division follows the local *x* and *y* directions of the support (red and yellow axes on the diagram).



If the button *Show all support forces* is down, the view is scaled to show all force crosses. If the button is up only crosses within the bounding rectangle of the footing are displayed. These forces act at the top of the footing and does not include the weight of the backfill, footing and blind concrete.

Reinforcement

On the *Reinforcement* tab reinforcement calculations can be activated.
Rebar steel grade, c_T and c_B top and bottom concrete covers can be entered (i.e., the least distance between the surface of embedded reinforcement and the outer surface of the concrete). The top two rows under *Diameter* and *Direction* represent the top rebars (the 1st row is the outer one, the 2nd row is the inner one), the bottom two rows represent the bottom rebars (the 3rd row is the inner one, the 4th row is the outer one). The actual scheme in local *x-z* view is displayed accordingly.



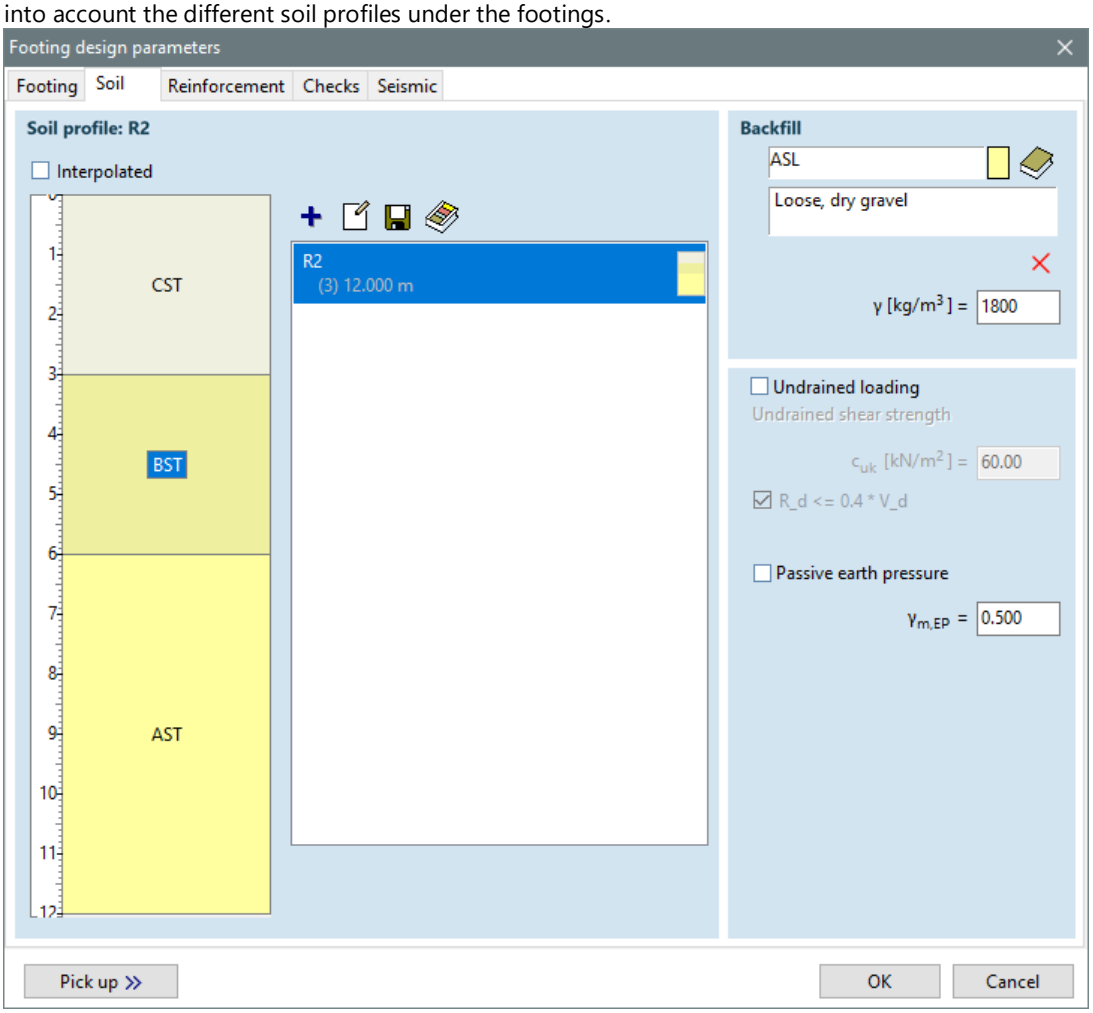
Punching reinforcement calculations can be activated only for simple plate footings.
For the parameters see **6.5.15 Punching analysis – RC3 module**

Soil

At the *Soil* tab you can specify the soil profile and the properties of the backfill.

Soil profile

The soil profile can be selected from the list of profiles defined in the model, imported from a model-independent soil profile library (**6.5.16.4 Soil profile library**) or created from scratch (**6.5.16.3 Soil profile editor**).
If borehole samples have been defined in the model (see **4.9.3 Add borehole samples – SOIL module**), the *Interpolated* option can be used to calculate the soil profile under the support from the stratigraphic model. If multiple footings are selected, the module performs the requested checks with the parameters entered in the dialog. If the *Interpolated* option is activated, the checking algorithm takes into account the different soil profiles under the footings.



Create a new soil profile
Displays the *Soil profile editor* where the soil profile can be entered and added to the profiles available in the model.

Edit soil profile
Displays the *Soil profile editor* with the current soil profile for editing. The existing soil profile can be overwritten or the modified soil profile can be added to the list.

Save soil profile to the library
The selected soil profile is saved to the model-independent *Soil profile library*. If there is already a soil profile in the library with this name, (1), (2), ... is appended.

Soil profile library
This library is available from any model. Multiple soil profiles can be selected and imported. If there is already a soil profile with the same name (1), (2), ... is appended.

Soil profiles can be deleted in the *Table Browser*. Select *Soil profiles* to see the soil profiles defined in the model. As deleting soil profiles may delete Winkler springs and/or footings, the name of the used soil profiles is displayed in bold. Deleting a used soil profile triggers a warning message.

Backfill

Enter the parameters of the soil used as backfill.

Opens the soil library to select a predefined material.

Undrained loading

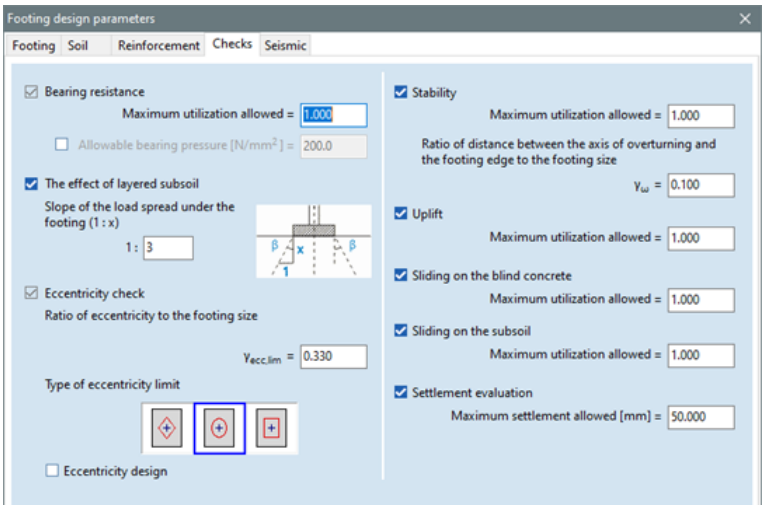
Under undrained loading there is no volume change, since water cannot escape. The soil is fully saturated, shear strength is a constant value that can be determined by experiments. In this case user must enter the c_{uk} shear strength.

Passive earth pressure

If this option is turned on the sliding resistance is increased taking into account the passive soil pressure. Active soil pressure increases the horizontal forces. These effects are usually neglected to be on the safe side. Activating this option requires extra watchfulness.
$\gamma_{m,EP}$ is the mobilization factor of passive earth pressure.

Checks

The different verification methods can be configured on the *Checks* tab. The verification of bearing resistance and the eccentricity are always checked (cannot be ignored).



Bearing resistance

Bearing resistance is calculated according to EN 1997-1 Annex D.
Drained condition:

$$R_{V,d} = \left(c' \cdot N_c \cdot b_c \cdot s_c \cdot i_c + q' \cdot N_q \cdot b_q \cdot s_q \cdot i_q + 0{,}5 \cdot \gamma' \cdot B' \cdot b_\gamma \cdot s_\gamma \cdot i_\gamma\right) \cdot A'$$

Undrained condition:

$$R_{V,d} = \left((\pi + 2) c_u \cdot b_c \cdot s_c \cdot i_c + q\right) \cdot A'$$

Allowable bearing pressure

The calculated stress is compared with the given value of allowable bearing pressure.

Examination of layered subsoil

If there are several different soil layers under the footing the critical layer may be a deeper one with less resistance. A utilization is calculated from the stress and resistance at the top of the layer using the slope of the load spread under the footing and the following formulas:

$$q_{E,d,i} = \frac{V_d}{A_i'} + (q_i' - q')$$

where

$A_i' = B_i' \cdot L_i'$

$B_i' = B' + 2 \cdot Cot\beta \cdot (|z_i| - D)$ and $L_i' = L' + 2 \cdot Cot\beta \cdot (|z_i| - D)$

$|z_i| - D$ is the distance between the bottom of the footing and the top of the layer

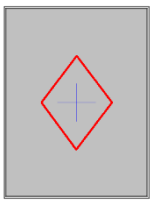
In the case of design, the program optimizes the footing sizes using an iteration process until the utilization falls below maximum value allowed.

$$\lambda_{R,v} = \frac{V_d}{R_{V,d}} \leq \lambda_{R,v,lim}$$

The program considers the maximum sizes as upper limits in the iteration. Therefore, it may happen the foundation does not meet the requirements with the maximum sizes.
If the calculated size of the footing is greater than the thickness of the foundation, the program shows a warning message.

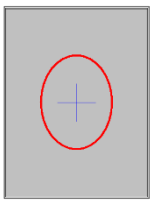
For rectangular footings $\gamma_{ecc} = 0.5$ means that the the bounding rectangle of the eccentricity limit curve has the dimensions $b_x/2$ and $b_y/2$, where b_x and b_y are the footing dimensions. The eccentrity factor is calculated according to the *Type of eccentricity limit:*

Linear (diamond),



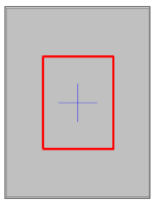
$$\gamma_{ecc} = \left|\frac{e_x}{b_x}\right| + \left|\frac{e_y}{b_y}\right|$$

Quadratic (ellipse)



$$\gamma_{ecc} = \sqrt{\left(\frac{e_x}{b_x}\right)^2 + \left(\frac{e_y}{b_y}\right)^2}$$

Axis of rotation (rectangle)



$$\gamma_{ecc} = \max\left(\frac{|e_x|}{b_x}; \frac{|e_y|}{b_y}\right)$$

For circular footings the eccentricity limit curve is a circle, $\gamma_{ecc} = e/r$, where $r = d_p/2$

Eccentricity check

The checkbox of the Eccentricity design can be switched off independently of the state of the main lock. Design procedure: footing size is increased until the eccentricity factor (γ_{ecc}) calculated from load combination eccentricities falls below the allowed limit: $\gamma_{ecc} \leq \gamma_{ecc,lim}$.

Stability

AXISVM calculates the moment of actions around the axes of overturning, sums up the stabilizing and destabilizing moments then checks the following:

$$\Delta_{EQU} = \left|\frac{M_{dst}}{M_{stb}}\right| \leq \Delta_{EQU,lim}$$

Uplift

If upward (tension) force acting on the footing, the program checks whether the self-weight of the footing can counterbalance the internal force: $G_{dst} \leq G_{stb}$

Sliding of the footing (or the blind concrete) on the soil

It is checked whether the horizontal force applied to the footing causes sliding failure between the footing and the soil using the following formula:

$$H_d \leq R_{d,Hs} + R_{p,d}$$

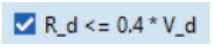
Where the sliding resistance is

Drained condition: $R_{d,Hs} = V_d tan\delta_d$

For cast-*in-situ* concrete the design value of the structure-ground interface friction angle can be considered equal to the critical state angle of the shearing resistance: $\delta_d = \varphi_{cv,d}$

Undrained condition: $R_{d,Hs} = A'c_{u,d}$ and $R_{d,Hs} \leq 0{,}4V_d$*

* If water or air can get between the footing and the soil EN 1997-1 6.5.3. (12)P requires an additional check. It can be activated on the *Soil* tab by checking the following option



Sliding of the footing on the blind concrete

It is checked whether the footing slides on the blind concrete using the following formula:

$$H_d \leq R_{d,Hb} + R_{p,d}$$

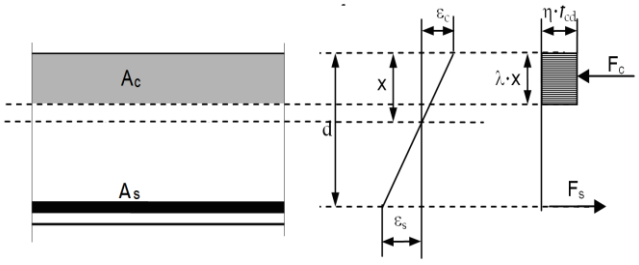
The sliding resistance being $R_{d,Hb} = V_d \frac{\mu_{cc}}{\gamma_\mu}$

where μ_{cc} is the friction coefficient between the footing and the blind concrete, γ_μ is the partial factor for the friction coefficient.

Reinforcement of the foundation base plate

If rebar positions and diameter are specified the module determines the necessary amount of top and bottom reinforcement in *x* and *y* direction according to the following diagram.

The minimum requirement is always taken into account.



The necessary rebar spacing is calculated from the rebar diameter.

Warnings and errors:
The program sends a warning if compression reinforcement is required or the calculated amount is more than the maximum allowed ($A_s > A_{s,max}$).

Calculating according to Eurocode 7

Eurocode 7 allows different design approaches (DA). These are certain combinations of partial factors for actions, material properties and resistances. Partial factor sets applied to actions are referred to as A1, A2, sets applied to material properties are M1, M2, sets applied to resistances are R1, R2, R3. (See *EN 1997-1, Annex A*) Each design approach combine these partial factor sets.

Design Approach		*Combination*	*Actions*	*Material properties*	*Resistances*
DA1	Combination 1	ULS	A1	M1	R1
	Combination 2	SLS	A2	M2	R1
DA2		ULS	A1	M1	R2
DA3		SLS	A2	M2	R3

The program checks A1+M1+R1 (DA1 / 1) and A1+M1+R2 (DA 2) for critical ULS combinations, A2+M2+R1 (DA1 / 2) and A2+M2+R3 (DA3) for critical SLS combinations.
So for each critical combination two results are calculated.
If design was performed for a user-defined load combination set this combination to ULS or SLS otherwise the footing may be overdesigned.
Bearing resistance is $q_{Rd} = s_\gamma \cdot \gamma' \cdot B' \cdot N_\gamma \cdot i_\gamma \cdot b_\gamma \cdot 0.5 + s_q \cdot q \cdot N_q \cdot i_q \cdot b_q + s_c \cdot c \cdot N_c \cdot i_c \cdot b_c$

Punching check

The module checks the shear resistance of the foundation ($v_{Rd,max}$), at the perimeter of the column and determines the necessary amount of shear reinforcement.
The calculation reduces the punching force by the soil reaction on the effective area (and within the critical punching line).
The punching check is passed if $v_{Ed} \le v_{Rd}$
Without shear reinforcement $v_{Rd} = \min\begin{Bmatrix} v_{Rd,c} \\ v_{Rd,max} \end{Bmatrix}$, with shear reinforcement $v_{Rd} = \min\begin{Bmatrix} v_{Rd,cs} \\ v_{Rd,max} \end{Bmatrix}$.
Warning and errors:

If $v_{Ed} \le v_{Rd,c}$, no shear reinforcement is necessary.
If $v_{Rd,max} > v_{Ed} > v_{Rd,c}$, shear reinforcement is necessary.
If $v_{Ed} > v_{Rd,max}$, the base plate fails due to punching. Plate thickness of column cross-section size should be increased.

If a stepped or sloped footing is designed, the size of the pedestal is determined checking the punching requirements so utilization for punching is not calculated.

Predicting the settlement of footing

AXISVM calculates the elastic settlement caused by additional stress in soil layers.
Loads cause the following stress at depth of *z* under the center of the centrally loaded rectangle of the footing (after *Boussinesq-Steinbrenner*):

$$\sigma_{z,a} = \sigma_{z,I} + \sigma_{z,II} + \sigma_{z,III} + \sigma_{z,IV}$$

$$\sigma_z = \frac{\sigma_0}{2\pi}\left\{\arctan\left[\frac{b}{z}\cdot\frac{a(a^2+b^2)-2az(R-z)}{(a^2+b^2)(R-z)-z(R-z)^2}\right]+\frac{bz}{b^2+z^2}\cdot\frac{a(R^2+z^2)}{(a^2+z^2)R}\right\}$$

where the distance between the characteristic point and the central axes are *0.37B*' and *0.37L*'.
a and *b* are the dimensions of the four parts of the loaded rectangle according to the following table:

	a	b
I.	$(0.5-0.37)\cdot L'$	$(0.5-0.37)\cdot B'$
II.	$(0.5+0.37)\cdot L'$	$(0.5-0.37)\cdot B'$
III.	$(0.5+0.37)\cdot L'$	$(0.5+0.37)\cdot B'$
IV.	$(0.5-0.37)\cdot L'$	$(0.5+0.37)\cdot B'$

If $a < b$, the two values are swapped.

σ_0 is the soil stress at the footing base plane caused by loads (including the self-weight of the footing and the backfill minus the weight of the removed soil above the base plane), and $R = \sqrt{a^2+b^2+z^2}$.

This stress calculation is valid for a homogeneous half space. In case of soil layers effective layer thicknesses must be calculated:

$$h_{i,eq} = h_i \cdot \left(\frac{E_{oed,i}}{E_{s,ref}}\right)^{1/3}$$

where

$h_{i,eq}$ is the effective thickness of the soil layer i
h_i is the thickness of the soil layer i
$E_{oed,ref}$ is the Young modulus of the reference soil layer
$E_{oed,i}$ is the Young modulus of the soil layer i

AXISVM breaks up the user defined soil layers into 10 cm sublayers and calculates the stress due to soil weight and the stress caused by loading at the bottom of the sublayer. The change in sublayer thickness is calculated according to the following formulas:

$$\Delta h_i = h_i \frac{\sigma_{ai}}{E_{oed,i}};\ \ \sigma_{ai} = \frac{\sigma_{i-1} + \sigma_i}{2}$$

σ_{ai} is the average stress caused by loading in sublayer i
σ_{i-1} is the average stress caused by loading at the top of sublayer i
σ_i is the average stress caused by loading at the bottom of sublayer i
$E_{oed,i}$: the Young modulus of the sublayer i

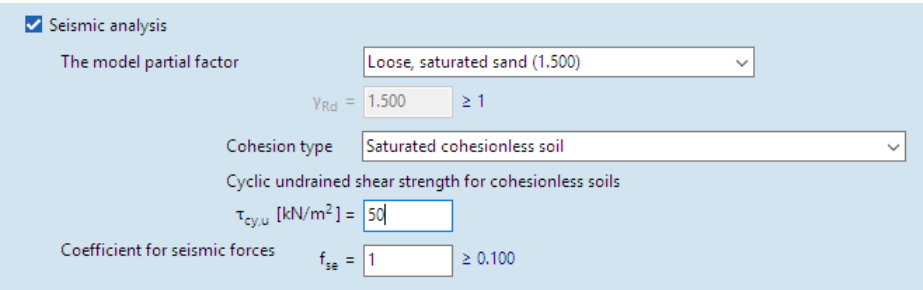
The predicted settlement at a given depth is calculated as the sum of the changes in sublayer thicknesses for the sublayers above the level:

$$s_m = \sum_{i=0}^{m} \Delta h_i$$

AXISVM calculates the limit depth, where $\sigma = 0.2 \cdot \sigma_{ob}$ (i.e., the extra stress caused by loading falls under the 20% of the stress due to soil self-weight.
If this condition is not met at the bottom of the layer structure a settlement estimation is made based on the settlement at this point and the stress ratio (> 0.1) is calculated.
If the stress caused by loading at the footing base plane is smaller than the stress due to the original soil layers settlement is not calculated.
AXISVM calculates the settlement for all load cases and SLS combinations. Stress and settlement functions are displayed for the selected load case. Settlement function *s(z)* is the total settlement of layers above *z*.

Seismic

If vibration analysis is completed and the seismic loads are generated seismic check of footings can be performed. Enter the following parameters on the *Seismic* tab:



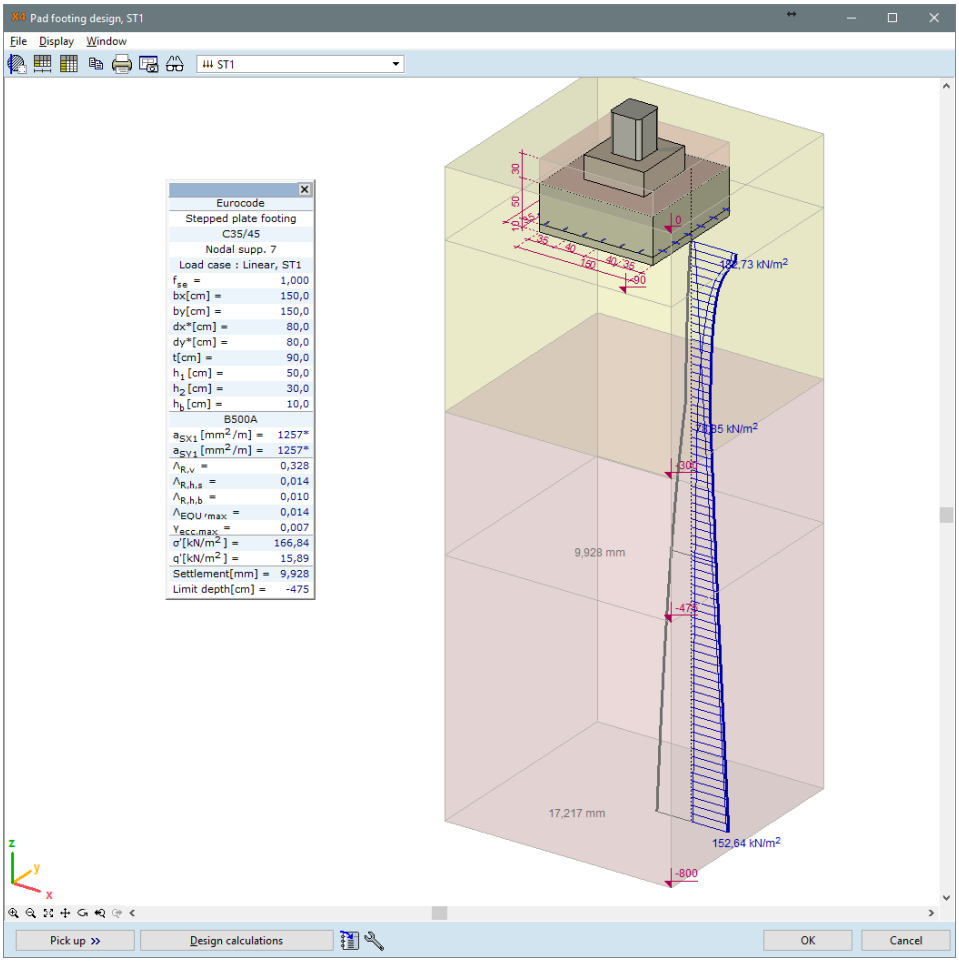
The seismic resistance of the footing is verified according to EN 1998-5 Annex F. Especially the value of the *Cyclic undrained shear strength for cohesionless soil* ($\tau_{cy,u}$) must be determined carefully.

SIA

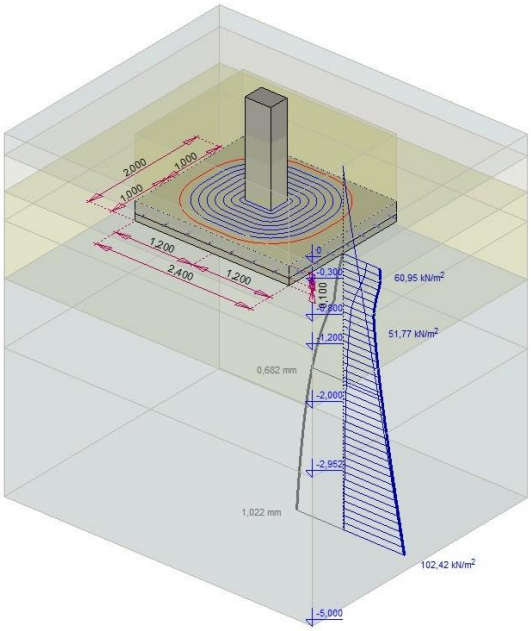
Geotechnical calculations are performed according to EN 1997-1 using the safety factors according to SIA. Structural calculations follow SIA 262, punching analysis is performed according to **6.5.15 Punching analysis – RC3 module.**

Results

The designed foundation will be displayed in top view with soil layers, punching circles and places dimension lines automatically. The 3D model can be zoomed in and out, shifted and rotated just like the main model.

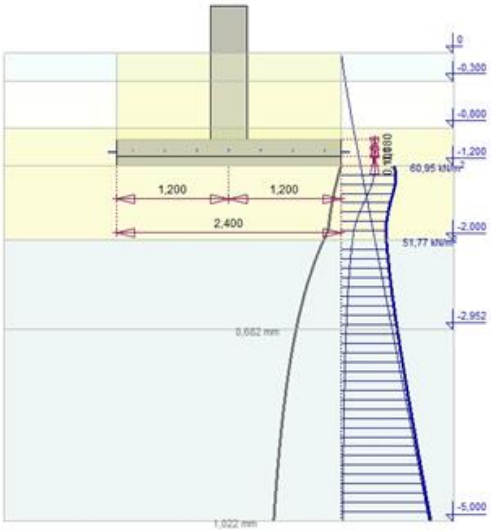


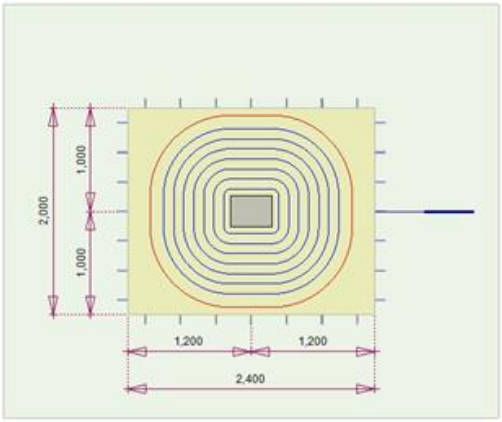
If the display of settlement is activated (see *Display parameters)* a thick blue diagram plots the total soil stress against depth. Thin diagrams show the stress due to loading and the self-weight of the soil. The first one is decreasing, the second one is increasing with depth. Horizontal lines show the sublayers. The grey diagram on the other side of the axis is the settlement function.



The settlement displayed in the info window is the value of the settlement function at the limit depth (where the stress caused by loading is 10% of the stress due to self-weight for the soil).

If this condition is not met at the bottom of the layer structure a settlement estimation is made based on the settlement at this point and the stress ratio (> 0.1) is calculated.





If stress caused by loading at the bottom of the layer structure is still more than 10% of the stress due to soil self-weight the limit depth cannot be determined as the further structure of the soil is unknown. In this case the info window displays the value of the settlement function at the bottom of the layer structure as >*value*.
To improve the estimation further soil layer information must be added.

Footing internal forces

This table displays the forces of the selected supports and the most important results including calculated geometry.
As support forces are calculated in the local system of the support the *x* and y directions are the local *x* and y directions of the support. If the supports are global, these are the global X and Y directions.

	$R_x, R_y, R_z,$ R_{xx}, R_{yy}, R_{zz}	support forces
	Type	the combination type (ULS or SLS) of custom load combinations
	Design approach	according to EN 1997-1, Section 2.4.7.3.4.
	γ_G	partial factor of the weight of blind concrete, footing and backfill
	V_d	design vertical action at the bottom of the foundation including the weight of the blind concrete, foundation and backfill)
	R_d	design bearing resistance
	q_{Ed}	design bearing pressure (V_d/A')
	R_d/A'	specific design bearing resistance
	$\Lambda_{R,V}$	soil utilization factor (V_d/R_d)
	$\Lambda_{R,V,max}$	soil utilization factor (the effect of layered subsoil)
	a_{sx1}	local *x* direction bottom reinforcement (if calculated)
	a_{sy1}	local *y* direction bottom reinforcement (if calculated)
	a_{sx2}	local *x* direction top reinforcement (if calculated)
	a_{sy2}	local *y* direction top reinforcement (if calculated)
	$\Lambda_{R,H,s}$	utilization based on blind concrete sliding on the soil
	$\Lambda_{R,H,b}$	utilization based on foundation sliding on the blind concrete
	Λ_{EQU}	utilization for stability
	Λ_{UPL}	utilization for uplift
EC, NTC	$v_{Ed0}/v_{Rd,max}$	
EC, NTC	$v_{Ed}/v_{Rd,c}$	
EC, NTC	$v_{Ed}/v_{Rd,cs}$	utilization based on punching (for simple plate footings)
SIA	V_d/V_{Rd}	
SIA	$V_d/V_{Rd,c}$	
SIA	$V_{d,s}/V_{Rd,s}$	
	Settlement	predicted settlement of the footing, with a > before, if the depth of the soil profile is smaller than the limit depth
	Load case	

Detailed internal forces Displays the data in the table of *Footing internal forces* plus the following results:



	e_x, e_y	load eccentricity in *x* and *y* direction
	B'	effective foundation width
	L'	effective foundation length
	A'	effective area of the foundation
	H_d	design value of the horizontal action
	$R_{d,H,s}$	sliding resistance between the soil and the bottom of the foundation
	$R_{d,H,b}$	sliding resistance between the blind concrete and the footing
	γ_{ecc}	eccentricity factor
	M_{dst}	destabilizing moment
	M_{stb}	stabilizing moment
	V_{dst}	design value of the combination of permanent and accidental vertical destabilizing actions
	G_{stb}	design value of the stabilizing permanent vertical actions
	Rebars x_a	rebar scheme in bottom *x* direction (if calculated)
	Rebars x_f	rebar scheme in top *x* direction (if calculated)
	Rebars y_a	rebar scheme in bottom y direction (if calculated)
	Rebars y_f	rebar scheme in top y direction (if calculated)
	ΔV_{ed}	the net upward force within the control perimeter considered
	u_0	length of the perimeter of the load area
	u_1	length of the basic control perimeter
EC, NTC	v_{Ed0}	maximum shear stress at the u_0 column perimeter
EC, NTC	v_{Ed}	maximum shear stress at the control perimeter
EC, NTC	$v_{Rd,max}$	maximum design shear resistance without punching shear reinforcement
EC, NTC	$v_{Rd,c}$	minimum design shear resistance without punching shear reinforcement
EC, NTC	$v_{Rd,cs}$	design shear resistance with punching shear reinforcement
EC, NTC	A_{sw}	shear reinforcement along the control perimeter
SIA	V_d	design punching force
SIA	$V_{d,s}$	$V_d - V_{Rd,c} > V_d/2$
SIA	$V_{Rd,max}$	maximum design shear resistance without punching shear reinforcement
SIA	$V_{Rd,c}$	design value of punching resistance based on concrete compressive strength
SIA	$V_{Rd,s}$	SIA 262 4.3.6.5.4
	Stress ratio	ratio of stress caused by loading and the stress due to self-weight of the soil (if limit depth is below the bottom of the layer structure its value is determined at that point and is greater than 0.1, otherwise it is 0.1)



Copies image to the Clipboard



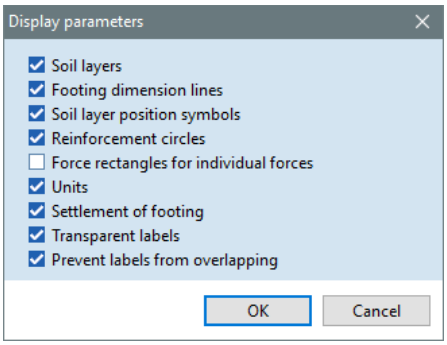
Prints image to the Clipboard



Saves the drawing into the Drawing Library



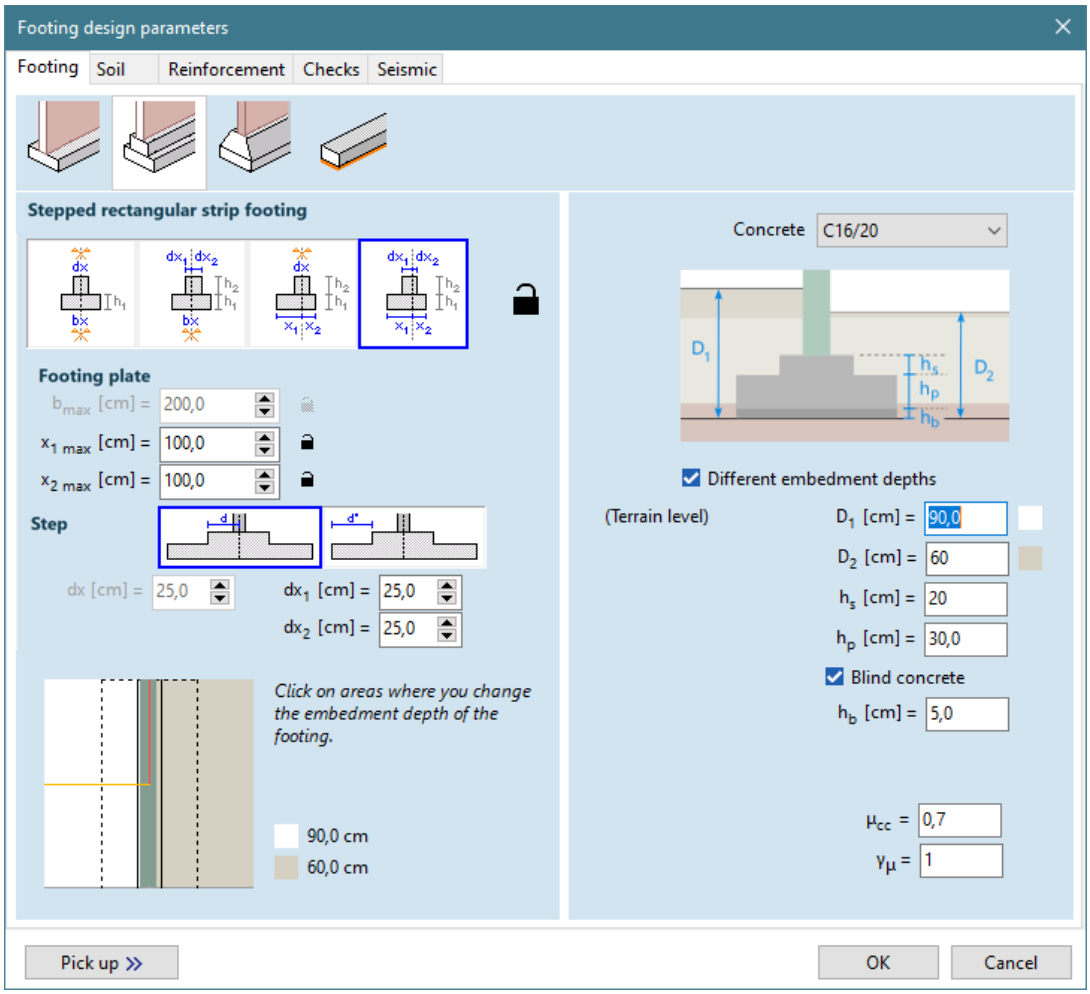
Display parameters
Turns on and off symbols of the drawing.



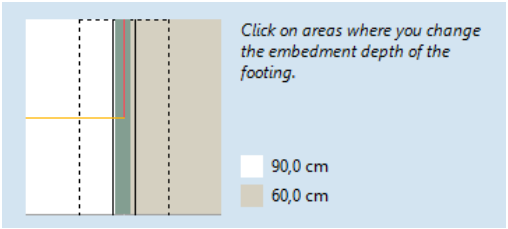
6.5.16.2 Strip footing design



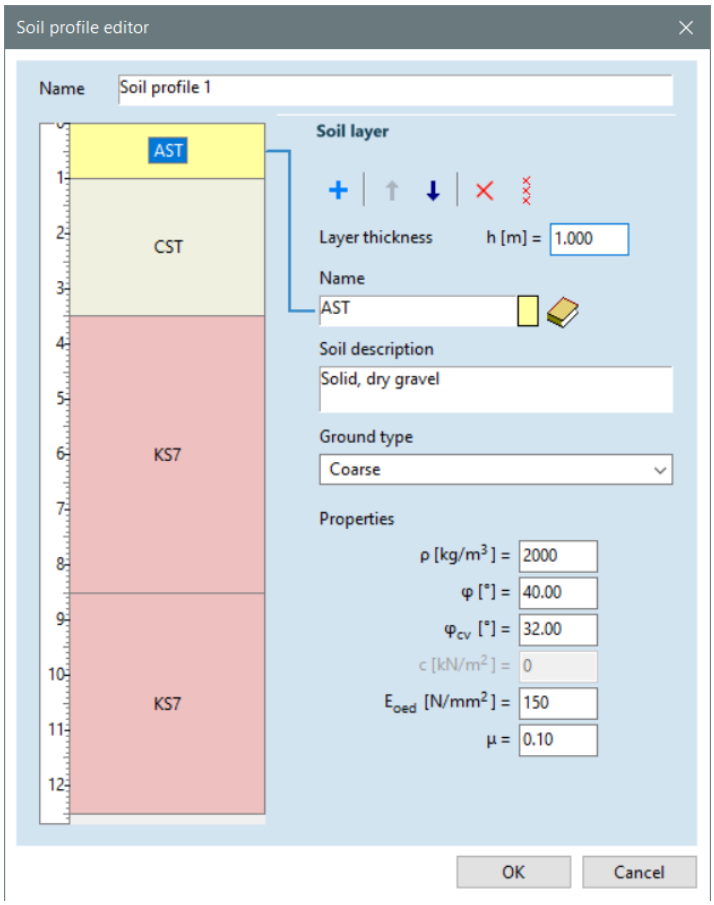
AXISVM can determine the necessary size and reinforcement of strip foundations (with or without pedestal), and can check the footing against sliding and punching according to Eurocode7. It determines the settlement of the foundation as well.
Strip footing design is similar to the pad footing design. Parameters describing the geometry of the strip section must be entered.
If there is at least one borehole sample in the model (**4.9.3 Add borehole samples – SOIL module**), it is possible to select *Interpolated* soil profile in the same way as for pad footings. In this case, the soil characteristic can vary along the strip footing. For this reason, the program will only show the maximum utilization along the footing.



If the option *Different embedment depths* is enabled, the footing divides the area into two halves and clicking on the left or right side changes the embedment depth there. Left or right is defined by the local *y* direction of the support (yellow axis on the diagram).



6.5.16.3 Soil profile editor



The editor can be used to edit or create a soil profile.
The *Name* field can also be used to rename an existing soil profile. After editing a soil profile of the model (or the library) you can choose to overwrite the original profile or add a new one to the model (or the library). Click on the soil diagram on the left to select a layer (or use the Up and Down keys). The parameters of the selected layer can be edited on the right.

Add a new soil layer
Adds a new layer at the bottom with the current thickness and parameters. The position of the layer can be set using the arrows.

Move up
Moves the layer to the top of the layer above.

Move down
Moves the layer to the bottom of the layer below.

Delete layer
Deletes the selected layer from the soil profile.

Delete all
Deletes all layers.

Soil layer properties

Enter *Layer thickness (h),* a short layer name, the layer colour (click on the coloured rectangle to edit), and a description of the soil. The soil profile diagram displays only the short name.

Soil parameters used in the calculations are:

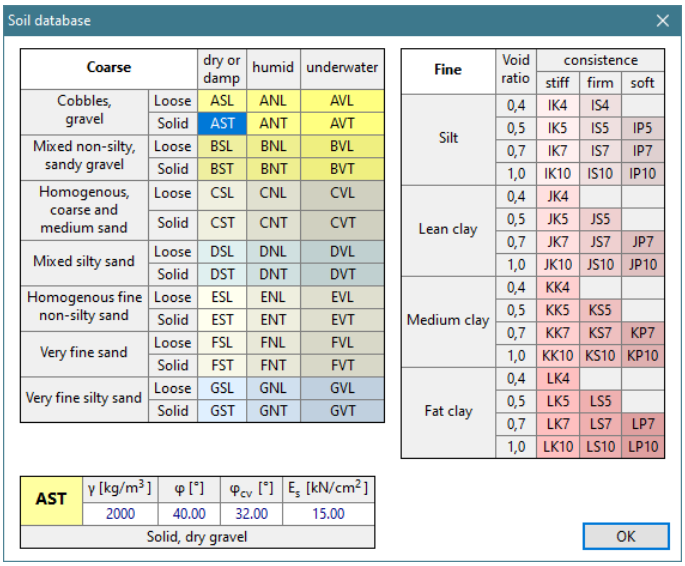
Soil type	Coarse, Coarse underwater, or Fine
γ [kg/m3]	Mass density
φ [°]	Internal angle of friction
φ cv [°]	Critical state angle of the shearing resistance (used in sliding check)
c [kN/m2]	Cohesion (only for fine soils)
E_{oed}	Compression modulus

Soil database

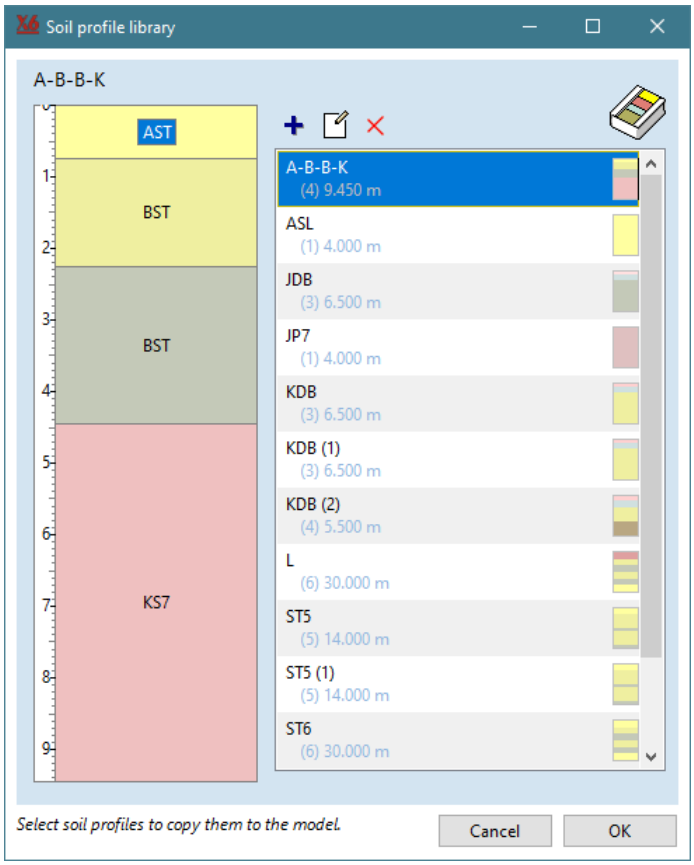


Clicking on the *Soil database* button opens a dialog to select a predefined material. The properties of the selected material is displayed at the bottom.

Clicking on *OK* or double-clicking on a cell copies the parameters of the selected soil to the current layer.



6.5.16.4 Soil profile library



Soil profiles saved in the Soil profile library are available from any model.
The list is in alphabetical order. The number of layers is shown in parentheses below the name of the profile, followed by the depth of the soil profile. A preview of the profile is displayed on the right.
Multiple selection is available in the usual way (Ctrl + click to add individual items to the selection, Shift + click to select everything from the selected item).



Create a new soil profile
The **Soil profile editor** is started and the layers built there are added to the library.



Edit soil profile
Allows editing in the *Soil profile editor*. The existing profile can be overwritten or the modified profile can be added to the library.



Delete soil profile from the library
Removes the selected profiles from the library.

6.5.17 Design of reinforced concrete cores and walls – RC5 module



Using RC5 module, reinforcement can be assigned to reinforced cores and walls and the design check of cores/walls subjected to bending and axial force can be performed.
Reinforcement can be assigned to virtual beams or virtual strips (see after clicking the icon on the left or clicking on the axis of a virtual beam/strip. The icon is visible if there is reinforced concrete domain, there is at least on virtual beam/strip and statical or dynamical results are available in the model.

After clicking on the icon, one virtual beam or strip can be selected in the course of selection procedure (see **2.16.1 Selection**). If the selection is invalid, the software warns with a warning message. The selection may be invalid in the following cases:

- Neither virtual beam, nor virtual strip has been selected.
- More than one virtual beam/strip has been selected.
- The selected virtual beam/strip has non-concrete domain(s).
- The selected virtual beam/strip is not continuous.
- The cross section of the virtual beam/strip is not supported (e.g., variable section).
- The selected virtual beam/strip is not vertical (max. 15° deviation is allowed).
- Automatic cross section is set to the selected virtual beam/strip but a rib connected to one of the domains is not concrete.

After a valid selection, the *Reinforced concrete cores and walls* window is opened where storeys and reinforcement can be assigned to cores and walls and the design check can be performed. Altogether, there are three tabs on the window: 1) Virtual beams; 2) Virtual strips; 3) Overall utilization.

Virtual beams can be used to design reinforced concrete cores with the assumption that sections remain approximately plane before and after loading. This assumption is likely not valid in case of large and short cores and in case of the presence of considerable warping torsional moment and a section that is sensitive to warping effects.
The module can check concrete wall sections (virtual strips) or cores (virtual beams) subjected to bending with axial force (*N-M* interaction). The shear check is available only for virtual strips, but the shear utilization cannot be summed up with the *N-M* (bending with axial force) utilization. In the case of complex cores, the shear check can only be performed for the separate virtual strips.
For eccentric internal forces *(N-M)* the design check of reinforced concrete cores is performed similarly to the design check of reinforced concrete columns. The N_x-M_y-M_z strength interaction diagram of the section is calculated and it is checked whether the point representing $N_{x,Ed}$, $M_{y,Ed}$, $M_{z,Ed}$ design forces is within the interaction diagram. During design check of core reinforcement, buckling length coefficient cannot be defined, since modern design codes (e.g., EN 1992-1-1) includes methods wherewith the effect of **imperfections** and **second order effects** can be taken into account on structural level. **In case of reinforced concrete cores, it is assumed that the integrated design internal forces include the effect of imperfections and second order effects.**

Virtual strips can be used by the design check of wall ends or wall segments considering possible buckling failures of the wall between floors. The program is able to calculate the additional eccentricities from imperfections and second order effects regarding a storey, based on the given buckling length coefficient. After sum with the initial eccentricities (calculated from the integrated internal forces), the program calculates the $N_{x,Ed}$, $M_{y,Ed}$, $M_{z,Ed}$ design forces. **It is assumed that the integrated design internal forces include the effect of imperfections and second order effects on structural level.**
The design check is performed similarly to the case of reinforced concrete cores that the N_x-M_y-M_z strength interaction diagram of the section is generated and it is checked whether the point representing $N_{x,Ed}$, $M_{y,Ed}$, $M_{z,Ed}$ design forces is within the interaction diagram.
On the overall utilization tab, design results of reinforced concrete cores calculated with virtual beams and design results of wall ends/segments calculated with virtual strips can be summed.

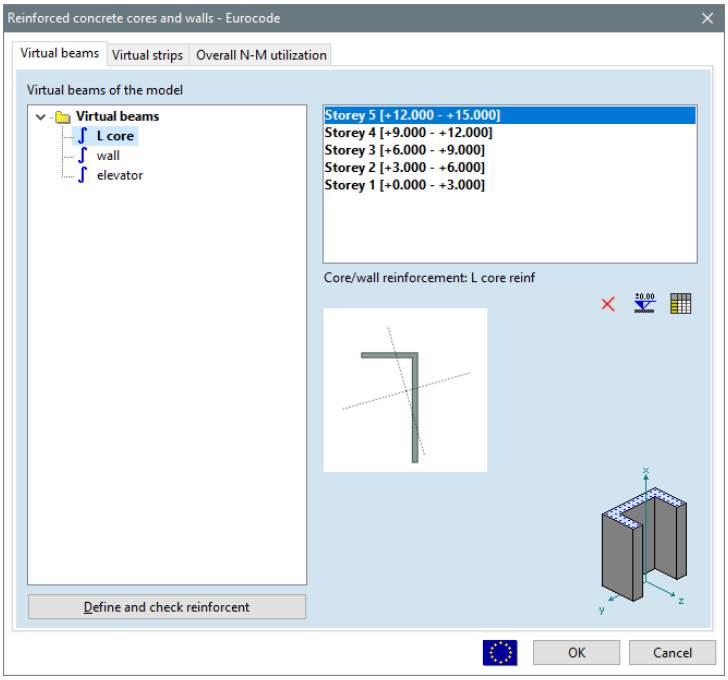
It may be necessary because the N_x-M_y-M_z strength interaction diagram of cores is generated without consideration the loss of stability of compressed wall ends and inner wall segments. The program supports the use of various summation rules as it can be seen in the following figures.

☞ **In case of warping-sensitive cores and the presence of considerable warping torsional moments, the section warps and it does not remain plane after loading. It is recommended to Ms perform the design check for wall segments of the core separately in such cases because the section cannot be designed as a whole.**

Virtual beams

Virtual beams of the model

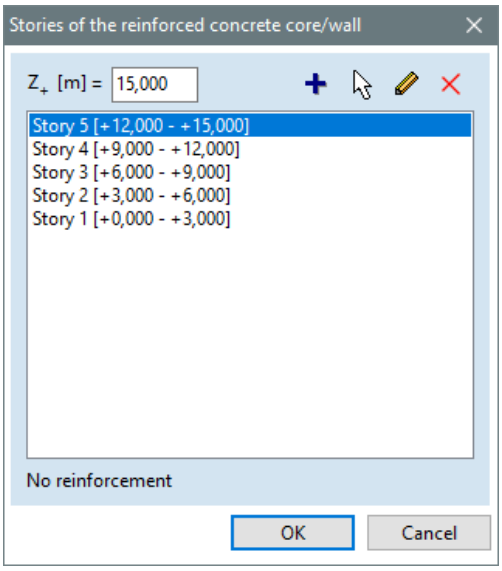
Virtual beams in the model are listed on the left. If at least storeys are assigned to the virtual beam, the name of the virtual beam is written bold letters. After clicking on a virtual beam written with bold letters, the storeys of the selected virtual beam are listed in the list-box on the right of the window.
If actual reinforcement is assigned to the storey, the storey is shown with bold letters.



If a virtual beam is selected which has no storeys, the list on the right remains empty and only the middle button is visible below the list. After clicking on this button, a new window opens where new storeys can be added, storeys can be deleted and modified.



Before definition of actual reinforcement, storeys need to be assigned to the virtual beams/strips. Using storeys, more economical and efficient reinforcement can be assigned to the virtual beam that follows the change in the internal forces.
Story needs to have intersection with the axis of the selected virtual beam/strip. The software fills the list of storeys with predefined model storeys (see **3.3.4 Storeys**) if there are any. The storeys assigned to a virtual beam or strips does not necessarily need to be identical with the storeys of the model.



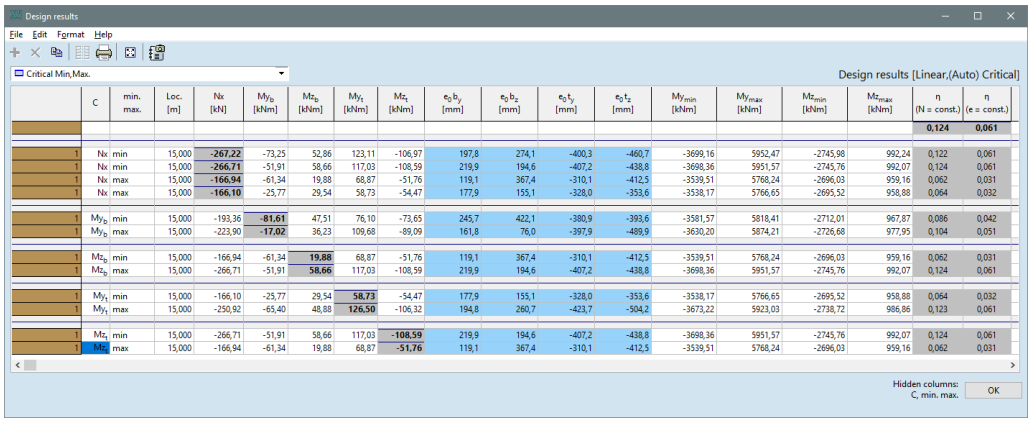
The storeys can be deleted (delete icon), modified (pencil) and new storeys can be added (+ and arrow). The storeys are saved to the virtual beam/strip. If storeys are assigned to the virtual beam/strip, the cross section will be displayed together with circles on the axis of virtual beam/strip indicating the boundaries of the defined storeys.



Reinforcement can be deleted from the selected storey with this button. It is visible only if reinforcement is assigned to the storey.



By clicking on table button, the design results of the selected storey of the selected virtual beam can be seen summarized in a table (see below). This button is visible only if reinforcement is assigned to the storey.



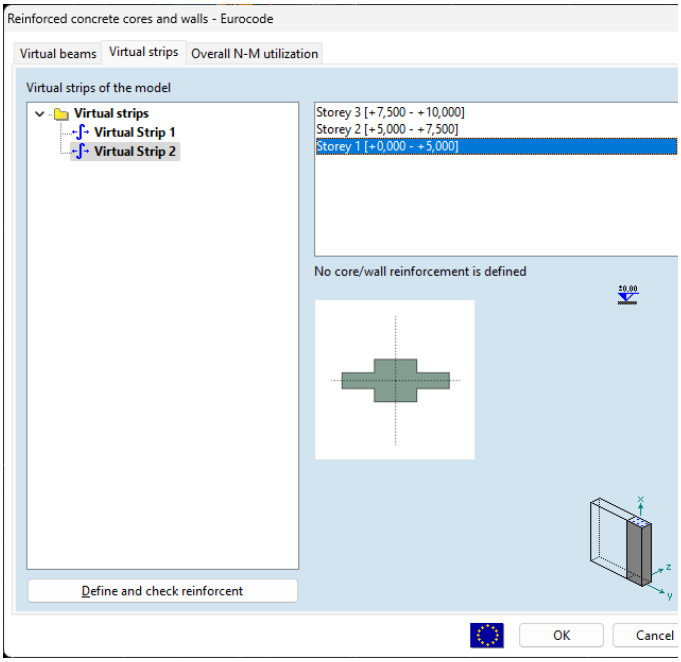
Define and check reinforcement

After clicking on the *Define and check reinforcement* button (or double clicking on a storey), a new dialog opens (see **6.5.17.1 Defining reinforcement**) where new reinforcement can be assigned to the storey or an existing reinforcement can be modified.

☞ **Instead of clicking on *Define* and check reinforcement button, alternatively the window for define and check reinforcement opens if you double-click on the selected storey.**

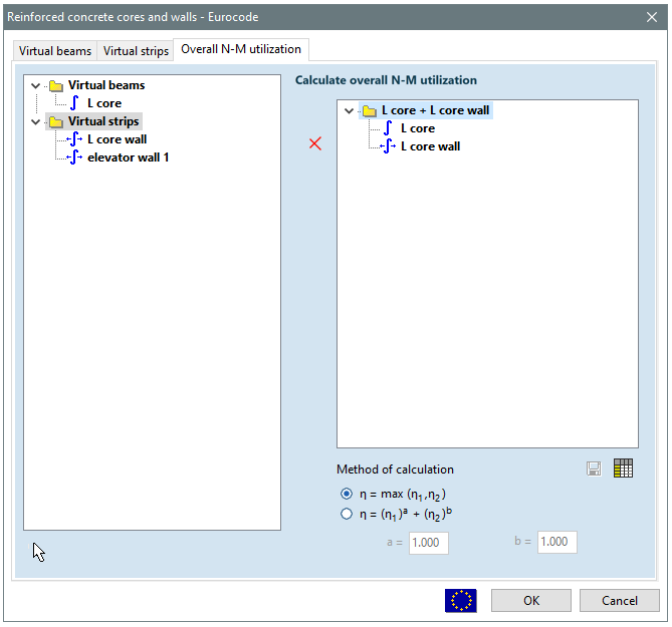
Virtual strips

On *Virtual strips* tab, the same functionalities and buttons are available than on Virtual beams tab. storeys and reinforcement can be assigned to virtual strips that can be used to model and design walls ends and segments.



Overall N-M utilization

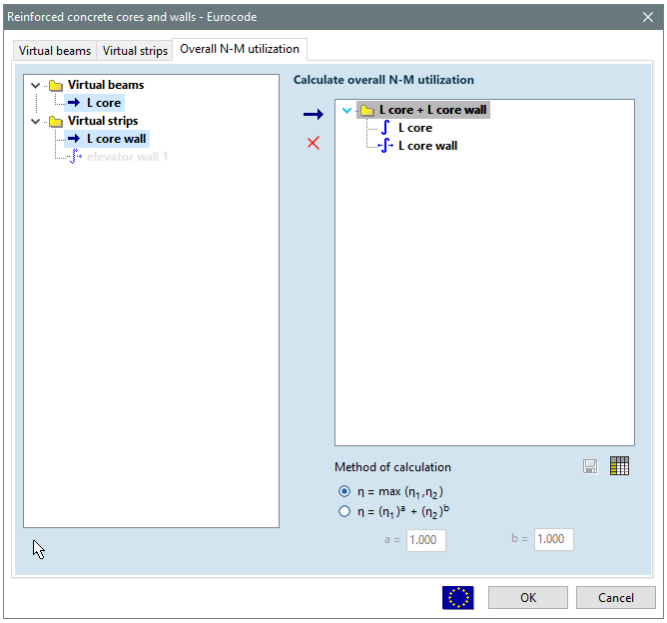
On *Overall N-M utilization* tab, virtual beam – virtual strip pairs can be created, deleted and modified that are used to calculate the overall utilization of reinforced cores and walls.



Virtual beams and virtual strips in the model

Virtual beams and virtual strips in the model that have storeys and reinforcement are listed on the left. If actual reinforcement is missing from at least one storey, the name of the virtual beam/strip is written with red letters. In this case, the overall utilization cannot be calculated for every storey.

The list supports the multiple selection which is available while Ctrl button is holding. To create a virtual beam – virtual strip pair, a virtual beam and a compatible virtual strip need to be selected. If a virtual beam is selected, the incompatible virtual strips will be displayed with light grey color.



☞ **A virtual beam and a virtual strip are compatible if they have the same number of storeys and their storeys are identical.**

Calculation of overall utilization

The created virtual beam – virtual strip pairs are listed on the right, below the *Calculate overall utilization* label.



This button is visible only if a virtual beam and a compatible virtual strip are selected in the list on the left. After clicking, a new virtual beam – virtual strip pair will be added to the list. The name is composed from the names of virtual beam and the virtual strip.



This button can be used to delete virtual beam – virtual strip pair from the list.

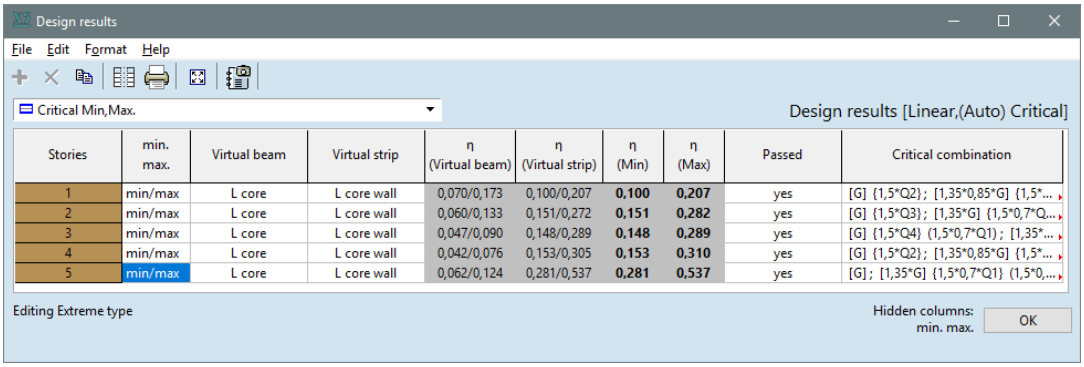


The combination rule of utilizations for the selected virtual beam – virtual strip pair can be defined below the list. The different combination rules are available.

1. $\eta = max(\eta_1, \eta_2)$
2. $\eta = (\eta_1)^a + (\eta_2)^b$

When creating a new virtual beam – virtual strip pair, the software sets the first combination rule automatically. To modify the combination rule of the selected virtual beam – virtual strip pair, after setting the new rule save button need to be clicked.

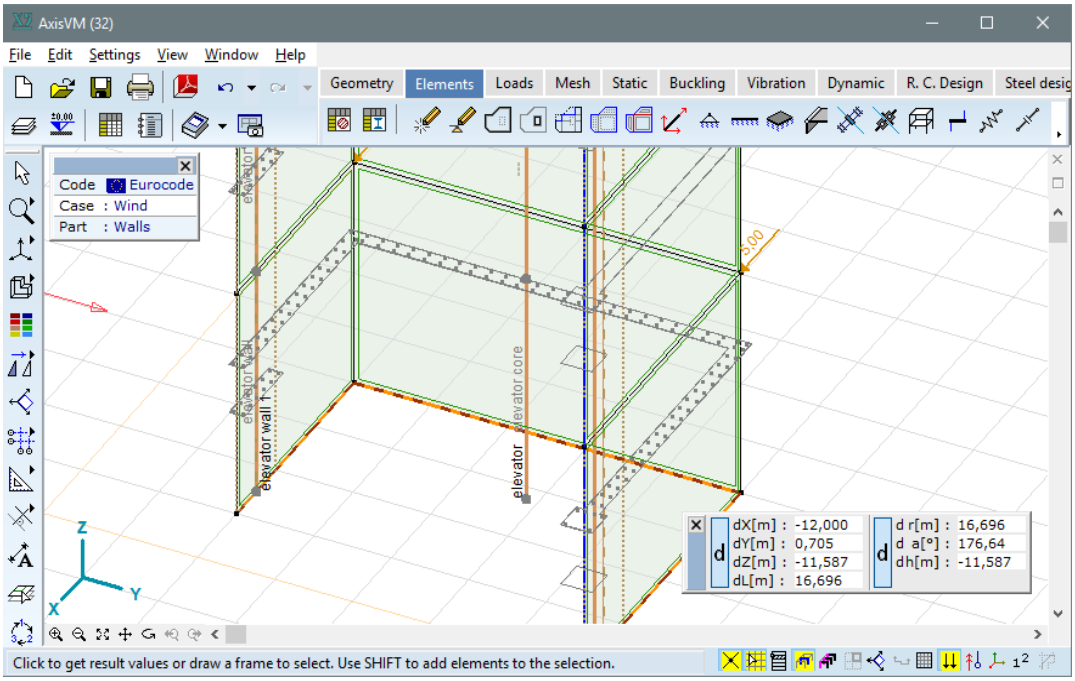
By clicking on table button, the design results of the selected virtual beam – virtual strip pair can be seen summarized in a table (see below).



Stories	min. max.	Virtual beam	Virtual strip	η (Virtual beam)	η (Virtual strip)	η (Min)	η (Max)	Passed	Critical combination
1	min/max	L core	L core wall	0,070/0,173	0,100/0,207	**0,100**	**0,207**	yes	[G] {1,5*Q2}; [1,35*0,85*G] {1,5*...
2	min/max	L core	L core wall	0,060/0,133	0,151/0,272	**0,151**	**0,282**	yes	[G] {1,5*Q3}; [1,35*G] {1,5*0,7*Q...
3	min/max	L core	L core wall	0,047/0,090	0,148/0,289	**0,148**	**0,289**	yes	[G] {1,5*Q4} {1,5*0,7*Q1}; [1,35*...
4	min/max	L core	L core wall	0,042/0,076	0,153/0,305	**0,153**	**0,310**	yes	[G] {1,5*Q2}; [1,35*0,85*G] {1,5*...
5	min/max	L core	L core wall	0,062/0,124	0,281/0,537	**0,281**	**0,537**	yes	[G]; [1,35*G] {1,5*0,7*Q1} {1,5*0,...

Display

If storeys and actual reinforcement are assigned to the virtual beam/strip, the cross section with reinforcement will be displayed together with circles on the axis of virtual beam/strip indicating the boundaries of the defined storeys. The name of the actual reinforcement assigned to the storey is displayed close to the middle of the storey on the axis of the virtual beam/strip.



☞ **With the association of storeys and actual reinforcement to virtual beams and virtual strips, practically, we assign reinforcement to non-existing elements (virtual beam and virtual strip) in order to integrate the stresses in domains and surfaces and to perform design check for the integrated internal forces. The given width of virtual strips or the defined storeys do not necessarily match to boundaries of domains or the finite element mesh. For this reason, reinforcement parameters and actual reinforcement are independent of reinforcement parameters (**see 6.5.1 Surface reinforcement parameters and reinforcement calculation - RC1 module**) and actual reinforcement (**see 6.5.2 Actual reinforcement**) assigned to surfaces. Contradiction between reinforcement parameters and actual reinforcements defined two different ways is not investigated. Rebar statistics in weight report (**see 3.2.13 Weight Report**) does not incorporate actual reinforcements assigned to virtual beams and strips.**

6.5.17.1 Defining reinforcement

Define and check

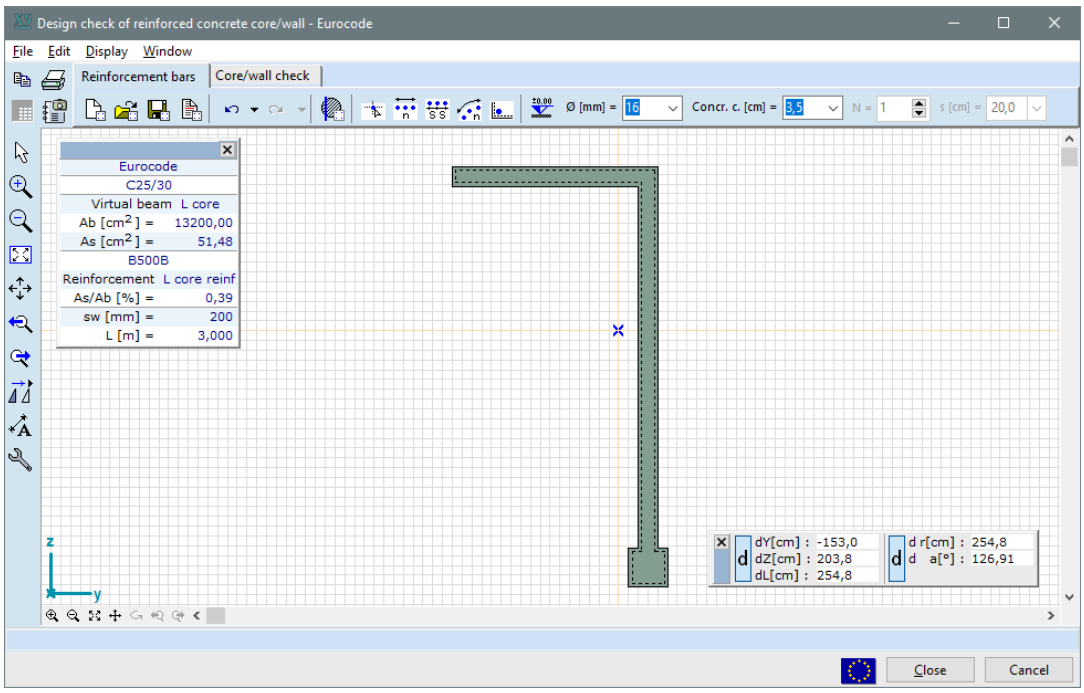
Parameters for reinforcement and actual reinforcement can be specified in the dialog (*Design check of reinforced concrete core/wall*) below. This dialog can be opened after clicking on *Define and check reinforcement* button on Virtual beams and Virtual strips tabs, respectively. The functionalities, dialogs and buttons are very similar to functionalities, dialogs and buttons described and presented by column reinforcement (see **6.5.4 Actual reinforcement against biaxial bending (column)** and **6.5.10 Column reinforcement – RC2 module).** On the page of *Reinfrocement bars*, the steel material, the concrete material, the out-of-plane buckling parameters (only in case of virtual strips) can be specified and rebars with certain diameter can be placed in the cross section.

Cross-section

In this dialog window the design cross-section can be seen. This design cross-section is inherited from the virtual beam/strip. By the definition of virtual beams and strips, the cross-section can be automatically created from the intersected surface elements or a custom cross-section can be defined by the user. The cross-section cannot be modified in this window, it is a property of the virtual beam/strip. The cross-section can be changed only on *Virtual beams* window (see **2.16.19 Virtual beams).**

Local coordinate system

The local coordinate system of the virtual beam/strip plays an important role in the design check procedure due to the fact that the internal forces are integrated from the stresses according to this coordinate system. The orientation of the cross-section displayed in *Design check of reinforced concrete core/wall* window is also consistent with this local coordinate system.



Add drawing to gallery

Saves the current drawing to the Drawings Library.

New reinforcement

Defines a new reinforcement.

Load

Loads previously defined reinforcement.

Save

Saves the reinforcement.

List of reinforcements

List of existing reinforcements.

Parameters

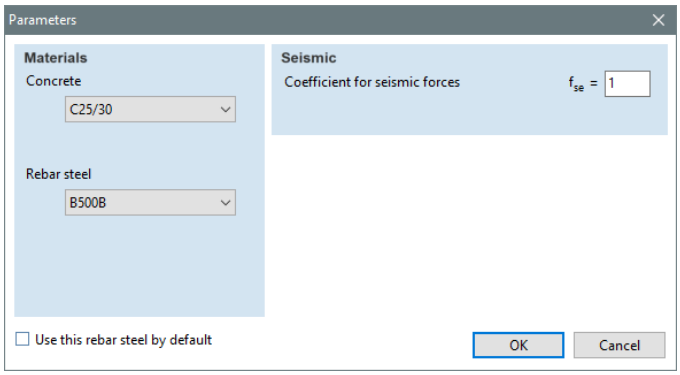
Lets you specify the parameters used in the course of design check.



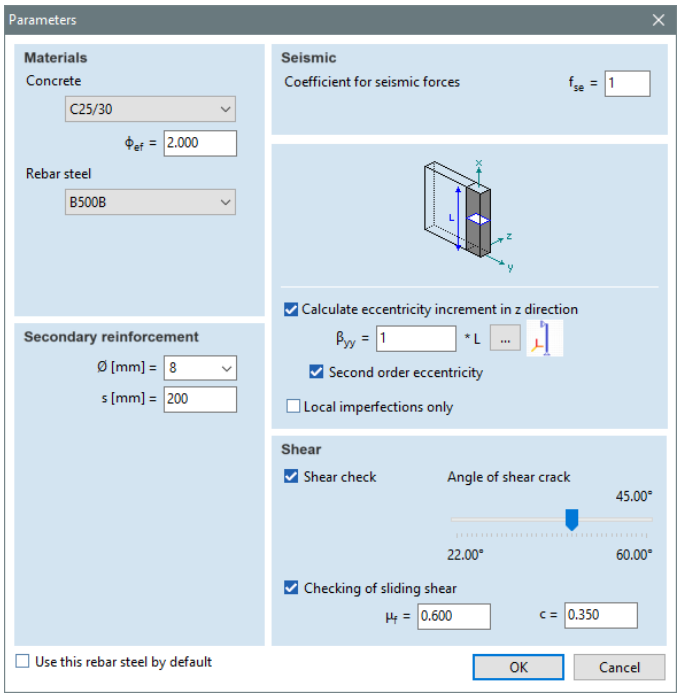
In case of virtual strips, β_{yy} buckling length factor can be specified. (β_{yy} related to buckling in *x-z* plane). It is also possible to select the buckling shape. The buckling length is calculates as the product of buckling length factor and the height of the wall in the selected storey.
The considered concrete class in the calculation can be altered compared to concrete class assigned to the structural element. It allows the user to evaluate structural safety with different concrete class without new static analysis. ϕ_{ef} is the so-called effective creep ratio according to Section 5.8.4 of EN 1992-1-1 standard.
You can see coefficient of seimic forces at **4.10.25 Seismic loads – SE1 module**.

In addition to the above settings, in the case of virtual strips, the diameter and spacing of the secondary reinforcement can be set, and the shear check can be requested. In addition to the shear check of the cross-section, the check of sliding shear can also be selected. The angle of the shear crack can be specified. Selecting sliding shear check, two parameters depending on the roughness of the interface can be set: μ_f and c.

Virtual beam



Virtual strip

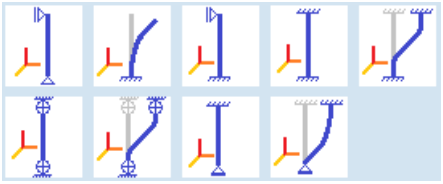


If global imperfections are taken into account by extra loads acting on the structure, they must be ignored in reinforcement calculation. To achieve this, check *Local imperfections only*. If a sway buckling shape is selected only a constant local geometric imperfection is considered (see **6.5.17.3 Calculation of eccentricity**).



Buckling shape

Second order eccentricities are calculated based on the selected buckling shape (see **6.5.17.3 Calculation of eccentricity**). Buckling length factors are changed accordingly, but can be overridden.



To position / by cover



Generates a reinforcement bar with a specified diameter to the location of the cursor.
If the cursor is on a corner or on the contour line, the reinforcement will be placed considering the concrete cover.

Spacing



Inserts evenly *N+1* new rebars between two selected points.

Spacing with constant rebar distance



Inserts new rebars between two selected points considering constant *s* rebar distance. If the distance between to end points is not integral multiple of *s* rebar distance, the distance between the last and the last but one rebar will be less than *s*.

Spacing on circular arch



Inserts evenly *N+1* new rebars between a selected starting point and an end point of a circular arch.
If an entire circle is drawn, the program inserts N rebars evenly around the circle.

Placing rebars at the corners



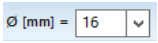
Places rebars in positive and negative corners (> 5°) if there is no rebar in the corner.

Storeys



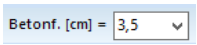
Opens a window where the storey can be selected.

Diameter

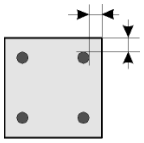


Lets you define or modify the diameter of a rebar.
To modify, select the rebars and than the enter the diameter or select a value from the list.

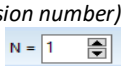
Concrete cover



Lets you define or modify the concrete cover.

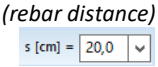


N (division number)



The division number which defines the number of rebars as *N+1*.

s (rebar distance)



The rebar distance that is considered in spacing with constant rebar distance.

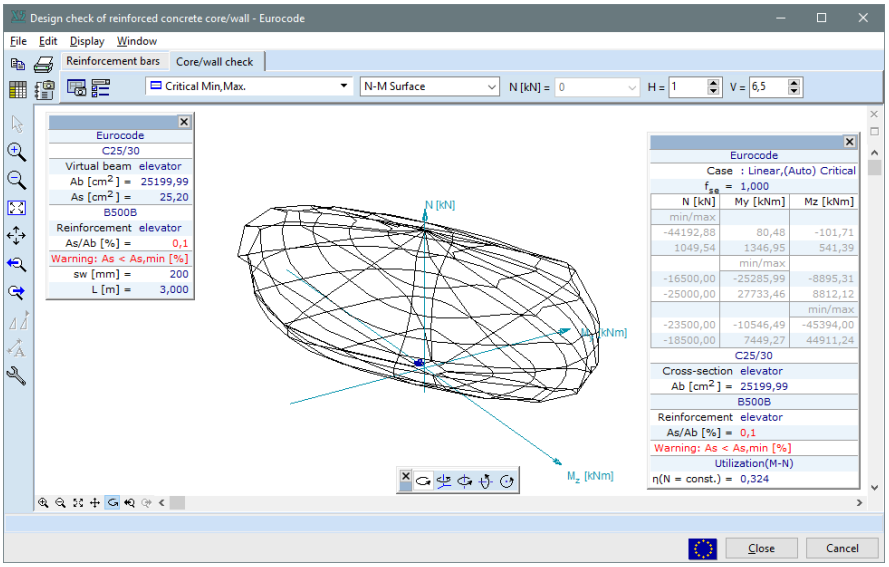
Geometrical transformations

Translation		Creates new rebars by copying existing ones by translation.
Rotation		Creates new rebars by copying existing ones by rotation.
Mirroring		Creates new rebars by mirroring existing ones.

6.5.17.2 Check reinforcement

Design codes

The design codes and national annexes in the software for which the design of reinforced concrete walls and cores can be performed are listed in **3.3.7 Design codes**.



Wall check

After selecting *Core/wall check* tab, the program performs the verification against bending with or without axial force or the shear check based on the cross-section properties and reinforcement parameters. In case of *N-M* interaction, the module determines the eccentricities coming from imperfection and second order effects based on the given buckling parameters and according to requirements of the current design code.
The program calculates $N_{x,Ed}, M_{y,Ed}, M_{z,Ed}$ design internal forces (see **6.5.17.3 Calculation of eccentricity**) and it checks if these points are within the interaction diagram.
The design check of reinforced concrete cores and walls is performed very similarly to the design check of reinforced concrete columns, thus diagrams, functionalities and buttons are very similar or identical that are introduced in previous sections (for further details see **6.5.10 Column reinforcement – RC2 module**).
The displayed diagram and its properties can be set in the Display Parameters window.

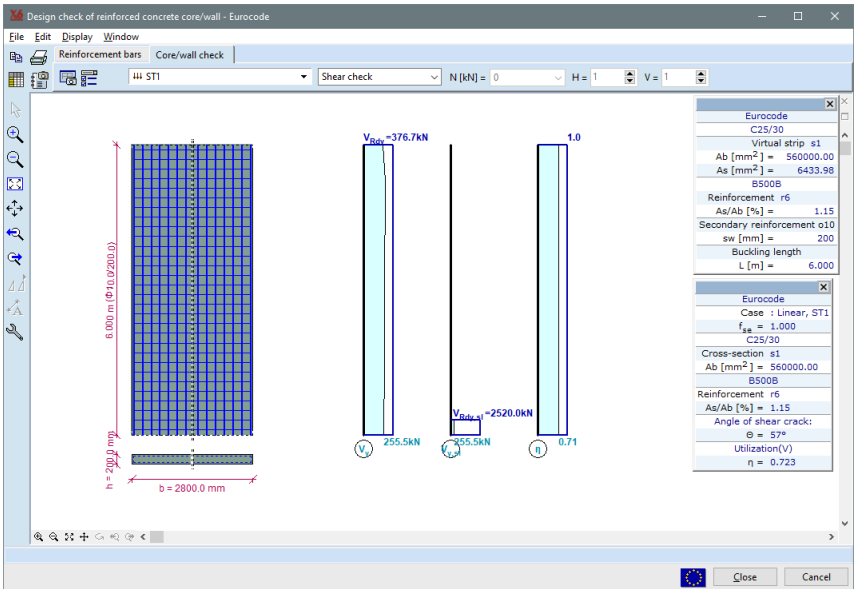
The shear check is only available for virtual strips, if the check of shear is requested among the parameters of the reinforcement parameters of the wall. The walls are only verified against the shear forces parallel to the plane of the wall.
In the result window, the following figures and diagrams are presented:

a) the side view of the wall strip and its reinforcement (in case of wider walls, only the end segments of the wall are shown)
b) the design shear force (V_y) and the shear resistance (blue line)
c) in the case of sliding shear, the design shear force at the bottom of the wall and the sliding shear resistance (blue line)
d) the maximum utilization (η) along the height of the wall.

☞ **In the case of ribbed walls, calculating the shear resistance, only the cross-section of the wall domain (without ribs) and its reinforcement is considered.**

☞ **The design check of walls and cores does not take into account fire design rules, elevated temperatures in the cross-section, spalling of concrete and degradation in stiffness and strength.**

Result of the shear check



Internal forces

The *Design results* table contains the maximum normal forces and bending moments at the top and bottom of the selected storey of the invetigated core/wall. The internal forces are integrated from the stresses and they are associated with the local coordinate system of the virtual beam/strip.

Detailing rules

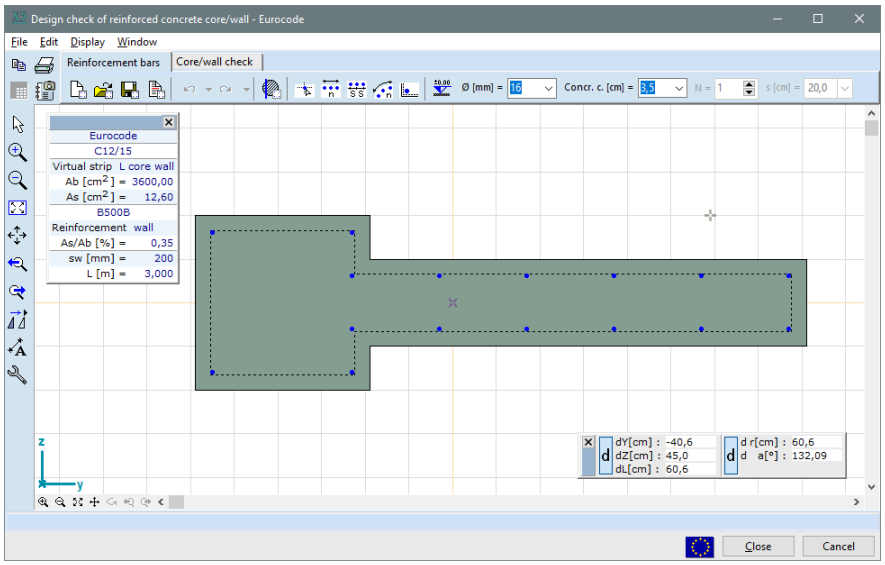
The software checks the most critical detailing rules according to the current design code. If any of these rules are violated a warning message appears.

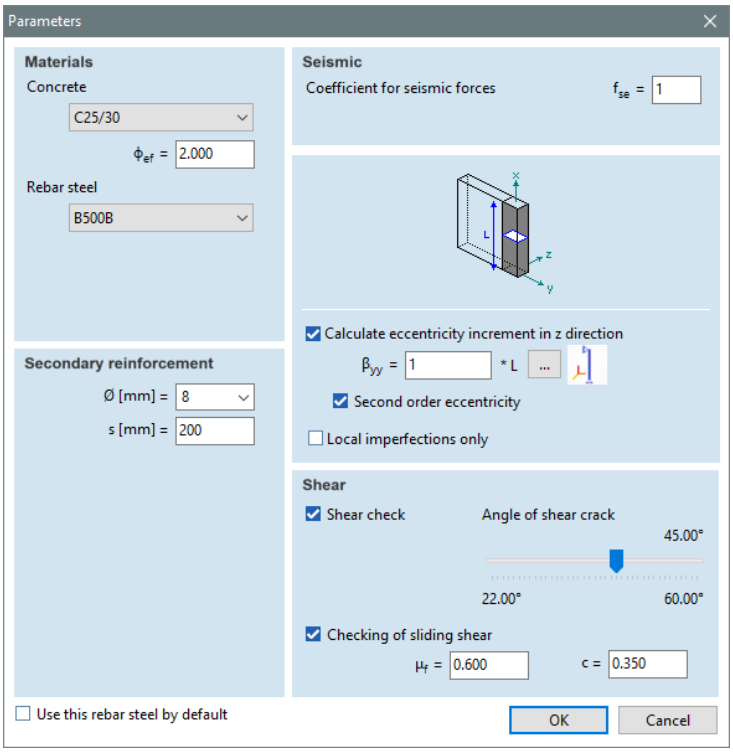
6.5.17.3 Calculation of eccentricity

Eccentricities due to imperfections and second order effects are calculated only in the course of design check of reinforcements assigned to virtual strips. These eccentricities are summed with the initial eccentricities calculates from the internal forces. In case of reinforcements assigned to virtual beams, the software does not calculate eccentricities due to imperfections and second order effects and the initial eccentricities are design eccentricities because modern design codes (e.g., EN 1992-1-1) includes methods wherewith the effect of **imperfections** and **second order effects** can be taken into account on structural level. It is assumed that the internal forces already represent these effects.

The calculation of eccentricities due to imperfections and second order effects in case of reinforcements assigned to virtual strips is performed similarly to the calculation presented by reinforced concrete columns (see **6.5.10 Column reinforcement – RC2 module**). The major difference is that eccentricities are calculated only in direction perpendicular to the plane of the wall (it is assumed that this direction is identical with local z axis).

☞ **In case of reinforcements assigned to virtual strips, it is assumed that the local z axis of the virtual strip is perpendicular to the wall. For this reason, it is assumed that the weak axis is the local y axis. Among the reinforcement parameters, the buckling length coefficient related to y weak axis can be specified by the user. Review of the orientation of the local coordinate system is recommended before performing design check.**





6.5.17.3.1 Check reinforcement according to Eurocode 2 (bending with axial force)

Design moments are $M_d = N_d \cdot e_d$

where N_d is the axial force and e_d is the design eccentricity.

- virtual beams: $e_d = e_e$
- virtual strips parallel to the plane: $e_d = e_e$
- virtual strips perpendicular to the plane: $e_d = e_e + e_i + e_2$

Minimal eccentricity: $max\left(20mm, \frac{h}{30}\right)$.

e_e is the **initial eccentricity** calculated from the first order axial force and bending moment.
Initial eccentricities at the ends of the investigated section:

$$e_e = M_I / N_d$$

Initial eccentricities at intermediate cross-section of the investigated section (equivalent eccentricity):

$$e_e = \max\begin{Bmatrix} 0.6 \cdot e_a + 0.4 \cdot e_b \\ 0.4 \cdot e_a \end{Bmatrix} \text{ and } |e_a| \geq |e_b|,$$

where e_a and e_b are the initial eccentricities at the ends of the investigated section.

e_i is the **eccentricity from geometric imperfections.**
$e_i = \alpha_h \Theta_0 \frac{l_0}{2}$, where l_0 is the buckling length, $\alpha_h = 2/\sqrt{l}$ and $\frac{2}{3} \le \alpha_h \le 1$, where l is the storey height.
Θ_0 – according to NA of the selected code.

e_2 is the **second order eccentricity.**
Second order eccentricity is considered if $\lambda \ge \lambda_{lim} = 20\frac{ABC}{\sqrt{n}}$ where

$$n = \frac{N_{Ed}}{A_c f_{cd}} \qquad A = \frac{1}{1+0.2\varphi_{ef}}, \qquad B = \sqrt{1+2\omega}, \qquad \omega = \frac{A_s f_{yd}}{A_c f_{cd}},$$

C is a constant that can be changed in *Design codes* window.

☞ **The calculation of λlim may be different according to NA of the selected code!**

In case of DIN EN 1992-1-1: $n < 0.41 \rightarrow \lambda_{lim} = \frac{16}{\sqrt{n}}; n > 0.41 \rightarrow \lambda_{lim} = 25.0$

In case of Italian Code: $\lambda_{lim} = \frac{25}{\sqrt{n}}$

In case of NS EN 1992-1-1 : $\lambda_n = \sqrt{\frac{n}{1+k_a\omega}} \ge \lambda_{n,lim} = 13A_\varphi$

The prescribed calculation method of $\boldsymbol{e_2}$ can be nominal curvature method or nominal stiffness method according to NA of the selected code. In some NAs, the application of both methods is allowed. In these cases, nominal curvature method is applied.

e_2 is the **second order eccentricity.**
Second order eccentricity is considered if $\lambda \ge \lambda_{lim} = 20\frac{ABC}{\sqrt{n}}$ where $n = \frac{N_{Ed}}{A_c f_{cd}}$

☞ **The calculation of λ_{lim} may be different according to NA of the selected code!**

In case of DIN EN 1992-1-1: $n < 0.41 \rightarrow \lambda_{lim} = \frac{16}{\sqrt{n}}; n > 0.41 \rightarrow \lambda_{lim} = 25.0$

In case of Italian Code: $\lambda_{lim} = 15.4\frac{C}{\sqrt{n}}$

The prescribed calculation method of $\boldsymbol{e_2}$ can be nominal curvature method or nominal stiffness method according to NA of the selected code. In some NAs, the application of both methods is allowed. In these cases, nominal curvature method is applied.

Nominal curvature method

$$e_2 = \frac{1}{r}\frac{l_0^2}{c}$$

where $\frac{1}{r} = K_r K_\varphi \frac{f_{yd}}{E_s \cdot 0.45 \cdot d'}$
$8 \le c \le \pi^2$ based on the first order bending moment diagram
$K_r = \min\left\{\frac{N_u' - N_{Ed}}{N_u' - N_{bal}}; 1.0\right\}$, $K_\varphi = \max\{1 + \beta\varphi_{ef}; 1.0\}$,
$\beta = 0.35 + \frac{f_{ck}}{200} - \frac{\lambda}{150}$, where f_{ck} is in N/mm2 unit
$d' = \frac{h}{2} + i_s$, where i_s is the radius of inertia of the longitudinal reinforcement

Nominal stiffness method

$$e_2 = (e_e + e_i)\left[1 + \frac{\beta}{N_B/N_d - 1}\right],$$

where $\beta = \frac{\pi^2}{c}$
N_B is the buckling load based on nominal stiffness
$EI = K_c E_{cd} I_c + K_s E_s I_s$ is the nominal stiffness, where
E_{cd} is the design value of the modulus of elasticity of concrete, see 5.8.6 (3)
I_c is the moment of inertia of concrete cross section
E_s is the design value of the modulus of elasticity of reinforcement, 5.8.6 (3)
I_s is the second moment of area of reinforcement, about the centre of area of the concrete
K_c is a factor for effects of cracking, creep etc, see 5.8.7.2. (2), (3)
K_s is a factor for contribution of reinforcement, see 5.8.7.2. (2), (3)

Eccentricities are determined in both bending planes. The program checks the following design situations:

Virtual beams

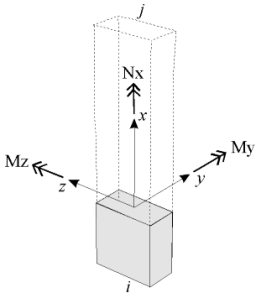
$$M_{dy} = N_d e_{ez}$$
$$M_{dz} = -N_d e_{ey}$$

Virtual strips

$$M_{dy} = N_d(e_{ez} + e_{iz} \pm e_{2z})$$
$$M_{dz} = -N_d e_{ey}$$

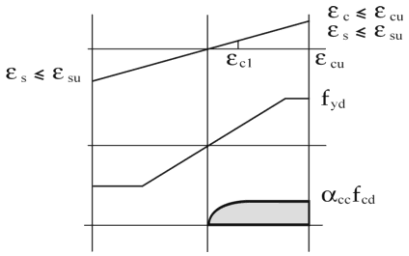
AXISVM checks whether the calculated design loads *(M_{dy}, M_{dz}, N_d)* are inside the N-M strength interaction diagram. If it is not satisfied in any of the design situations, the core/wall with the given cross-section and reinforcement fails.

On *N-M_R* strength interaction diagrams and on critical eccentricity curves, points represent these design loads. Custom force and moment values can also be entered in the table. These points will be displayed in the *N-M_R* strength interaction diagrams and in the critical eccentricity curves. Signs of the forces and moments are determined according to the picture.



Material models

σ, ε relations for concrete and steel:



6.5.17.3.2 Check reinforcement according to SIA 262 (bending with axial force)

Design moments are $M_d = N_d \cdot e_d$

where N_d is the axial force and e_d is the design eccentricity.

- virtual beams: $e_d = e_{1d}$
- virtual strips parallel to the plane: $e_d = e_{1d}$
- virtual strips perpendicular to the plane: $e_d = e_{1d} + e_{0d} + e_{2d}$

e_{0d}: **increment due to geometric imperfections**

$e_{0d} = \max\left\{\alpha_i \frac{l_{cr}}{2}; \frac{d}{30}\right\}$

where $\frac{1}{200} \geq \alpha_i = \frac{0.01}{\sqrt{l}} \geq \frac{1}{300}$

l_{cr} is the buckling length, l is the actual length, d is the effective height of the cross-section.

e_{1d}: is the **initial eccentricity** calculated from the first order force and moment.

Initial eccentricities at the ends of the investigated section:

$$e_{1d} = M_I / N_d$$

Initial eccentricities at intermediate cross-section of the investigated section (equivalent eccentricity):

$$e_{1d} = \max\begin{Bmatrix} 0.6 \cdot e_a + 0.4 \cdot e_b \\ 0.4 \cdot e_a \end{Bmatrix} \; and \; |e_a| \geq |e_b|,$$

where e_a and e_b are the initial eccentricities at the ends of the investigated section.

$e_{2d} = \chi_d \frac{l_{cr}^2}{\pi^2}$,

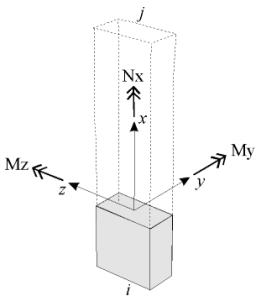
where $\chi_d = \frac{2 f_{yd}}{E_s (d - d')}$ if approximate curvature option is checked, otherwise $\chi_d = \frac{\varepsilon_{sd} - \varepsilon'_{sd}}{d - d'} + \frac{|\varepsilon_{c,\infty}|}{d}$.

Eccentricities are determined in both bending planes. The program checks the following design situations:

Virtual beams	Virtual strips
$M_{dy} = N_d e_{1z}$	$M_{dy} = N_d (e_{1z} + e_{0z} \pm e_{2z})$
$M_{dz} = -N_d e_{1y}$	$M_{dz} = -N_d e_{1y}$

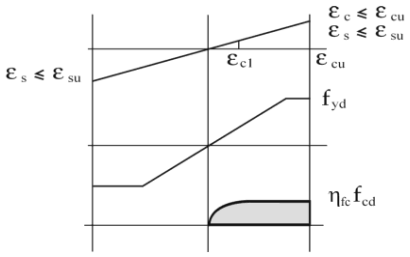
AXISVM checks whether the calculated design loads *(M_{dy}, M_{dz}, N_d)* are inside the N-M strength interaction diagram. If it is not satisfied in any of the design situations, the core/wall with the given cross-section and reinforcement fails.

On *N-MR* strength interaction diagrams and on critical eccentricity curves, points represent these design loads. Custom force and moment values can also be entered in the table. These points will be displayed in the *N-MR* strength interaction diagrams and in the critical eccentricity curves. Signs of the forces and moments are determined according to the picture.



Material models

σ, ε relations for steel and concrete:



6.5.17.4 Shear verification of reinforced concrete walls

General principles, limitations

In case of virtual strips, the module provides the ability to calculate the shear resistance and to determine the shear utilization of the wall strips. The shear utilization cannot be summed up by the utilization coming from N-M interaction. In the case of cores consisting of several wall strips, the strips can only be checked separately.
In the calculation, it is supposed that the vertical and horizontal reinforcement has an adequate anchorage.

The shear check consists of two parts:

1) The module calculates the shear strength of the concrete cross-section and the applied reinforcement, considering the angle of shear crack set among the reinforcement parameters.
   The cross-sectional area of the ribs connected to the wall domain and their reinforcements are not considered in the calculation.
   The analysis can only be applied to walls for which the angle of the shear crack is considered to be valid.
2) The module calculates the sliding shear resistance at the bottom of the walls, if this type of check is requested.

The maximum shear utilization of a storey is calculated based on the result of the above two check.

Lever arm

During the calculation of shear resistance of a reinforced concrete section, the level arm (z) of internal forces plays an important role. The level arm is interpreted as the distance between the center of tension reinforcement and the center of compression zone. In case of beams, z is approximately selected 0.9d, where d is the effective depth. However, if axial force is present, this approximation cannot be used. The program automatically calculates the level arm of internal forces with the consideration of the axial force in the section.

6.5.17.4.1 Shear and sliding shear check according to Eurocode 2

Shear resistance of concrete

$$V_{Rd} = k_d \tau_{cd} b_w d$$
$$k_d = \frac{1}{1 + 3\varepsilon_v d}$$
$$\varepsilon_v = 1.5 \frac{f_{sd}}{E_s}$$

where:

- τ_{cd} is the shear stress limit ($\tau_{cd} = 0{,}3\sqrt{f_{ck}}/\gamma_c$),
- b_w is the smallest width of the cross-section in the tensile area (circular section 0.8D),
- d is the effective depth.

Shear resistance of concrete

$$V_{Rd,c} = \left[C_{Rd,c} k (100 \rho_l f_{ck})^{1/3} + k_1 \sigma_{cp}\right] b_w d \geq \left(v_{min} + k_1 \sigma_{cp}\right) b_w d$$

where:

- ρ_{sl} ratio of tensile reinforcement,
- b_w is the smallest width of the cross-section in the tensile area (circular section 0.8D),
- d is the effective depth,
- σ_{cp} is the axial stress in the cross-section due to loading or prestressing (compression is positive) $<0.2 f_{cd}$,
- $C_{Rd,c}$, v_{min} and k_1 parameters that are regulated in the National Annex. The values of these factors can be set in the Design codes window under the Reinforced concrete tab (see **3.3.7 Design codes**).

Shear resistance of horizontal reinforcement

Rectangular cross section:

$$V_{Rd,s} = \frac{A_{sw}}{s} z f_{ywd} \cot\theta$$

where:

- A_{sw} is the cross-sectional area of the shear reinforcement,
- s is the spacing of stirrups or the pitch of spiral stirrup,
- z is the level arm,
- f_{ywd} is the design yield strength of the shear reinforcement,
- θ is the angle of shear crack (set among the reinforcement parameters).

Maximum design shear resistance

$$V_{Rd,max} = \alpha_{cw} b_w z \nu_1 f_{cd} \frac{\cot\theta + \cot\alpha}{1 + \cot^2\theta}$$

where:

- α_{cw} is a coefficient taking account of the state of the stress in the compression chord,
- ν_1 is a strength reduction factor for concrete cracked in shear,
- α is the angle between shear reinforcement and the beam axis perpendicular to the shear force (in case of stirrups α=90°).

Sliding shear resistance

The analysis is based on the rules of section 6.2.5 in Eurocode 1992-1-1 considering the following formula and parameters. The sliding shear resistance at the bottom of the wall:

$$V_{Rd,sl} = A_w \left[c f_{ctd} + \mu \sigma_n + \rho f_{yd} (\mu \sin\alpha + \cos\alpha)\right] \leq 0.5 \nu f_{cd}$$

where:

- A_w is the cross-sectional area of the domain associated with the virtual strip (without the cross-sectional area of the ribs connected to the domain)
- c and μ are factors which depend on the roughness of the interface
- f_{ctd} is the design tensile strength
- σ_n is the stress per unit area caused by the minimum external normal force across the interface that can act simultaneously with the shear force, positive for compression, such that $\sigma_n < 0.6 f_{cd}$, and negative for tension. When σ_n is tensile $c \cdot f_{ctd}$ should be taken as 0
- $\rho = A_{s0}/A_i$,

where:

- A_{s0} is the area of reinforcement crossing the interface, including the reinforcements of the ribs connected to the domain
- A_i is the area of the joint, including the cross-sectional are of the ribs connected to the domain
- f_{yd} is the design yield strength of reinforcement, α = 90° (vertical reinforcement)
- ν is a strength reduction factor

6.5.17.4.2 Shear and sliding shear check according to SIA 262

Shear resistance of the secondary reinforcement

Rectangular cross section:

$$V_{Rd,s} = \frac{A_{sw}}{s} z f_{yd} \cot \alpha$$

where:

- A_{sw} is the cross-sectional area of the shear reinforcement,
- s is the spacing of stirrups or the pitch of spiral stirrup,
- z is the level arm,
- f_{sd} is the design yield strength of the shear reinforcement,
- α is the angle of shear crack,

Maximum design shear resistance

$$V_{Rd,c} = b_w z k_c f_{cd} \sin \alpha \cos \alpha$$
$$\varepsilon_1 = \varepsilon_x + (\varepsilon_1 + 0.001) \cot^2 \alpha$$

Sliding shear resistance

The analysis is based on the rules of section 4.3.4.3.2 in SIA 262 considering the following formula and parameters. The sliding shear resistance at the bottom of the wall:

$$V_{Rd,sl} = b_w \cdot z(k_{c\tau}\tau_{cd} - k_{c\sigma}\sigma_d) \leq 0.3 f_{cd}$$

where:

- b_w is the thickness of the domain associated with the virtual strip (thickness of the wall), the rib cross-sections are not considered,
- z is the lever arm of internal forces (considering the cross-section of the domain associated with the virtual strip, but without the cross-section of the ribs)
- $k_{c\sigma}$ is the factor for reduction of normal stress,
- $k_{c\tau}$ is the factor for reduction of shear stress limit,
- τ_{cd} is the dimensioning value of shear stress limit,
- f_{cd} is the dimensioning value of concrete compressive strength,
- σ_d is the normal stress across the interface, calculated by the following formula

$$\sigma_d = \frac{-|V_d| \cot \alpha}{z \cdot b_w} + \frac{N_d}{l_w b_w}$$

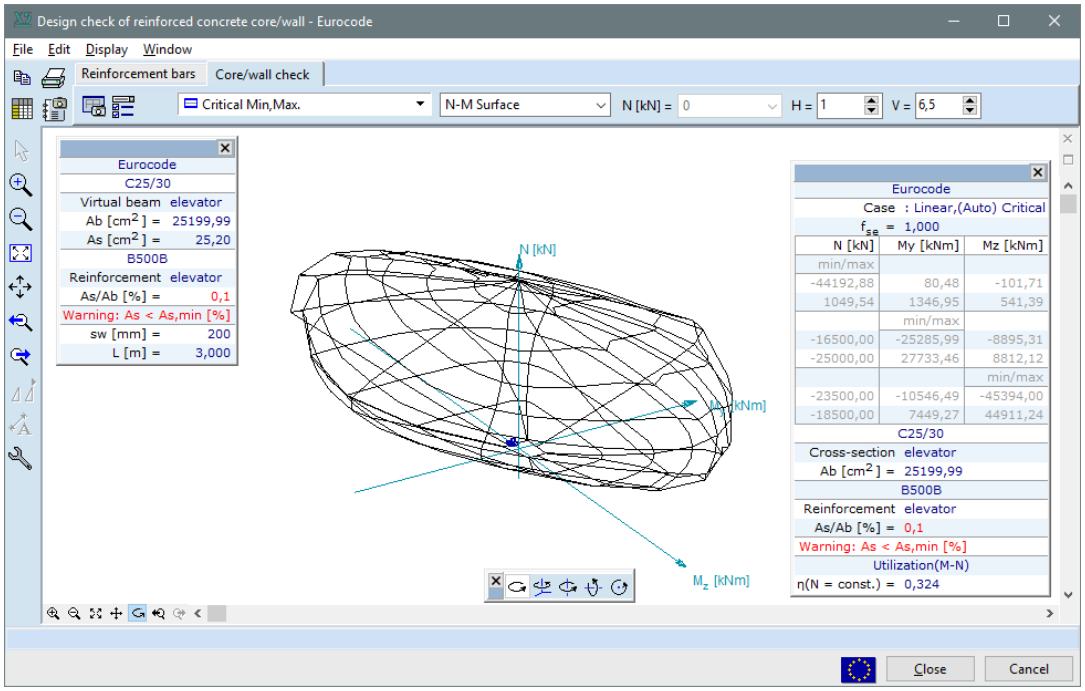
where:

- N_d is the dimensioning value of normal force acting on the cross-section of the virtual strip. It is negative in case of compression.
- V_d is the dimensioning value of shear force,
- α is the compression field inclination,
- l_w is the length of the shear wall (length of the virtual strip).

6.5.17.5 Results

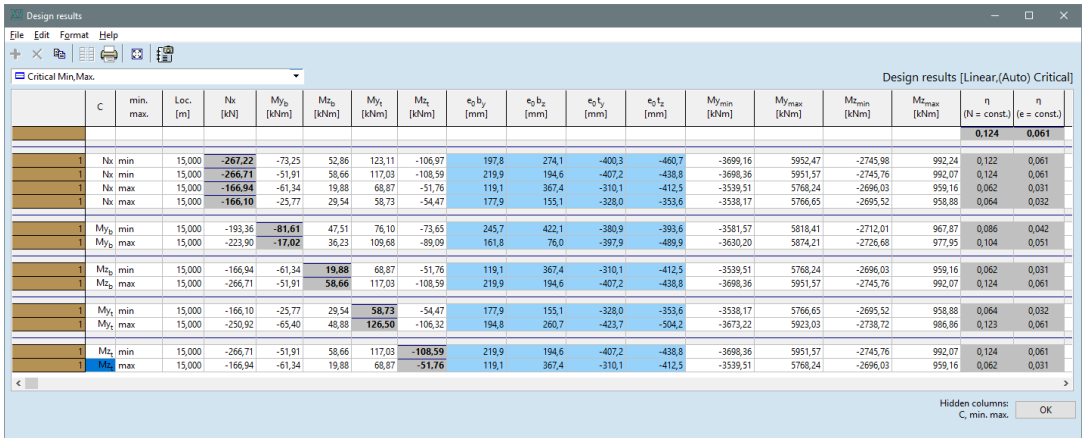
The design results of reinforced concrete cores and walls are available in many ways in the software.

Drawings



Design check drawings (N_x-M_y-M_z, N_x-M_y, N_x-M_z *and* M_y-M_z strength interaction diagrams, critical eccentricity curves, shear check) can be saved into the Drawings Library by clicking *Save to Drawings Library* button (see **3.6.10 Drawings Library**).

Tables



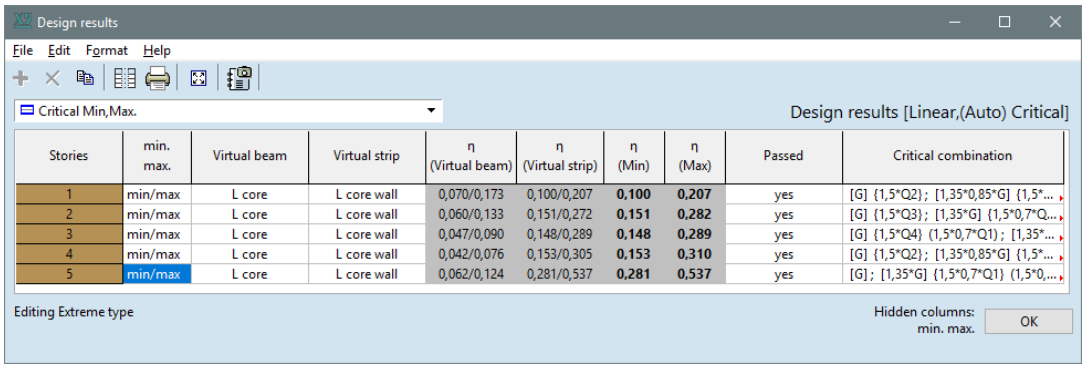
Design results table is available via *Design check of reinforced concrete core/wall* and *Reinforced concrete cores and walls* windows, as well.

If the table is opened from the Design check dialog, the presented results depend on the active design check drawing. (If the active result view is N_x-M_y-M_z, N_x-M_y, N_x-M_z *and* M_y-M_z strength interaction diagrams or critical eccentricity curves, then only the result of *N-M* interaction is summarized. If the active result view is Shear check (available only for virtual strips), only the shear utilization is displayed.)

The table is editable if it is opened through *Design check of reinforced concrete core/wall* window. In this case, columns can be turned on/off and new custom force and moment values can also be added to the table. If the table is opened from *Reinforced concrete cores and walls* window, the table is not editable.

In this case, the load case/combination, envelope or automatic critical load combination considered in the design check can be selected from the dropdown list. After changing the load situation, the design checks are performed automatically, that can take a few seconds.

Design results table for overall utilizations (available only for *N-M* interaction) can be opened only through *Reinforced concrete cores and walls* window. The table summarizes the design results for the virtual beam and the virtual strip and it shows the overall utilization as well.
If the utilization cannot be calculated due to any reason, that is marked with ***???*** symbol. If the calculated utilization exceeds 1.0, the utilization is displayed with red color.
The overall utilization is the utilization of *N-M* interaction. Shear check results of strips are not taken into account.



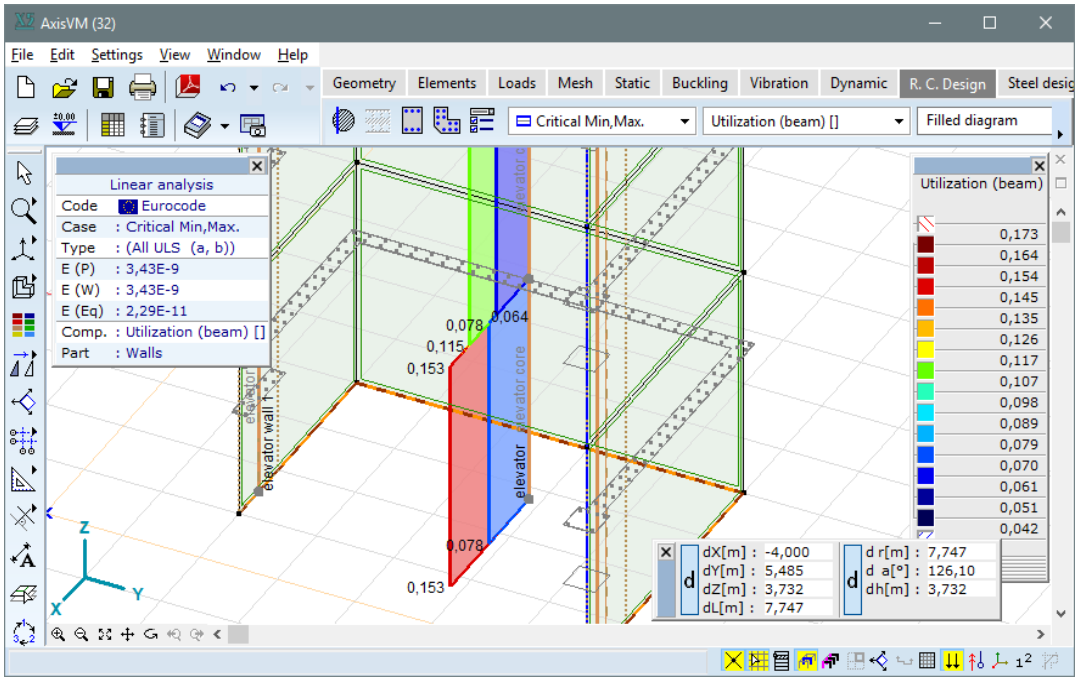
Stories	min. max.	Virtual beam	Virtual strip	η (Virtual beam)	η (Virtual strip)	η (Min)	η (Max)	Passed	Critical combination
1	min/max	L core	L core wall	0,070/0,173	0,100/0,207	**0,100**	**0,207**	yes	[G] {1,5*Q2}; [1,35*0,85*G] {1,5*...
2	min/max	L core	L core wall	0,060/0,133	0,151/0,272	**0,151**	**0,282**	yes	[G] {1,5*Q3}; [1,35*G] {1,5*0,7*Q...
3	min/max	L core	L core wall	0,047/0,090	0,148/0,289	**0,148**	**0,289**	yes	[G] {1,5*Q4} {1,5*0,7*Q1}; [1,35*...
4	min/max	L core	L core wall	0,042/0,076	0,153/0,305	**0,153**	**0,310**	yes	[G] {1,5*Q2}; [1,35*0,85*G] {1,5*...
5	min/max	L core	L core wall	0,062/0,124	0,281/0,537	**0,281**	**0,537**	yes	[G]; [1,35*G] {1,5*0,7*Q1} {1,5*0,...

Result components

If there are storeys and actual reinforcements assigned to virtual beams/strips in the model and static (linear/non-linear) or dynamic results are available, new result components become available on R. C. Design tab related to design check results of cores and walls:

- *Utilization (beam)* – if there are storeys and actual reinforcements assigned to virtual beams in the model,
- *Utilization (strip)* – if there are storeys and actual reinforcements assigned to virtual strips in the model,
- *Overall utilization* – if at least virtual beam – virtual strip pair is defined in *Reinforced concrete cores and walls* window.
- *N*-M *utilization (Virtual strip)* - if there are storeys and actual reinforcements assigned to virtual strips in the model,
- *V utilization (Virtual strip)* - if there are storeys and actual reinforcements assigned to virtual strips in the model and shear check is requested.

(Overall utilization results are displayed on virtual strip of virtual beam – virtual strip pairs.). The overall utilization is the utilization of *N-M* interaction. Shear check results of strips are not taken into account.



Results in the Table Browser

Design results are also available in the Table Browser (*see* 2.9 Table Browser), where the results of every reinforced concrete wall are available in summary tables.

6.5.18 Stress-strain analysis of reinforced concrete structures – RC6 module

Stresses and strains within reinforced concrete elements are important in many cases (e.g., serviceability checks, fatigue analysis). The RC6 module allows stress-strain analysis of columns, beams and surface elements and virtual beams with actual reinforcement for the internal forces, taking into account the effect of actual reinforcement, cracks in concrete and nonlinear behaviour of the materials. Using this module, in addition to stresses and strains, the crack width can also be determined for arbitrary cross-sections and internal forces.
The aim of the analysis is to obtain the strain and the curvatures of the section for given internal forces. Due to the fact that this problem is not continuous and not linear, an iteration is required to find the equilibrium.

☞ **Analysing beams and columns, RC2 module is a prerequisite to use RC6 module because an actual column or beam reinforcement is a requirement (see 6.5.12 Beam reinforcement parameters (uniaxial bending) and (6.5.4 Actual reinforcement against biaxial bending (column)).**

☞ **RC1 module is a prerequisite to use RC6 module for the analysis of surface elements. Only normal (solid), ribbed, composite ribbed slabs and slab with custom stiffness matrix can be analysed with this module.**

☞ **The stress-strain analysis does not take into account fire design rules, elevated temperatures in the cross-section, spalling of concrete and degradation in stiffness and strength.**

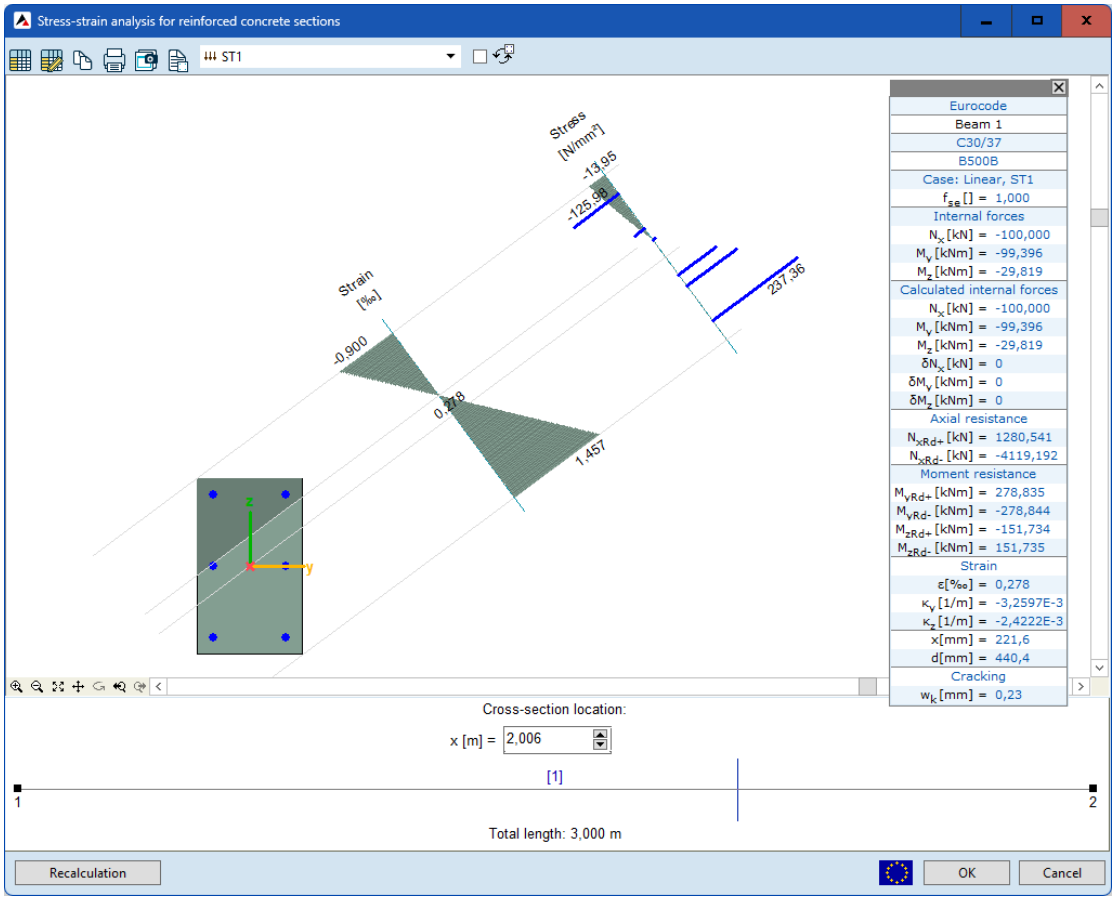
6.5.18.1 Stress-strain analysis of reinforced concrete sections



Click on the icon, then select one (and only one) reinforced concrete structural member (see 2.16.1 Selection). The icon is active if there is at least one reinforced concrete structural member (beam, rib, truss) or virtual beam/strip made of concrete domains and ribs in the model with actual reinforcement. The following selection problems trigger a warning message

- No structural member has been selected
- More than one structural member has been selected
- The material of the selected structural member is not concrete, or no actual reinforcement has been defined for the member
- The selected structural member has a composite cross-section

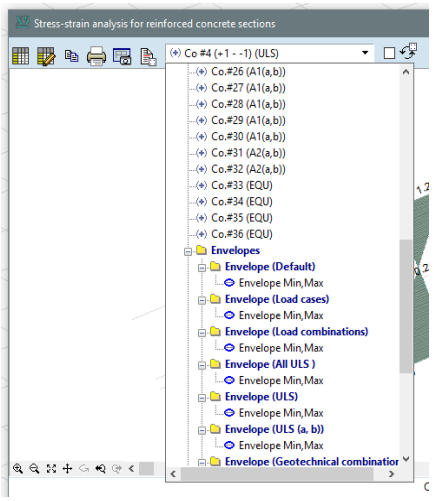
If the selection is accepted, the *Stress-strain analysis for reinforced concrete sections* dialog appears where the stress-strain analysis can be performed for any section along the member.



Load case, load combination, envelope or critical combination

You can select a case from the drop-down list to display:

- Load case, load combination
- The k-th increment of a nonlinear analysis
- Envelope display
- Critical combination



Position of the cross-section

Stress-strain analysis is performed for internal forces in a given cross-section. The location of the cross-section can be set with the slider at the bottom of the window.



Result table - see **6.5.18.1.3 Results**



Result table (custom internal forces) - see **6.5.18.1.3 Results**



Copies image to the Clipboard.



Prints image.



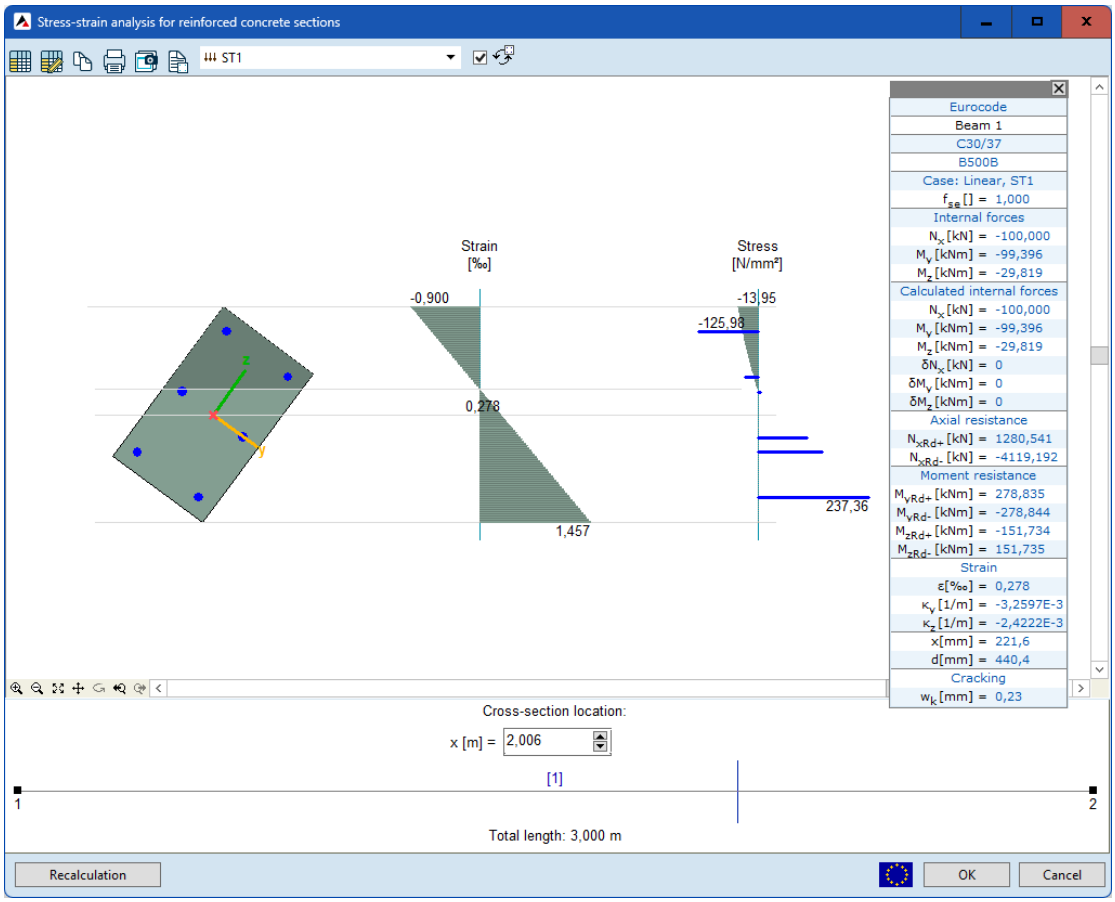
Saves the drawing into the Drawings Library (see **3.6.10 Drawings Library** and **3.6.11 Save to Drawings Library**)



Stress-strain analysis parameters - see **6.5.18.1.1 Stress-strain analysis parameters**



Displaying stress and strain results on a rotated cross-section.



Recalculation

It may happen that the heuristic algorithm does not find the equilibrium or its precision is not adequate. In these cases, stress-strain analysis can be repeated by clicking on the *Recalculation* button.

Virtual beams and strips

If a virtual beam or virtual strip is selected, the software is able to analyse the section and obtain the strains and stresses in it after the definition of reinforcement. In the case of virtual beams/strips with reinforced concrete wall or core reinforcement (see **6.5.17 Design of reinforced concrete cores and walls – RC5 module**), the stress-strain analysis is performed for the reinforcement defined in RC5 module. In the case of walls and cores, neither the reinforcement nor the storeys can be modified. Only in RC5 module, if necessary.

If no wall/core reinforcement is attached to a virtual beam/strip, actual reinforcement needs to be defined before in the window opens before stress-strain analysis. The buttons and functionalities of this window can be found in the description of RC5 module.



Three new buttons can be found in stress-strain analysis window compared to the case when a reinforced concrete column or beam is selected. The button on the left defines a new section boundary to the virtual beam or strip. These sections can have different reinforcement but the reinforcements need to be constant in a section. Section boundaries can be modified later by dragging them with the mouse. The second button can be used to delete the active section (where the vertical line indicating the current position is located). The third button is to change/define the reinforcement of the section.

6.5.18.1.1 Stress-strain analysis parameters



The parameters considered in stress-strain analysis can be entered in the following dialog.

Materials

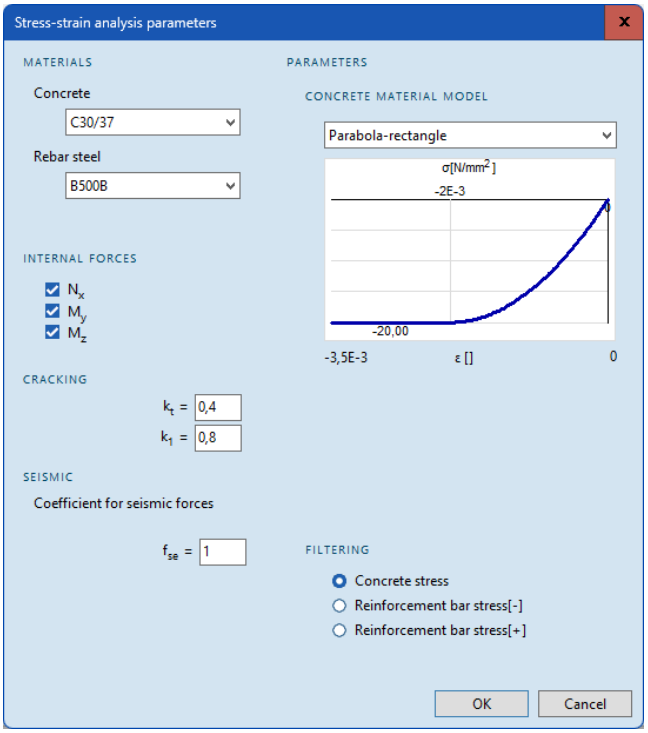
Concrete and rebar steel grade can be specified.

Internal forces

Internal force components can be turned on/off. Only the checked components are considered in stress-strain analysis. At least one component must be selected.

Seismic

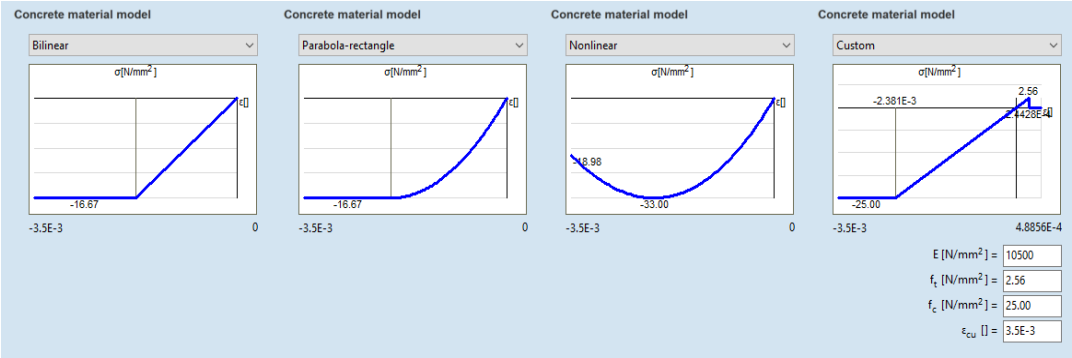
For coefficient of seimic forces see **4.10.25 Seismic loads – SE1 module**



Concrete material model

The nonlinear concrete material model can be selected from a dropdown list. The available material models are different for Eurocode and SIA standards, respectively.

Eurocode



Bilinear

Concrete material model according to section 3.1.7. (2) of EN 1992-1-1

$\sigma = f_{cd}$ ha $\varepsilon < -\varepsilon_{c3}$

$\sigma = f_{cd}\frac{\varepsilon}{\varepsilon_{c3}}$ ha $0 \geq \varepsilon \geq -\varepsilon_{c3}$

Parabola-rectangle

Concrete material model according to section 3.1.7. (2) of EN 1992-1-1

$\sigma = f_{cd}$ ha $\varepsilon < -\varepsilon_{c2}$

$\sigma = f_{cd}\left[1-\left(1-\frac{\varepsilon}{\varepsilon_{c2}}\right)^{n}\right]$ ha $0 \geq \varepsilon \geq -\varepsilon_{c2}$

Nonlinear

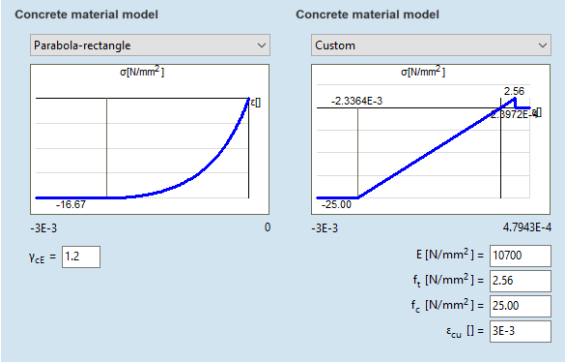
Concrete material model according to section 3.1.5. (1) of EN 1992-1-1

$$\frac{\sigma}{f_{cm}} = \frac{k\eta - \eta^2}{1 + (k-2)\eta}$$

Custom

Parameters of the material model are specified by the user. It can be used to consider tension strength or creep effect on concrete stiffness.

SIA



Parabola-rectangle

Concrete material model according to section 4.2.1.6. of SIA 262

$$\frac{\sigma_c}{f_{cd}} = \frac{k_\sigma \zeta - \zeta^2}{1 + (k_\sigma - 2)\zeta}$$

☞ **Design codes specify bilinear, parabola-rectangle and nonlinear material models especially for the design of cross-sections. For this reason, the stiffness of concrete is not necessarily equal to the stiffness used by hand calculations. If a custom material model is selected for concrete, the Young's modulus of the concrete can be entered. This material model can be used if tension strength or creep effect is considered.**

☞ **The behaviour of steel is described by bilinear material model.**

Filtering the results

If envelope or critical combination results are selected in the *Stress-strain analysis for reinforced concrete sections* dialog, the analysis is performed for multiple load cases and combinations. The results of these analyses can be seen in the result tables, however, only the results of the critical case/combination are presented graphically and listed in the info window. The presented case/combination can be critical in terms of stress in concrete or in reinforcement. Set the required option under *Filtering*.

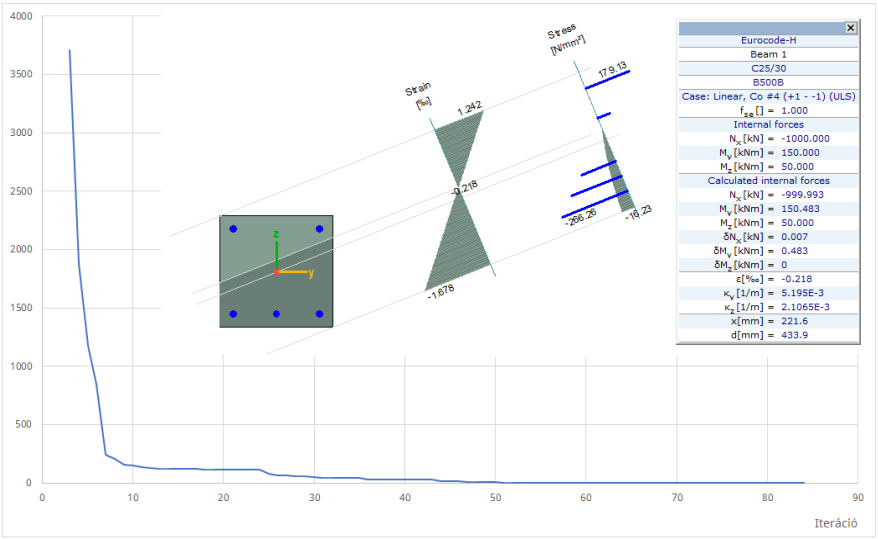
6.5.18.1.2 Stress-strain analysis

The design codes and national annexes in the software for which the stress-strain analysis can be performed are listed in 3.3.7 Design codes.

☞ **RC6 module can use parallel computing if envelope or critical combination is selected, thus it is recommended to turn on the option of multiple threads (*Settings / Preferences / Analysis*).**

Iteráció

Due to the fact that the problem is nonlinear and not continuous, an iteration need to be performed to find the equilibrium. The RC6 module uses a heuristic algorithm based on a genetic algorithm.



☞ **In some cases, it may happen that the heuristic algorithm does not find the equilibrium or its precision is not adequate (the accepted maximal difference in axial force is 1% or 1 kN and in bending moment is 1 kNm). Stress-strain analysis can be repeated by clicking on *Recalculation* button.**

Integration

Internal forces compatible with strains are calculated with the help of so-called fiber model through numerical integration based on ε normal strains, κy and κz curvatures. The considered concrete material model and rebar steel grade can be specified by the user.

6.5.18.1.3 Results

Results

Results of the stress-strain analysis are presented in result tables and as a diagram in the main window. Possible errors in the stress-strain analysis:

- the algorithm does not find the equilibrium, the stress-strain analysis should be repeated
- the internal forces exceed the resistance of the cross-section

Envelope and critical combination

The results of the critical case/combination are presented as a diagram and listed in the info window if envelope or critical combination is selected. The name of the load case/combination is shown above the drawing.

Info window

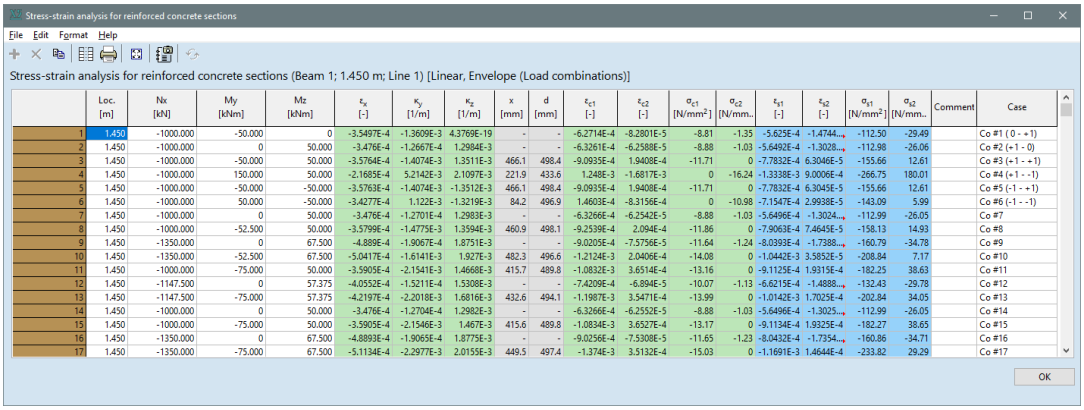
f_{se} – coefficient of seismic forces
N_x – axial force
M_y – bending moment
M_z – bending moment
N_x – axial force (integrated)
M_y – bending moment (integrated)
M_z – bending moment (integrated)
δN_x – difference between input and integrated axial force
δM_y – difference between input and integrated bending moment
δM_z – difference between input and integrated bending moment
N_{xRd+} – tension resistance of the cross-section
N_{xRd-} – compression resistance of the cross-section
M_{yRd+} – bending resistance considering N_x axial force
M_{yRd-} – bending resistance considering N_x axial force
M_{zRd+} – bending resistance considering N_x axial force
M_{zRd-} – bending resistance considering N_x axial force
ε – strain at the centroid
κy – curvature (*y* axis)
κz – curvature (*z* axis)
x – height of compression zone
d – effective height

Eurocode
L core
C20/25
B500B
Case: Linear, Co #1 (ULS)
f_{se} [] = 1,000
Internal forces
N_x [kN] = -1029,31
M_y [kNm] = -226,93
M_z [kNm] = -8,88
Calculated internal forces
N_x [kN] = -1029,31
M_y [kNm] = -226,93
M_z [kNm] = -8,88
δN_x [kN] = 0
δM_y [kNm] = 0
δM_z [kNm] = 0
Axial resistance
N_{xRd+} [kN] = 2219,60
N_{xRd-} [kN] = -19573,96
Moment resistance
M_{yRd+} [kNm] = 7838,10
M_{yRd-} [kNm] = -4802,60
M_{zRd+} [kNm] = -3894,28
M_{zRd-} [kNm] = 1086,44
Strain
ε[‰] = -0,056
κ_y [1/m] = -7,6833E-6
κ_z [1/m] = 1,0223E-5

Result tables

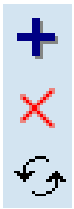


By clicking on tables button, a result table opens with the detailed results of stress-strain analysis.





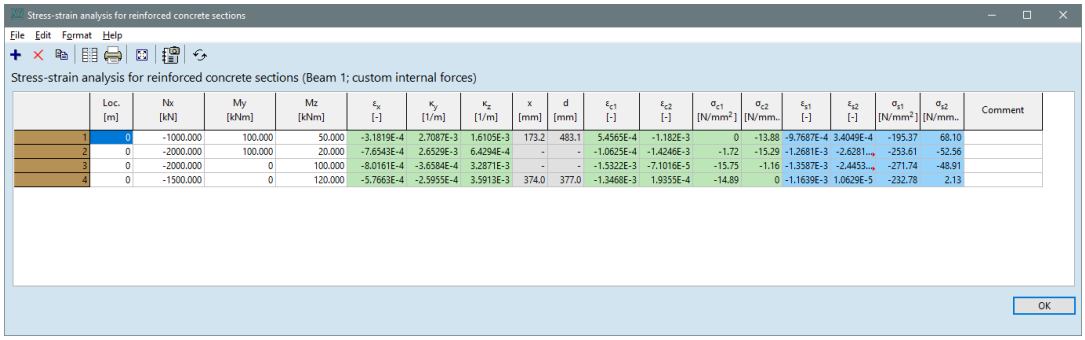
Custom internal forces can be given in the result table opened by clicking on this icon on the left (see 2.9 Table Browser). Stress strain analysis results for custom internal forces are available only in this result table.



New row

Delete rows

Recalculation



6.5.18.2 Stress-strain analysis of surface elements

Stress-strain analysis

In the case of surface elements, the basic principles of the analysis and the functioning are similar as described in the previous sections of beams and column elements. Only the main features and differences are detailed below.

Type of surface elements

The following type of slab can be analysed with the module:

normal (solid) slab

ribbed slab

composite ribbed slab

slab with custom stiffness matrix

☞ **Trapezoidal steel deck, Airdeck and hollow core slab cannot be analyzed with the module.**

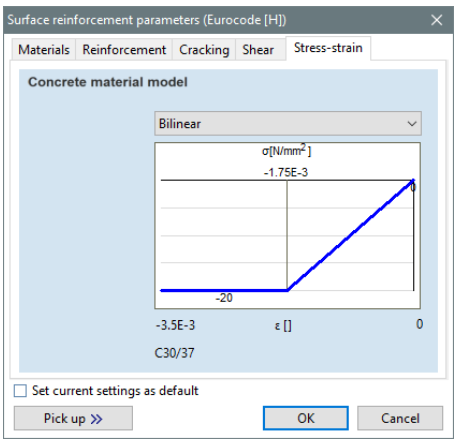
☞ **The module can only perform the analysis for slabs that have applied reinforcement, and the reinforcement is defined local x-y reinforcement direction. Slabs with custom reinforcement directions cannot be verified.**

Parameters



The applied concrete material model can be set on the *Stress-strain* tab of the *Surface reinforcement parameters* window. The automatic (non-custom) models use the parameters of the concrete set on the *Materials* tab.

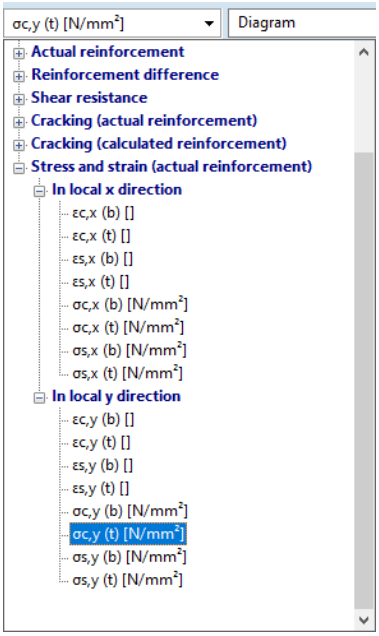
For the details of the available concrete material models, see 6.5.18.1.1 Stress-strain analysis parameters.



Result components

The available result components of the stress-strain analysis are listed in the *Stress and strain (actual reinforcement)* component group. The results are calculated for the actual, visible model or parts of the main window. Due to the iteration required, the calculation can take a considerable amount of time, strongly depending on the number of finite elements applied. Partitioning of the model can help.

These result components are available only if the model contains a reinforced surface element on which the module can be perform the analysis and the element has actual reinforcement defined.



The following result components are available in the local *x* and *y* direction:

Strain

εc (b)	Strain in the concrete at the bottom of the section
εc (t)	Strain in the concrete at the top of the section
εs (b)	Strain in the centre of gravity of the applied bottom reinforcement
εs (t)	Strain in the centre of gravity of the applied top reinforcement

Stress

σc (b)	Stress in the concrete at the bottom of the section
σc (t)	Stress in the concrete at the top of the section
σs (b)	Stress in the centre of gravity of the applied bottom reinforcement
σs (t)	Stress in the centre of gravity of the applied top reinforcement

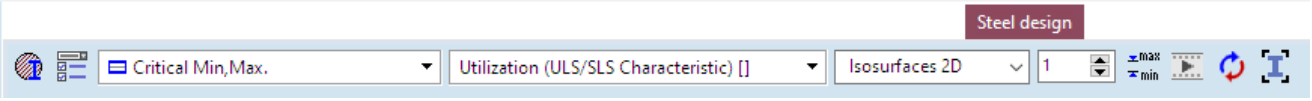
The results can be displayed on the surface element or a section segment with the usual representation modes.

Result tables

The results of the surface elements and the section segments are summarized in the *Result tables*.



6.6 Steel design



Itt tartok

AXISVM modules offer the following design methods for steel bar members:

SD1 – Design of hot-rolled and welded cross-sections

6.6.1 Steel beam design according to Eurocode 3 – SD1 module

SD3 – Design of cold-formed cross-sections

6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 -SD3 module

SD8 – Steel beam fire design

6.6.3 Steel beam fire design according to Eurocode 3 – SD8 module

SD9 – Steel cross-section optimization

6.6.2.4 Steel cross-section optimization - SD9 module

6.6.1 Steel beam design according to Eurocode 3 – SD1 module

EUROCODE 3

The steel beam design module can be applied to the following shapes:

Rolled I shapes	Tee shapes
Welded I shapes	Rectangular (solid) shapes
Box shapes	Round (solid) shapes
Pipe shapes	Arbitrary shapes, some checks are not performed
Single-symmetric I shapes	

Among elements with cross-section class 4, single- and double-symmetric I shaped, rectangular T and box shaped cross-sections can be designed with this module. Effective section properties are calculated in the cases of uniform compression and uniform bending. These properties can be found in the *Table Browser* under *Steel design,* in the table *Design resistances,* or in the pop-up window after clicking on the element:

A_{eff} area of the effective cross section when subjected to uniform compression

$e_{N,y}$ the shift of the y neutral axis when the cross-section is subjected to uniform compression (will be zero if the section is symmetric to axis y). Negative shift will cause a negative $\Delta My = N \cdot e_{N,y}$ moment in the actual cross-section.

$W_{eff,min}$ elastic section modulus (corresponding to the fiber with the maximum elastic stress) of the effective cross section when subjected only to moment about the relevant axis.

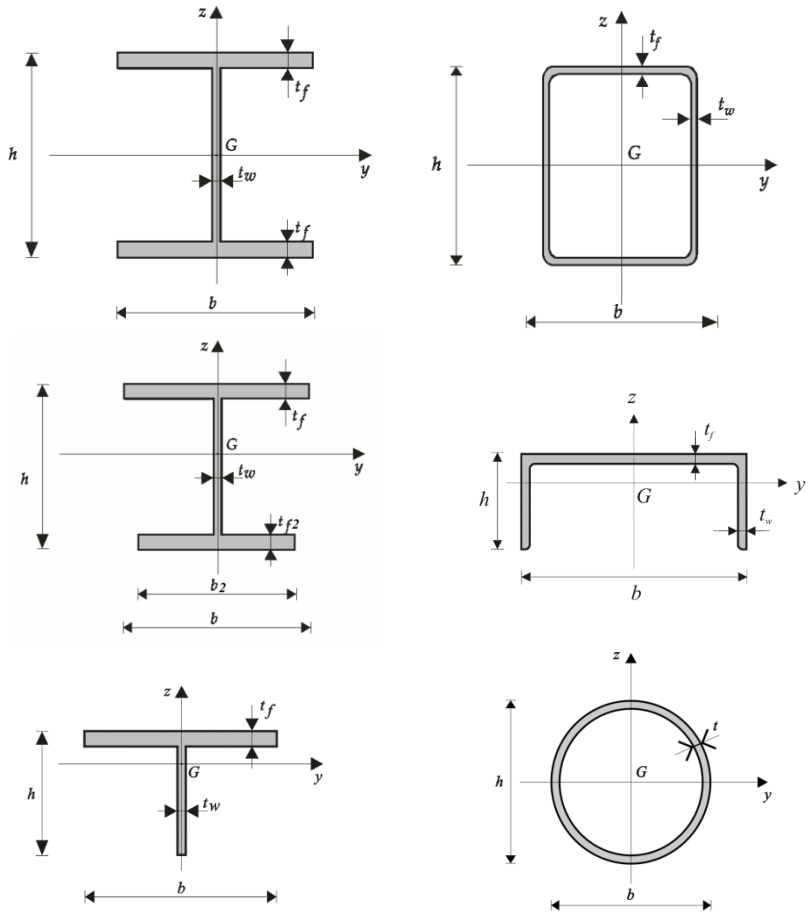
$W_{eff,(-),min}$ refers to sections where the moment is negative

$W_{eff,(+),min}$ refers to those where the moment is positive

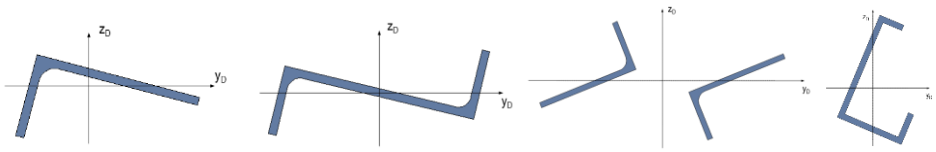
It is important to know, that these section properties are calculated only when the section is in class 4. It might happen that there is no stress causing plate buckling, but the properties will still be available in the *Table Browser*. Non-uniform members (variable cross-sections) in class 4 are calculated only if α of the non-rectangular panels are not greater, than 10 degrees (EN 1993-1-5: 2.3).

It is assumed that the cross-sections do not have holes in them and are made of plates with a thickness less than or equal to 40 mm.
The cross section should be constant or tapered. It is also assumed that the loads on single-symmetric cross-sections act in the plane of symmetry, that is the plane of bending. For general shapes with no plane of symmetry only *Axial Force-Bending-Shear (N-M-V)* and *Compression-Bending-Buckling (N-M-Buckling)* is checked.

☞ **AXISVM performs only the checks listed below. All the other checks specified in the design code like constrained torsion, strutting forces, joints, etc. has to be completed by the user.**



For cross-sections having principal axes non-coincident with the local axes the program performs checks in a design coordinate system y_D-z_D, which is aligned to the principal directions.

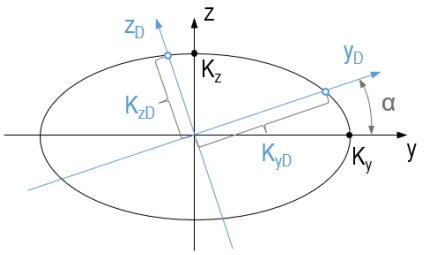


The K_{y_D} K_{z_D} buckling length factors for the design axes are interpolated along an ellipse fitted to the buckling length factors for the local *y*-*z* axes:

$$t = \operatorname{arctg}\left(\frac{K_y}{K_z}\operatorname{tg}\,\alpha\right)$$

$$K_{y_D} = \sqrt{\left(K_y \cos t\right)^2 + \left(K_z \sin t\right)^2}$$

$$K_{z_D} = \sqrt{\left(K_y \sin t\right)^2 + \left(K_z \cos t\right)^2}$$



Classification of cross-section

The program performs the classification of the cross-section based on EN 1993-1-1, Table 5.2. Whenever possible, it determines the class based on the actual stress distribution of combined compression and bending. The classification is conducted at several cross-sections along the length of the member, and the highest class is assigned to the member. If there is no compression in any point of the cross-section, it is automatically classified as Class 1. In envelope and critical combinations classification is performed according to the internal forces of the combination. The program determines the elastic stress distribution induced by internal forces when evaluating Class 3 and 4 sections, and the plastic stress distribution when evaluating Class 1 and 2 sections, for any combination of axial load, biaxial moment, and bimoment. For Class 4 cross-sections, the classification is performed through an iterative process that accounts for the displacement of the centroid.

Checks

Axial Force-Bending-Shear [N-M-V] (EN 1993-1-1, 6.2.1, 6.2.8)
Compression-Bending-Buckling (flexural in plane or torsional) [N-M-Buckl.] (EN 1993-1-1, 6.3.3)
Axial force-Bending-Lateral Tors. Buckling [N-M-LTBuckl.] (EN 1993-1-1, 6.3.3)
Shear /y [Vy] (EN 1993-1-1, 6.2.6)
Shear /z [Vz] (EN 1993-1-1, 6.2.6)
Web Shear-Bending-Axial Force [Vw-M-N] (EN 1993-1-1, 6.2.1, 6.2.8)

Resistances

Plastic resistance (axial) [$N_{pl,Rd}$] (EN 1993-1-1, 6.2.4)
Effective resistance (when subjected to uniform compression) [$N_{eff,Rd}$] (EN 1993-1-1, 6.2.4)
Elastic shear resistance /y axis [Vel,y,Rd] (EN 1993-1-1, 6.2.6)
Elastic shear resistance /z axis [Vel,z,Rd] (EN 1993-1-1, 6.2.6)
Plastic shear resistance /y axis [$V_{pl,y,Rd}$] (EN 1993-1-1, 6.2.6)
Plastic shear resistance /z axis [$V_{pl,z,Rd}$ (EN 1993-1-1, 6.2.6)
Shear web buckling [$V_{b,Rd}$] (EN 1993-1-5, 5.2-3)
Elastic moment Resistance /yy [$M_{el,y,Rd}$] (EN 1993-1-1, 6.2.5)
Elastic moment Resistance /zz [$M_{el,z,Rd}$] (EN 1993-1-1, 6.2.5)
Plastic moment Resistance /yy [$M_{pl,y,Rd}$] (EN 1993-1-1, 6.2.5)
Plastic moment Resistance /zz [$M_{pl,z,Rd}$] (EN 1993-1-1, 6.2.5)
Moment resistance for effective cross-section subjected to ending around axis y [$M_{pl,y,Rd}$] (EN 1993-1-1, 6.2.5)
Moment resistance for effective cross-section subjected to bending around axis z [$M_{pl,z,Rd}$] (EN 1993-1-1, 6.2.5)
Minimal buckling (flexural in plane or torsional) resistance [$N_{b,Rd}$] (EN 1993-1-1, 6.3.1)
Lateral torsional buckling resistance [$M_{b,Rd}$] (EN 1993-1-1, 6.3.2, ENV 1993-1-1, Appendix F1.2)

This information is given by the program as auxiliary results. The checks are mostly defined by interaction formulae. The definition and the detailed conditions of the application of the variables contained by the equations can be found in the design code.

In the following, $N_{Rk} = f_y A$; $M_{y,Rk} = f_y W_y$ and $M_{z,Rk} = f_y W_z$
where $W_y = W_{pl,y}$ and $W_z = W_{pl,z}$ for class 1 or 2 cross sections,
$W_y = W_{el,y}$ and $W_z = W_{el,z}$ for class 3 cross sections and
$W_y = W_{eff,y}$ and $W_z = W_{eff,z}$ for class 4 cross sections.

Axial Force-Bending-Shear

The member can be in tension or in compression. The check is performed on the basis of EN 1993-1-1, 6.2.1 (7).

$$\frac{N_{Ed}}{N_{Rk}/\gamma_{M_0}} + \frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rk}/\gamma_{M_0}} + \frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{z,Rk}/\gamma_{M_0}} \leq 1$$

$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$: it differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis *y*.
$\Delta M_{z,Ed} = N_{Ed} \cdot e_{N,z}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis z.

High shear

If the shear force is greater than 50% of the shear resistance, the effect of shear force is considered as detailed below.

For section class 1. and 2. allowance is made on the resistance moment accoding to EN 1993-1-1, 6.2.8.

For section class 3. and 4. stresses are calculated and the general and conservative formula in EN 1993-1-1, 6.2.1 (5) is applied. This is done for section types: I, T, C, box and pipe. For other section types (L shape, rectangular and round sold shapes, and user defined shapes) the effect of hight shear has to be calculated by the user.

☞ **For cross-section class 4, the program carries out the normal-force-shear-bending-torsion interaction analy-sis, using von Mises stress calculation based on the effective cross-section.**

Plastic resistance check

For I, pipe and box shaped sections in section class 1. and 2., the resistance check is performed according to EN 1993-1-1 6.2.10. Allowance is made for the effect of both shear force and axial force on the resistance moment. Besides resistance check of pure axial force and pure shear force, the following criteria should be satisfied:

$$\frac{M_{y,Ed}}{M_{N,y,Rd}} \leq 1; \frac{M_{z,Ed}}{M_{N,z,Rd}} \leq 1$$

where $M_{N,y,Rd}$,$M_{N,z,Rd}$ are reduced moment resistances based on the effect of shear force and axial force (EN 1993-1-1 6.2.8. and 6.2.9.1). For pipe sections, the reduced moment is calculated as follows:

$$M_{N,y,Rd} = 1.04 \cdot \left(1 - \rho - \frac{n^{1.7}}{(1-\rho)^{0.7}}\right);\ \ n = \frac{N_{Ed}}{N_{pl,Rd}};\ \ \rho = \left(2\frac{V_{Ed}}{V_{pl,z,Rd}} - 1\right)^2$$

$$M_{N,z,Rd} = 1.04 \cdot \left(1 - \rho - \frac{n^{1.7}}{(1-\rho)^{0.7}}\right);\ \ n = \frac{N_{Ed}}{N_{pl,Rd}};\ \ \rho = \left(2\frac{V_{Ed}}{V_{pl,y,Rd}} - 1\right)^2$$

For bi-axial bending the criterion in EN 1993-1-1 6.2.9.1. (6) should be satisfied:

$$\left[\frac{M_{y,Ed}}{M_{N,y,Rd}}\right]^{\alpha} + \left[\frac{M_{z,Ed}}{M_{N,z,Rd}}\right]^{\beta} \le 1$$

For 7 DOF beams with a symmetric or double symmetric I or C cross-sections another term representing the effect of the flange bending (bimoment) due to warping is added to these formulas [41; 3.1.1 and 3.1.2]:

$$\frac{N_{Ed}}{N_{Rk}/\gamma_{M_0}} + \frac{M_{y,Ed}}{M_{y,Rk}/\gamma_{M_0}} + \frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M_0}} + \frac{M_{w,Ed}}{M_{f,Rk}/\gamma_{M_0}} \le 1$$

$$\frac{M_{w,Ed}}{M_{N,f,Rd}} \le 1$$

$$\left[\frac{M_{y,Ed}}{M_{N,y,Rd}}\right]^{\alpha} + \left[\frac{M_{z,Ed}}{M_{N,z,Rd}} + \frac{M_{w,Ed}}{M_{N,f,Rd}}\right]^{\beta} \le 1$$

where $M_{f,Rk}$ and $M_{N,f,Rd}$ is the memnt resistance of the weaker flange, $M_{w,Ed}$ is an additional bending moment on flanges due to the B_{Ed} bimoment:

$$M_{w,Ed} = \frac{B_{Ed}}{h - \frac{t_f}{2} - \frac{t_{f_2}}{2}}$$

Compression-Bending-Buckling

The check is based on EN 1993-1-1, 6.3.3 (6.61) and (6.62):

$$\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M_1}} + k_{yy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rk}/\gamma_{M_1}} + k_{yz}\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \le 1$$

$$\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M_1}} + k_{zy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rk}/\gamma_{M_1}} + k_{zz}\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \le 1$$

$$(\chi_{LT} = 1.0)$$

$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis y.
$\Delta M_{z,Ed} = N_{Ed} \cdot e_{N,z}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis z.

Axial Force-Bending-Lateral Torsional Buckling

When determining the lateral-torsional buckling resistance, it is assumed that the cross section is constant and symmetric to the local *z* axis. It is also assumed the the loads act in the plane of symmetry, that is the plane of bending (except for C/U cross sections, see below). The value of *k* (ENV 1993-1-1, F1.2) is taken equal to K_z (buckling length factor). The weak axis should be the local z axis.

The check is based on the form of equations (6.61) and (6.62) of EN 1993-1-1, 6.3.3 :

$$\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M_1}} + k_{yy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} M_{y,Rk}/\gamma_{M_1}} + k_{yz}\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \le 1$$

$$\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M_1}} + k_{zy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} M_{y,Rk}/\gamma_{M_1}} + k_{zz}\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \le 1$$

$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$: it differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis y.

$\Delta M_{z,Ed} = N_{Ed} \cdot e_{N,z}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis z

χ_{LT} is calculated according to EN 1993-1-1 6.3.2.2 or 6.3.2.3.

As a default, the determination of the $k_{yy}, k_{yz}, k_{zy}, k_{zz}$ interaction factors is based on EN 1993-1-1, Appendix B Method 2 (Tables B.1 and B.2). Using Method 1 of Appendix A can be set in the Standard window.

The equivalent uniform moment factors C_{my}, C_{mz}, C_{mLT} are listed in Table B.3.

The moment diagram section between the support points is selected based on the buckling length.

- If $K > 1$, the moment diagram should be taken into account for a section that is longer than the member, so the program uses the most conservative value $C_{mz} = 1$.
- If $0.5 < K \leq 1$, the program uses the moment diagram of the design member.
- If $K < 0.5$, it is assumed that there is at least one intermediate support. In this case, the program takes into account a section of the moment diagram around the minimum and maximum. The length of the section is the buckling length. The higher of these two values will be the moment-factor.
- If $K = 0.5$, the highest C_m obtained from the previous two methods is used.

For tensile axial force, the check is performed using the effective moments based on ENV 1993-1-1, 5.5.3.

The lateral torsional check of beams with U or C cross sections is based on the method of Snijder et al. (*Design rules for lateral torsional buckling of channel sections subject to web loading*, Stahlbau, Vol. 77, 2008). The method assumes that the eccentric load is parallel to the web and acts in the plane of the web. The method was derived for beams with UPE cross section in which the span-to-height ratio is between 15 and 40.

The stability interaction formula is valid for beams that have an axis of symmetry in their plane of bending. For C and U cross sections, the results are only informative.

☞ **Compared to earlier versions of the software, the lateral torsional buckling of beams with C or U cross section is no longer checked in the plane of symmetry, but it is checked for an eccentric load that is parallel to the web, and that goes through the center of the web.**

For 7DOF beams with a symmetric or double symmetric I or C cross-section the interaction formula includes the effect of flange bending (bimoment) due to warping according to the formula A.1 in the Annex A of EN 1993-6 [41]:

$$\frac{M_{y,Ed}}{\chi_{LT} M_{y,Rk}/\gamma_{M_1}} + C_{mz}\frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} + k_{wLT}k_{zwLT}k_{\alpha LT}\frac{M_{w,Ed}}{M_{f,Rk}/\gamma_{M_1}} \leq 1$$

Shear /y The check is performed on the basis of EN 1993-1-1, 6.2.6.

$$\frac{V_{y,Ed}}{V_{c,y,Rd}} \leq 1$$

Elastic sections:

$$V_{c,y,Rd} = \frac{I \cdot t}{S}\frac{f_y}{\sqrt{3}\gamma_{M_0}}$$

For supported sections see tables below.

Plastic or non-supported sections:

$$V_{c,y,Rd} = A_v\frac{f_y}{\sqrt{3}\gamma_{M_0}}$$

Shear /z The check is performed on the basis of EN 1993-1-1, 6.2.6.

$$\frac{V_{zEd}}{\min(V_{czRd}, V_{bRd})} \leq 1$$

$V_{b,Rd} = V_{bw,Rd}$: The resistance is calculated with the contribution of the web, but not the flanges.

Elastic sections:

$$V_{c,z,Rd} = \frac{I \cdot t}{S}\frac{f_y}{\sqrt{3}\gamma_{M_0}}$$

For supported sections see tables below.

$$V_{c,z,Rd} = A_v\frac{f_y}{\sqrt{3}\gamma_{M_0}}$$

☞ **In case of 2L, 2U, 2I and 4L sections the elastic/plastic shear cross-section is equal to twice / four times of a single cross-section. This is a conservative approach assuming that the constituting elements are closely fitted.**

☞ **No formula can be applied to arbitrary shapes. The value is taken from the cross-section table: Ay, Az.**

Web Shear-Bending-Axial Force

The check is performed for cross-sections with web (I and box sections) based on EN 1993-1-5 7.1, 6.2.8, 6.2.9 assuming that the web is parallel to the local z axis.

$$\frac{M_{Ed}}{M_{pl,Rd}} + \left(1 - \frac{M_{fRd}}{M_{pl,Rd}}\right) \cdot \left(2\frac{V_{Ed}}{V_{bw,Rd}} - 1\right)^2 \leq 1$$

In case of high shear force or high axial force formulas in EN 1993-1-1 6.2.8, 6.2.9 are applied.

Basic section types

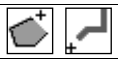
Section type		*N-M-V Stress*	*N-M-Buckling*	*N-M-LT buckling*	*Shear Vy*	*Shear Vz*	*Shear buckling*	*Effective section*
I		✓	✓	✓	✓	✓	✓	✓
Single symmetric I		✓	✓	✓	✓	✓	✓	✓
T		✓	✓	✓	✓	✓	—	✓
Box		✓	✓	✓	✓	✓	✓	✓
Welded box		✓	✓	✓	✓	✓	✓	✓
Pipe		✓	✓	✓	✓	✓	—	—
L		✓	✓	—	✓	✓	—	✓
L equal		✓	✓	*if bending acts in the plane of symmetry + Mcr is required*	✓	✓	—	✓
U		✓	✓	*approximative*	✓	✓	—	✓
C		✓	✓	*approximative*	✓	✓	—	✓
Round		✓	✓	✓	✓	✓	—	—
Rectangular		✓	✓	✓	✓	✓	—	—

Double-sections [(*)]

Section type		*N-M-V Stress*	*N-M-Stability*	*N-M-LT buckling*	*Shear Vy*	*Shear Vz*	*Shear buckling*	*Effective section*
2I		✓	✓	—	✓	✓	—	✓
2I *if a=0 → welded box*		✓	✓	✓	✓	✓	✓	✓
2L		✓	✓	—	✓	✓	—	✓
2L *if a=0 → T*		✓	✓	✓	✓	✓	—	✓
2U opened][		✓	✓	—	✓	✓	—	✓
2U opened][*if a=0 → I*		✓	✓	✓	✓	✓	✓	✓
2U closed []		✓	✓	—	✓	✓	—	✓
2U closed [] *if a=0 → Box*		✓	✓	✓	✓	✓	✓	✓

2LX		✓	✓	—	✓	✓	—	✓
4X		✓	✓	—	✓	✓	—	✓
4X *if a=0 → X*		✓	✓	—	✓	✓	—	✓

Other section types

Section type		*N-M-V Stress*	*N-M-Stability*	*N-M-LT buckling*	*Shear Vy*	*Shear Vz*	*Shear buckling*	*Effective section*
Z		✓	✓	—	✓	✓	-	✓
J		—						
Asymmetric C		—						
Asymmetric Z		—						
S		—						
X		✓	✓	✓	✓	✓	—	✓
Arc		✓	✓	—	✓	✓	—	—
Half circle		✓	✓	—	✓	✓	—	—
Reg. polygon shape		✓	✓	—	✓	✓	—	✓
Wedged I (***)		✓	✓	✓	✓	✓	✓	✓
Double wedged I (***)		✓	✓	✓	✓	✓	✓	✓
Complex/ Other (**)		✓	✓	—	✓	✓	—	—

(*) For double-section types, it is assumed that the connection between the two elements is continuous (EN 1993-1-1 6.4.4: Closely spaced built-up members), and the buckling check is performed accordingly. If the distance between the two sections is zero, the program will assume that the connection between the elements is continuous and will replace the two with one section (I, T, X or box). The connection needs to be calculated by the user.

(**) These sections are designed only if local coordinates are the same as principal directions.

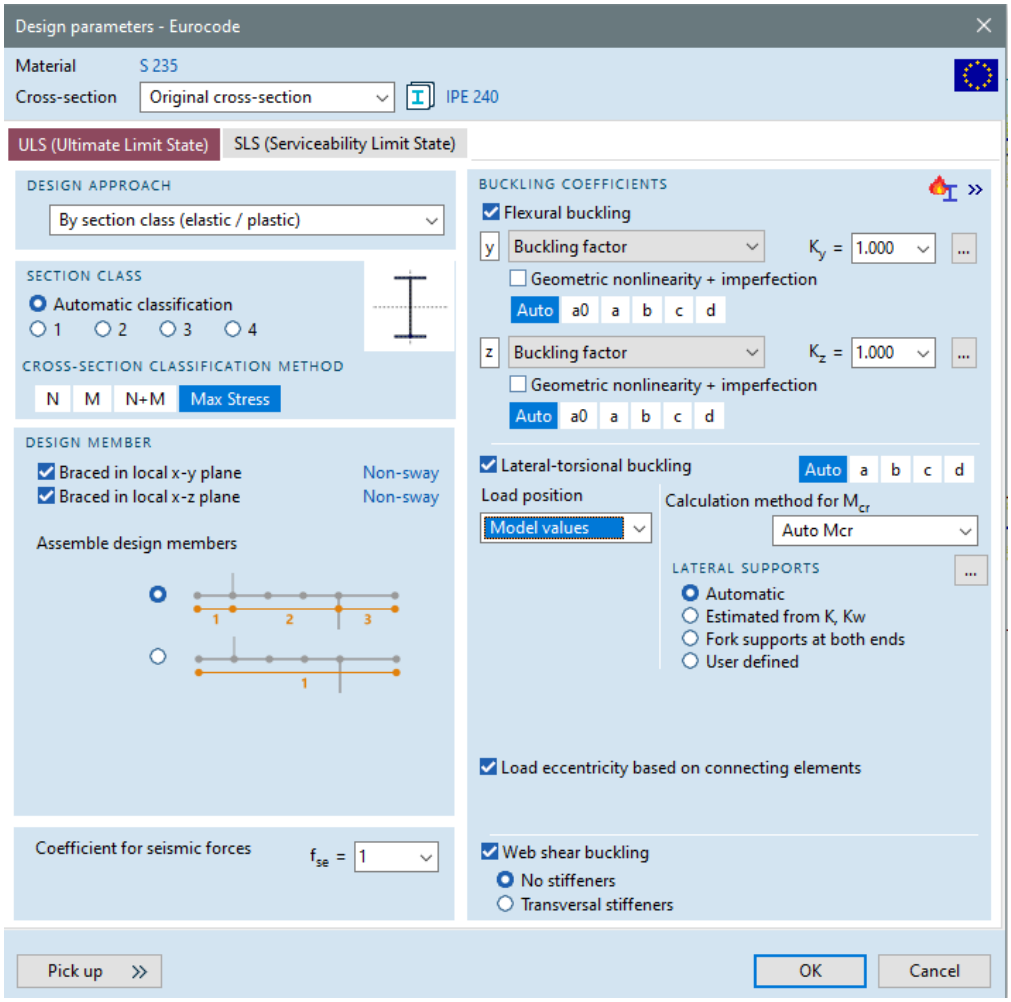
(***) Calculate with the total height of the cross-section, stiffening effect of the middle flanges is neglected.

☞ **If the manufacturing process of the section is cold-formed or other, the member is not designed.**

6.6.1.1 ULS design parameters

For the design based on Eurocode 3, the following design parameters should be defined and assigned to the structural members to perform ULS checks:

ULS (Ultimate Limit State)



Cross-section

For the steel design, a cross-section size can be defined, that is different from the *original cross-section* used for the static calculation. If the cross-section size is modified, the steel design checks will be performed for the original internal forces, but with the modified cross-section size. Similarly, in case of designing for fire, the original fire reduction factors are used (see **6.6.3 Steel beam fire design according to Eurocode 3 – SD8 module**). When viewing the results, a modified steel design member is indicated by a * after the name of the cross-section, to distinguish it from the original cross-section. In the drop-down menu, we can select from the cross-sections available in the model.

Clicking on this icon will allow us to load further cross-sections from the Cross-section library or by defining a parametric shape. On the right side of this icon, the original cross-section sizes are displayed.

Clicking on the *Replace cross-sections* icon updates the actual cross-section of the selected members to the cross-section entered in the Design parameters dialog.

Design approach

By section class (elastic/plastic): both elastic and plastic design methods are allowed, depending on the section class of the structural member.

Elastic design: all checks use elastic design methods. Resistances are calculated from elastic cross-section properties; in Class 4 effective cross-section properties are used.

Section class

Automatic classification classifies the cross-section by the actual stress values.

Classification method

N M N+M Max. stress

Classification method controls the considered forces during the automatic classification
N: Axial forces only; M Bending moments only; N+M: Axial forces and bending moments; Max Stress: Cross-section with maximum stress

Design member bracing

If the design member has a sway frame buckling mode in local *x-y* or *x-z* plane the respective bracing must be turned off. These settings affect the automatic flexural buckling calculation (AutoNcr), and in case of lateral torsional buckling the lateral supports for the AutoMcr method. Furthermore, they affect equivalent uniform moment factors *Cmy* and *Cmz* of the stability interaction check (EN 1993-1-1 Annex B: Method 2: Table B.3).

Classification method

Classification method controls the considered forces during the automatic classification
N: Axial forces only; M Bending moments only; N+M: Axial forces and bending moments

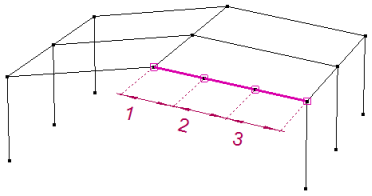
Assemble design members

The program assembles design members from the selected elements before performing design calculations. Design members consist of finite elements with the same material, same type (truss, beam or rib) and local system orientation. Design members must have constant or linearly varying cross-section and excentricity along the whole member. Finite elements must be on the same line.

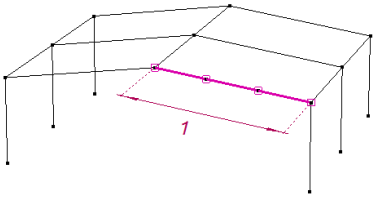
☞ **Steel design members are not the same as the structural members**
(see 3.2.14 Assemble structural members)

The program allows two methods to define design members as follows:

Any node of a selection set of finite elements where another finite element is connected will become an end point of a design member within the selection set of finite elements.



The finite elements in the selection set become one design member irrespective of other finite elements connecting to its nodes.



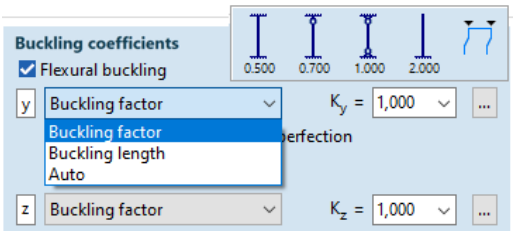
Factor for seismic forces see **4.10.25 Seismic loads – SE1 module**

Stability parameters

AXISVM performs checks against buckling, lateral-torsional buckling and web shear buckling. Each check can be activated separately by clicking on the checkbox before its name. For example, if it is sure that there is no need to check lateral-torsional buckling that part of the check can be deactivated and no parameters has to be specified.

6.6.1.1.1 Buckling (flexural)

To determine the K_y and K_z buckling resistance around *y* and *z* axes (in the local *x-z* and *y-z* plane) end support conditions must be defined.
There are three ways to specify buckling behaviour: *Buckling factor*, *Buckling length* and *Auto*.
Buckling factor or buckling length can be entered directly or can be taken from a shape obtained from buckling analysis of the model clicking the ... button.



Buckling factor

K_y, K_z buckling factors (effective length factors). The buckling factor will be multiplied by the length of the design members and not the total length of the selected elements. Note that connecting elements might lead to separation of a selected member into multiple design members (see *Assemble design members* above)!
If a buckling analysis of the model has been performed (**6.2 Buckling**) AXISVM can determine buckling factors from a selected shape.

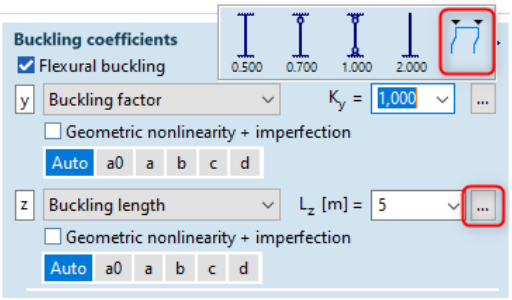
Buckling length

L_y, L_z buckling lengths. The entered buckling length will be used regardless of the length of the design member. This often leads to more straightforward design in case of complex structures.

Selecting a buckling shape

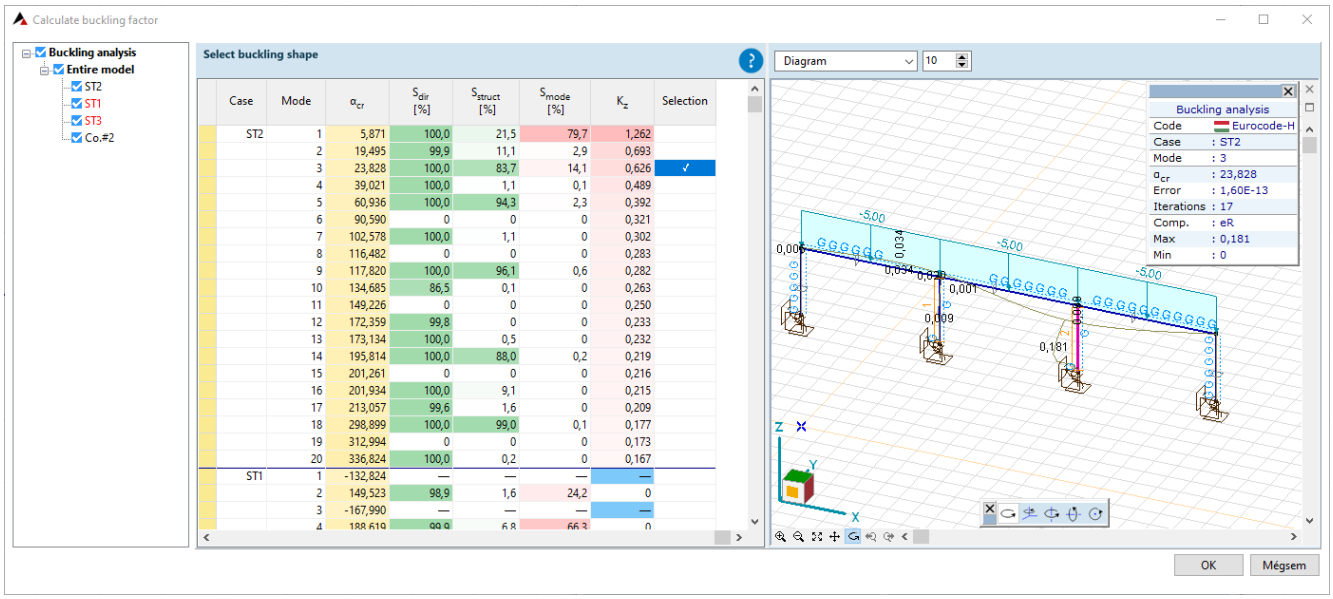


Both the buckling factor and the buckling length can be obtained from buckling results. The S_{dir}, S_{struct}, S_{mode} sensitivity factors help to select the relevant case.
The pop-up window has three parts. The tree on the left displays the load cases and combinations for which the buckling analysis was completed.
Load cases / combinations where the maximum of the calculated α_{cr} critical load multipliers is less than 25 are displayed in **red**.



These buckling results may be incomplete. The number of shapes calculated may have been set too low, or there may have been too many similar (possibly local) buckling shapes.
Load cases / combinations where the buckling results were calculated in X7 or earlier versions are displayed in **orange**. These cases allow only limited use as pre-X8 version analyses did not calculate sensitivity factors for shapes.
The model is displayed on the right with the first (or the selected) buckling shape. The buckling shape can be selected from a table having the following columns:

Case	Load case or load combination
Mode	Buckling shape's number
α_{cr}	Critical load multiplier
S_{dir}	Directional sensitivity factor
S_{struct}	Structural sensitivity factor
S_{mode}	Buckling mode shape sensitivity factor
Ky / Ly	Buckling factor / buckling length
Selection	Click in the "selection" column to select the buckling shape



How to select the proper buckling shape?

High S_{mode} values shows which buckling shapes are recommended to investigate. In a simple case, the one with the smallest eigenvalue is the appropriate one. A reduced S_{dir} or S_{struct} value indicates that the buckling shape is non-pure or non-local.
Before selection it is recommended to check the buckling shapes visually.

Limitations

The calculation can be applied for only one selected design member. Buckling analysis is required to obtain the sensitivity factors.

☞ **If buckling results were calculated with previous versions, sensitivity indexes are not be calculated, only *Ky / Ly* values from Euler formulae.**

Sensitivity factors

Directional sensitivity factor

$$S_{dir} = \frac{W_{e,d,i}}{W_{e,i}}$$

It shows the fraction of the strain energy of the *design member* due to buckling about a certain axis. For pure buckling about the certain axis, the value is 100%.

A reduced value indicates the presence of buckling in the other plane or flexural torsional buckling. In the latter case, the displacement in the plane may also belong to a lateral torsional buckling.

Structural sensitivity factor

$$S_{struct} = \frac{W_{e,d,i}}{W_{tot,i}}$$

It shows the fraction of the total strain energy of the *whole structure* due to the buckling of the design member about a certain axis. For local buckling of the design member, the value is 100%. A reduced value indicates that the loss of stability is not or not only due to the selected design member.

Buckling mode shape sensitivity factor

$$S_{mode} = \frac{W_{e,d,i}}{\sum_{(i+)} W_{e,d,i}}$$

The ratio of the strain energy of the design member in the certain buckling shape to the sum of the strain energy for the calculated buckling shapes with positive eigenvalue. A high value indicates that the deformation of the design member about the buckling axis is significant in that shape. This value is the best indicator which buckling shapes are recommended to investigate.

Mathematical appendix

Strain energies

Elastic strain energy stored in the whole structure in the i-th buckling mode shape:

$$W_{tot,i} = \boldsymbol{u}_i^T \boldsymbol{K} \boldsymbol{u}_i$$

Elastic strain energy stored in the design member in the i-th buckling mode shape:

$$W_{e,i} = \boldsymbol{u}_{e,i}^T \boldsymbol{K}_e \boldsymbol{u}_{e,i}$$

Elastic strain energy stored in the design member due to deflection about the certain buckling axis in the i-th buckling mode shape:

$$W_{e,d,i} = \boldsymbol{u}_{e,i}^T \boldsymbol{K}_{e,d} \boldsymbol{u}_{e,i}$$

Notations:

Indices

i	Index of the buckling mode shape.
e	Quantity associated with the design member under consideration
$d = \{y, z\}$	Quantity associated with the certain buckling axis (*y* or *z*).

Vectors

$\boldsymbol{u}_i$	Displacement vector of the whole structure in the i-th buckling mode shape.
$\boldsymbol{u}_{e,i}$	Displacement vector of the design member in the i-th buckling mode shape.

Matrices

$\boldsymbol{K}$	Elastic stiffness matrix of the whole structure
$\boldsymbol{K}_e$	Elastic stiffness matrix of the design member.
$\boldsymbol{K}_{e,d}$	„Reduced" elastic stiffness matrix of the design member. It contains only the entries associated with the translational and rotational degrees of freedom in a certain buckling plane.

If the *Steel design* tab is selected the calculated *Ky* or *Kz* buckling length factors can be displayed by selecting *Ky* or *Kz* as a result component from the *Resistances* group.

Auto

If this option is selected the buckling length of design members is calculated automatically. The so-called AutoN_{cr} method determines buckling length based on the model geometry and the distribution of internal forces in the model. The buckling length of each design members is calculated after considering the stabilizing effect of connected other members. This method is based on the rules recommended by the *European Convention for Constructional Steelwork (ECCS TC8: Rules for Member Stability in EN 1993-1-1: Background documentation and design guidelines)*.

The original method was developed for vertical columns of simple frames. The improved algorithm of AXISVM can handle any three-dimensional structures but for special geometries the error can be considerable. In such cases it is recommended to check if the calculated buckling length values are within the expected range. For complex structures it is also advisable to determine the critical load parameter solving an eigenvalue problem (see **6.2 Buckling**) or to perform a nonlinear analysis with material nonlinearity and geometric imperfection.

The buckling length heavily depends on whether the design member has a sway buckling mode. Check the *Braced in local x-y / x-z plane* checkboxes according to the sensitivity of the member to second-order effects. It is important to recognize for instance that columns of a portal frame typically buckle in sway mode in-plane, while its beam does not (because both end points are supported by the columns).

AXISVM takes into account the effect of all beam elements (including steel beams with no steel design parameters defined or non-steel beams). As a conservative assumption, all columns are considered to be in sway mode while all beams are in non-sway mode by default.

The algorithm takes into account hinges, rigid and semi-rigid connections. Nonlinear connections are represented by their initial stiffness. Nodal supports and constrained nodal degrees of freedom are also taken into account. Due to constrained nodal DOFs planar structures can show very small buckling length for out of plane buckling. This is not an error but the consequence of the constrained motion at the end points of finite elements. For planar structures it is recommended to specify the buckling length for out of plane buckling or to model the actual supports instead of constraining nodal DOFs.

The buckling length of design members is heavily affected by the distribution of internal forces. As these forces are different in each load case and combination, the calculated buckling length also depends on the selected load case or combination! Calculation speed can be increased in the conceptual design phase by neglecting the influence of internal force distribution; uncheck the *Take N into account* checkbox in the *Design Parameters* dialog to do so. When the internal forces are not taken into account, the design parameter under consideration is assumed to have uniform normal force distribution, while other members are assumed unloaded.

Limitations

This buckling length calculation method can determine the critical load parameter only for structures made of truss, beam or rib elements. Design members are considered to have piecewise constant cross-section. The calculation method is not available for tapered beams yet. Effects from other types of structural components (e.g., plates, springs, rigid bodies, line or surface supports) are ignored. Only nodal supports are considered; the influence of line and surface supports is not taken into account.

Additional information on the automatic flexural buckling coefficient calculation tool is available from the menu *Help / AutoNcr Guide*.

Model the second order effects with nonlinear analysis

If the internal loads already contain the influence of second order effects (accordingly we defined imperfect geometry or imperfect loads in case of geometric non-linear analysis), verifying the strength of the cross-section is sufficient. If this option is checked buckling analysis runs with $\chi_x = \chi_y = 1$ values.

Buckling of design members

Buckling of design members (Eurocode-D) [Linear, 1.35*ST1 + 1.35*Wand Wt + 1.35*Floor + 1.50*Nutzlast (ULS), (acr = 2.173)]

	Design member	Length [m]	N_{min} [kN]	N_{cr} [kN]	$L_{y,cr}$ [m]	$L_{z,cr}$ [m]	$K_{y,cr}$	$K_{z,cr}$	λ
	21(2–13) E: Dimensions	0.700	—	—	—	—	—	—	—
	22(16–15) E: Dimensions	4.000	—	—	—	—	—	—	—
	23(14–13) E: Dimensions	4.000	—	—	—	—	—	—	—
	24(9–4)	0.200	-107.975	234.609	3.951	3.951	19.753	19.753	130.086
	25(4–8)	3.100	-64.848	140.902	5.098	5.098	1.644	1.644	167.859
	26(11–2)	0.200	-107.975	234.609	3.951	3.951	19.753	19.753	130.086
	27(2–7)	3.100	-64.848	140.902	5.098	5.098	1.644	1.644	167.859
	28(12–3)	0.200	-107.975	234.609	3.951	3.951	19.753	19.753	130.086
	29(3–6)	3.100	-64.848	140.902	5.098	5.098	1.644	1.644	167.859
	30(10–1)	0.200	-107.975	234.609	3.951	3.951	19.753	19.753	130.086
	31(1–5)	3.100	-64.848	140.902	5.098	5.098	1.644	1.644	167.859
	32(8–5) E: Cold f.	5.000	—	—	—	—	—	—	—
	33(8–7) E: Dimensions	4.000	—	—	—	—	—	—	—

If a buckling analysis has been completed for the structure a summary of buckling of design members is displayed in the *Table Browser* (e.g., *RESULTS / Linear analysis / Steel design / Buckling / Load case or combination / Mode 4*). Nonlinear results are available only if geometrical nonlinearity was ignored and the final load factor is 1.0. Another table (*...Steel design / Buckling / Minimum Ncr)* displays the smallest critical force (N_{cr}) for each design member with other buckling data.

If design calculations failed for a certain member the cause of the error (wrong dimensions, wrong type etc.) is also displayed in the *Design member* column (just like in the table of *Utilization*).

The critical force is calculated from the Euler formula

$$N_{cr} = \frac{\pi^2 EI}{L_{cr}{}^2} = N_{min}\alpha_{cr}$$

The buckling length of the equivalent Euler beam is $L_{cr} = K_{cr}L$, where L is the physical length of the design member and Kcr is the buckling factor.

α_{cr}	Critical load parameter
N_{min}	The smallest normal force along the beam
N_{cr}	The critical force
$L_{y,cr}$	Buckling length about the y axis (in z direction)
$L_{z,cr}$	Buckling length about the z axis (in y direction)
$K_{y,cr}$	Buckling factor about the y axis
$K_{z,cr}$	Buckling factor about the z axis
λ	Slenderness

6.6.1.1.2 Lateral-torsional buckling

K_ω is a factor related to the constraints against warping. Its value must be between 0.5 and 1.

- if warping is not constrained it is 1.0.
- if warping is constrained at both ends of the beam, it is 0.5.
- if warping is constrained at one of the ends of the beam, it is 0.7.

See in detail: Appendix F1 of ENV 1993-1-1.

Z_a is the load position relative to the center of gravity (a signed value).

Load position can be set to *As defined* or *Custom* in the dropdown list. *As defined* means that the load position is determined from the eccentricity of a concentrated or distributed beam load. Nodal loads are applied in the center of gravity of the cross-section. If *Custom* is chosen, load position must be selected from the following options by radio buttons: *Top, Center of gravity, Bottom* or *Custom* (entering the Z_a value). The analysis assumes that the local z axis is within the plane of symmetry and that the forces causing lateral torsional buckling act parallel to this axis.

Calculation of M_{cr} (critical moment)

Two options are available to calculate the critical moment of the lateral-torsional buckling *(M_{cr})*.

1.) By formula

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{(kL)^2}\left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(kL)^2 GI_t}{\pi^2 EI_z} + \left(C_2 z_g - C_3 z_j\right)^2} - \left(C_2 z_g - C_3 z_j\right)\right]$$

Meaning of the parameters can be found in the literature or in the Appendix F1.2 of ENV 1993-1-1. The value of C_1, C_2, C_3 parameters depend on the shape of the moment curve and the *k* factors.

To enter parameter values, choose *Custom C1, C2, C3*.

In certain cases, C_1 can be calculated automatically. Choose *C1 Lopez formula* from the *Calculation method for Mcr* combo. **This option is not available** if the steel structural member is a cantilever or $k > 1$

C_2 must be entered if external loads are applied to the structural member and the point of application is not coincident with the shear center of the cross section. In case of a single-symmetric cross-section C_3 shall also be entered. C parameter values can be set using ENV 1993-1-1, F1.2.

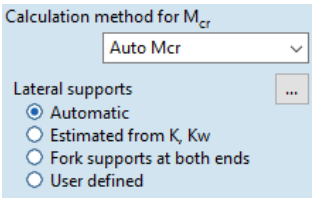
Calculating M_{cr} is also possible according to the method given in the national appendix of Dutch standard (NEN-EN 1993-1-3 Bijlage NB.NB). Based on the beam's moment diagram and its internal load the C1 and C2 coefficients are automatically calculated in a conservative way according to the given procedures and tables in the appendix (*Calculated C1,C2 (NL NAD)).* In some load cases and support conditions, it is hard to automatically choose between any of the categories covered in the appendix, therefore we let the user define C1 and C2 (*User-defined C1, C2 (NL NAD)).*

2.) The AutoMcr method

This method makes a separate finite element model for each design member and calculates *Mcr* directly for each load combination making C_1, C_2, C_3 unnecessary but increasing calculation time. This method handles variable cross-sections and cantilevers as well. The finite element sub-model of a beam contains at least 30 finite elements where each node has four degrees of freedom essential to determine lateral torsional buckling: 1) *v,* lateral shift in local *y* direction, 2) Θ_x torsion, 3) Θ_z lateral rotation, 4) *w,* warping. This method builds the beam stiffness form two parts: the first one is linear, the second one has geometric nonlinearity.

It applies loads with their eccentricity then reduces the calculation to an eigenvalue problem. The method is developed for bending constant cross-sections in their plane of symmetry, so for variable cross-sections the program creates the appropriate number of finite elements. [See Yvan Galea: *Moment critique de deversement elastique de poutres flechies presentation du logiciel ltbeam,* CTICM, 2003]

6.6.1.1.3 Lateral supports



AXISVM determines the support conditions for the sub-model automatically by default. Alternatively, the following methods may be chosen:
Estimated from k, kw: the position of the supports is estimated;
Fork supports at both ends;
User defined: the user may edit/define the supports.

The five stiffness components of a lateral supports are $S, R_y, R_{xx}, R_{zz}, R_w$.
It the mentioned option above was set to *Automatic*, the program determines the support conditions as follows:
Model support: based on the supports along the beam defined earlier in the *Elements* tab; the R_y, R_{xx} and R_{zz} values are directly taken from the support stiffness values, while $R_w = 0$. The program can import the shear field stiffness (S) set for line elements. In its absence, the stabilizing effect of the sheeting can also be entered manually in the table.

Connecting elements: line and shell elements directly connected to the designed member provide some support against lateral torsional buckling. The estimated support stiffness values are summarized in the *AutoMcr Guide*. See *Help / AutoMcr Guide*

Limitations
The sub-model does not take into account the effect of nodal DOFs.

Without having proper supports the sub-model has no stability against lateral torsional buckling. To avoid this kind of instability the following conditions must be met: 1) R_y is nonzero in at least one point and R_y or R_{zz} is nonzero in another one. 2) R_{xx} is nonzero in at least one point.
If the first condition is not met, the default settings are applied: R_y and R_{xx} is rigid at both ends. It is an approximation of the $k = k_w = 1$ in the ENV formula. For a cantilever beam, the default setting is a support on one end with rigid R_y, R_{xx} and R_{zz}.

If the second condition is not met, i.e., no torsional support was defined, the default setting is to make R_{xx} rigid at one end.



Lateral support conditions can be edited by clicking the ... button.
A dialog appears with a table of lateral supports of the design members created from the selection.
Design members are listed on the left. The table shows the lateral supports of the selected item.
The last item of the list is *Same supports on selected elements*. Selecting this item, the table shows only the common supports on the selected design members. Adding or deleting supports will change the support configuration of all selected design members.



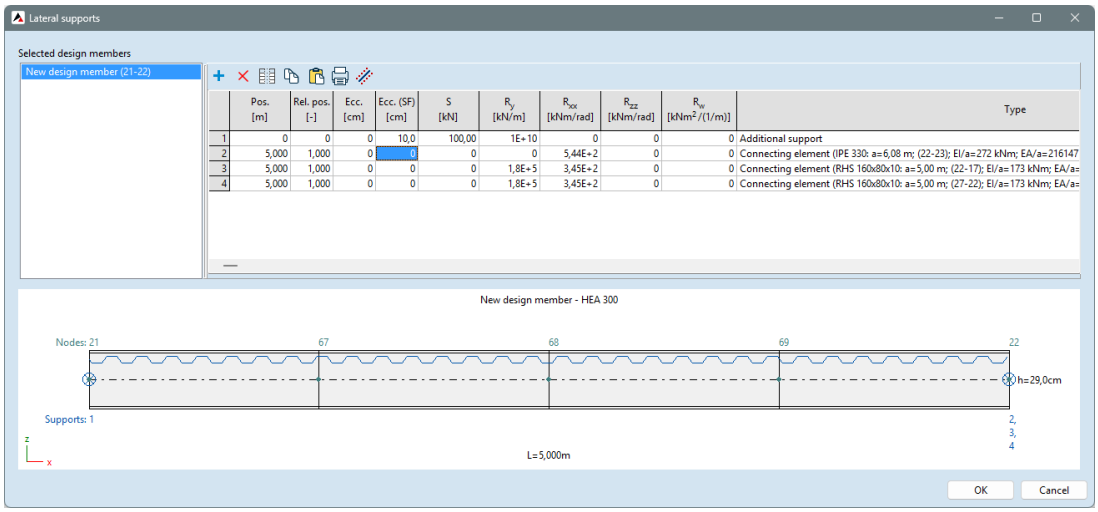
Add new support.
Adds a new line to the table and allows entering support properties.



Delete selected supports.
Deletes selected rows of the table.

The following properties can be edited: support position, eccentricity of the support in *z* direction relative to the center of gravity of the cross-section, stiffness components. The last column shows the support type:

Additional support was defined by the user.
Support from model refers to a nodal support connected to the design member.
Connecting element refers to a truss, beam or rib element connecting to the design member.
Default support refers to supports created automatically when the first condition was not met.
Default torsional support refers to supports created automatically when the second condition was not met.

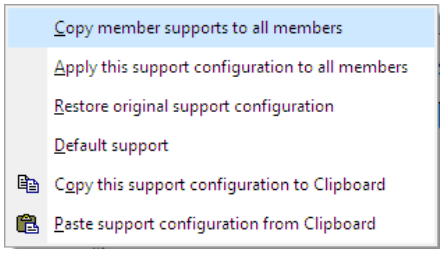


The table displays connecting elements even if the angle between their axis and the design member axis is greater than 15°, but only if the direction vector of the connecting element has a nonzero component in the local *y* axis of the design member (lateral support).
Some table fields also allow entering formulas containing numbers and certain notations:

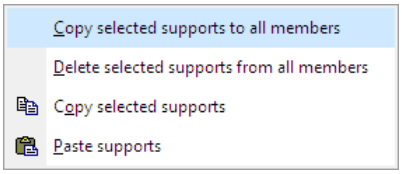
Pos. *L* is the length of the design member. So relative position can be entered like *L/2* or *2*L/3*.
Exc. *h* is the cross-section height. So, eccentricity can be entered like *h/2* or *2*h/3*.
R... Stiffness components: *m* will be interpreted as 1E+10.
In case of a connecting element with a length of *a* and an *E*I* stiffness *6*EI/a* can be entered.

Point supports are linked to a specific position, whereas shear field stiffness is assigned to the segment between two consecutive positions and is interpreted as continuous support. Accordingly, entering shear field stiffness in the last row is not required.

Right click on a list item on the left for additional operations

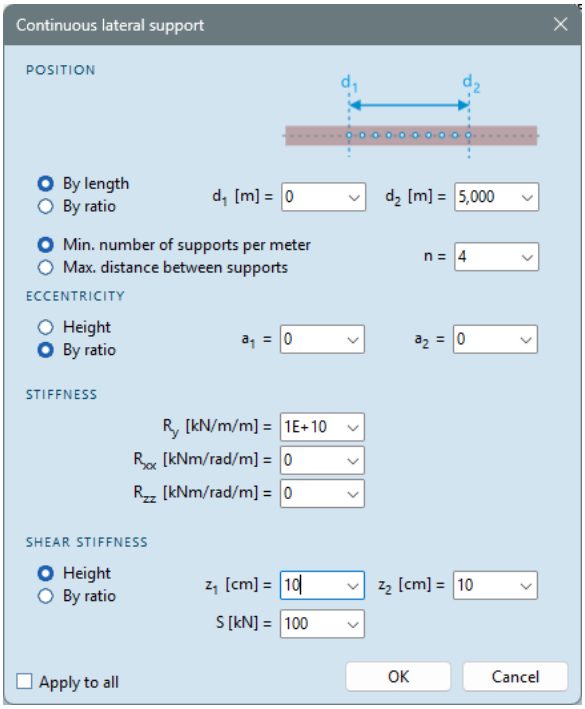


Right click on a table row for additional operations





Continuous lateral support
If a plate (e.g., metal roof panel) is connected to the designed member, its stiffness can be considered as a continuous lateral support.
In the *Continuous lateral support* window, the position, the eccentricity and the stiffness of the plate can be defined. By checking *Apply to all*, the support will be added to all the selected steel design members. Otherwise, it will be added only to the design member, that is selected in the *Lateral support* window.
The continuous support is converted to individual supports by the program. The density of the individual supports can be defined by the user (*Min. number of the supports per meter, n*). The individual supports on the end of the plate have half of the stiffnes, as those in the middle.



☞ **Changing the *Assemble design members* option will redefine design members so lateral supports will be reset to default values and all modifications will be lost.**

Load eccentricity based on connecting elements

The Auto Mcr procedure locates connected elements and determines the magnitude of the load transferred from these elements based on the variation of the shear force. If the connected elements are linked eccentrically, the program reads the magnitude of the eccentricity to account for second-order effects. In the case of custom load placement, it calculates with the transferred loads at the user-defined position. If this option is turned off, the software will not take into account the second-order effect of loads transferred from other elements.

AXISVM Version 12 estimated support conditions from k_z and k_w values. Models created with this version will appear with *Estimated from kz, kw setting*.
In order to help using and understanding the *AutoMcr* method and defining the lateral supports correctly, guidelines and examples can be found in the *AutoMcr Guide*. See *Help / AutoMcr Guide*

Load position

Z_a is the z coordinate of the point of application of the transversal load (relative to the center of gravity of the cross-section), based on ENV 1993-1-1, Figure F1.1. It is a signed value and must be defined as the ratio of this distance to the height of the cross-section. The positions of the center of gravity and the top or bottom of the cross section can also be chosen by radio buttons.

Web Shear Buckling

For shapes with webs, the web can be supported or not with stiffeners:
No stiffeners: assumes no transversal stiffeners along the structural member.
Transversal stiffeners: In any cases the program assumes that there are transversal stiffeners (non-rigid end post) at the ends of the structural members (e.g., at the supports).

You can see coefficient of seimic forces at **4.10.25 Seismic loads – SE1 module.**

6.6.1.2 SLS design parameters

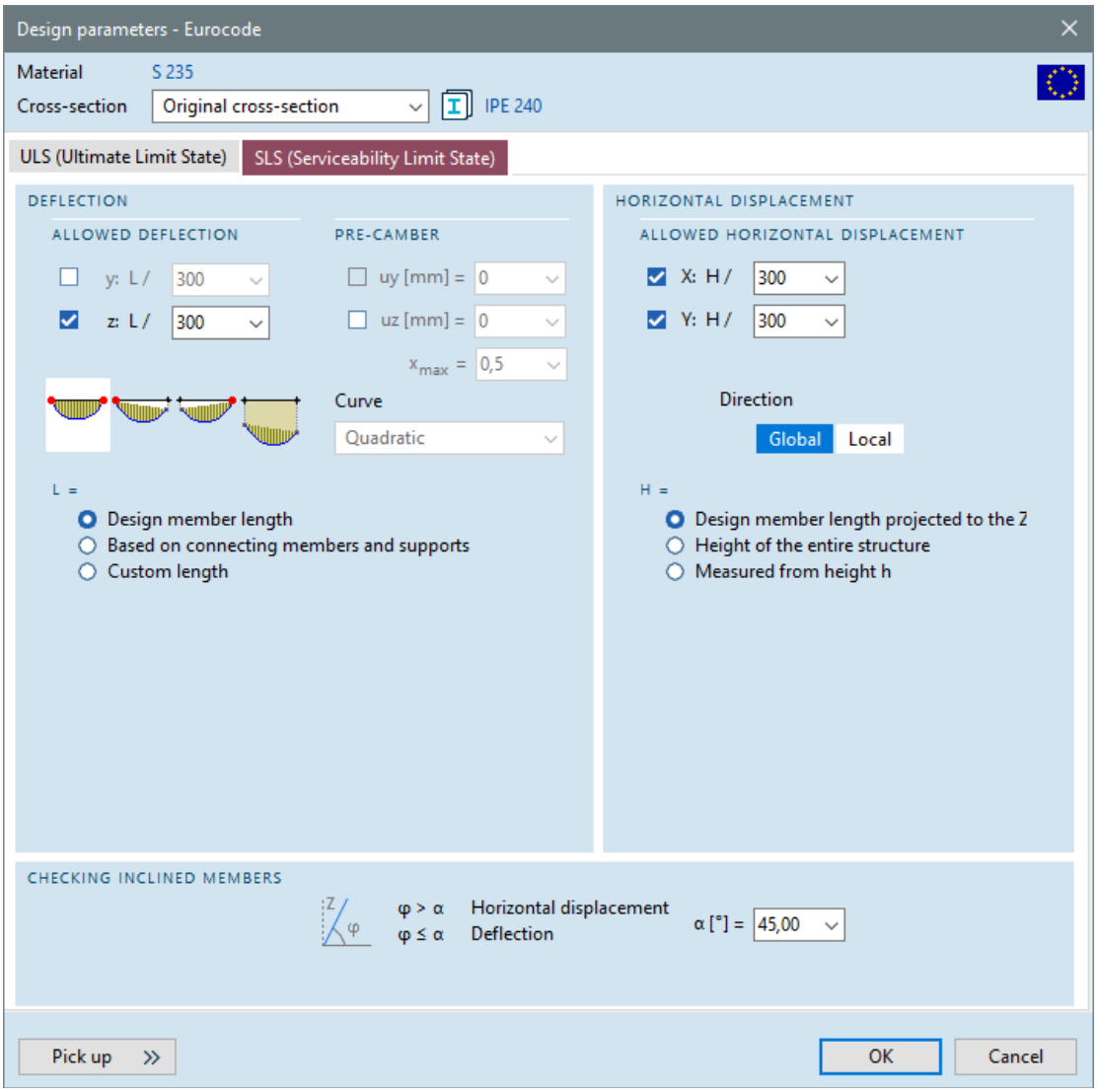


SLS

The parameters for the serviceability limit state (SLS) checks can be set on the SLS tab in the Design parameters window.

SLS

(Serviceability Limit State)

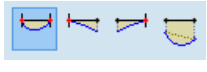


The program checks the deflections of beams and the horizontal displacements of columns. Inclined steel design members are categorized by the angle between their centerline and the horizontal plane. If this angle is above α = 45°, then the member is considered a column, and if it is below α = 45°, then it is considered a beam. This α value can be modified at *Checking inclined members*.

☞ **If the critical load combinations are generated automatically, the program performs the SLS checks in the characteristic load combinations.**

Vertical deflections

Deflections can be checked according to the *z* or *y* local axis directions of the steel design member. The check in each direction can be activated/deactivated by checking/unchecking the box before the limit values. The deflection limit values shall be set according to the length (L) detailed below.



Deflections can be considered in several ways: based on the actual displacements or based on the displacements relative only to the left, only to the right or to both end points of the design member.

Length (L)

Deflections of a design member and the deflection limits (e.g., L/300) are based on one of the chosen length values below:
Design member length: L is equal to the length of the design member;
Based on connecting members and supports: If the Assemble design members parameter in the ULS tab is set to the first option (members are broken at connecting elements and supports), then this length is equal to the design member's length, and thus this is the same as the length above. If it is set to the second option (the selected beams are checked as one design member), then the program checks if there

are any supports or connecting members along the design member in the direction of the deflection check that prevents the deflection of the design member. Between these, the SLS check is performed for each section of the member. For example, checking a 15-meter-long beam, if there are supporting columns in the end points and at one-third of the length of the beam, then the program performs SLS checks separately for the first 5-meter-long section and the remaining 10-meter-long section of the beam. *Custom length:* any value can be entered.

Pre-camber

Pre-camber can be defined for the design members, where the pre-camber amounts (*uz* and *uy*) are interpreted in the local coordinate-system. The shape of the pre-camber can be set with three parameters: *Quadratic* or *Linear* curves may be chosen, for which the maximum value *(u)* and its location along the member *(x_{max})* shall also be defined by the user. As an exception, if the actual displacements are set by the user (e.g., only a section of a beam is selected as a design member), then the pre-camber values are constant along the length. If the length L is set to the design member length or to a custom length, the pre-camber is zero in the end points of the design members. If L is *Based on connecting members and supports*, the pre-camber will be zero in the end points and at the supports found by the program in the relevant direction.

☞ **It is important to note, that the pre-camber value set here has an effect in the SLS checks only, by reducing the previously calculated deflections that is based on the original shape.**

Horizontal displacements

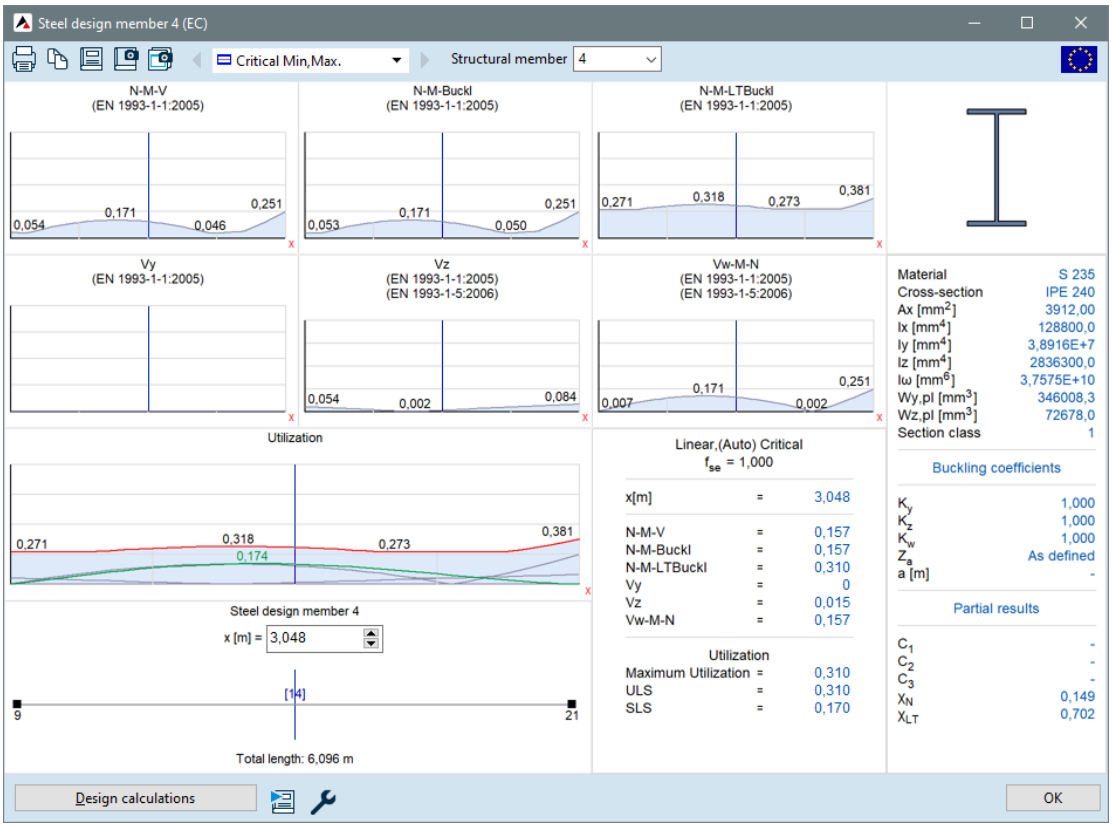
Horizontal displacements can be checked in two directions at a time, according to the global or local coordinate system. If the global coordinate system is chosen, the program checks the displacements in horizontal X and Y axis directions. If the local coordinate system is chosen, then the horizontal components of the deflections (*wx* and *wy*) are checked in the local *z* and *y* directions of the design member.
The height *(H),* along which the program performs displacement checks, can be the whole length of the structure, or only the height of the design member or the level. In the first case, the actual deflections are considered, while in the latter cases, only the relative displacements along the considered height are checked. In case of inclined members, the height is defined as the vertical component of the design member length. The height can be set by the user:

- *Design member length* (relative horizontal displacements);
- *Height of the entire structure* (actual horizontal displacements);
- *Measured from height h* (actual horizontal displacements).

6.6.1.3 Diagrams and design calculations

Diagrams

You can display the diagrams corresponding to all individual checks and their envelope by clicking on the steel design member. Results for any position of any steel design member in any load case or combination can be obtained by setting the combo boxes and dragging the tracking line. If a check cannot be performed with the current cross-section the respective diagram is replaced by a cross.



Design calculations

Clicking on the *Design calculations* button a report of the calculation details can be displayed. All strength and stability checks appear as formulas completed with substituted actual values and references to the design code.
The report consists of the six basic interaction checks listed above and several partial results which make it easier to follow the calculations and provide useful details for cross-section optimization.
The partial results are:

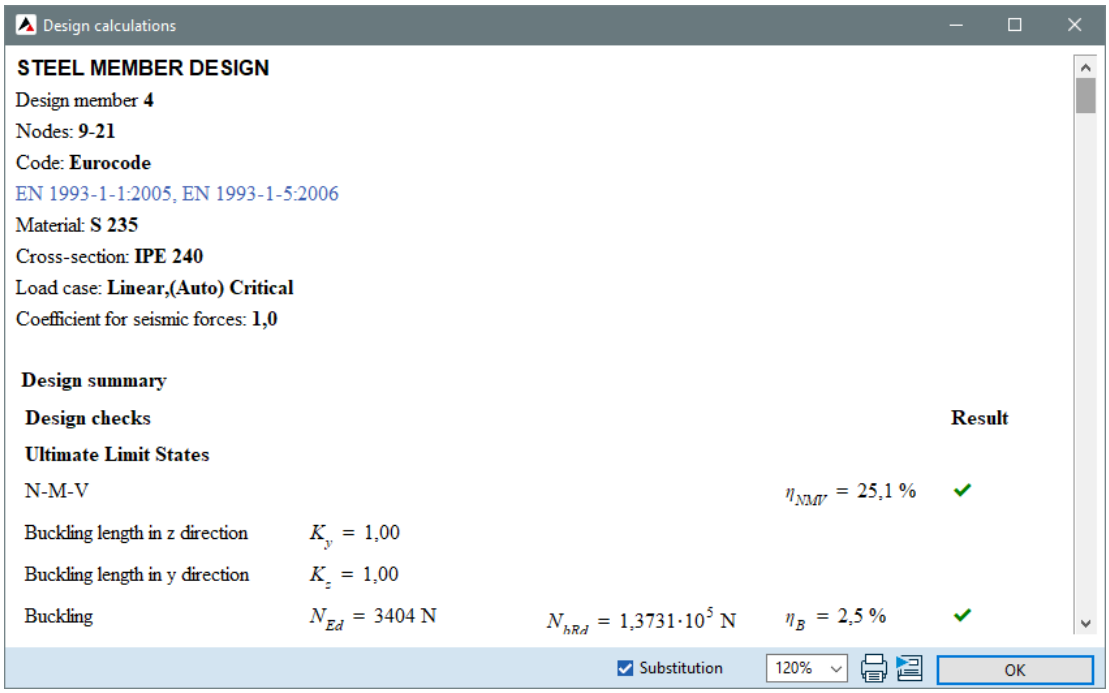
1. *Axial Plastic Resistance*
2. *Plastic Moment Resistance about the y axis*
3. *Plastic Moment Resistance about the z axis*
4. *Plastic Shear Resistance in z direction*
5. *Bending-shear interaction check*
6. *Bending-axial force interaction check*
7. *Flexural Buckling Resistance*
8. *Lateral-Torsional Buckling Resistance*



Clicking on the *Settings* icon beside the *Design calculations* button allows setting the basic units for force and length used in the design calculations. Important results also appear converted to standard AXISVM units (see **3.3.8 Units and Formats**).
A short summary of design calculations is available, which can be printed on its own or together with the detailed calculations.

Details of calculations

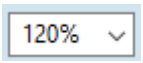
The details of calculations according to the current design code are displayed as a multi-page document. References to sections and formulas of the design code appear in blue.



This window can be resized.

Substitution

Substitution into formulae can be turned on / off. Eliminating substitution makes the report somewhat shorter.



Select the font size of the report.



Prints the design calculation.



Clicking on this icon adds the design calculation to the current report.

6.6.1.4 Steel cross-section optimization - SD9 module



SD9 module requires SD1 module.

Cross-section optimization of steel structures makes steel design members previously defined and designed more efficient by fine-tuning cross-section dimensions and reducing self-weight.

☞ **Optimization checks the design members for the same internal forces ignoring stiffness changes due to changing dimensions. In certain structures recalculation of the model may show considerable changes in internal force patterns. In these cases, several consecutive optimizations may find the more efficient structure.**

Optimization uses the steel design parameters previously assigned to the design members.

Cross-section types suitable for optimization are: I, asymmetric I, 2I, rectangular, T, C, 2U shapes and pipes. Variable cross-sections cannot be optimized.

Optimization groups

The first step of the optimization is to create optimization groups from the existing steel design members. Each member of an optimization group must have the same cross-section and optimization will assign the same cross-section to the group members. The actual optimization can be started from the second tab *(Optimization)*.

The list of optimization groups (see it on the left side of the *Design optimization groups* tab) shows the common cross-section and the number of design members within the group (<*n*>). Select a group and set the optimization parameters on the right (see below).

Objective of optimization

Objective of optimization can be (1) minimum weight, (2) minimum height or (3) minimum width.

This defines the objective function. The process will search for the cross-section with a utilization <1 for all group members and closest to the objective. This cross-section is called *optimized cross-section*. The objective is reached separately for each group.

A maximum of utilization can also be set. Limiting the utilization can help when optimizing statically indeterminate structures where cross-section modifications can cause big changes in internal forces.

Optimization checks

It is possible to ignore certain checks during the optimization process. All strength checks are always performed but checks for flexural buckling, lateral torsional buckling, web buckling and SLS can be de-activated.

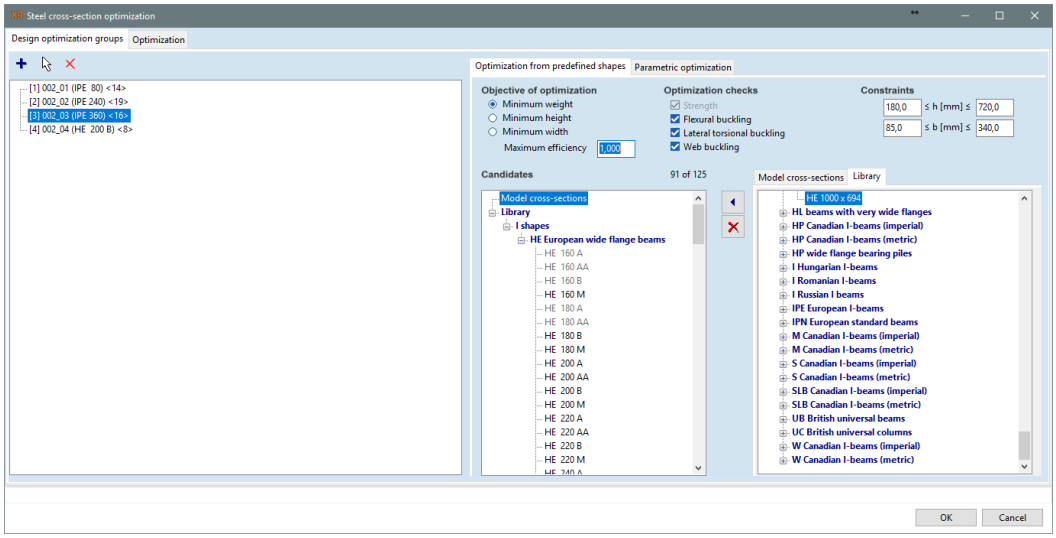
Optimization types

There are two ways to define the range of cross-sections to be checked. *Optimization from predefined shapes* works on a given number of cross-sections while *Parametric optimization* finds the optimal shape within different geometry parameter ranges.

Optimization from predefined shapes

This method finds the optimal cross-section from a given number of predefined shapes. Candidates can be selected from model cross-sections and from the library. Candidates must have the same cross-section type as the original cross-section of the group.

The range of candidates can be reduced by setting *Constraints*. Only cross-sections between the limits for height and width will be used as candidates (other sections will appear greyed).



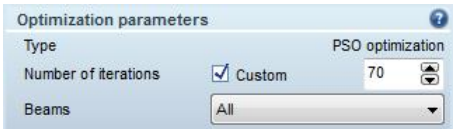
If a group contains more than one design member all members will be checked. Members are checked along their entire length. Not all candidates will be checked. The program analyses only those necessary for finding the global optimum.

Parametric optimization

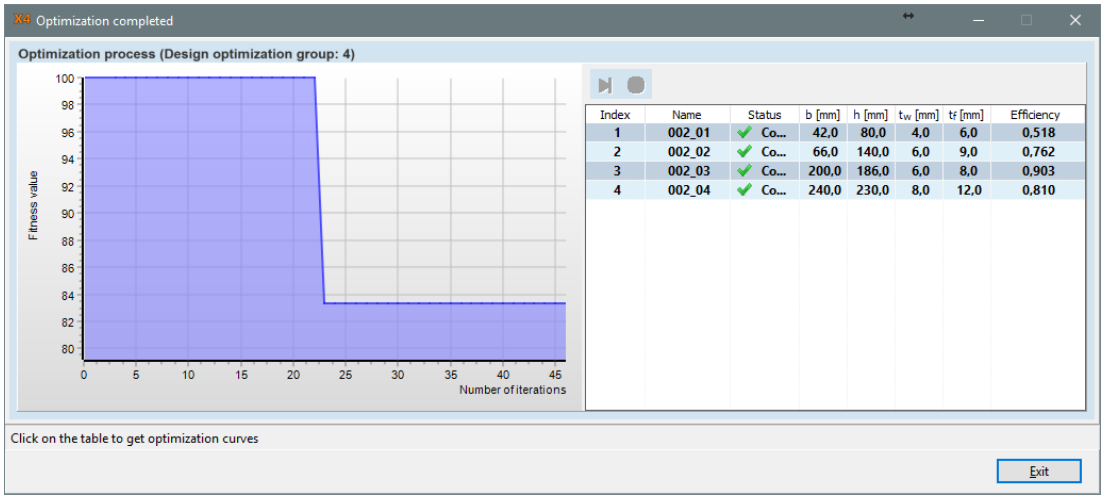
This method finds the optimal cross-section within different geometry parameter ranges.
Many different optimum search algorithms are known and used successfully for optimizing frame structures. Due to the nonlinearity of the problem and the large number of local optimums it is hard to find a global optimum with pure mathematics. It is even harder if the optimization has to perform not only strength checks but also stability analysis.
AXISVM uses the so-called *Particle Swarm Optimization* (PSO), a stochastic computational method for finding optimum. It is an evolutionary algorithm developed in the 1990s.
The PSO process runs for a given number of iterations and due to its stochastic nature, it can find multiple local optimums. The number of iterations is determined by the program trying to balance running time and the fullest possible mapping of the search-space. Moreover, if the algorithm finds no changes in the result after a long period it assumes that it is the global optimum and quits. In multi-threaded mode the search space is partitioned among the threads.

The algorithm estimates the necessary number of iterations and selects the optimization method. It can be either a simple linear search or a PSO optimization. The decision is based on the following parameters: (1) size of the search space, (2) iteration step size, (3) number of fixed parameters, (4) the objective of the optimization, (5) single- or multi-threaded mode (see *Settings / Preferences / Analysis*) (6) range of beams taken into account (all or *x%* of the most efficient beams). Reducing the number of beams taken into account make the calculation faster but reduces the precision. This is useful if optimization groups consist of many members with different utilization and the utilization comes from a check which is also performed in the optimization.

In case of PSO optimization the user can set a custom *Number of iterations,* checking *Custom*.



The calculation can start parallel search processes using all processor cores going through the search space faster. A typical graph for the optimization process is like this:



The blue line shows the decreasing population diversity on a reversed scale, the filled graph displays the fitness value in the percentage of the initial (the lower the better). If fitness value does not change through many iterations and the population diversity is small we can accept the current result by jumping to the optimization of the next group or pressing **Esc** to quit optimization.



Skip to the next group. Stops the optimization of the current group and jumps to the next one.

The current status is considered to be the result.



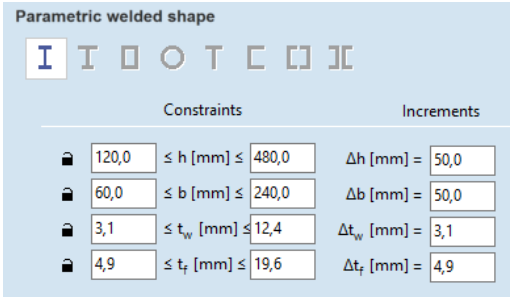
Stop the optimization. Stops the entire optimization. The current status of each group is considered to be the result. Same as pressing the **Esc** key.

The advantage and drawback of this algorithm is its stochastic nature. Running the optimization multiple times for the same problem can lead to slightly different results. This tendency is stronger in large search-spaces. For small search spaces like pipes within a narrow size range a simple linear search is made (analysing all candidates for finding the best one).

Setting constraints

Cross-section parameter constraints and increments can be defined. Clicking on the lock icon locks the parameter to its original value. To set any parameter to a desired value set the lower and upper bound to the same value.

The algorithm tries to find a cross-section which is as close to the maximum utilization as possible and is closest to the selected objective of optimization.

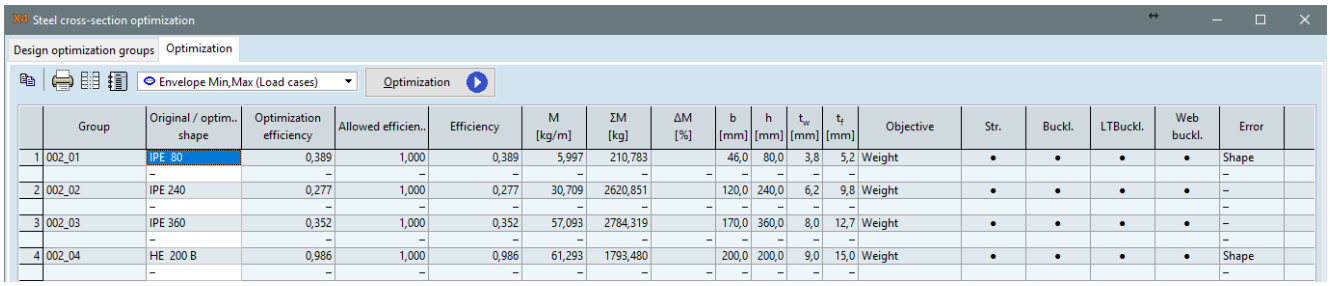


Too big intervals and/or too small increments makes the search space extremely large and as a result, the calculation time increases and/or the convergence slows down. So, it is important to set the ranges around the estimated optimum.

If an optimization group contains multiple design members, the overall utilization will be the highest utilization of the members. Therefore, it is not recommended to place members with very different length or internal forces into the same optimization group.

Optimization

After setting the parameters go to the *Optimization* tab to run the optimization for the selected load case, combination, envelope or critical combination.



	Group	Original / optim.. shape	Optimization efficiency	Allowed efficien..	Efficiency	M [kg/m]	ΣM [kg]	ΔM [%]	b [mm]	h [mm]	t_w [mm]	t_f [mm]	Objective	Str.	Buckl.	LTBuckl.	Web buckl.	Error
1	002_01	IPE 80	0,389	1,000	0,389	5,997	210,783		46,0	80,0	3,8	5,2	Weight	•	•	•	•	Shape
		–	–	–	–	–	–	–	–	–	–	–						–
2	002_02	IPE 240	0,277	1,000	0,277	30,709	2620,851		120,0	240,0	6,2	9,8	Weight	•	•	•	•	–
		–	–	–	–	–	–	–	–	–	–	–						–
3	002_03	IPE 360	0,352	1,000	0,352	57,093	2784,319		170,0	360,0	8,0	12,7	Weight	•	•	•	•	–
		–	–	–	–	–	–	–	–	–	–	–						–
4	002_04	HE 200 B	0,986	1,000	0,986	61,293	1793,480		200,0	200,0	9,0	15,0	Weight	•	•	•	•	Shape
		–	–	–	–	–	–	–	–	–	–	–						–

The table displays the group parameters and the results of the optimization (weight per length unit, weight reduction, width and height. The *Opt.* column can be used to control which group is to be optimized.

If optimization was based on predefined shapes a dropdown list can be opened with all checked and usable cross-sections sorted from higher utilization to lower.

The *Replace* column controls which cross-sections are to be replaced. Clicking on the *Replace cross-sections* button will actually replace the cross-sections in the selected groups.

☞ **Cross-section optimization may be a time-consuming task depending on the range of candidates and the size of groups so in case of a large search space (e.g., parametric optimization) it is recommended not to choose the slower *AutoMcr* method for lateral buckling calculations.**

6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 -SD3 module

EUROCODE 3

This module focuses on the proper analysis and design of cold-formed steel sections. It is mainly developed to design the following types of cross-sections:

Z cross-sections
C and U cross-sections

This module covers the design of cold-formed steel members according to the design procedures described in Eurocode 3 Part 1-3 (EN 1993-1-3). The module was specifically developed for the design of purlins and girts, namely elements that are subjected dominantly to bending moments and shear forces with limited axial force. Among thin-walled elements, plain and lipped C-sections, plain and lipped Z-sections can be designed with this module. Stiffeners are also allowed. Effective section properties are calculated to account for both local and distortional buckling. As thin sections are highly susceptible to instability, the module evaluates distortional buckling by isolating the edge stiffeners (lips) and treating them as a compression strut on an elastic foundation, automatically applying a reduced effective thickness if distorsional buckling is detected. It applies an iterative procedure to determine the reduced properties of the cross-section due to local and distortional buckling. The following properties are determined for each cross-section along the length of the member being designed.

Average yield strength

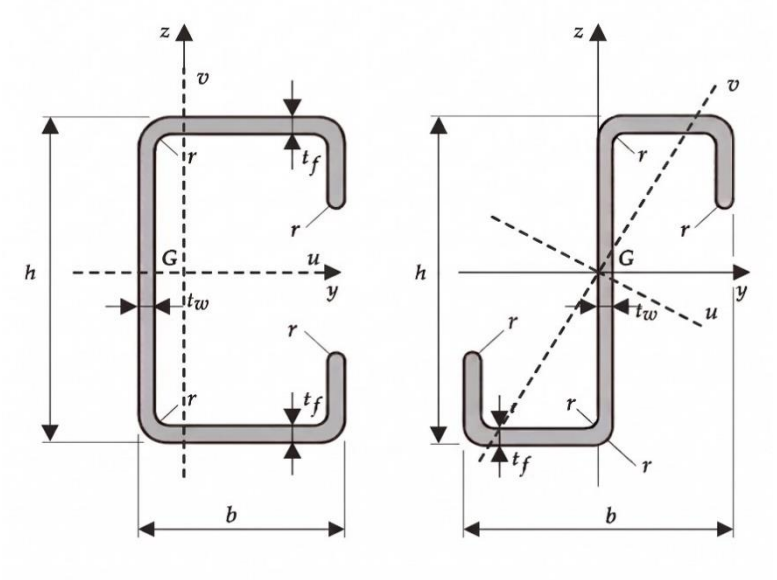
In general, the basic yield strength f_{yb} should be used. However, the average yield strength f_{ya} of a cross-section due to cold forming can be determined using the formula 3.1 provided in section 3.2.2 of EN 1993-1-3. f_{ya} can be used in the following cases: (i) cross-section resistance of an axially loaded tension member. (ii) cross-section resistance of a fully effective cross-section of an axially loaded compression member. (iii) moment resistance of a cross-section with fully effective flanges.

Steel core thickness

The steel core thickness t_{cor} should be used as design thickness, as specified by Eq.(3.3) of EN 1993-1-3, depending on the tolerance. The provisions provided for design according to Eurocode is to be used with the given ranges of core thickness $0.45\ mm \leq t_{cor} \leq 15\ mm$ for sheetings and members and $0.45_{mm} \leq t_{cor} \leq 4\ mm$ for connections. Ranges of core thickness may be given in National Annex.

AXISVM performs only the checks listed below. All the other checks specified in the design code such as the design of purlin-rafter cleat, and design of bolts against pull through, etc. have to be completed by the user.

☞ **This design module can only be used with constant (non-tapered) cross-sections.**



For C and Z cross-sections having principal axes non-coincident with the local axes the program performs checks in a design coordinate system y_D-z_D, which is aligned to the principal directions, as shown in Section 6.6.1.

Classification of cross-section

The program performs the classification of the cross-section based on EN 1993-1-1, Table 5.2. Whenever possible, it determines the class based on the actual stress distribution of combined compression and bending. The classification is conducted at several cross-sections along the length of the member, and the highest class is assigned to the member. The effect of local and distorsional buckling is taken into account in determining the class of cold-formed steel members. The local effects are accounted for using effective cross-sectional properties, calculated based on the effective widths of the elements susceptible to local buckling. The effective widths of elements (b_{eff}) are determined based on the largest compressive stress σ_{Ed} in the relevant element, which is calculated based on the previously determined effective cross-section assuming $\sigma_{com,Ed} = f_{yb}/\gamma_{M0}$. Iterative procedure is performed using the internal forces and moments determined using the effective cross-section properties from the pervious iteration.

Checks

Axial Force-Bending-Shear [N-M-V]	(EN 1993-1-3, 6.1.6, 6.1.8, 6.1.9, 6.1.10)
Compression-Bending-Buckling (flexural in plane or torsional) [N-M-Buckl.]	(EN 1993-1-3:2024, 8.2.5)
Connection	(EN 1993-1-3, 8.3)
Shear /y [Vy]	(EN 1993-1-3, 6.1.5)
Shear /z [Vz]	(EN 1993-1-3, 6.1.5)
Support Reaction-Bending [Fz-My]	(EN 1993-1-3, 6.1.11)
Buckling of the free flange	(EN 1993-1-3, 10.1.4)

Resistances

Plastic resistance (axial) [$N_{pl,Rd}$]	(EN 1993-1-3, 6.1.2)
Effective resistance (when subjected to uniform compression) [$N_{eff,Rd}$]	(EN 1993-1-3, 6.1.3)
Elastic shear resistance /y axis [Vel,y,Rd]	(EN 1993-1-3, 6.1.5)
Elastic shear resistance /z axis [Vel,z,Rd]	(EN 1993-1-3, 6.1.5)
Elastic moment Resistance /yy [$M_{el,y,Rd}$]	(EN 1993-1-3, 6.1.4)
Elastic moment Resistance /zz [$M_{el,z,Rd}$]	(EN 1993-1-3, 6.1.4)
Elastic-Plastic moment Resistance /yy [$M_{pl,y,Rd}$]	(EN 1993-1-3, 6.1.4)
Elastic-Plastic moment Resistance /zz [$M_{pl,z,Rd}$]	(EN 1993-1-3, 6.1.4)
Moment resistance for effective cross-section subjected to bending around axis y [$M_{pl,y,Rd}$]	(EN 1993-1-3, 6.1.4)
Moment resistance for effective cross-section subjected to bending around axis z [$M_{pl,z,Rd}$]	(EN 1993-1-3, 6.2.2, 6.2.3)
Minimal buckling (flexural in plane or torsional) resistance [$N_{b,Rd}$]	(EN 1993-1-3, 6.3.1)
Lateral torsional buckling resistance [$M_{b,Rd}$]	(EN 1993-1-3, 6.2.4, EN 1993-1-1, 6.3.2)
Local transverse force resistance [Rw,Rd]	(EN 1993-1-3, 6.1.7)

This information is given by the program as auxiliary results. The checks are mostly defined by interaction formulae. The definition and the detailed conditions of the application of the variables contained by the equations can be found in the design code.

In the following, $N_{c,Rd} = f_{yb}A/\gamma_{M_0}$; $N_{t,Rd} = f_{yb}A/\gamma_{M_0}$; $M_{y,Rd} = f_{yb}W_y/\gamma_{M_0}$ and $M_{z,Rd} = f_{yb}W_z/\gamma_{M_0}$ where
$A = A_g$ for class 1-3 cross-sections,
$A = A_{eff}$ for class 4 cross-sections,
If $A_{eff} = A_g$, $N_{c,Rd}$, $M_{y,Rd}$, $M_{z,Rd}$ can be determined according to EN 1993-1-3 6.1.3 and 6.1.4 with average yield strength f_{ya}.
$W_y = W_{pl,y}$ and $W_z = W_{pl,z}$ for class 1 or 2 cross sections,
$W_y = W_{el,y}$ and $W_z = W_{el,z}$ for class 3 cross sections and
$W_y = W_{eff,y}$ and $W_z = W_{eff,z}$ for class 4 cross sections.
A_{eff} area of the effective cross section when subjected to uniform compression
W_{eff} elastic section modulus (corresponding to the fiber with the maximum elastic stress) of the effective cross section when subjected only to moment about the relevant axis.

Axial Force-Bending-Shear

Combined tension and bending
The check is performed on the basis of EN 1993-1-3, 6.1.8 for combined tension and bending.

$$\frac{N_{Ed}}{N_{t,Rd}} + \frac{M_{y,Ed}}{M_{cy,Rd,ten}} + \frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{cz,Rd,ten}} \leq 1$$

where $N_{t,Rd}$ is the design resistance of a cross-section for uniform tension, $M_{cy,Rd,ten}$ design moment resistance for maximum tensile stress if subject only to moment about the y-y axis, and $M_{cz,Rd,ten}$ is the design moment resistance for maximum tensile stress if subject only to moment about the z-z axis.

If $M_{cy,Rd,com} \leq M_{cy,Rd,ten}$ or $M_{cz,Rd,com} \leq M_{cz,Rd,ten}$ the following criterion should also be satisfied, where $M_{cy,Rd,com}$ is the moment resistance for the maximum compressive stress in a cross-section that is subject only to moment about the relevant axis.

$$\frac{M_{y,Ed}}{M_{cy,Rd,com}} + \frac{M_{z,Ed}}{M_{cz,Rd,com}} - \frac{N_{Ed}}{N_{t,Rd}} \leq 1$$

Combined compression and bending:
The check is performed on the basis of EN 1993-1-3, 6.1.9 for combined compression and bending.

$$\frac{N_{Ed}}{N_{c,Rd}} + \frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rd,com}} + \frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{z,Rd,com}} \leq 1$$

$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$: it differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis *y*.

$\Delta M_{z,Ed} = N_{Ed} \cdot e_{N,z}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis z.

If $M_{cy,Rd,ten} \leq M_{cy,Rd,com}$ or $M_{cz,Rd,ten} \leq M_{cz,Rd,com}$ the following criterion should also be statisfied, where $M_{cy,Rd,com}$ is the moment resistance for the maximum compressive stress in a cross-section that is subject only to moment about the relevant axis.

$$\frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rd,ten}} + \frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{z,Rd,ten}} - \frac{N_{Ed}}{N_{c,Rd}} \leq 1$$

High shear
If the shear force is greater than 50% of the shear resistance, the effect of shear force is considered and the following equations should be satisfied:

$$\frac{N_{Ed}}{N_{c,Rd}} + \frac{M_{y,Ed}}{M_{y,Rd}} + (1 - \frac{M_{f,Rd}}{M_{pl,Rd}})(\frac{2V_{z,Ed}}{V_{z,w,Rd}} - 1) \leq 1$$

$$\frac{N_{Ed}}{N_{c,Rd}} + \frac{M_{z,Ed}}{M_{z,Rd}} + (\frac{2V_{y,Ed}}{V_{y,w,Rd}} - 1) \leq 1$$

where $M_{f,Rd}$ is the moment of resistance of a cross-section consisting of the effective area of flanges only, and $M_{pl,Rd}$ is the plastic moment of resistance of the cross-section. $V_{z,w,Rd}$ and $V_{y,w,Rd}$ are the design shear resistance of the web and the flanges, respectively.

☞ *Axial force-shear-bending-torsion*
If the loads applied eccentric to the shear center of the cross-section, the effect of torsion should be taken into account. The stresses due to axial force and bending moments are based on the effective cross-section. However, for shear stresses due to transverse shear forces, the shear stress due to St. Venant torsion and the direct stresses and shear stresses due to warping are based on the gross cross-section. The calculations are performed based on EN 1993-1-3, 6.1.6.

The following conditions must be statisfied:

$$\sigma_{tot,Ed} \leq \frac{f_y}{\gamma_{M0}}$$

$$\tau_{tot,Ed} \leq \frac{f_y/\sqrt{3}}{\gamma_{M0}}$$

$$\sqrt{\sigma_{tot,Ed}^2 + 3\,\tau_{tot,Ed}^2} \leq \frac{f_y/\sqrt{3}}{\gamma_{M0}}$$

where
$\sigma_{tot,Ed}$ is the design total direct stress, calculated on the relevant effective cross-section
$\tau_{tot,Ed}$ is the design total shear stress, calculated on the gross cross-section.

Axial force-Bending-Buckling

The check is based on 1993-1-1:2024, 8.2.5 and 8.2.6 for bending and axial compression and bending and axial tension, respectively.

Bending and axial compression:

The design value of the resistance against combined axial force and combined bending should satisfy the following criteria:

For major principal axis buckling:

$$\left(\omega_{x,y}\frac{N_{Ed}}{\chi_y N_{c,Rd}}\right)^{\alpha_y} + \left(\omega_{x,LT}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} M_{cy,Rd}}\right)^{\beta_y} + \left(\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{cz,Rd}}\right)^{\delta_y} \leq 1$$

For minor principal axis buckling:

$$\left(\omega_{x,z}\frac{N_{Ed}}{\chi_z N_{c,Rd}}\right)^{\alpha_z} + \left(\omega_{x,LT}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} M_{cy,Rd}}\right)^{\beta_z} + \left(\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{cz,Rd}}\right)^{\delta_z} \leq 1$$

Additionally, for asymmetrical sections:

$$\left(\omega_{x,y}\frac{N_{Ed}}{\chi_y N_{c,Rd}} - \frac{N_{Ed}}{N_{c,Rd}}\right)^{\alpha_y} - \frac{N_{Ed}}{N_{c,Rd}} + \left(\frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{cy,Rd}}\right)^{\beta_y} + \left(\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{cz,Rd}}\right)^{\delta_y} \leq 1$$

$$\left(\omega_{x,z}\frac{N_{Ed}}{\chi_z N_{c,Rd}} - \frac{N_{Ed}}{N_{c,Rd}}\right)^{\alpha_z} - \frac{N_{Ed}}{N_{c,Rd}} + \left(\frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{cy,Rd}}\right)^{\beta_z} + \left(\frac{M_{z,Ed} + \Delta M_{z,Ed}}{M_{cz,Rd}}\right)^{\delta_z} \leq 1$$

where
χ_y, χ_z are the reduction factors for flexural buckling. In case of torsional-flexural buckling, the relevant reduction factor flexural buckling χ_y or χ_z should be replaced with χ_{TF}according to EN1993-1-1:2006, 6.2.3.
χ_{LT} is the reduction factor for lateral-torsional buckling according to EN1993-1-1:2006, 6.2.4.
$\alpha_y, \alpha_z, \beta_y, \beta_z, \delta_y, \delta_z$ are exponents in the interaction formulas, determined according to Table 8.9 in EN1993-1-1:2024.
$\omega_{x,y}, \omega_{x,z}, \omega_{x,LT}$ are the interpolation factors according to Table 8.10 in EN1993-1-1:2024. Conservatively, a value of 1.0 can be used for these interpolation factors. The factors have the form $\omega_{x,i} = \chi_i + (1 - \chi_i)\sin\frac{\pi x_s}{l_{cr,i}}$, where i is the relevant buckling axis, x_s is the distance from the cross-section under consideration to a simple support or a point of contraflexure. $l_{cr,i}$ is the buckling length for the relevant buckling mode.
$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis y.
$\Delta M_{z,Ed} = N_{Ed} \cdot e_{N,z}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis z.

Bending and axial tension:
The shown formulas for bending and compression are used for bending and tension, as well. However, $N_{c,Rd}$ is replaced by $N_{t,Rd}$, and $\chi_y = \chi_z = \chi_{LT} = 1.0$.

Connection

The design of connections is performed based on section 8.3 of EN 1993-1-3, 8.3. The design is performed for bolted connections only.

Bolts loaded by shear:

Bearing resistance:
The bearing resistance of bolts is determined according to Table 8.4 of EN 1993-1-3, as follows:

$$F_{b,Rd} = 2.5\ \alpha_b k_t f_u\ d\ t/\gamma_{M2}$$

where d is the nominal diameter of the fastener, $f_u$and t are the ultimate strength and thickness of the thinner plate, $\alpha_b$is the smallest of 1.0 or $e_1/3d$. e_1 is the distance to edge bolt in direction of the load transfer.

$$k_t = (0.8\ t + 1.5)/2.5 \text{ for } 0.75\ mm \le t \le 1.25mm;\ k_t = 1.0 \text{ for } t > 1.25\ mm$$

Shear resistance:
For strength grades 4.6, 5.6 and 8.8:

$$F_{v,Rd} = 0.6 f_{ub} A_s/\gamma_{M2}$$

For strength grades 4.8, 5.8, 6.8 and 10.9:

$$F_{v,Rd} = 0.5 f_{ub} A_s/\gamma_{M2}$$

where A_s is the tensile stress area of a fastener, and $f_{ub}$is the ultimate tensile strength of the fastener material. The shear resistance $F_{v,Rd}$ must be equal to or greater than 1.2 times the bearing strength.

Net-section resistance:

$$F_{n,Rd} = (1 + 3r(d_0/u - 0.3)A_{net} f_u/\gamma_{M2} \ \le\ A_{net} f_u/\gamma_{M2}$$

Bolts loaded by tension:

Tension resistance:

$$F_{t,Rd} = 0.9 f_{ub} A_s/\gamma_{M2}$$

Pull-through resistance: must be determined by testing.

The design is performed by comparing the resultant force on the bolt against the relevant resistance, depending on the type of connection under design.

The resultant force is calculated for the in-plane shear forces, as follows:

$$F_{Ed} = \sqrt{F_x^2 + F_y^2}$$

where F_x and F_y are calculated for each bolt depending on the location of the bolt, the number of bolts, the arrangement of the bolts, and the type of each connection.

The utilisation ratio is calculated per bolt as

$$\eta = F_{Ed}/F_{b,Rd}$$

The utilisation ratio is calculated for the bolts in the web, the flange and at the end of the overlap part.

Shear /y

The check is performed on the basis of EN 1993-1-3, 6.1.5.

$$\frac{V_{y,Ed}}{V_{b,y,Rd}} \leq 1$$

$$V_{b,y,Rd} = \frac{A_f f_{bv}}{\gamma_{M0}}$$

where f_{bv} is the shear strength considering buckling according to Table 6.1 of EN 1993-1-3.
A_f is area of the flanges

Shear /z

The check is performed on the basis of EN 1993-1-1, 6.1.5.

$$\frac{V_{zEd}}{V_{b,z,Rd}} \leq 1$$

$$V_{b,z,Rd} = \frac{\frac{h_w}{\sin\phi} t f_{bv}}{\gamma_{M0}}$$

where f_{bv} is the shear strength considering buckling according to Table 6.1 of EN 1993-1-3.
$h_w$is the web height between the midlines of the flanges.
ϕ is the slope of the web relative to the flanges.

The shear strength is determined based on the relative web slenderness, as follows:
For webs without longitudinal stiffeners:

$$\bar{\lambda}_w = 0.346 \frac{s_w}{t} \sqrt{\frac{f_{yb}}{E}}$$

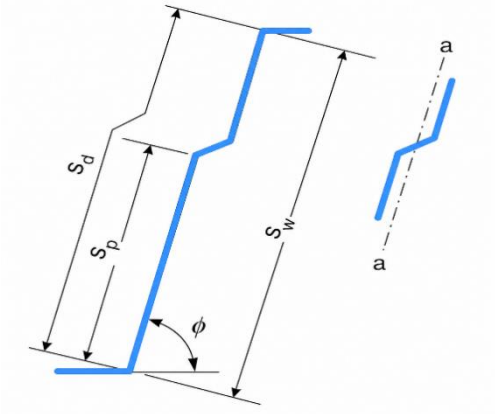
For webs with longitudinal stiffeners:

$$\bar{\lambda}_w = 0.346 \frac{s_d}{t} \sqrt{\frac{5.34\, f_{yb}}{k_\tau E}}, \bar{\lambda}_w \geq 0.346 \frac{s_p}{t} \sqrt{\frac{f_{yb}}{E}}$$

with

$$k_\tau = 5.34 + \frac{2.10}{t} \left(\frac{\sum I_s}{s_d}\right)^{\frac{1}{3}}$$

where
I_s is the second moment of area of the individual longitudinal stiffener about the axis a-a parallel to the adjacent plates.
$s_d$is the total developed slant height of the web.
$s_p$is the slant height of the largest plane element in the web.
$s_w$is the slant height of the web, between the midpoints of the corners.



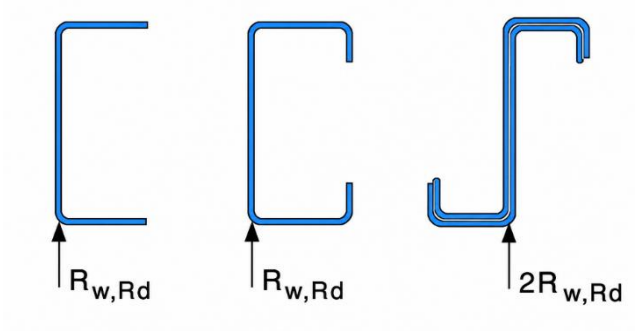
Combined bending moment and support reaction

The check is performed for cross-sections with single web according to EN 1993-1-3, 6.1.11. Cross-sections subject to the combined action of bending moment M_{Ed} and transverse force due to local load or support reaction F_{Ed} must satisfy the following condition:

$$\frac{M_{Ed}}{M_{c,Rd}} + \frac{F_{Ed}}{R_{w,Rd}} \leq 1.25$$

where

$M_{c,Rd}$ is the moment resistance of the cross-section, as illustrated earlier.
$R_{w,Rd}$ is the appropriate value of the local transverse resistance of the web, according to EN 1993-1-3, 6.1.7.
Separate checks are performed for $\frac{M_{Ed}}{M_{c,Rd}} < 1$ and $\frac{F_{Ed}}{R_{w,Rd}} \leq 1$, as well.
Cross-sections with single unstiffened web are checked using EN 1993-1-3, 6.1.7.2.



The cross-sections should satisfy the following limits:

$$\frac{h_w}{t} \leq 200$$

$$\frac{r}{t} \leq 6$$

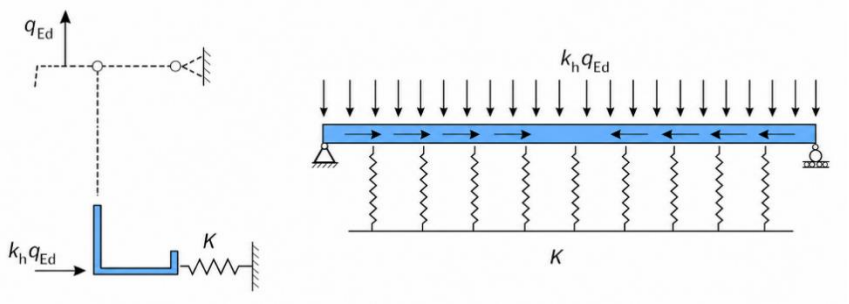
$$45 \leq \phi \leq 90$$

Cross-sections with stiffened webs are checked using EN 1993-1-3, 6.1.7.4.

Buckling of the free flange

The check is performed for beams restrained by sheeting according to EN 1993-1-3, 10.1.4.

The design of purlins subjected to axial force and transverse load the resistance of the cross-section is verified by superposing the stresses due to in-plane bending $M_{y,Ed}$, the axial force N_{Ed}, and an equivalent lateral load $q_{h,Ed}$ acting on the free flange, due to torsion and lateral bending.



The maximum stress in the cross-section should satisfy the following:

Restrained flange:

$$\frac{1}{\frac{f_y}{\gamma_M}}\left(\frac{M_{y,Ed}}{W_{eff,y}} + \frac{N_{Ed}}{A_{eff}}\right) \leq 1.0$$

Free flange:

$$\frac{1}{\frac{f_y}{\gamma_M}}\left(\frac{M_{y,Ed}}{W_{eff,y}} + \frac{N_{Ed}}{A_{eff}} + \frac{M_{fz,Ed}}{W_{fz}}\right) \leq 1.0$$

where

A_{eff} is the effective area of the cross-section for only uniform compression.

$M_{fz,Ed}$ is the bending moment in the free flange due to the lateral load $q_{h,Ed}$.

$W_{eff,y}$ is the effective section modulus of the cross-section for only bending about the y-y axis.

W_{fz} is the gross elastic section modulus of the free flange plus the contributing part of the web for bending about the z-z axis. The contributing part is defined as 1/5 or 1/6 of the height of the web, depending on the type of the cross-section.

The equivalent lateral load $q_{h,Ed} = k_h q_{Ed}$, where k_h is be obtained as indicated in figure 10.3 of EN 1993-1-3. The program calculates an equivalent uniform distributed load q_{Ed} depending on the shear diagram for each supported member.

The lateral bending moment $M_{fz,Ed}$ is determined, as follows:

$$M_{fz,Ed} = \kappa_R M_{0,fz,Ed}$$

where

$M_{0,fz,Ed}$ is the initial lateral bending moment in the free flange without any spring support.

κ_R is a correction factor for the effective spring width, according to Table 10.1 of EN 1993-1-3 for the relevant location and boundary conditions, using the value of the coefficient R of the spring support given by:

$$R = \frac{KL^4}{\pi^4 E I_{fz}}$$

where

I_{fz} is the second moment of area of the gross-section of the free flange plus the contributing part of the web for bending about the z-z axis.

K is the lateral spring stiffness defined as $\frac{1}{\frac{1}{C_{D,A}}+\frac{1}{C_{D,C}}}$, where $C_{D,A}$ and $C_{D,C}$ are the rotational stiffness between the sheeting and the purlin and the rotational stiffness corresponding to the flexural stiffness of the sheeting.

L is the distance between supports.

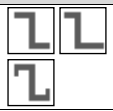
Buckling of the free flange:

The buckling of the flange in compression is checked, as follows:

$$\frac{1}{\chi_{LT}}\left(\frac{M_{y,Ed}}{\frac{f_y}{\gamma_M}W_{eff,y}} + \frac{N_{Ed}}{\frac{f_y}{\gamma_M}A_{ff}}\right) + \frac{M_{fz,Ed}}{\frac{f_y}{\gamma_M}W_{fz}} \leq 1.0$$

The reduction factor χ_{LT} is the reduction factor for lateral torsional buckling, determined using the critical elastic moment using the AutoMcr, designed to take into account the unsymmetry of the C and Z sections.

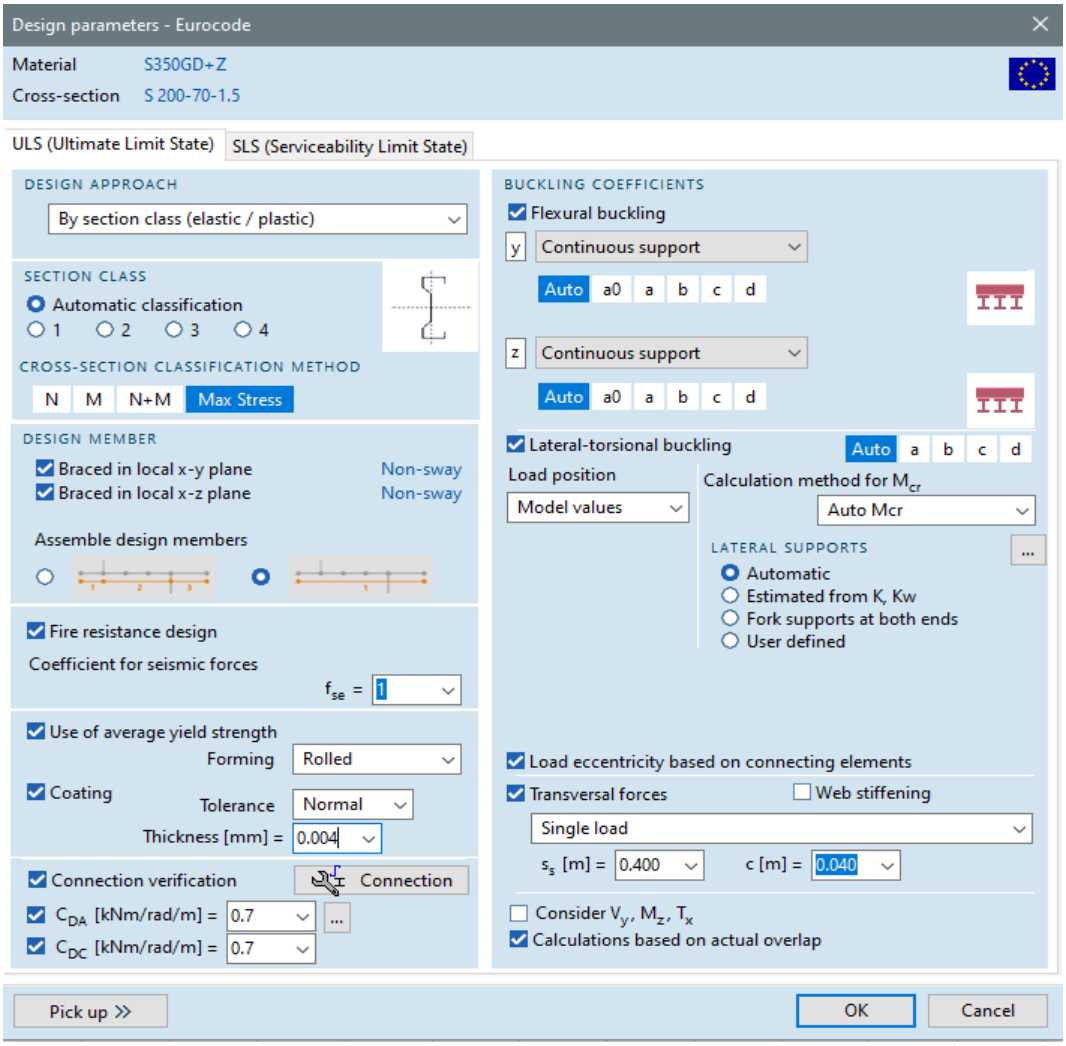
Supported section types

Section type		*N-M-V Stress*	*N-M-Stability*	*Connection Design*	*Shear Vy*	*Shear Vz*	*Fz-My Stress*	*Free Flange*	*Effective section*
Asymmetric Z		✓	✓	✓	✓	✓	✓	✓	✓
Asymmetric C		✓	✓	✓	✓	✓	✓	✓	✓

6.6.2.1 ULS design parameters

For the design of cold-formed steel sections based on EN 1993-1-3, the following design parameters should be defined and assigned to the structural members to perform ULS checks:

ULS (Ultimate Limit State)



Design approach — The same design approaches apply to cold-formed steel, as defined in **6.6.1.1 ULS design parameters**

Section class — Automatic classification classifies the cross-section by the actual stress values.

Design member bracing — The same classification methods apply, as defined in **6.6.1.1 ULS design parameters**

Classification method — If the design member has a sway buckling mode in local *x-y* or *x-z* plane the respective bracing must be turned off. This option will affect the obtained buckling results, depending on the chosen buckling shape. See **6.6.1.1 ULS design parameters**.

Assemble design members — The program assembles design members from the selected elements before performing design calculations. Design members consist of finite elements with the same material, same type (truss, beam or rib) and local system orientation. Finite elements must be on the same line. **It is recommended to assemble members that form a continuous beam system, to ensure proper design of the member.** (see **6.6.1.1 ULS design parameters**)

Fire resistance design — **Enable fire resistance design for members under defined fire effect and for accidental load combinations.**

Coefficient for seismic forces — Factor for seismic forces see **4.10.25 Seismic loads – SE1 module**

Stability parameters AXISVM performs checks against buckling, lateral-torsional buckling and web shear buckling. Each check can be activated separately by clicking on the checkbox before its name. For example, if it is sure that there is no need to check lateral-torsional buckling that part of the check can be deactivated and no parameter has to be specified.

Use of average yield strength Use the average yield strength (f_{ya}) instead of the basic yield strength (f_{yb}) for von Mises check and for the cases mentioned demonstrated in EN 1993-1-3, 3.2.2. The rolling type must be chosen, either roll forming or other methods of forming.

Coating If the cross-section thickness defined in the cross-section editor includes the coating thickness, this option must be enabled to reduce the cross-section thickness, and obtain the design value, according to EN 1993-1-3. The tolerance and the thickness must be defined according to EN 1993-1-3, 3.2.4.

Connection verification The connection verification allows to define the type of the connection, layout and properties of bolts (see 6.6.2.1.1 Connection verification).

C_{DA} and C_{DC} C_{DA} and C_{DC} define the rotational stiffness of the connection between the sheeting and the purlin and the rotational stiffness corresponding to the flexural stiffness of the sheeting, respectively. The value of C_{DA} and C_{DC} must be defined by the user. The value of C_{DA} can be calculated automatically by filing the number of sheet-to-purlin fasteners per meter length of the purlin, as defined in EN 1993-1-3, 10.1.5.2(7).

Transverse forces Performs transversal force check on the supports of the design member, according to EN 1993-1-3, 6.1.7. The following must be defined to perform the check:
- The web stiffening defines whether there is a type of stiffening to the web of the member under design, such as cleats that prevents the buckling of the web.
- The support and load configuations:
(i) Single load (EN 1993-1-3, 6.1.7.2(3), Figure 6.7a).
(ii) Opposing loads (EN 1993-1-3, 6.1.7.2(3), Figure 6.7b).
(iii) Single load + prevented web rotation (EN 1993-1-3, 6.1.7.2(4), Figure 6.7a).
(iv) Opposing loads + prevented web rotation (EN 1993-1-3, 6.1.7.2(4), Figure 6.7b).
- s_s is the nominal length of the stiff bearing
- c is the shortest distance from the member edge to the beginning of the supported region.

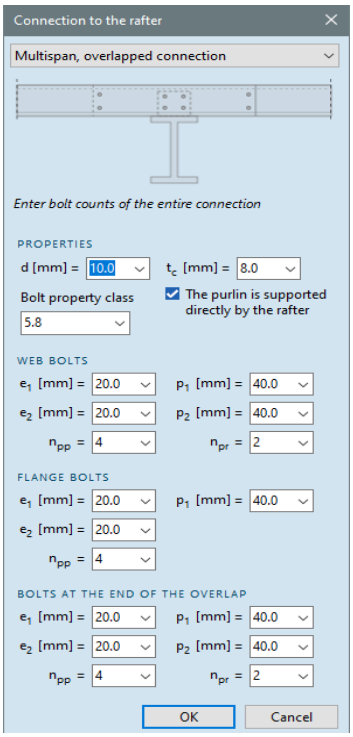
Consider V_y, M_z, T_x The values of the horizontal shear (V_y), horizontal bending moment (M_z), Saint-Venant torsion and warping torsion may be considered zero due to the supporting effect utilized by the trapezoidal sheeting or sandwich panel.

Calculations based on actual overlap In the overlap zone, the software can calculate the properties of the cross-section formed by the combination of the overlapping and original sections both approximately and exactly. The exact calculation significantly increases the computational time. In the approximate calculation, the software simply sums the required cross-sectional properties of the two sections.

Connection See the next chapter.

6.6.2.1.1 Connection verification

Purlin-rafter connection



Multiple types of connections are available

Multispan, overlapped connection
Multispan, continuous connection
Standard hinged connection
Shear plate hinged connection.

The type of the connection must be chosen according to the member configuration. The connection must satisfy the conditions specified in Table 8.4 of EN 1993-1-3.

Properties

The diameter (d) and the property class of the bolts, and the cleat thickness (t_c) must be defined.
For members that are directly supported by the rafter the vertical force on the bolts can be considered zero ($F_y = 0$) for gravity loading.

Web bolts

The bolts in the web of the member near the connection.
The following parameters must be given:
e_1 is the end distance from the centre of the fastener to the adjacent end of the connection part in the direction of load transfer.
e_2 is the end distance from the centre of the fastener to the adjacent end of the connection part in the direction perpendicular to the direction of load transfer.
p_1 is the spacing centre-to-centre of fasteners in the direction of load transfer.
p_2 is the spacing centre-to-centre of fasteners in the direction perpendicular to the direction of load transfer.
n_{pp} is the total number of bolts in the web of member near the connection.
n_{pr} is the number of bolt rows in the web of the member near the connection.

Flange bolts

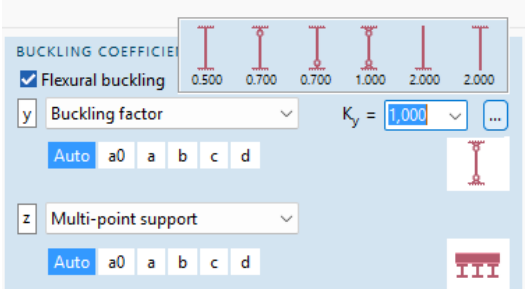
The bolts in the flange of the purlin. Similar parameters must be defined as defined in web bolts. However, n_{pp} is the total number of bolts in the flange of member near the connection.
Only the total number of bolts is needed in this case. The design is only performed to the member under design; the supporting rafter and cleat must be checked separately.

Bolts at the end of the overlap

The bolts at the end overlap region. This type of bolts is only applicable in the case of Multispan, overlapped connection. Similar parameters must be defined as defined in web bolts, However, n_{pp} is the total number of bolts in the end part of the overlap between two members. n_{pr} is the number of bolt rows in the end part of the overlap between two members.

6.6.2.1.2 Buckling (flexural)

To determine the K_y and K_z buckling resistance around *y* and *z* axes (in the local *x-z* and *y-z* plane) end support conditions must be defined.
There are three ways to specify buckling behaviour: *Buckling factor, Buckling length* and *Multi-point support*..



Buckling factor K_y, K_z buckling factors (effective length factors). The buckling factor will be multiplied by the length of the design members and not the total length of the selected elements. Note that connecting elements might lead to separation of a selected member into multiple design members (see Assemble design members above)!

Buckling length L_y, L_z buckling lengths. The entered buckling length will be used regardless of the length of the design member.

☞ **For the buckling factor and buckling length it is necessary to select a buckling shape, as the interaction of axial-bending-buckling depends on selected buckling shape.**
For buckling factors of 0.7 and 2.0, particular attention must be paid to the correct positioning of the hinge and free end at the beginning or end of the member.

Multi-point support The buckling shape is automatically determined for the member, assuming a sinusoidal wave with a half wavelength equal to the member length between supports. The recommended option for the design of purlins.

6.6.2.1.3 Lateral-torsional buckling

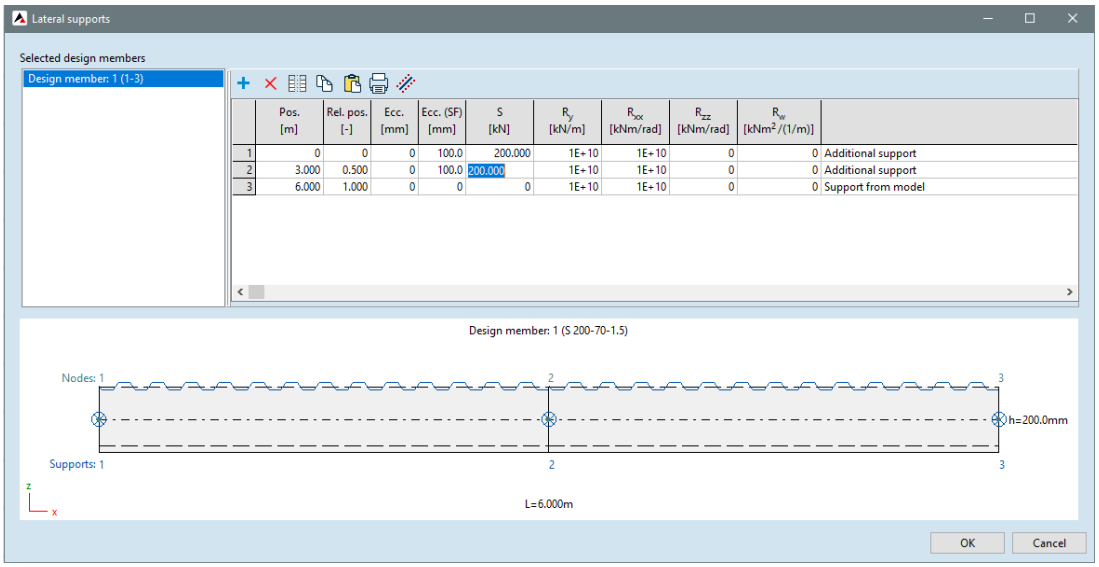
See 6.6.2.1.3 Lateral-torsional buckling

6.6.2.1.4 Lateral supports

Automatic lateral supports



For the detailed definition of lateral supports see 6.6.2.1.4 Lateral supports
The lateral support conditions can be edited by clicking the ... button.
A dialog appears with a table of lateral supports of the design members created from the selection.
Design members are listed on the left. The table shows the lateral supports of the selected item.
The last item of the list is *Same supports on selected elements*. Selecting this item, the table shows only the common supports on the selected design members. Adding or deleting supports will change the support configuration of all selected design members.



Shear field The dialog allows the shear field effect on the member to be defined. The *Ecc. (SF)* and *S* columns allow the definition of the eccentricity and magnitude of the shear field, respectively. Shear fields defined during the modelling stage are automatically displayed in this dialog, where their magnitude and the location can be edited. A shear field is defined at a specific node and applied on the entire member length up to the next node. The shear field is considered active only when it acts on the top or bottom flange, with a maximum tolerance of 10 mm. Otherwise, it is not considered.

6.6.2.2 SLS design parameters



The parameters for the serviceability limit state (SLS) checks can be set on the SLS tab in the Design parameters window. (See 6.6.1.2 SLS design parameters)

The effect of the effective cross-section on the calculated horizontal and vertical displacements is accounted for by applying an amplification to the displacement values based on the effective cross-section.

6.6.2.3 Diagrams and design calculations

Diagrams

You can display the diagrams corresponding to all individual checks and their envelope by clicking on the steel design member. Results for any position of any steel design member in any load case or combination can be obtained by setting the combo boxes and dragging the tracking line. If a check cannot be performed with the current cross-section the respective diagram is replaced by a cross. (See 6.6.1.3 Diagrams and design calculations)



Design calculations

Clicking on the *Design calculations* button a report of the calculation details can be displayed. All strength and stability checks appear as formulas completed with substituted actual values and references to the design code.

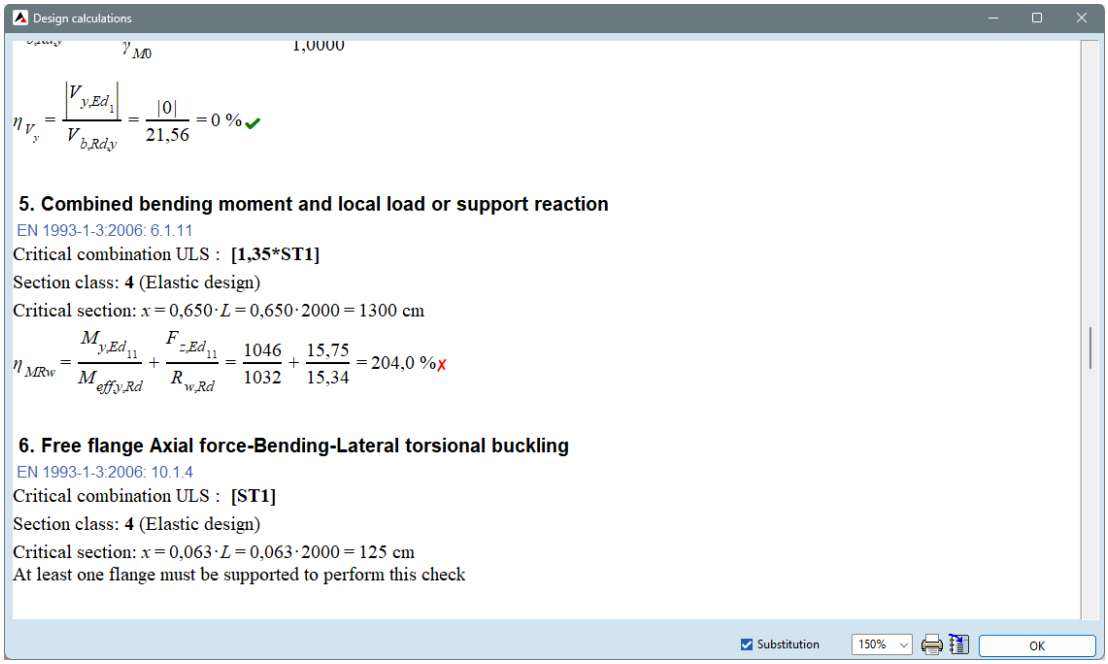
The report consists of the seven basic interaction checks listed above and several partial results which make it easier to follow the calculations and provide useful details for cross-section analysis.
The partial results are:

Axial elastic Resistance
Elastic Moment Resistance about the y axis
Elastic Moment Resistance about the z axis
Elastic Shear Resistance in z direction
Local transverse resistance
Bending-axial force interaction check
Flexural Buckling Resistance
Lateral-Torsional Buckling Resistance

Clicking on the Settings icon beside the *Design calculations* button allows setting the basic units for force and length used in the design calculations. Important results also appear converted to standard AXISVM units (see **3.3.8 Units and Formats**).

Details of calculations

The details of calculations according to the current design code are displayed as a multi-page document. References to sections and formulas of the design code appear in blue.



6.6.2.4 Steel cross-section optimization - SD9 module

Cold formed, thin-walled steel beams **cannot be optimized by SD9 module**.

6.6.3 Steel beam fire design according to Eurocode 3 – SD8 module

Standards

AXISVM performs steel fire design according to different national codes. The design codes and national annexes for which this functionality is available are listed in **3.3.7 Design codes**.

Fire design of steel structures according to EN 1993-1-2

Steel beam fire design (SD8 module) is based on the "ordinary" steel beam design (SD1 module; see **6.6.1 Steel beam design according to Eurocode 3 – SD1 module) (SD3 module: see 6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 -SD3 module**), so **SD1 or SD3 module is a prerequisite to use SD8 module**.
EN 1993-1-2 (EC3-1-2) discusses fire design of steel structures. EC3-1-2 is very close to EN 1993-1-1, so we note only the difference.
Steel beam fire design can be performed in AXISVM if
1) the selected load combination includes a fire load case (**4.10.1 Load cases, load groups**)
2) if a fire effect has been assigned to any of the selected elements (**4.10.32 Fire effect on steel line elements – SD8 module**) in that fire load case. If different fire effects have been assigned to elements of a design member, an error message appears and fire design cannot be performed.
If the selection contains elements without fire effect the ordinary steel design will be performed on them.
Certain steel fire design parameters also have to be assigned to the elements (see below).

Member analysis

EC3-1-2 discusses different methods: member analysis, analysis of a part of the structure and analysis of the entire structure. Member analysis is the most widespread verification method due to its simplicity. Analysis of a part or whole of the structure usually requires complex nonlinear numerical models and nonlinear statical analysis.

Guiding principles of member analysis according to EC3-1-2:

- Design member internal forces can be calculated with linear static analysis
- „Only the effects of thermal deformations resulting from thermal gradients across the cross-section need to be considered. The effects of axial or in-plain thermal expansions may be neglected." (EC3-1-2, 2.4.2 (4))
- „The boundary conditions at supports and ends of member may be assumed to remain unchanged throughout the fire exposure." (EC3-1-2, 2.4.2 (5))

SD8 performs member analysis. It has to be checked if that structure/structural part can be verified with member analysis according to the guidelines of EC3-1-2.

In certain cases, analysis of a part of the structure can be also executed in AXISVM if 1) boundary conditions are set carefully, 2) fire effects are taken into account as thermal load and 3) we consider the guidelines in section 2.4.3 of EC3-1-2.

$k_{y,\theta}$, k_E, $k_{y,\theta,4}$

According to the EC3-1-2, the following reduction factors for mechanical properties of carbon steels are considered in steel fire design in AXISVM:

- $k_{y,\theta}$ – reduction factor for yield strength (Class 1, 2 and 3 cross-sections), relative to yield strength at 20 °C
- $k_{y,\theta,4}$ – reduction factor for yield strength (Class 4 cross-section), relative to yield strength at 20 °C
- k_E – reduction factor for Young's modulus, relative to Young's modulus at 20 °C

It has to be noted that in case of the design of stainless-steel different reduction factors need to be used.

θ_s [°C]	$k_{y,\theta}$	$k_{y,\theta,4}$	k_E
20	1.000	1.000	1.000
100	1.000	1.000	1.000
200	1.000	0.890	0.900
300	1.000	0.780	0.800
400	1.000	0.650	0.700
500	0.780	0.530	0.600
600	0.470	0.300	0.310
700	0.230	0.130	0.130
800	0.110	0.070	0.090
900	0.060	0.050	0.0675
1000	0.040	0.030	0.045
1100	0.020	0.020	0.0225
1200	0.000	0.000	0.000

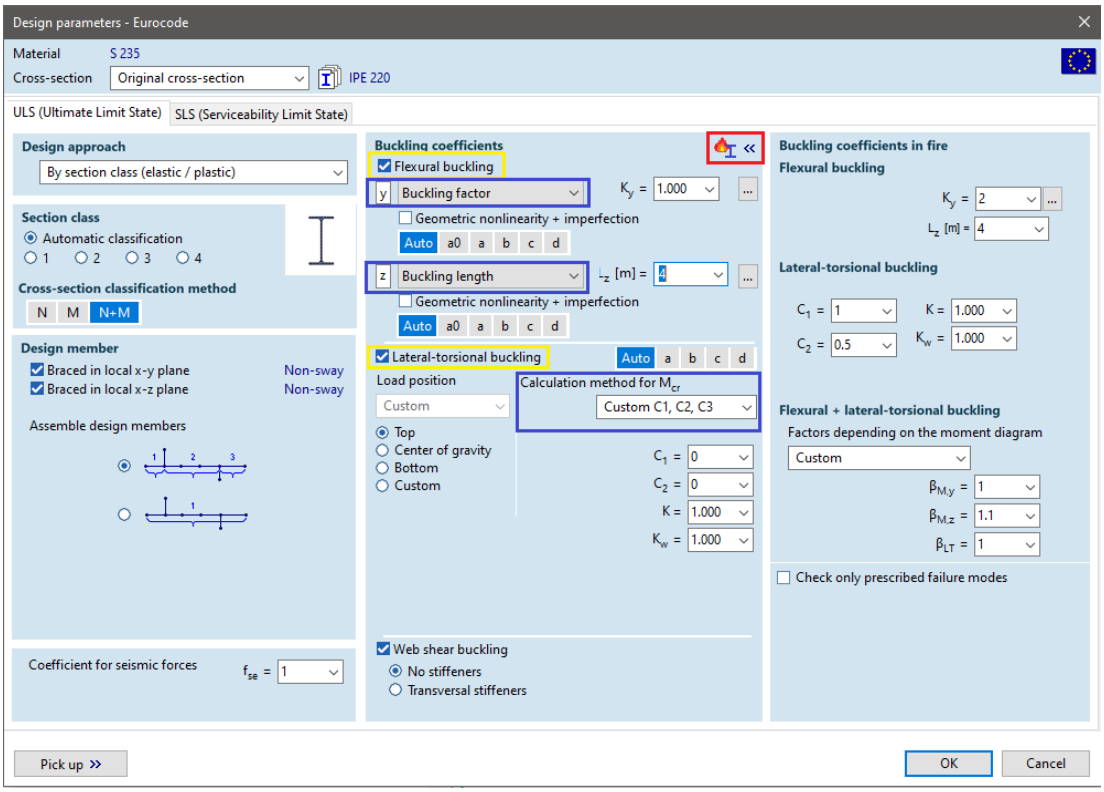
6.6.3.1 Fire design of hot-rolled steel design members

Steel fire design parameters



For the fire design based on EC3-1-2, fire design parameters should be defined and assigned to the structural members. Cick the button displayed below in red rectangle.
This button is visible only if
1) SD8 module is part of the configuration and
2) steel fire design according to the current design code is supported.
The button adds an additional panel on the right side. If it remains hidden, fire design of the element will not be performed.





Buckling coefficients in fire

Flexural buckling

Lateral-torsional buckling

K_ω: factor related to the constraints against warping can be different on elevated temperature.
If *Lateral-torsional buckling* is checked, resistance calculation and verification are performed on both normal and elevated temperature.

Mcr – critical bending moment

Different parameters can be defined for the consideration and calculation of critical bending moment. The way of calculation of critical bending moment and load position are the same as in normal temperature design (blue rectangle).
If *AutoMcr* is selected, automatically calculated or user defined lateral supports are taken into account by the calculation of M_{cr}.

Flexural buckling + Lateral-torsional bucklings

Verification against flexural torsional buckling is performed if *Flexural buckling* and *Lateral-torsional buckling* are checked and the stability parameters are defined. The verification method is explained below. Equivalent uniform moment factors ($\beta_{M,y}$, $\beta_{M,z}$ *and* $\beta_{M,LT}$ of EC3-1-2; Fig. 4.2) need to be defined. It is possible to calculate these factors automatically based on the bending moment diagram.

Check only prescribed failure modes

EN 1993-1-1 and EN 1993-1-2 standards are similar, but EN 1993-1-2 does not discuss verification in several failure modes (e.g., strength failure caused by internal forces in interaction with each other or shear web buckling failure). According to the relevant literature, EN 1993-1-1 verification formulae can be used in these cases but with reduced yield strength and Young's modulus. Nevertheless, these verifications are not prescribed by the code.

If *Check only prescribed failure modes* is checked, only EC3-1-2 design checks are performed.

Classification of cross-sections

AXISVM classifies cross-sections according to Table 5.2 of EN 1993-1-1. In case of fire design, the ε parameter is calculated according to EC3-1-2:

$$\varepsilon = 0.85\sqrt{\frac{235}{f_y}}$$

Design checks in fire

SD8 performs the following design checks:

Check only prescribed failure modes



Axial force-Bending-Shear [N-M-V] (EN 1993-1-1, 6.2.1, 6.2.9.3) (EN 1993-1-2, 4.2.1-4.2.3, Annex E)

Compression-Bending-Buckling (flexural in plane or torsional) [N-M-Buck] (EN 1993-1-2, 4.2.3.5, Annex E)

Axial force-Bending-Lateral Tors. Buckling [N-M-LTBuckl] (EN 1993-1-2, 4.2.3.5, Annex E)

Shear /y [Vy] (EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Shear /z [Vz] (EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Check only prescribed failure modes



Axial force-Bending-Shear [N-M-V] (EN 1993-1-1, 6.2.1, 6.2.8, 6.2.9.3) (EN 1993-1-2, 4.2.1-4.2.3, Annex E)

Compression-Bending-Buckling (flexural in plane or torsional) [N-M-Buck] (EN 1993-1-2, 4.2.3.5, Annex E)

Axial force-Bending-Lateral Tors. Buckling [N-M-LTBuckl] (EN 1993-1-2, 4.2.3.5, Annex E)

Shear /y [Vy] (EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Shear /z [Vz] (EN 1993-1-5, 5.1-5.3) (EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Web shear-Bending-Axial force [Vw-M-N] (EN 1993-1-1, 6.2.1, 6.2.8, 6.2.9) (EN 1993-1-5, 5.1-5.3, 7.1)

(EN 1993-1-2, 4.2.1-4.2.3, Annex E)

Resistances

Plastic resistance (axial) [Npl,fi,Rd] (EN 1993-1-2, 4.2.3.1, Annex E)

Effective resistance (when subjected to uniform compression) [Neff, fi,Rd] (EN 1993-1-2, 4.2.3.1, Annex E)

Elastic Shear Resistance /y axis [Vel,fi,y,Rd] (EN 1993-1-1, 6.2.6, EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Elastic Shear Resistance /z axis [Vel,fi,z,Rd] (EN 1993-1-1, 6.2.6, EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Plastic Shear Resistance /y axis [Vpl,fi,y,Rd] (EN 1993-1-1, 6.2.6, EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Plastic Shear Resistance /z axis [Vpl,fi,z,Rd] (EN 1993-1-1, 6.2.6, EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Shear Web Buckling [Vb,fi,Rd] (EN 1993-1-5, 5.1-5.3, EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Elastic Moment Resistance /yy [Mel,fi,y,Rd] (EN 1993-1-2, 4.2.3.4, Annex E)

Elastic Moment Resistance /zz [Mel,fi,z,Rd] (EN 1993-1-2, 4.2.3.4, Annex E)

Plastic Moment Resistance /yy [Mpl,fi,y, Rd] (EN 1993-1-2, 4.2.3.3, Annex E)

Plastic Moment Resistance /zz [Mpl,fi,z,Rd] (EN 1993-1-2, 4.2.3.3, Annex E)

Moment Resistance for effective cross-section subjected to bending around axis y [Meff,fi,y, Rd] (EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Moment Resistance for effective cross-section subjected to bending around axis z [Meff,fi,z,Rd] (EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Minimal Buckling (flexural in plane or torsional) Resistance [Nb,fi,Rd] (EN 1993-1-2, 4.2.3.2, Annex E)

Lateral Torsional Buckling Resistance [Mb,fi,Rd] (EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E)

Critical temperature [θ_{cr}] (EN 1993-1-2)

Axial force-Bending-Shear

Check only prescribed failure modes



Elastic design check (Class 3 cross-sections)

General formula:

$$\frac{N_{Ed}}{A\frac{f_y k_{y,\theta}}{\gamma_{M_{fi}}}} + \frac{M_{y,Ed}}{W_{el,y}\frac{f_y k_{y,\theta}}{\gamma_{M_{fi}}}} + \frac{M_{z,Ed}}{W_{el,z}\frac{f_y k_{y,\theta}}{\gamma_{M_{fi}}}} \leq 1$$

If shear force is greater than 50% of the shear resistance or torsional moment is present, the following yield condition is checked (only for I, T, C, box and pie sections):

$$\left(\frac{\sigma_{x,Ed}}{f_y k_{y,\theta}/\gamma_{M_{fi}}}\right)^2+\left(\frac{\sigma_{z,Ed}}{f_y k_{y,\theta}/\gamma_{M_{fi}}}\right)^2-\left(\frac{\sigma_{x,Ed}}{f_y k_{y,\theta}/\gamma_{M_{fi}}}\right)\left(\frac{\sigma_{z,Ed}}{f_y k_{y,\theta}/\gamma_{M_{fi}}}\right)+3\left(\frac{\tau_{Ed}}{f_y k_{y,\theta}/\gamma_{M_{fi}}}\right)^2\le 1$$

Elastic design check (class 4 cross-sections)

General formula:

$$\frac{N_{Ed}}{A_{eff}\frac{f_y k_{y,\theta,4}}{\gamma_{M_{fi}}}}+\frac{M_{y,Ed}+\Delta M_{y,Ed}}{W_{eff,y}\frac{f_y k_{y,\theta,4}}{\gamma_{M_{fi}}}}+\frac{M_{z,Ed}+\Delta M_{z,Ed}}{W_{eff,z}\frac{f_y k_{y,\theta,4}}{\gamma_{M_{fi}}}}\le 1$$

If the shear force is greater than 50% of the shear resistance or torsional moment is present, the following yield condition is checked (only for I, T, C, box and pie sections):

$$\left(\frac{\sigma_{x,Ed}}{f_y k_{y,\theta,4}/\gamma_{M_{fi}}}\right)^2+\left(\frac{\sigma_{z,Ed}}{f_y k_{y,\theta,4}/\gamma_{M_{fi}}}\right)^2-\left(\frac{\sigma_{x,Ed}}{f_y k_{y,\theta,4}/\gamma_{M_{fi}}}\right)\left(\frac{\sigma_{z,Ed}}{f_y k_{y,\theta,4}/\gamma_{M_{fi}}}\right)+3\left(\frac{\tau_{Ed}}{f_y k_{y,\theta,4}/\gamma_{M_{fi}}}\right)^2\le 1$$

$\Delta M_{y,Ed}=N_{Ed}\cdot e_{N,y}$: it differs from zero only when the cross section is in Class 4 and the original cross-section is asymmetric to axis *y*.

$\Delta M_{z,Ed}=N_{Ed}\cdot e_{N,z}$ differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis z.

Plastic design check

For I, pipe and box sections in section Class 1 and 2, the resistance check is performed according to EN 1993-1-1 6.2.10 and EC3-1-2. Allowance is made for the effect of both shear force and axial force on the resistance moment. Beyond the resistance check of pure axial force and pure shear force, the following criteria should also be satisfied:

$$\frac{M_{y,Ed}}{M_{N,fi,y,Rd}}\le 1;\ \frac{M_{z,Ed}}{M_{N,fi,z,Rd}}\le 1$$

where $M_{N,fi,y,Rd}$, $M_{N,fi,z,Rd}$ are reduced moment resistances based on the effect of shear force and axial force (EN 1993-1-1 6.2.8. and 6.2.9.1). For pipe sections, the reduced moment is calculated as follows:

$$M_{N,fi,y,Rd}=1.04\cdot\left(1-\rho-\frac{n^{1.7}}{(1-\rho)^{0.7}}\right);\ n=\frac{N_{Ed}}{N_{pl,fi,Rd}};\ \rho=\left(2\frac{V_{Ed}}{V_{pl,fi,z,Rd}}-1\right)^2$$

$$M_{N,fi,z,Rd}=1.04\cdot\left(1-\rho-\frac{n^{1.7}}{(1-\rho)^{0.7}}\right);\ n=\frac{N_{Ed}}{N_{pl,fi,Rd}};\ \rho=\left(2\frac{V_{Ed}}{V_{pl,fi,y,Rd}}-1\right)^2$$

For bi-axial bending the criterion in EN 1993-1-1 6.2.9.1. (6) should be satisfied (I, box and pipe shaped cross-sections):

$$\left[\frac{M_{y,Ed}}{M_{N,fi,y,Rd}}\right]^{\alpha}+\left[\frac{M_{z,Ed}}{M_{N,fi,z,Rd}}\right]^{\beta}\le 1$$

Other cross-section types:

$$N_{pl,fi,Rd}=\frac{Af_y k_{y,\theta}}{\gamma_{M_{fi}}};\ M_{pl,fi,y,Rd}=\frac{W_{pl,y}f_y k_{y,\theta}}{\gamma_{M_{fi}}};\ M_{pl,fi,z,Rd}=\frac{W_{pl,z}f_y k_{y,\theta}}{\gamma_{M_{fi}}}$$

$$\frac{N_{Ed}}{N_{pl,fi,Rd}}+\frac{M_{y,Ed}}{M_{pl,fi,y,Rd}}+\frac{M_{z,Ed}}{M_{pl,fi,z,Rd}}\le 1$$

If the shear force is greater than 50% of the shear resistance, the bending moment resistance is reduced considering the effect of shear force according to EN 1993-1-1 6.2.8. and EC3-1-2 ($M_{V,fi,y,Rd}$, $M_{V,fi,z,Rd}$).

Check only prescribed failure modes



Elastic design check (class 3 cross-sections)

$$\frac{N_{Ed}}{A\frac{f_y k_{y,\theta}}{\gamma_{M_{fi}}}}+\frac{M_{y,Ed}}{W_{el,y}\frac{f_y k_{y,\theta}}{\gamma_{M_{fi}}}}+\frac{M_{z,Ed}}{W_{el,z}\frac{f_y k_{y,\theta}}{\gamma_{M_{fi}}}}\le 1$$

$$\frac{V_{y,Ed}}{V_{el,fi,y,Rd}}\le 1;\ \frac{V_{z,Ed}}{V_{el,fi,z,Rd}}\le 1$$

Elastic design check (class 4 cross-sections)

$$\frac{N_{Ed}}{A_{eff}\frac{f_y k_{y,\theta,4}}{\gamma_{M_{fi}}}}+\frac{M_{y,Ed}+\Delta M_{y,Ed}}{W_{eff,y}\frac{f_y k_{y,\theta,4}}{\gamma_{M_{fi}}}}+\frac{M_{z,Ed}+\Delta M_{z,Ed}}{W_{eff,z}\frac{f_y k_{y,\theta,4}}{\gamma_{M_{fi}}}}\le 1$$

$$\frac{V_{y,Ed}}{V_{el,fi,y,Rd}}\le 1;\ \frac{V_{z,Ed}}{V_{el,fi,z,Rd}}\le 1$$

Plastic design check

$$\frac{N_{Ed}}{N_{pl,fi,Rd}}+\frac{M_{y,Ed}}{M_{pl,fi,y,Rd}}+\frac{M_{z,Ed}}{M_{pl,fi,z,Rd}}\le 1$$

$$\frac{V_{y,Ed}}{V_{pl,fi,y,Rd}}\le 1;\ \frac{V_{z,Ed}}{V_{pl,fi,z,Rd}}\le 1$$

Compression-Bending-Buckling

According to EN 1993-1-2, 4.2.3.5 (4.21a, c):

Elastic design check (class 3 cross-sections)

$$\frac{N_{Ed}}{\chi_{min,fi}Ak_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}+k_y\frac{M_{y,Ed}}{W_{el,y}k_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}+k_z\frac{M_{z,Ed}}{W_{el,z}k_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}\le 1$$

$$k_y=1-\frac{\mu_y N_{Ed}}{\chi_{y,fi}Ak_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}\le 3;\ k_z=1-\frac{\mu_z N_{Ed}}{\chi_{z,fi}Ak_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}\le 3$$

Elastic design check (class 4 cross-sections)

$$\frac{N_{Ed}}{\chi_{min,fi}A_{eff}k_{y,\theta,4}\frac{f_y}{\gamma_{M_{fi}}}}+k_y\frac{M_{y,Ed}+\Delta M_{y,Ed}}{W_{eff,y}k_{y,\theta,4}\frac{f_y}{\gamma_{M_{fi}}}}+k_z\frac{M_{z,Ed}+\Delta M_{z,Ed}}{W_{eff,z}k_{y,\theta,4}\frac{f_y}{\gamma_{M_{fi}}}}\le 1$$

$$k_y=1-\frac{\mu_y N_{Ed}}{\chi_{y,fi}A_{eff}k_{y,\theta,4}\frac{f_y}{\gamma_{M_{fi}}}}\le 3;\ k_z=1-\frac{\mu_z N_{Ed}}{\chi_{z,fi}A_{eff}k_{y,\theta,4}\frac{f_y}{\gamma_{M_{fi}}}}\le 3$$

Plastic design check

$$\frac{N_{Ed}}{\chi_{min,fi}Ak_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}+k_y\frac{M_{y,Ed}}{W_{pl,y}k_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}+k_z\frac{M_{z,Ed}}{W_{pl,z}k_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}\le 1$$

$$k_y=1-\frac{\mu_y N_{Ed}}{\chi_{y,fi}Ak_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}\le 3;\ k_z=1-\frac{\mu_z N_{Ed}}{\chi_{z,fi}Ak_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}\le 3$$

where

$$\mu_y=\left(2\beta_{M,y}-5\right)\bar{\lambda}_{y,\theta}+0.44\beta_{M,y}+0.29\le 0.8$$

$$\mu_z=\left(1.2\beta_{M,y}-3\right)\bar{\lambda}_{z,\theta}+0.71\beta_{M,z}-0.29\le 0.8$$

The calculation of $\chi_{y,fi}$ and $\chi_{z,fi}$ reduction factors is according to EC3-1-2 4.2.3.2.

Axial force-Bending-Lateral-torsional buckling

According to EN 1993-1-2, 4.2.3.5 (4.21b, d):

Elastic design check (class 3 cross-sections)

$$\frac{N_{Ed}}{\chi_{z,fi}Ak_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}+k_{LT}\frac{M_{y,Ed}}{\chi_{LT,fi}W_{el,y}k_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}+k_z\frac{M_{z,Ed}}{W_{el,z}k_{y,\theta}\frac{f_y}{\gamma_{M_{fi}}}}\le 1$$

$$k_{LT} = 1 - \frac{\mu_{LT} N_{Ed}}{\chi_{z,fi} A k_{y,\theta} \frac{f_y}{\gamma_{M_{fi}}}} \leq 3;\ k_z = 1 - \frac{\mu_z N_{Ed}}{\chi_{z,fi} A k_{y,\theta} \frac{f_y}{\gamma_{M_{fi}}}} \leq 3$$

Elastic design check (class 4 cross-sections)

$$\frac{N_{Ed}}{\chi_{z,fi} A_{eff} k_{y,\theta,4} \frac{f_y}{\gamma_{M_{fi}}}} + k_{LT} \frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT,fi} W_{eff,y} k_{y,\theta,4} \frac{f_y}{\gamma_{M_{fi}}}} + k_z \frac{M_{z,Ed} + \Delta M_{z,Ed}}{W_{eff,z} k_{y,\theta,4} \frac{f_y}{\gamma_{M_{fi}}}} \leq 1$$

$$k_{LT} = 1 - \frac{\mu_{LT} N_{Ed}}{\chi_{z,fi} A_{eff} k_{y,\theta,4} \frac{f_y}{\gamma_{M_{fi}}}} \leq 3;\ k_z = 1 - \frac{\mu_z N_{Ed}}{\chi_{z,fi} A_{eff} k_{y,\theta,4} \frac{f_y}{\gamma_{M_{fi}}}} \leq 3$$

Plastic design check

$$\frac{N_{Ed}}{\chi_{z,fi} A k_{y,\theta} \frac{f_y}{\gamma_{M_{fi}}}} + k_{LT} \frac{M_{y,Ed}}{\chi_{LT,fi} W_{pl,y} k_{y,\theta} \frac{f_y}{\gamma_{M_{fi}}}} + k_z \frac{M_{z,Ed}}{W_{pl,z} k_{y,\theta} \frac{f_y}{\gamma_{M_{fi}}}} \leq 1$$

$$k_{LT} = 1 - \frac{\mu_{LT} N_{Ed}}{\chi_{z,fi} A k_{y,\theta} \frac{f_y}{\gamma_{M_{fi}}}} \leq 3;\ k_z = 1 - \frac{\mu_z N_{Ed}}{\chi_{z,fi} A k_{y,\theta} \frac{f_y}{\gamma_{M_{fi}}}} \leq 3$$

where

$$\mu_{LT} = 0.15 \bar{\lambda}_{z,\theta} \beta_{M,LT} - 0.15 \leq 0.9$$

$$\mu_z = (1.2 \beta_{M,y} - 3) \bar{\lambda}_{z,\theta} + 0.71 \beta_{M,z} - 0.29 \leq 0.8$$

The calculation of $\chi_{LT,fi}$ and $\chi_{z,fi}$ reduction factors is according to EC3-1-2 4.2.3.2 and 4.2.3.3.

Shear /y [Vy]

According to EN 1993-1-1, 6.2.6, EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E:

Elastic design check (class 3 cross-sections)

$$\frac{V_{y,Ed}}{V_{el,fi,y,Rd}} \leq 1;\ V_{el,fi,y,Rd} = \frac{I_z \cdot t}{S_z} \frac{f_y k_{y,\theta}}{\sqrt{3} \gamma_{M_{fi}}}$$

Elastic design check (class 4 cross-sections)

$$\frac{V_{y,Ed}}{V_{el,fi,y,Rd}} \leq 1;\ V_{el,fi,y,Rd} = \frac{I_z \cdot t}{S_z} \frac{f_y k_{y,\theta,4}}{\sqrt{3} \gamma_{M_{fi}}}$$

For supported sections see tables at the end of chapter **6.6.1 Steel beam design according to Eurocode 3 – SD1 module**

Unsupported cross-section types or plastic design:

$$\frac{V_{y,Ed}}{V_{pl,fi,y,Rd}} \leq 1;\ V_{pl,fi,y,Rd} = A_v \frac{f_y k_{y,\theta}}{\sqrt{3} \gamma_{M_{fi}}}$$

Shear /z [Vz]

Check only prescribed failure modes

☐

AXISVM performs the following design check:

$$\frac{V_{zEd}}{\min(V_{z,fi,Rd}, V_{b,fi,Rd})} \leq 1$$

$V_{b,fi,Rd}$ shear web buckling resistance (for I and box cross-sections) is based on the shear resistance of webs. The calculation of $V_{z,fi,Rd}$ is shown below.

No guidelines may be found in EC3-1-2 about verification against shear web buckling in fire. In this case, the verification is performed according to sections 5.1-5.3 of EN 1993-1-5 with reduced yield strength and Young's modulus due to the elevated steel temperature (EC3-1-2). Furthermore, research results of André Reis, Nuno Lopes, Paulo Vila Real are also considered by this verification (A. Reis, N. Lopes and P. Vila Real: *Numerical study of steel plate girders under shear loading at elevated temperatures,* Journal of Constructional Steel Research, 117 (2006) 1-12).

Class 1, 2 and 3 cross-sections:

$$V_{b,fi,Rd} = \chi_{w,\theta} h_w t_w \frac{f_y k_{y,\theta}}{\sqrt{3}\gamma_{M_{fi}}}$$

Class 4 cross-sections:

$$V_{b,fi,Rd} = \chi_{w,\theta} h_w t_w \frac{f_y k_{y,\theta,4}}{\sqrt{3}\gamma_{M_{fi}}}$$

where

$$\chi_{w,\theta} = 1.2 \quad \bar{\lambda}_{w,\theta} < \frac{0.61}{1.2}$$
$$\chi_{w,\theta} = 0.1 + \frac{0.55}{\bar{\lambda}_{w,\theta}} \quad \bar{\lambda}_{w,\theta} \geq \frac{0.61}{1.2}$$

$$\bar{\lambda}_{w,\theta} = \frac{h_w}{37.4 t_w \varepsilon \sqrt{k_\tau}}; \varepsilon = 0.85\sqrt{\frac{235}{f_y}}$$

Check only prescribed failure modes



According to EN 1993-1-1, 6.2.6, EN 1993-1-2, 4.2.3.3, 4.2.3.4, Annex E:

Elastic design check (class 3 cross-sections)

$$\frac{V_{z,Ed}}{V_{el,fi,z,Rd}} \leq 1;\ V_{el,fi,z,Rd} = \frac{I_y \cdot t}{S_y} \frac{f_y k_{y,\theta}}{\sqrt{3}\gamma_{M_{fi}}}$$

Elastic design check (class 4 cross-sections)

$$\frac{V_{z,Ed}}{V_{el,fi,z,Rd}} \leq 1;\ V_{el,fi,z,Rd} = \frac{I_y \cdot t}{S_y} \frac{f_y k_{y,\theta,4}}{\sqrt{3}\gamma_{M_{fi}}}$$

For supported sections see tables at the end of chapter **6.6.1 Steel beam design according to Eurocode 3 – SD1 module**

Unsupported cross-section types or plastic design:

$$\frac{V_{z,Ed}}{V_{pl,fi,z,Rd}} \leq 1;\ V_{pl,fi,z,Rd} = A_v \frac{f_y k_{y,\theta}}{\sqrt{3}\gamma_{M_{fi}}}$$

Web shear-Bending-Axial force

In case of I and box cross-sections, this interaction design check is also performed according to section 7.1 of EN 1993-1-5 and sections 6.2.8, 6.2.9 of EN 1993-1-1 with reduced yield strength and Young's modulus due to the elevated steel temperature (EC3-1-2).

Check only prescribed failure modes



$$\frac{M_{Ed}}{M_{pl,fi,Rd}} + \left(1 - \frac{M_{f,fi,Rd}}{M_{pl,fi,Rd}}\right) \cdot \left(2\frac{V_{Ed}}{V_{b,fi,,Rd}} - 1\right)^2 \leq 1$$

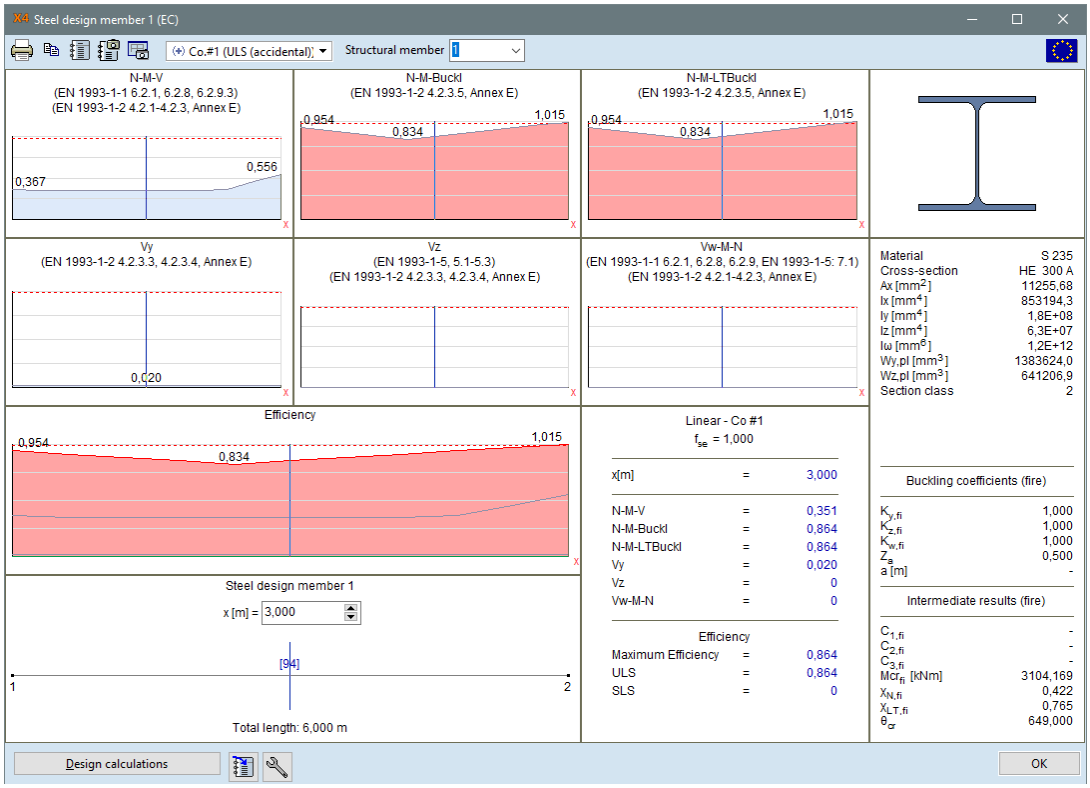
where $M_{f,fi,Rd}$ is the plastic bending resistance of the flanges.
If *Check only prescribed failure modes* is checked, this interaction design check is not performed.

Critical temperature

In many cases the critical temperature (the steel temperature where failure of the element occurs) also has to be determined. For example, the thickness of intumescent coating fire protection is selected on the basis of section factor (A/V) and critical temperature.
AXISVM can calculate the critical temperature for the above listed failure modes. The critical temperature of the element is the minimum of the critical temperatures for the different failure modes. The investigated range is 20 – 1000 °C. So, if AXISVM displays 20 °C as critical temperature in case of an element, it means that the utilization is high and the safety of the element against failure cannot be confirmed even at room temperature in accidental design situation based on the regulations of EC3-1-2. In this case, it is very likely that the resistance is not adequate in persistent design situation neither. If AXISVM displays 1001 °C as critical temperature in case of an element, it means that the utilization is very low and the critical temperature is above 1000 °C.

Diagrams

Click on the steel design member to display the diagrams all individual checks and their envelope. Results for any position of any steel design member in any load case or combination can be obtained by setting *x* or dragging the tracking line. If a check cannot be performed with the current cross-section the respective diagram is replaced by a cross.
If the selected or the critical combination contains fire load case and fire design was performed for the selected steel design member, buckling coefficients in fire and critical temperature are shown.



Design calculations

Click on *Design calculations* button to see the design details. The calculation can be saved into the documentation. In case of fire design, the type of fire curve, the design temperature and the critical temperature are also shown.

11. Shear(z) (strength check):

EN 1993-1-1: 6.2.6, EN 1993-1-2: 4.2.3.3, 4.2.3.4

Fire curve: **Parametric fire curve**

Design temperature $\theta = 510.652$ °C

Critical section: $x = 0.00 \cdot L = 0.00 \cdot 600.000 = 0$ cm

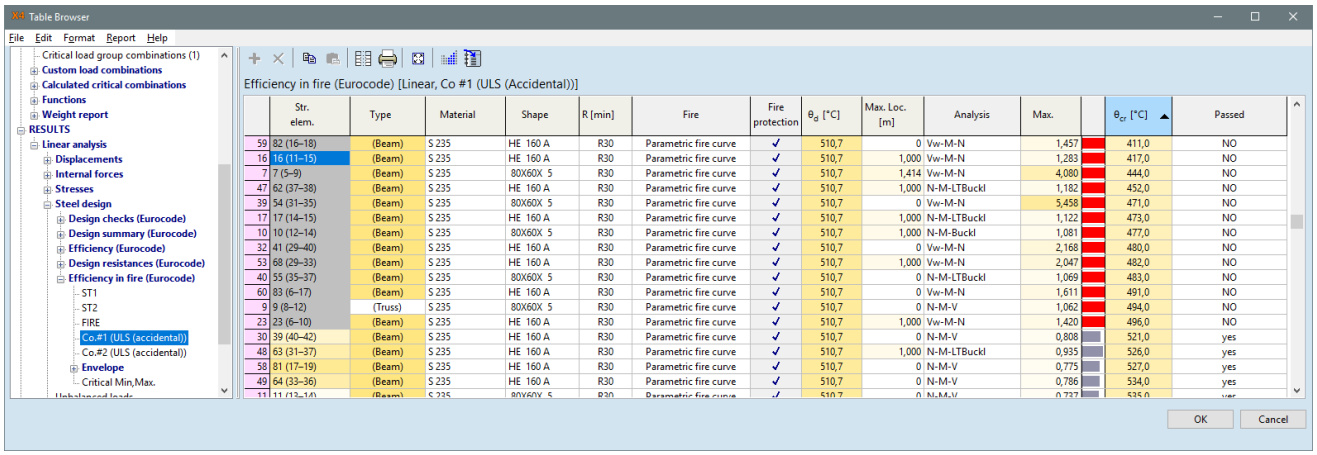
$$V_{pl,fi,Rd,z} = \frac{A_{V,z} \cdot f_y \cdot k_{y,\theta}}{\sqrt{3} \cdot \gamma_{MFi}} = \frac{15.887 \cdot 23.500 \cdot 0.747}{\sqrt{3} \cdot 1} = 161.012 \text{ kN} \quad (6.18, 4.16)$$

$$\eta_{V_{z,fi}} = \frac{|V_{z,Ed_1}|}{V_{pl,fi,Rd,z}} = \frac{|(-54.000)|}{161.012} = 33.5\ \% \quad (6.17) \qquad \textbf{passed}$$

Critical temperature $\theta_{cr} = 692.000$ °C

Result table

Among steel design result tables, *Utilization in fire* summarizes the fire design results in order to help the communication with fire safety engineers.



Besides the basic initial parameters, the following parameters are listed: required time of fire resistance (e.g., R30), type of selected fire curve, design steel temperature, critical failure mode, utilization and critical temperature.

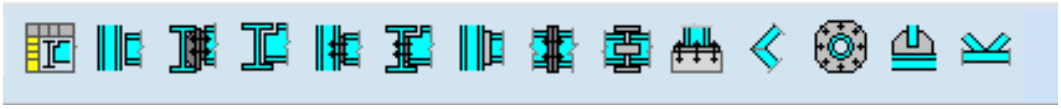
Cross-section optimization

If SD9 module is installed, steel cross-section optimization can be performed for load combinations that contain fire load case with respect to steel fire design rules. During the optimization, design steel temperature is not recalculated automatically. For this reason, the utilization may be slightly different after analysis of the optimized structural configuration.
See **6.6.1.4 Steel cross-section optimization - SD9 module**

6.6.3.2 Fire design of cold-formed, thin-walled steel design members

In the case of cold-formed, thin-walled steel members (typically purlins and girts), fire resistance capacity cannot be verified by bending load-bearing action. Due to their small wall thickness, these elements heat up very rapidly, leading to extremely large deflections within a few minutes. However, due to these large deflections, the purlins begin to behave as tensioned cables and are capable of carrying by catenary action.
To verify catenary load-bearing capacity, the application of recommendations/design guidelines issued by the CTICM (*Centre Technique Industriel de la Construction Métallique*), a French industrial technical and research center specializing in metal and steel construction, is widely accepted in professional practice: *C. Renaud, S. Sakji: CTICM - Steel frameworks: Method of proving quarter hour stability (R15)*
In the ULS design parameters window (*see* **6.6.2 Design of cold-formed, thin-walled steel purlins and girts according to Eurocode 3 -SD3 module**), the fire resistance design switch must be activated; no additional parameters need to be set. If design is performed for an accidental load combination involving fire action, and fire load parameters has been defined for the steel element in question (**4.10.32 Fire effect on steel line elements – SD8 module**), the software automatically performs the fire resistance design based on the CTICM recommendation. If the fire resistance requirement differs from 15 minutes, the software includes an extended version of the CTICM design models that can handle this. The check covers the strength verification of the tensioned cable itself, and the maximum equilibratable load value is also determined based on the resistance of the connection. Calculation details can be accessed by clicking the Design calculation button.

6.6.4 Steel connection design - SC1 module



This module is developed by GammaCAD sp. z o.o., Poland GammaCAD www.gammacad.pl

Verification of steel connections

The SC1 module is to verify bolted and welded steel connections by checking if they meet the requirements for structural joints specified in the design code.
The module uses the "component method" defined and supported by the relevant design codes for the verification.

Design codes

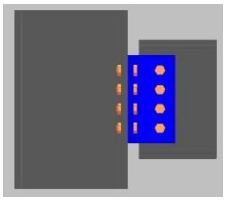
Steel connection verification is supported according to
EN 1993-1-1 (2005): Eurocode 3: Design of steel structures - Part 1-1: General rules and rules for buildings
EN 1993-1-8 (2012): Eurocode 3: Design of steel structures - Part 1-8: Design of joints

The range of supported steel joint types, restrictions and instructions to define joints are included in a PDF document: *Steel connections – SC1 application* is available on our website. **Download >>**

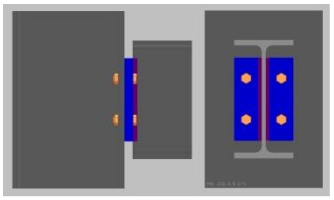
Joint types

Available types

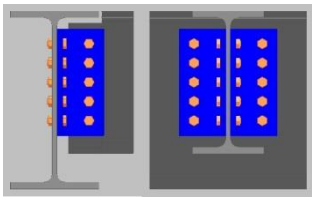
a) Column – beam (semi-rigid)



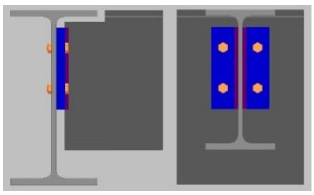
b) Column – beam (nominally pinned)



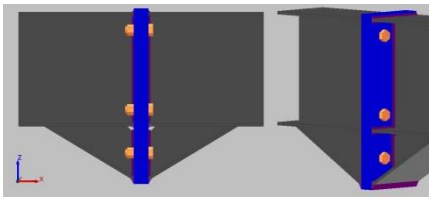
c) Girder-beam (semi-rigid)



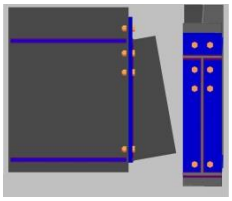
d) Girder – beam (nominally pinned)



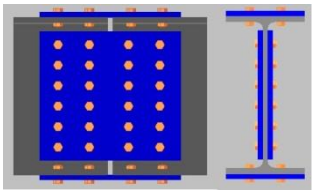
e) Beam-beam (end plate)



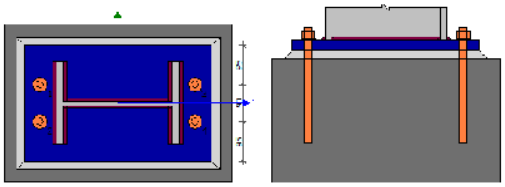
f) Beam-column (end plate)



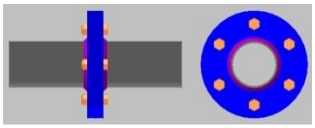
g) Beam splice



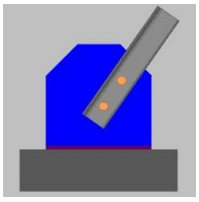
h) Column - base



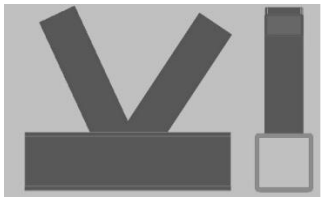
i) Pipe splice



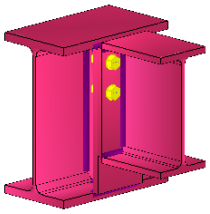
j) Gusset plate (single truss)



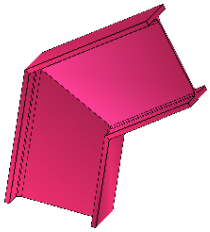
k) Tube node



l) Grider-beam rigid connection

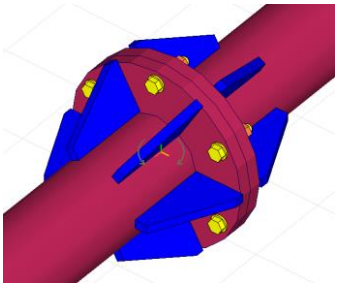


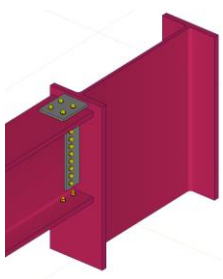
m) Knee welded connection



Rendered view

Parameters of joints are saved into the AXISVM model. Joint details are displayed in the rendered view.

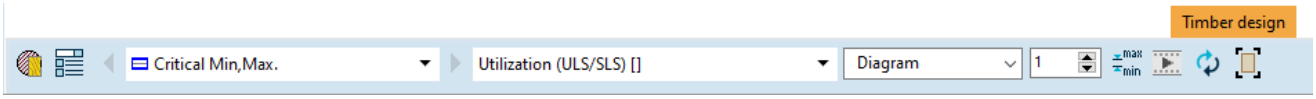




Further information

Further details are available in the SC1 manual. **Download >>**

6.7 Timber design



The following design methods are available for timber design members:

TD1 – Timber beam elements
6.7.1 Timber beam design – TD1 module

TD8 – Faszerkezetek ellenőrzése tűzhatásra
6.7.2 Timber beam fire design – TD8 module

TD9 – Faszerkezetek szelvényoptimalizálása
6.7.3 Timber cross-section optimization – TD9 module

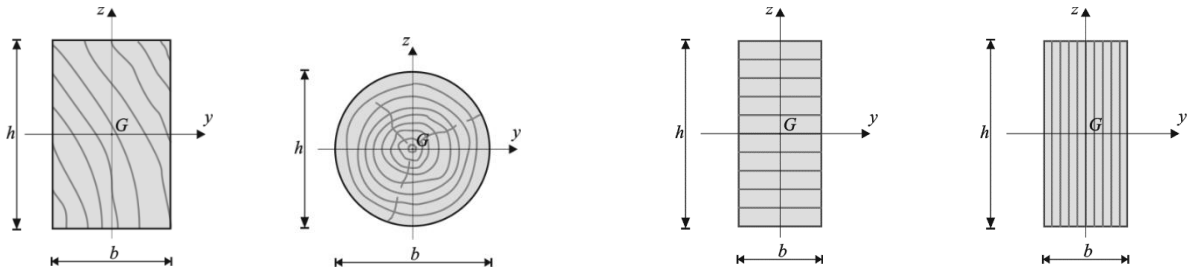
6.7.1 Timber beam design – TD1 module

EUROCODE 5

The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

The timber beam design module can be applied to the following cross-sections and timber materials:

a) Rectangle for solid timber, Glued laminated timber (Glulam) and for Laminated veneer lumber (LVL)
b) Round for solid timber



Solid timber (softwood, hardwood) *Glulam* *LVL*

Material properties

The material database contains the solid, Glulam and LVL timber material properties according to the related EN standard. (Solid timber: EN338, Glulam: EN 1194)

Characteristic strength	*Notation*
Bending strength	$f_{m,k}$
Tensile strength parallel to grain	$f_{t,0,k}$
Tensile strength perpendicular to grain	$f_{t,90,k}$
Compression strength parallel to grain	$f_{c,0,k}$
Compression strength perpendicular to grain in *y* direction*	$f_{c,90,k,y}$
Compression strength perpendicular to grain in *z* direction*	$f_{c,90,k,z}$
Shear strength perpendicular to the grain in y direction*	$f_{v,k,y}$
Shear strength perpendicular to the grain in z direction*	$f_{v,k,z}$

**In case of solid and Glulam timber $f_{v,k,z} = f_{v,k,y} = f_{v,k}$ and $f_{c90,k,z} = f_{c90,k,y} = f_{c90,k}$*

Modulus of elasticity	*Notation*
Mean value parallel to grain	$E_{0,mean}$
Mean value perpendicular to grain	$E_{90,mean}$
5% value of modulus parallel to grain	$E_{0,05}$
Mean value of shear modulus	G_{mean}

Density	*Notation*
Characteristic value of density	ρ_k

Mean value of density	ρ_{mean}

Partial factor	Notation
Partial factor for material	γ_M

Size effect factor	Notation
for LVL timber	s

Timber classes

Timber elements must have a service class. Service class can be set in the line elements definition dialog, at Service Class field. See **4.9.10 Line elements**

Service classes (EN 1995-1-1, 2.3.1.3):

Service class 1 – where the average moisture content in most softwoods will not exceed 12%. This corresponds to a temperature of 20°C and a relative humidity of the surrounding air only exceeding 65% for a few weeks per year.

Service class 2 – where the average moisture content in most softwoods will not exceed 20%. This corresponds to a temperature of 20°C and a relative humidity of the surrounding air only exceeding 85% for a few weeks per year.

Service class 3 – where the average moisture content in most softwoods exceeds 20%.

Design strength and other design properties of the timber materials depend on the service class.

Load-duration classes

Timber design module requires information on the load duration. So, if a timber material has been defined in the model load case duration class can be entered.

See **4.10.1 Load cases, load groups**

Design strength components

The design values of strength are calculated from the characteristic values of strength according to the following formulas:

In case of $f_{t,90,d}, f_{c,0,d}, f_{c,90,d}, f_{v,d}$ (Solid, Glulam, LVL timbers): $f_d = k_{mod} f_k / \gamma_M$

In case of $f_{m,d}$ (Solid, Glulam, LVL timbers): $f_d = k_{mod} k_h f_k / \gamma_M$

In case of $f_{t,0,d}$ (Solid and Glulam timbers): $f_d = k_{mod} k_h f_k / \gamma_M$

In case of $f_{t,0,d}$ (LVL timber): $f_d = k_{mod} k_l f_k / \gamma_M$

where

k_{mod} is a modification factor (EN 1995-1-1, 3.1.3)

k_h is the depth factor (EN 1995-1-1, 3.2, 3.3, 3.4)

k_l is the length factor for LVL timber (EN 1995-1-1, 3.4)

f_k is the characteristic strength

γ_M is the partial factor of material (EN 1995-1-1, Table 2.3)

☞ **The values of k_{mod} factor can be set in the *Design codes* window under the *Timber* tab (see 3.3.7 Design codes).**

k_h factor

The $f_{m,k}$ and $f_{t,0,k}$ characteristic strength values are determined for a reference depth of beam. In case of solid and Glulam timber if the depth (h) of the cross-section less than the reference value, the design strength is multiplied with the following factor.

Solid timber: $k_h = \min\left\{\left(\frac{150}{h}\right)^{0.2}; 1.3\right\}$ (if $\rho_k \leq 700$ kg/m^3)

Glulam: $k_h = \min\left\{\left(\frac{600}{h}\right)^{0.1}; 1.1\right\}$

In case of LVL timber if the depth (h) of the cross-section not equal to the reference value, the design strength is multiplied with the following factor.

LVL: $k_h = \min\left\{\left(\frac{300}{h}\right)^{s}; 1.2\right\}$

(where s is the size effect exponent, h is the cross-section depth in mm.

Reference depths for solid timber is 150 mm, for Glulam it is 600 mm, for LVL it is 300 mm.

k_l factor

The $f_{t,0,k}$ characteristic strength value of LVL timber is determined for a reference length of beam. If the length of the beam is not equal to the reference length, the design strength is multiplied by the following factor.

$$k_l = \min\left\{\left(\frac{3000}{l}\right)^{s/2}; 1.1\right\}$$

(where s is the size effect exponent).

l is the beam length in mm. Reference length: 3000 mm.

Stiffness values for analysis

Analysis type	Modulus (SLS)	Modulus (ULS)
First order, linear elastic	$E_{mean,fin} = \frac{E_{mean}}{1 + k_{def}}$	$E_{mean,fin} = \frac{E_{mean}}{1 + \psi_2 k_{def}}$

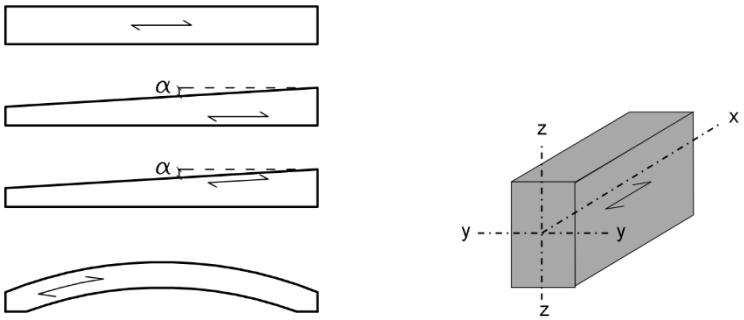
	$G_{mean,fin} = \frac{G_{mean}}{1 + k_{def}}$	$G_{mean,fin} = \frac{G_{mean}}{1 + \psi_2 k_{def}}$
Second order, linear elastic	$E_d = E_{mean}/\gamma_M$	$E_d = E_{mean}/\gamma_M$
	$G_d = G_{mean}/\gamma_M$	$G_d = G_{mean}/\gamma_M$
Frequency	E_{mean}, G_{mean}	E_{mean}, G_{mean}

Conservative way $\psi_2 = 1$ is used.

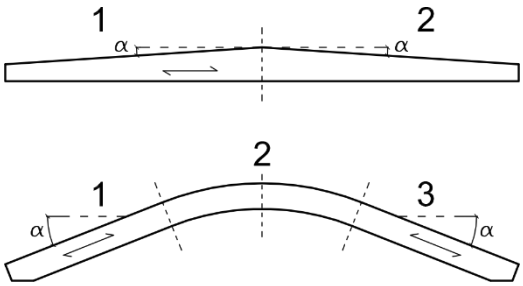
Design assumptions

- There is no hole or other weakening in the beams.
- The cross-section constant (rectangle, round) or linear changing depth along the beam (tapered beam).
- The beam elements that the module can check are classified into two main types and summarized in the figure below.
  1. 'simple' beam: beams with straight-axis; single tapered beam; simple curved beam
  2. 'complex' beam: double tapered beam; curved beam. These beams can only be of laminated-glued material (GLULAM).
- The types of beam elements that the module can check are summarized in the figure below.
- The grain parallel to the beam *x* axis.
- In case of tapered beam, the grain parallel one of the longitudinal edges.
- The dominant bending plane is the *x*-*z* plane of the beam (moment about *y* axis).
- $I_y \geq I_z$
- In case of Glulam the laminates are parallel to the *y* axis of the cross-section.
- in case of LVL the laminates are parallel to the *z* axis of the cross-section.

'simple' beam types



'complex' beam types



Modeling 'complex' beams

In the case of 'complex' type beams, the program can assemble the complete structural element from two or three "simple" parts. For this reason, when assembling the structural elements (see **6.7.1.1 ULS design parameters**), the user must select the option *'Do not break member at side element connections or nodal supports'* to have the program assemble the beam into one design memb er. (In the case of the option *'Break member at side element connections or nodal supports'*, the program will split the selected but continous elements into several design members.)
For each type, the individual elements must be in the same plane and the complex beam must have a symmetrical or near symmetrical design. The local *z*-axis of the cross-section must not deviate by more than ± 5° from the plane defined by the local *x*-axis of the beam and the global *Z*-axis.

A double tapered beam is always composed of two simple beam parts with the same but opposite slope. The continuity of the cross-section at the joining node must be ensured and the elements must share the same local system. Only the *'h'* beam height can change along the beam and the ridge point of the girder must be at the top (no inverted girder can be specified).
A complex curved beam shall consist of three parts. The central element is curved, the two end parts are straight, the cross-section is constant. The continuity of the section and the local system of

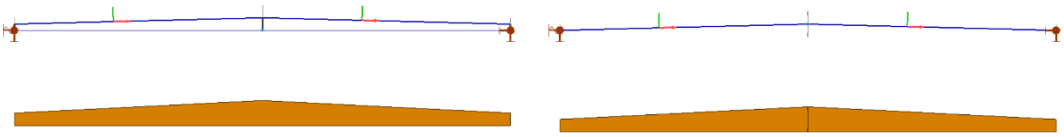
elements must be ensured at connecting nodes and the straight elements must join tangentially to the curved member part. The angle formed by the two straight parts shall not less than 90°.

The double tapered beam should be modelled by one of the following options:

- The modeling axes of the complex beam parts are aligned, the form of the beam is given by eccentricity (see **4.9.10.2.1 Eccentricity of beams and ribs**).
- The modeling axis of the two beam parts are not in a straight line.

In both cases, the edge of the complex beam opposite the sloped edge must be continuous.

The two different modelling options:



Checks

Normal force-Bending [N-M]	*(EN 1995-1-1, 6.2.3, 6.2.4)*
Compression-Bending-Buckling (in plane) [N-M-Buckling]	*(EN 1995-1-1, 6.3.2)*
Normal force-Bending-Lateral tors. buckling [N-M-LT buckling]	*(EN 1995-1-1, 6.3.3)*
Shear /y -Torsion /x [Vy-Tx]	*(EN 1995-1-1, 6.1.7, 6.1.8)*
Shear (y) - Shear(z) - Torsion (x) [Vy- Vz-Tx]	*(EN 1995-1-1, 6.1.7, 6.1.8)*
Moment /y - Shear /z (tensile stress perpendicular to the grain) [My-V z]	*(EN 1995-1-1, 6.4.3)*

Calculated parameters

$\lambda_{rel,y}$ Relative slenderness ratio (*y*) /in *z-x* plane of the beam/ []	
$\lambda_{rel,z}$ Relative slenderness ratio (*z*) /in *y-x* plane of the beam/ []	
$k_{c,y}$ Buckling (instability) factor (*y*) /in *z-x* plane of the beam/ []	*(EN 1995-1-1, 6.3.2)*
$k_{c,z}$ Buckling (instability) factor (*z*) /in *x-y* plane of the beam/ []	*(EN 1995-1-1, 6.3.2)*
k_{crit} Lateral torsional buckling factor []	*(EN 1995-1-1, 6.3.3)*
k_h Depth factor []	*(EN 1995-1-1, 3.2, 3.3, 3.4)*
k_{mod} modification factor []	*(EN 1995-1-1, 3.1.3)*
$\sigma_{t,90,d}$ (tensile stress perpendicular to the grain) [N/mm^2]	*(EN 1995-1-1, 6.4.3)*

☞ **AXISVM performs the following checks only. All the other checks specified in the design code like supports, joints, etc. has to be completed by the user.**

Normal force-Bending

The design value of normal force can be tension or compression.
Tension and moment (EN 1995-1-1, 6.2.3)

$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + \frac{\sigma_{m,y,d}}{f_{m,y,d}} + k_m \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$
$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + k_m \frac{\sigma_{m,y,d}}{f_{m,y,d}} + \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

Compression and moment (EN 1995-1-1, 6.2.4)

$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,y,d}}{f_{m,y,d}} + k_m \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$
$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + k_m \frac{\sigma_{m,y,d}}{f_{m,y,d}} + \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

where $k_m = 0.7$ in case of rectangle cross-section, $k_m = 1$ in all other cases.

Compression-Moment-Buckling

(EN 1995-1-1, 6.3.2)

$$\frac{\sigma_{c,0,d}}{k_{c,y}f_{c,0,d}} + \frac{\sigma_{m,y,d}}{f_{m,y,d}} + k_m\frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

$$\frac{\sigma_{c,0,d}}{k_{c,z}f_{c,0,d}} + k_m\frac{\sigma_{m,y,d}}{f_{m,y,d}} + \frac{\sigma_{m,z,d}}{f_{m,z,d}} \le 1$$

where,

$k_{c,y}$ Buckling (instability) factor (*y*) /in *z-x* plane of the beam/ (EN 1995-1-1, 6.3.2)

$k_{c,z}$ Buckling (instability) factor (*z*) /in *x-y* plane of the beam/ (EN 1995-1-1, 6.3.2)

In case of tension force the $f_{c,0,d}$ is replaced with $f_{t,0,d}$ and $k_{c,y} = k_{c,z} = 1$

Normal force-Bending-LT buckling

For lateral torsional buckling, check the program assumptions that the beam is bending in *z-x* plane (about *y* axis). If there is simultaneous M_z moment on the beam and the compression stress from M_z reach the 3% of the $f_{c,0,d}$ a warning message appears.

Bending only (EN 1995-1-1, 6.3.3)

$$\frac{\sigma_{m,d}}{k_{crit}f_{m,d}} \le 1$$

Compression and moment (EN 1995-1-1, 6.3.3)

$$\left(\frac{\sigma_{m,d}}{k_{crit}f_{m,d}}\right)^2 + \frac{\sigma_{cd}}{k_{c,z}f_{c,0,d}} \le 1$$

Tension and bending

In case of small tension and bending that lateral torsional buckling could be occur, however there is no rule in EC5 for this case.

The following conservative check is used.

$$\frac{|\sigma_{mt,d}|}{k_{crit}f_{m,d}} \le 1;$$

$$\sigma_{mt,d} = \frac{M_d}{W_y} + \frac{N_d}{A} < 0$$

where k_{crit} is the lateral buckling factor according to the following

$\lambda_{rel,m} \le 0.75$ $\quad k_{crit} = 1$

$0.75 < \lambda_{rel,m} \le 1.4$ $\quad k_{crit} = 1.56 - 0.75 \cdot \lambda_{rel,m}$

$1.4 < \lambda_{rel,m}$ $\quad k_{crit} = 1/\lambda_{rel,m}^2$

Shear-Torsion

There is no rule in EC5 for case of simultaneous shear force and torsional moment.

In this case the program uses the interaction formula according to DIN EN 1995-1-1

Shear(y) , Shear(z) and torsion

$$\max\left[\frac{\tau_{v,y,d}}{f_{v,d}}; \frac{\tau_{v,z,d}}{f_{v,d}}; \frac{\tau_{tor,d}}{k_{shape}f_{v,d}} + \left(\frac{\tau_{v,y,d}}{f_{v,d}}\right)^2 + \left(\frac{\tau_{v,z,d}}{f_{v,d}}\right)^2\right] \le 1$$

where,

k_{shape} is a factor for the shape of cross-section.

For round cross-section $k_{shape} = 1.2$,

for rectangular cross-section $k_{shape} = \min\{1 + 0.15 \cdot h/b\,; 2.0\}$

Moment-Shear

In case of curved beams, the program checks the tensile stress perpendicular to the grain from M_y and V_z forces. (EN 1995-1-1, 6.4.3.)

Moment(*y*)-Shear(*z*)

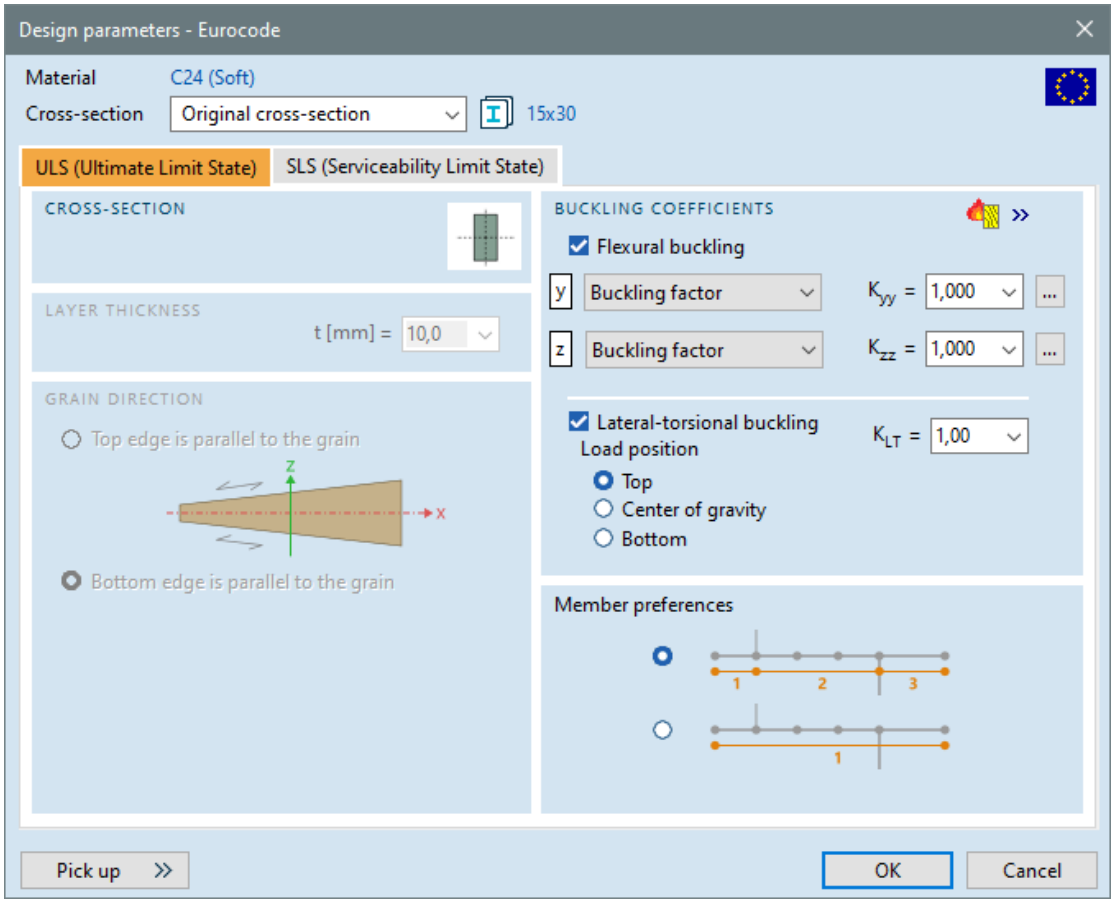
$$\frac{\tau_d}{f_{v,d}} + \frac{\sigma_{t,90,d}}{k_{dis}k_{vol}f_{t,90,d}} \le 1$$

k_{dis} is a factor which takes into account the effect of the stress distribution in the apex zone ($k_{dis} = 1.4$ for double tapered and curved beams)

k_{vol} is a volume factor $k_{vol} = [V_0/V]^{0.2}$

6.7.1.1 ULS design parameters

For the design based on Eurocode 5, the following design parameters should be defined and assigned to the design members:



Cross-section

For timber design, a cross-section can be defined, that is different from the *original cross-section* used for the static calculation. If the cross-section size is modified, design checks will be performed for the original internal forces, but with the modified cross-section size. Similarly, in case of designing for fire, the original fire reduction factors are used (see **6.7.2 Timber beam fire design – TD8 module**). When viewing the results, a modified design member is indicated by a * after the name of the cross-section, to distinguish it from the original cross-section. In the drop-down menu, we can select from the cross-sections available in the model.

Clicking on this icon will allow us to load further cross-sections from the *Cross-section library* or by defining a parametric shape. On the right side of this icon, the original cross-section sizes are displayed.

Clicking on the *Replace cross-sections* icon updates the actual cross-section of the selected members to the cross-section entered in the Design parameters dialog.

Layer thickness

In case of Glued laminated timber (Glulam) arcs thickness of one layer has to be defined.

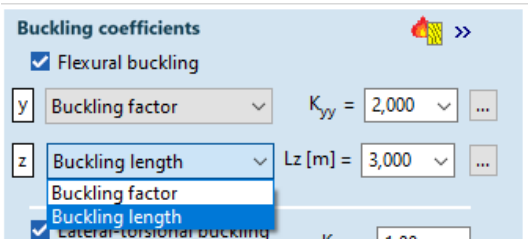
Grain direction

Set of grain direction in case of tapered beam. The grain direction can be parallel with the top edge or with the bottom edge. The top edge lays in the *+z* direction of the cross-section.

Stability parameters

Flexural buckling

There are two ways to specify buckling behaviour. The first is the buckling factor and the other is the buckling length.
Buckling factor or buckling length can be entered directly or can be taken from a shape obtained from buckling analysis of the model clicking the ... button just like in steel design. For details see **6.6.1.1.1 Buckling (flexural)**



K_y, K_z: buckling length factors corresponding to the *y* and *z* axis, respectively.

$$K_y = l_{ef,y}/l\,; \quad K_z = l_{ef,z}/l$$

where l is the member length
$l_{ef,y}$ is the buckling length in *x-z* plane of the member
$l_{ef,z}$ is the buckling length in *x-y* plane of the member
The checkbox can be used to activate the buckling check.

Ly, Lz *buckling lengths*. The entered buckling length will be used regardless of the length of the design member. This often leads to more straightforward design in case of complex structures.

Lateral torsional buckling

K_{LT}: lateral buckling length factors corresponding to the *z* axis.

$$K_{LT} = l_{ef}/l$$

where l is the member length
l_{ef} is the lateral buckling length of the member corresponding to the *z* axis.

When the load not applied to the centre of gravity, the program modifies the lateral buckling length according to the following:

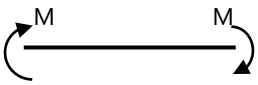
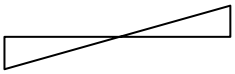
- if the load is applied to the compression edge of the member the l_{ef} is increased by $2h$
- if the load is applied to the tension edge of the member the l_{ef} is decreased by $0.5h$

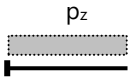
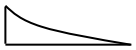
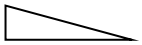
The checkbox can be used to activate the lateral torsional buckling check.

Common values of K_{LT} factor.
(Some of these values can be found in EN 1995-1-1, Table 6.1)

Loading type (direct load)	*My moment distribution between the lateral supports*	*Lateral support condition (in x-y plane)*
pz		0.9
Fz		0.8
Fz Fz, ¼ ½ ¼		0.96
pz		0.42
Fz		0.64

Loading type (no direct load)	*My moment distribution between the lateral supports*	*Lateral support condition (in x-y plane)*
M M		1.0
M ½ M		0.76
M 0		0.53
M ½ M		0.37

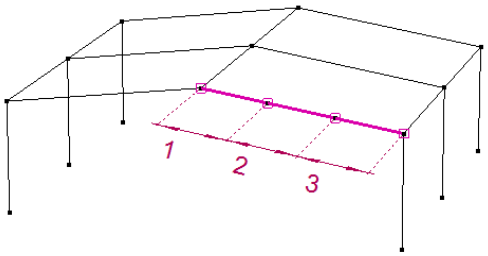
		0.36

Loading type (cantilever)	*My moment distribution*	*Lateral support condition (in x-y plane)*
		0.5
		0.8

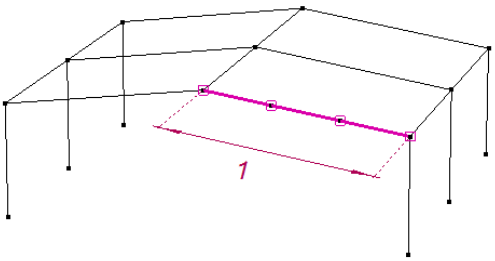
Design members

The design is performed on design members that can consist of one or more finite elements (beams and/or ribs). A group of finite elements can become a design member only if the finite elements in the group satisfy some requirements checked by the program: to be located on the same straight line or arc, to have the same material, cross-section and to have joining local coordinate systems.
The program allows two methods to define design members as follows:

Any node of a selection set of finite elements where another finite element is connected will become an end point of a design member within the selection set of finite elements.



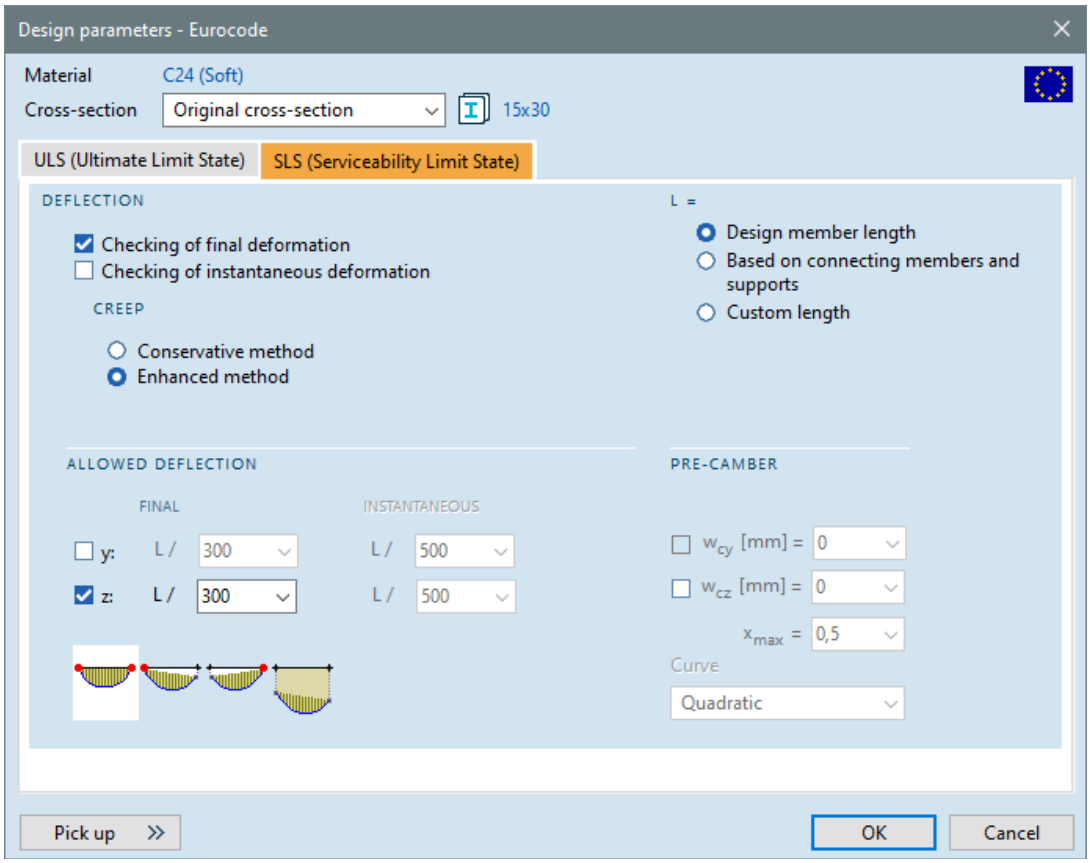
The finite elements in the selection set become only one design member irrespective of other finite elements connecting to its nodes.



6.7.1.2 SLS design parameters

The parameters for the serviceability limit state (SLS) checks can be set on the SLS tab in the Design parameters window. The program checks the deflections of beams.
The required parameters and the type of the SLS checks according to Eurocode and Swiss (SIA) standards are different.

Eurocode parameters



In the case of Eurocode standard, the final (u_{fin}) and the instantaneous deformation can be checked by the module. The check is only available for the results of a linear static calculation.
In the static analysis, the modulus of elasticity is decreased (divided by the value of $1+k_{def}$) considering the effect of creep. The SLS check is based on the results of the static analysis, but the deformation values are determined by the followings:

- Instantaneous deformation: the results of the static result is divided by $(1+k_{def})$ (another static analysis with short-term modulus of elasticity is not executed).
- To calculate the accurate value of final deformation (considering the elements having different creep values and the effect of the stiffness of connecting structural elements) would lead to complicated solutions in many cases. Two different static analyses would be required (for the short and long-term loads with the effect of creep) to combine the displacement results. As an approximation, the module offers two different solutions detailed below:

  a) In the case of the *Conservative method*, the final deformation is determined by the characteristic SLS combination and the displacement results of the static analysis (with the lowered modulus of elasticity) are considered as the final value of the determination. This conservative method is based on the recommendation of the book: *Jack Porteous: Structural Timber Design to Eurocode 5*. This solution approaches the exact deformation value from the safe side because of the applied characteristic combination.

  b) In the case of the *Corrected method*, the final deformation is recalculated. The module performs a correction calculation using existing static results but takes into account the short and long-term effects of the loads. The result is calculated by the summation of the deformation results of each load case considering the related combination factors.
  This recalculation is done for each design member separately without new static analysis. For this reason, this method also an approximation but gives better approximation

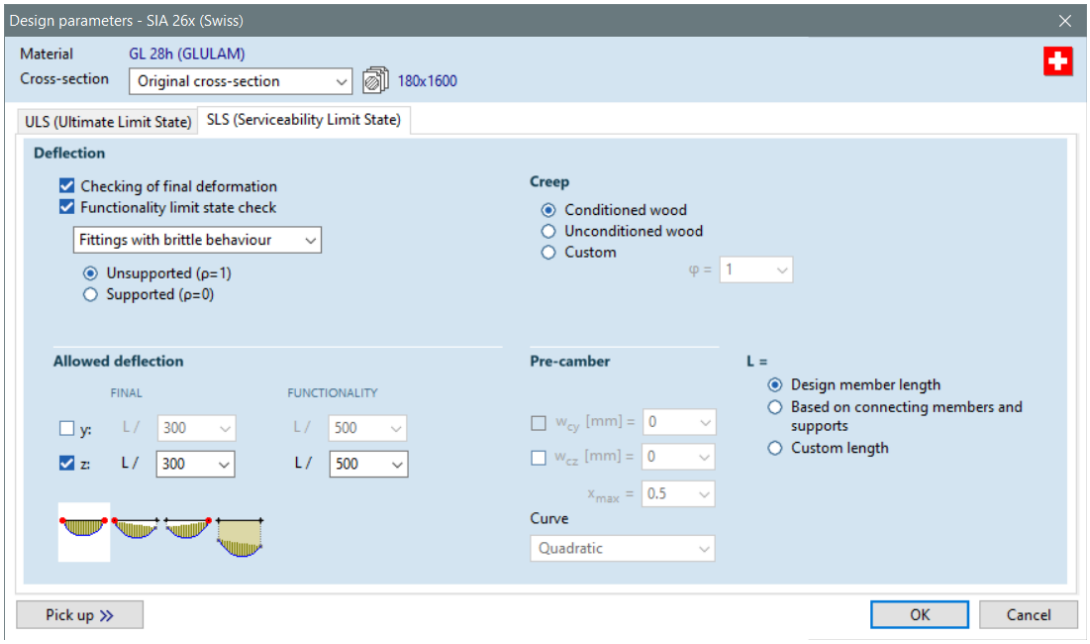
compared to the result of the *Conservative method*.
The recalculation can only be executed on characteristic load combinations or on simple load cases. In the case of custom load combination, the factor of ψ_0 comes from the factors of the given load combination for the variable loads (it means that the given factor should be between 0 and 1. The ψ_2 combination factor is taken out from the load group properties of the variable load.

☞ **The Spanish standard is an exception to the above. The module checks the active deflection (u_{act} see AN/UNE-EN 1995-1-1: AN.4) and the instantaneous deflection (uinst) for linear static calculations. The final value of deflection (ufin) calculated by the conservative method is given by the model displacement components, but is not included in the design calculation and in the Tables of the design members.**

EC Spanish

The instantaneous deflection is calculated using the general method for Eurocode (see above).
The calculation of the active deflection is similar to the determination of the final deflection value as described above: the program generates the deflection value by superposing the deflection results for the load cases, but taking into account the combination factors to be considered for the active deflection. The program can only determine the active deflection value based on characteristic combinations or individual load cases.

SIA parameters



The creep value is determined by the two classifications (preconditioned and unconditioned wood) as defined in *Table 5 of SIA 265:2021*, or the user can specify a specific value.

In the case of SIA (Swiss) standard, the final deformation (u_{fin}) and the usability conditions detailed below can be verified. The check is only available for linear static calculation. Different deflection criteria can be set for each check.

- To get the value of the final deformation, the result displacement of the static analysis is multiplied by the factor of $1+\varphi$. The check is interpreted for a quasi-permanent load combination.
- The following three usability conditions can be verified by the module (only one of these can be requested). The checks follow the requirements of Table 3 of SIA 260. The result is calculated by the summation of the deformation results of each load case considering the related combination factors. This recalculation is done for each design member separately without a new static analysis. For this reason, it can only be considered as an approximate method.

  a) *Fittings with brittle behaviour:*
  The check can only be interpreted for SLS characteristic load combination. The structure can be considered as supported or unsupported. In the case of the supported structure, the deformation due to permanent loads occurs after fittings are installed. (Unsupported means that the deformation due to permanent loads occurs before fittings are installed.)

  b) *Fittings with ductile behaviour:*
  The check can only be interpreted for SLS frequent load combination. The structure can be considered as supported or unsupported. In the case of the supported structure, the deformation due to permanent loads occurs after fittings are installed (unsupported means, that the deformation due to permanent loads occurs before fittings are installed).

  c) *Use and operation:*
  The check can only be interpreted for frequent load combination.

☞ **If the critical load combinations are generated automatically, the SLS check is performed based on the SLS combinations type required by the design code.**

Deflections can be checked according to the *z* or *y* local axis directions of the timber design member. The check in each direction can be activated or deactivated by checking the box in front of the limit values. The deflection limit values shall be set according to the length (*L*) detailed below.



Deflections can be considered in several ways: based on the actual displacements or based on the displacements relative only to the left, only to the right or to both end points of the design member.

Length (L)

Deflections of a design member and deflection limits (e.g., *L*/300) are based on one of the following length values:

Design member length: L is equal to the length of the design member;

Based on connecting members and supports: if the *Assemble design members* parameter in the ULS tab is set to the first option, then this length is equal to the design member's length, and thus this is the same as the length above. If it is set to the second option (the selected members are checked as one design member), then the program looks for supports or connecting members along the design member in the direction of the deflection check that can prevent the deflection of the design member and defines segments between these supported nodes. The SLS check is performed for each segment. For example, if a 15 m long beam has supporting columns at the end points and at one-third of the beam, the program performs SLS checks separately for the first 5 m segment and the remaining 10 m segment of the beam.

Custom length: any value can be defined.

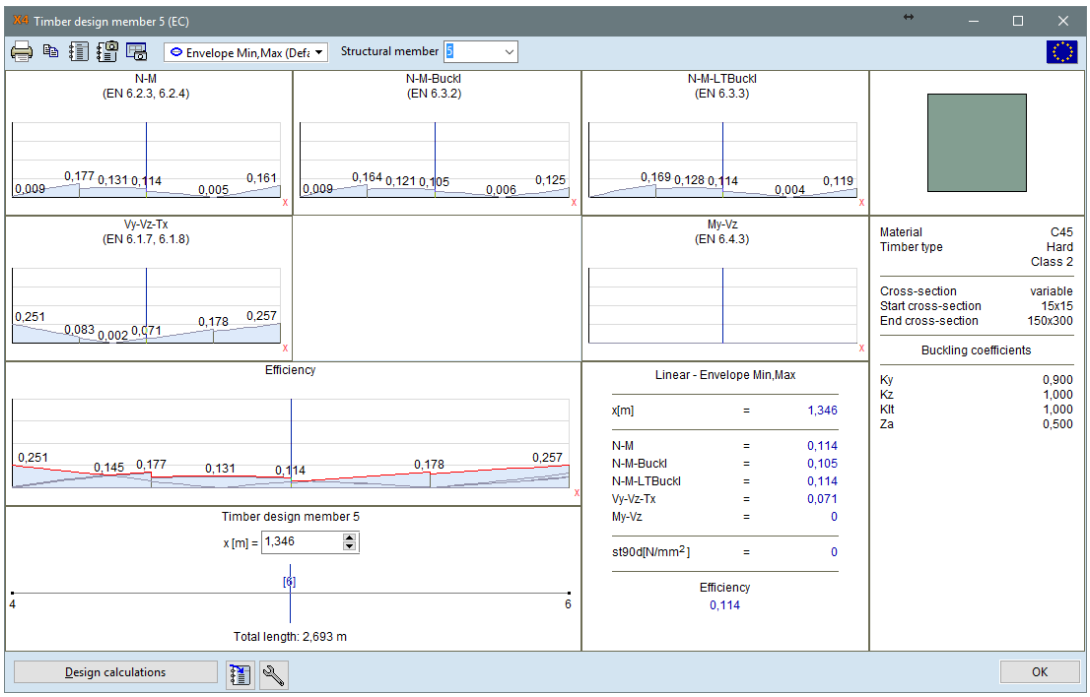
Pre-camber

Pre-camber can be defined for the design members, where the pre-camber values (w_{cz} and w_{cy}) are defined in local coordinates. The shape of the pre-camber can be set with three parameters: linear or second-order curves may be chosen, the maximal value (w) and its location along the member (x_{max}) must also be defined. If deflection is based on actual displacements (e.g., only a section of a beam is selected as a design member), then the pre-camber values are constant along the length. If the length *L* is set to the design member length or to a custom length, pre-camber is zero in the end points of the design members. If the *Based on connecting members and supports* is selected, pre-camber will be zero at the end points and at the supported nodes in the relevant direction.

☞ **It is important to note, that the pre-camber value set here has an effect in the SLS checks only, reducing the previously calculated deflection based on the original shape.**

Result diagrams

By clicking on a design member the program displays the diagrams corresponding to all the checks.



Design calculations

Clicking on the *Design calculations* button a report of the calculation details can be displayed. All strength and stability checks appear as formulas completed with substituted actual values and design code references.

Clicking on the *Settings* icon beside the *Design calculations* button allows setting the basic units for force and length used in the design calculations. Important results also appear converted to standard AXISVM units (see **3.3.8 Units and Formats**). A short summary of design calculations is available, which can be printed on its own or together with the detailed calculations.
For further details see **6.6.1 Steel beam design according to Eurocode 3 – SD1 module**

6.7.2 Timber beam fire design – TD8 module

Standards

AXISVM performs timber fire design according to many national codes. The design codes and national annexes for which this functionality is available are listed in **3.3.7 Design codes**.

Fire design of timber structures according to EN 1995-1-2

Timber beam fire design (TD8 module) is based on methods of general timber beam design (TD1 module; see **6.7.1 Timber beam design – TD1 module**), **thus TD1 module is a prerequisite to use TD8**.
EN 1995-1-2 (EC5-1-2) discusses fire design of timber structures. This section describes the differences between the general design and fire design.
Timber beam fire design can be performed in AXISVM if
1) the selected load combination includes a fire load case (see **4.10.1 Load cases, load groups**)
2) if a fire effect has been assigned to any of the selected elements (see **4.10.33 Fire effect on timber line elements – TD8 module)** in that fire load case.
If different fire effects have been assigned to elements of a design member fire design cannot be performed and an error message appears.
For selected elements without fire effect the general timber design will be performed. Timber fire design parameters also has to be assigned to the elements (see below).

Member analysis

EC5-1-2 discusses different methods: member analysis, analysis of a part of the structure and analysis of the entire structure. Member analysis is the most widespread verification method due to its simplicity. Analysis of a part or whole of the structure usually requires complex nonlinear numerical models and nonlinear statical analysis.

Guiding principles of member analysis according to EC5-1-2:

- Design member internal forces can be calculated with linear static analysis;
- The boundary conditions at supports may be assumed to be constant with time. (EC5-1-2, 2.4.2 (4)).

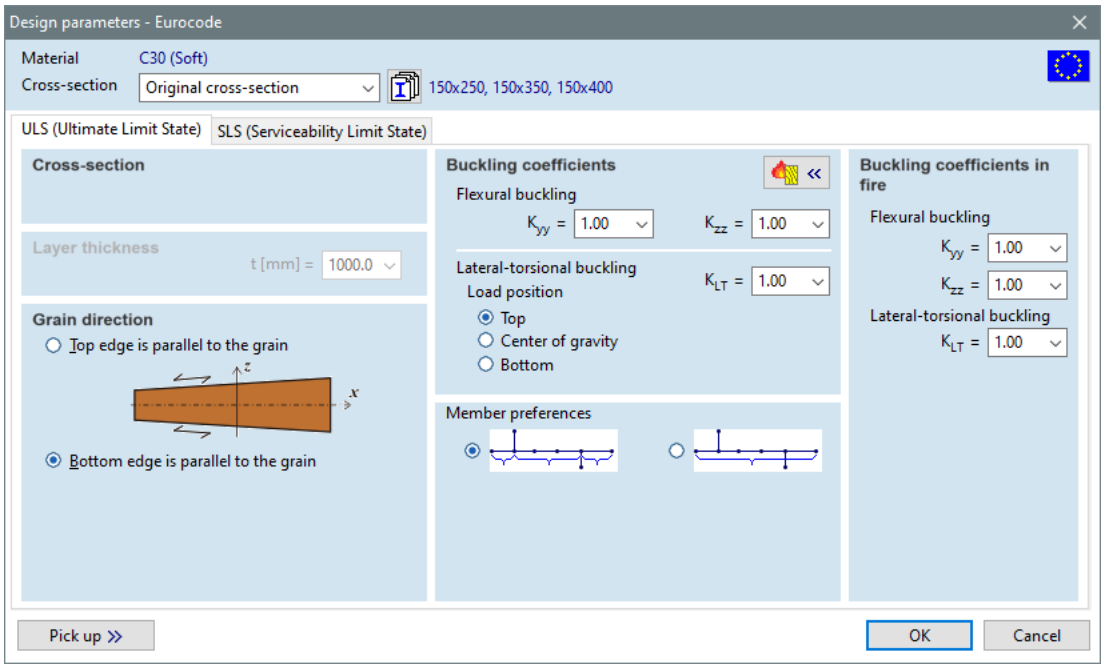
TD8 module performs member analysis. It has to be checked whether the structure/structural part can be verified with member analysis according to the guidelines of EC5-1-2.

Timber fire design parameters



Fire design based on EC5-1-2 requires fire design parameters defined and assigned to structural members. This button is available only if 1) TD8 module is part of the configuration and 2) the current design code supports timber fire design. The button adds an additional panel on the right side. If it remains hidden, fire design of the element will not be performed.





Buckling coefficients in fire

It is possible to define different buckling length coefficients in fire because deflection of the element is often prevented by other elements that may lose their stiffness or resistance in shorter period of time. The definition/calculation of buckling length is the same as in case of normal temperature design.

Reduced cross-section method

EC5-1-2 provides two different methodologies to calculate the mechanical resistance in fire:

- reduced cross-section method,
- reduced properties method.

In the software, the reduced cross-section method is implemented. As a first step, the so-called effective charring depth is calculated on the following way using the notional charring depth ($d_{char,n}$; see **4.10.33 Fire effect on timber line elements – TD8 module**)

$$d_{ef} = d_{char,n} + k_0 d_0$$

where $d_0 = 7mm$, k_0 is a coefficient

The cross-section is reduced with this effective charring depth on the exposed sides. This reduced cross-section is called effective cross-section.

Design checks in fire

The TD8 module, similarly to the TD1 module, performs the following design checks:

Axial force-Bending [N-M]	*(EN 1995-1-1, 6.2.3, 6.2.4)*
Compression-Bending-Buckling (in plane) [N-M-Buckling]	*(EN 1995-1-1, 6.3.2)*
Axial force-Bending-Lateral tors. buckling [N-M-LT buckling]	*(EN 1995-1-1, 6.3.3)*
Shear (y) - Shear(z) - Torsion (x) [Vy- Vz-Tx]	*(EN 1995-1-1, 6.1.7, 6.1.8)*
Moment /y - Shear /z (tensile stress perp. to the grain) [My-V z]	*(EN 1995-1-1, 6.4.3)*

The details of the calculation are presented in the previous section (see **6.7.1 Timber beam design – TD1 module**), thus we note only the differences in this section:

1. different strength needs to be considered,
2. different stability parameters are considered.

Calculation of strength of timber

According to EC5-1-2, the timber strength is calculated on the following way:

$$f_{d,fi} = k_{mod,fi} \frac{f_{20}}{\gamma_{M,fi}}$$

$$f_{20} = k_{fi} f_k$$

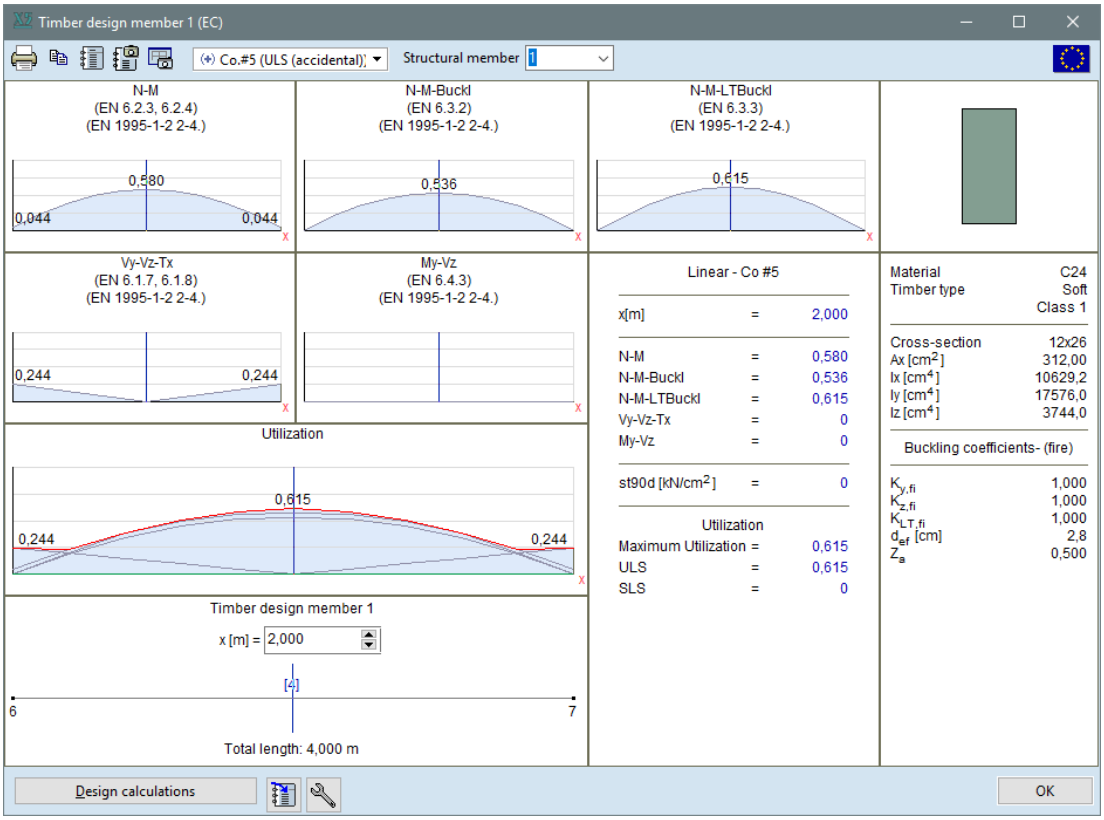
where
f_{20} is the 20% fractile strength at normal temperature
$k_{mod,fi} = 1$ is the modification factor for fire
k_{fi} is a coefficient, $\gamma_{M,fi}$ is a partial factor

Stability checks

Buckling length *($K_{y,fi}$* and *$K_{z,fi}$)* and lateral torsional buckling length *($K_{LT,fi}$)* coefficient given in *Design parameters window* are used to evaluate the flexural buckling and lateral torsional buckling resistances.

Result diagrams

Click on a timber design member to display diagrams for all individual checks and their envelope. Results for any position of any timber design member in any load case or combination can be obtained by setting *x* or dragging the tracking line. If design check cannot be performed with the current cross-section, the diagram is replaced by a cross.
If the selected or the critical combination contains fire load case and fire design was performed for the selected design member, buckling coefficients in fire and charring depth are also shown.



Design calculations

Click on *Design calculations* button to see the design details. The calculation can be saved into the documentation. In case of fire design, the type of fire curve and the calculated effective charring depth are also shown.

2. Bending (y)

EN 1995-1-1: 6.1.6
EN 1995-1-2: 2-4.

Fire curve: **Parametric fire curve**
Required time of fire resistance: **R30**
Effective charring depth: $d_{ef} = d_{char,n} + k_0 \cdot d_0 = 2{,}13 + 1 \cdot 0{,}70 = 2{,}83$ cm
Critical section: $x = 0{,}50 \cdot L = 0{,}50 \cdot 400{,}00 = 200{,}00$ cm

$$\sigma_{m,y,d,fi} = \frac{|M_y|}{W_{y,fi}} = \frac{|(-704{,}00)|}{438{,}15} = 1{,}61 \text{ kN/cm}^2$$

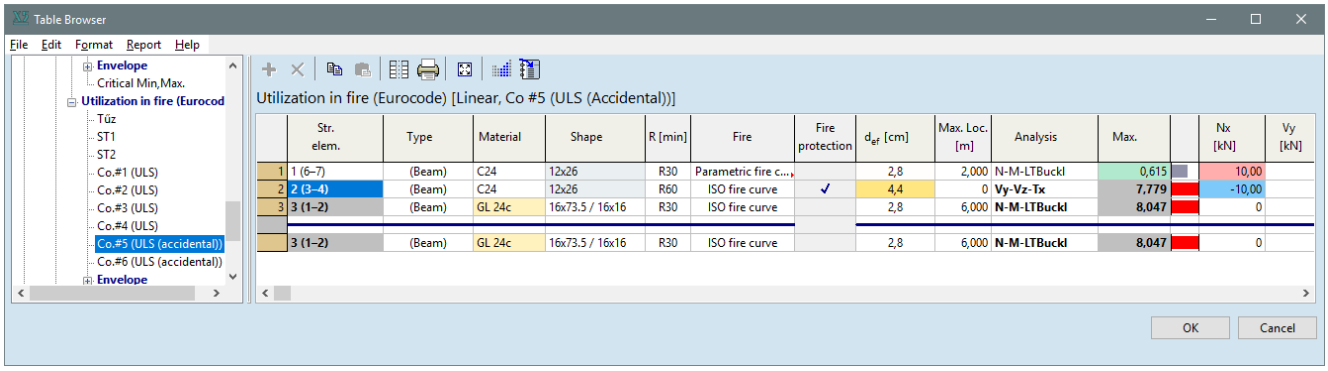
$k_{h,y} = 1$ (3.1)

$$f_{m,y,d,fi} = \frac{k_{mod,fi} \cdot k_{h,y} \cdot f_{m,20}}{\gamma_{M,fi}} = \frac{1 \cdot 1 \cdot 3{,}00}{1} = 3{,}00 \text{ kN/cm}^2$$

$$\eta_{M_y,fi} = \frac{\sigma_{m,y,d,fi}}{f_{m,y,d,fi}} = \frac{1{,}61}{3{,}00} = 53{,}6\ \%$$ **passed**

Result table

Among timber design result tables, *Utilization in fire* summarizes the fire design results in order to help the communication with fire safety engineers.



Cross-section optimization

If TD9 module is present, timber cross-section optimization can be performed for load combinations that contain fire load case with respect to fire design rules.

6.7.3 Timber cross-section optimization – TD9 module



TD9 module requires TD1 module.

Cross-section optimization of timber structures makes timber design members previously defined and designed more efficient by fine-tuning cross-section dimensions and reducing self-weight.

☞ **Optimization checks the design members for the same internal forces ignoring stiffness changes due to changing dimensions. In certain structures recalculation of the model may show considerable changes in internal force patterns. In these cases, several consecutive optimizations may find the more efficient structure.**

Optimization uses the timber design parameters previously assigned to the design members.

Cross-section types suitable for optimization are: rectangle, rounded rectangle and circular shapes. Variable cross-sections cannot be optimized.

For the details of the optimization see **6.6.1.4 Steel cross-section optimization - SD9 module**

6.8 Design of CLT domains – CLT modul

Design codes

There is no currently valid, overall design regulation for CLT panels. For further details on the detailed calculation procedure and the references see the "*CLT Theory and Design Guide*".

Material properties

There is a dedicated CLT material in the material library with predefined stiffness and strength properties. If the user wishes, the material properties can be arbitrarily modified in the material library associated with the model.

Characteristic values of material parameters	*Denotement*
Bending strength	$f_{m,k}$
Tensile strength parallel to the grain direction	$f_{t,0,k}$
Tensile strength perpendicular to the grain direction	$f_{t,90,k}$
Compressive strength parallel to the grain direction	$f_{c,0,k}$
Compressive strength perpendicular to the grain direction (*y*)	$f_{c,90,k}$
Compressive strength perpendicular to the grain direction (*z*)	$f_{c,90,k}$
Torsional strength	$f_{T,k}$
Shear strength in y direction	$f_{v,k}$
Shear strength in z direction	$f_{v,k}$
Rolling shear strength	$f_{r,k}$*

**assumed as 1.0 N/mm², independently of the strength class*

Stiffness values	*Denotement*
Mean value of Young's modulus parallel to the grain direction	$E_{0,mean}$
Mean value of the Young's modulus perpendicular to the grain direction	$E_{90,mean}$
Young's modulus parallel to the grain direction for the significance level of 0.05	$E_{0,05}$
Mean value of the shear modulus	G_{mean}
Mean value of the rolling shear modulus	$G_{R,mean}$*

**it is assumed that $G_{R,mean}/G_{mean} = 0{,}1$ holds, independently of the strength class*

Density	*Denotement*
Apparent density	ρ_k
Mean value of the density	ρ_{mean}

Partial safety factor	*Denotement*
Partial safety factor of the material	γ_M

Service classes

See in chapter **6.7.1 Timber beam design – TD1 module**

Load duration classes

See in chapter **6.7.1 Timber beam design – TD1 module**

Characteristic value of strength

The stress values are to be determined independently due to bending, normal and shear actions.

$\sigma_{mx,t}$ – relevant value of the normal stress in x direction due to bending, on the upper half of the region (the side of te region associated with the positive direction of the local z axis).

$\sigma_{mx,b}$ – relevant value of the normal stress in x direction due to bending, on the lower half of the region (the side of the region associated with the negative direction of the local z axis).

$\sigma_{my,t}$ – relevant value of the normal stress in y direction due to bending, on the upper half of the region (the side of the region associated with the positive direction of the local z axis).

$\sigma_{my,b}$ – relevant value of the normal stress in y direction due to bending, on the lower half of the region (the side of te region associated with the negative direction of the local z axis).

σ_{nx} – relevant value of the normal stress in x direction due to normal forces.

σ_{ny} – relevant value of the normal stress in y direction due to normal forces.

$\tau_{n,xy}$ – relevant value of in plane shear stress from normal force.

$\tau_{m,xy}$ – relevant value of in plane shear stress from twisting moment.

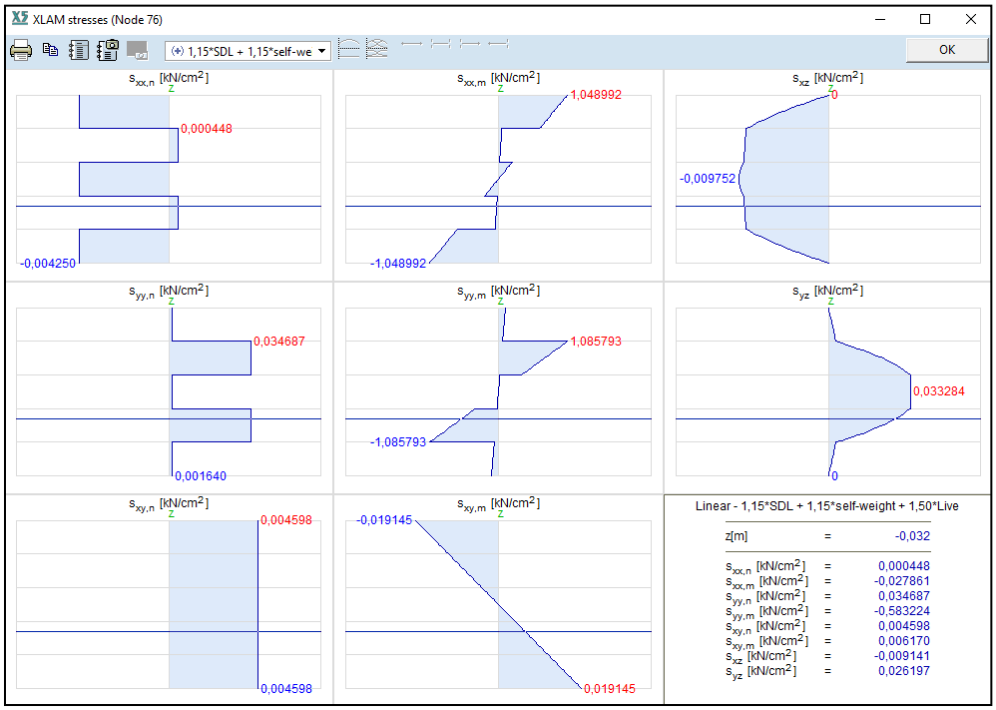
$\tau_{xz,max}$ – relevant shear stress acting in direction z on the plane whose normal is in direction x.

$\tau_{yz,max}$ – relevant shear stress acting in direction z on the plane whose normal is in direction y.

$\tau_{rx,max}$ – relevant rolling shear stress acting in direction z on the plane whose normal is in direction x.

$\tau_{ry,max}$ – relevant rolling shear stress acting in direction z on the plane whose normal is in direction y.

When clicking on a node, one can see the distribution of each calculated stress component through the thickness:



Design value of strength

The design values of strength can be calculated according to the formula:

$f_{c0d}, f_{c90d}, f_{vd}, f_{rd}$: $f_d = k_{mod} f_k / \gamma_M$

$f_{m90d}, f_{t0d}, f_{t90d}$: $f_d = k_{sys} k_{mod} f_k / \gamma_M$

f_{m0d}: $f_d = k_{sys} k_{mod} k_{fin} f_k / \gamma_M$

where,

k_{mod} modification factor for duration of load and moisture content (EN 1995-1-1, 3.1.3)

k_{sys} system strength factor

k_{fin} bending strength reduction factor

f_k charasteristic value of strength

γ_M partial safety factor of the material (EN 1995-1-1, Table 2.3)

k_{sys} factor

$k_{sys} = \min\{1 + 0.025 \cdot n; 1.2\}$ (n denotes the number of layers, but $n > 1$)

Interactions

Normal force-Bending moment in grain direction [M-N-0]
Normal force-Bending moment perpendicular to grain direction [M-N-90]
Shear-Torsion [V-T]
Normal force – Rolling shear [Vr-N]

Normal force – Bending moment in grain direction (M-N-0)

In case of tension

$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + \frac{|\sigma_{m,0,d}|}{f_{m,0,d}} \leq 1$$

In case of compression

$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{|\sigma_{m,0,d}|}{f_{m,0,d}} \leq 1$$

Normal force – Bending moment perpendicular to grain direction (M-N-90)

In case of tension

$$\frac{\sigma_{t,90,d}}{f_{t,90,d}} + \frac{|\sigma_{m,90,d}|}{f_{m,90,d}} \leq 1$$

In case of compression

$$\left(\frac{\sigma_{c,90,d}}{f_{c,90,d}}\right)^2 + \frac{|\sigma_{m,90,d}|}{f_{m,90,d}} \leq 1$$

Shear – Torsion (V-T)

$$\frac{|\tau_{T,d}|}{f_{T,d}} + \left(\frac{\tau_d}{f_{v,d}}\right)^2 + \left(\frac{\tau_{r,d}}{f_{r,d}}\right)^2 \leq 1$$

Rolling shear – Normal force (Vr-N)

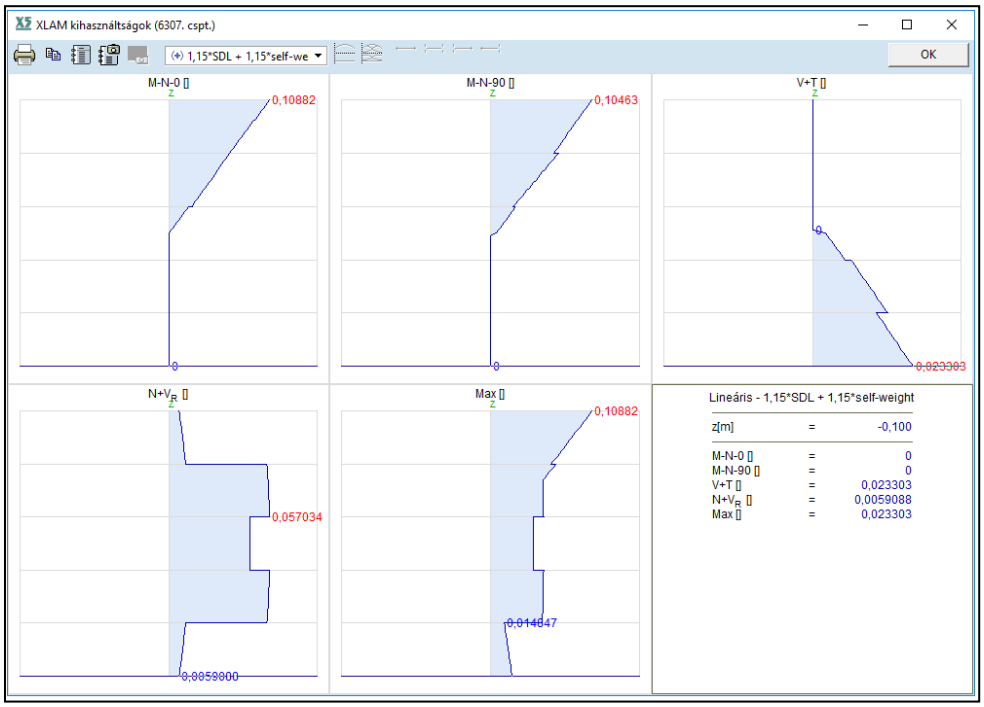
In grain direction

$$\frac{\sigma_{t,90,d}}{f_{t,90,d}} + \frac{|\tau_{r,d}|}{f_{r,d}} \leq 1\,; \quad \frac{|\sigma_{c,90,d}|}{f_{c,90,d}} + \frac{|\tau_{r,d}|}{f_{r,d}} \leq 1$$

Perpendicular to grain direction

$$\frac{\sigma_{t,0,d}}{f_{t,0,d}} + \frac{|\tau_{r,d}|}{f_{r,d}} \leq 1\,; \quad \frac{|\sigma_{c,0,d}|}{f_{c,0,d}} + \frac{|\tau_{r,d}|}{f_{r,d}} \leq 1$$

When clicking on a node, one can see the distribution of the calculated utilization component through the thickness:



6.9 Masonry wall design – MD1 module



MD1 module allows performing the typical analysis of unreinforced masonry walls and to specify their design parameters required for the design check. An overall and complex verification process is implemented for masonry walls subjected to mainly vertical and in-plane shear loading.
It provides a solution for the global analysis of one or multi-storey masonry walls with connecting joints. Joints usually has considerable influence on the load-bearing capacity of the walls.
The verification is available for Eurocode, NTC and SIA standards. The design codes and national annexes in the software for which this functionality is available are listed in **3.3.7 Design codes**.

6.9.1 Complex analysis of unreinforced masonry walls

General principles

The following types of analysis are available in the module:

A) *Stability analysis of walls subjected to mainly vertical loads.* This check takes into account only the eccentricities perpendicular to the plane of the wall, irrespective of the width of the wall strip. Two-directional eccentricity cannot be analysed.
B) *Complex stability check* handling two-directional normal force eccentricity coupled with in-plane shear check.

The A) and B) checks use different user-defined design sections (see **6.9.1.3**). Check type *B* is only performed if shear check is requested in the Masonry wall strip parameters dialog (see **6.9.1.2 Parameters of a masonry wall strip**)

Only unreinforced masonry walls can be checked assuming that the wall has constant wall thickness along a storey, only single leaf wall can be specified.

The masonry walls can be checked according to *EN 1996-1-1 (chapter 6.1)*, *NTC (chapter 4.5.6.2)* and *SIA 266 (chapter 4.3.1 and 4.4.)* considering the following assumptions and application limits.

The above design codes assume that the walls are horizontally supported by the slabs, so standalone or cantilevered walls cannot be analysed.

Various parameters can be added to the model (see *Joint settings*) which can result additional bending moments on the structural model (multi-storey walls with various wall thickness, joint settings, the effect of eccentric slab reaction force).

These effects are not negligible in the calculation of the load-bearing capacity of the elements, but accurate modelling may require significantly complicated models in some cases. To facilitate the design check, two approximative models are provided (see **Calculation of additional bending moments**). In all cases, the user should decide if these solutions are applicable.

The MD1 module works with virtual strips (see **2.16.19 Virtual beams**). Design parameters are assigned to virtual strips and the check can only be performed on the integrated results of the strips.

Virtual strips can include multiple storeys, so the whole wall segment can be analysed as a complete structural system, but storeys can have different design parameters.

Chapters **6.9.1.7…6.9.1.10** serves information about standard specific limitations.

☞ **If the model contains construction stages, the wall segment can only be analysed in the last construction phase or in the final state. In those phases, the virtual strip representing the wall segment must be continuous.**

6.9.1.1 Definition of masonry walls and their parameters

Design parameters



Click on the *Design parameters* icon, to specify the storeys of the masonry wall strip and check the wall in the *Masonry walls* dialog.

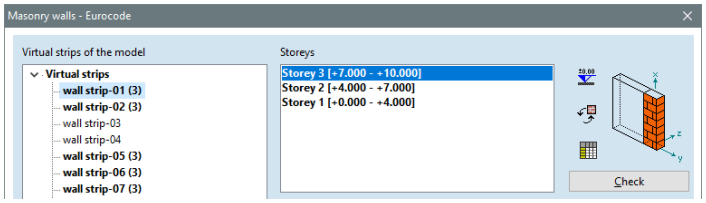
Click on the icon or on the axis of a virtual strip to assign design parameters (see **2.16.19 Virtual beams**). The icon is enabled only if there is at least one virtual strip in the model with static or dynamic results.

First the virtual strip must be selected (see **2.16.1 Selection**). Only one virtual strip can be selected, its domains cannot have materials other than masonry, the strip must be continuous, the domains cannot be tapered and must be vertical with a maximum deviation of 15°. If any of these criteria is not met a warning message appears otherwise the *Masonry walls* dialog is displayed and the selected virtual strip will be selected from the list of masonry virtual strips.

☞ ***Masonry walls* dialog opens automatically when clicking on a (valid) virtual strip in the model.**

Virtual strips

The *Masonry walls* dialog shows a list of *Virtual strips of the model*. These are the existing masonry virtual strips. Selecting a virtual strip displays the list of storeys assigned to the selected strip *(Storeys)*. Selecting a storey displays the design parameters assigned to the selected storey.
As design parameters are assigned to storeys the first step is define storeys for the selected strip.
If a virtual strip already contains storey(s), then its title is shown in bold. The number in the parentheses indicates the number of storeys assigned to the virtual strip.

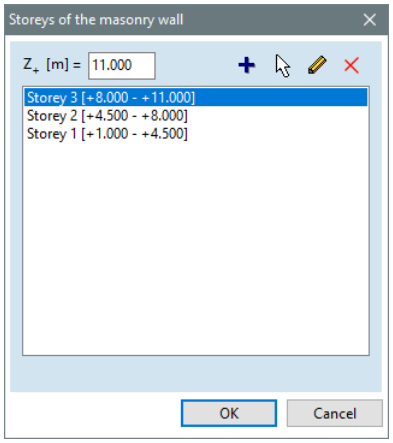


Storeys

The first step is to assign storeys to the strips. Design checks are performed on storeys. The set levels are identified as the levels of the slabs which support the wall laterally.



Click on the *Edit storeys* icon to define storeys for the virtual strip.



Each storey must intersect the axis of the selected virtual beam/strip. If the model has storeys (see **3.3.4 Storeys**) they are imported by default but storeys assigned to the masonry virtual strips do not need to be identical with the storeys of the model.

The storeys can be deleted (delete icon), modified (pencil) and new storeys can be added (+ and arrow). The storeys are saved to the virtual strip. If storeys are assigned to the virtual strip, the cross-section and vertical section (see **6.9.1.5 Display of masonry walls in the model**) of the wall will be displayed in the main window as well. Storey boundaries will be displayed as circles on the axis of the virtual strip.

Automatic design parameters

After setting the storeys, the following parameters are taken from the model based on the properties of the domains associated with the virtual strip.

- *The masonry material of each storey.*
  If the virtual strip passes through different masonry (wall) domains, only one of the materials is assigned to the strip (only one type of material can be considered on each storey). The assigned material can be overridden among the *Design parameters*.
  The moduli of elasticity (applied for the design check) are identified examining the local coordinate system of the domain. If neither axes of the local coordinate system coincides with the vertical direction, then the axis closest to the vertical direction is used to determine the parameter. The assigned moduli of elasticity can be overridden in the *Design parameters* window (see **6.9.1.3**).

- *The wall thickness of each storey.*
  If the virtual strip passes through different masonry (wall) domains, only one of the thickness parameters is assigned to the strip (only constant wall thickness can be considered in the verification). The thickness of the wall cannot be modified later, the property of the domain concerned has to be changed if necessary.
- *The width of the wall,* which is the same as the width of the virtual strip.
  The load-bearing capacity is calculated by the consideration of the total width.
- *The type of the (concrete) slab connected to the wall strips and its thickness.*
  The type of the slab can be *Continuous* and *One-side slab*. If no connecting concrete slab has been found the value *Slab is not defined* appears (see 6.9.1.3).

  Remark: the available design codes assume that the walls are supported laterally by the slabs. So, the value *'Slab is not defined'* only indicates that no supported slab has been found in the model.

  If more than one domain is connected to the virtual strip, the thickness of the thinnest slab is applied.
  The type of the slab and its thickness can be modified later independently of the model in the *Masonry joint* window.
- *Design cross-sections* (4 or 6 different positions, see later) based on the connecting structures.

☞ **If properties of slabs connecting to the storeys are overridden changes in the domain parameters no longer update the masonry parameters. These shall be updated by the user by deleting the masonry parameters and redefining it again, or the parameters has to be changed manually.**

Restore automatic design parameters



Some or all of the automatic design parameters of the selected storey of the actual virtual strip can be restored with this button. These are

- Masonry material (and its parameters),
- Wall thickness,
- Masonry joints (top and bottom joint, with all the reference cross-sections),
- Design parameters (all the data displayed in the *Design parameters* dialog except the material).

For details, see 6.9.1.3.

Table of global design parameters



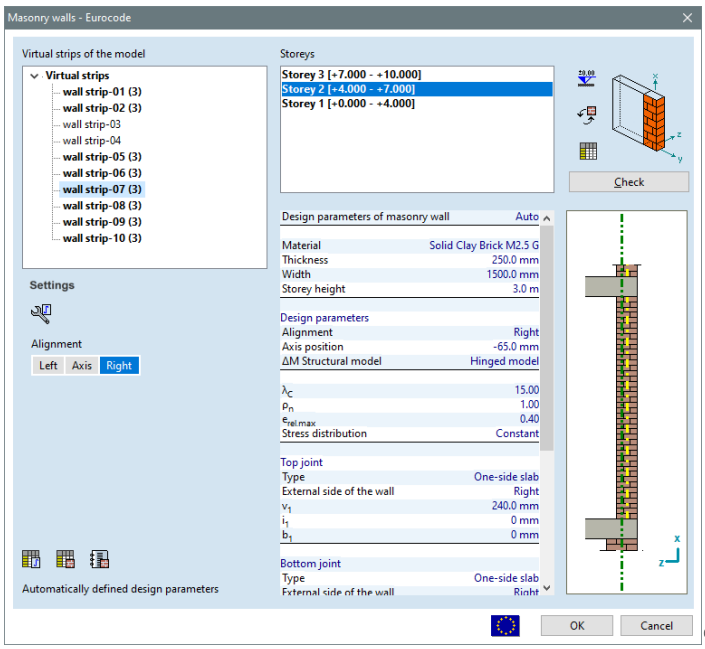
Clicking on the *Table of global design parameters* icon, the *design parameters* of the *virtual strips* can be seen summarized in a table.

For details, see 6.9.1.6 Results.

Check

Click on the *Check* button to perform design check of the storeys.

For details, see 6.9.1.4 Design check of masonry walls, definition of design- and joint parameters



☞ **The name of a storey can be changed by right-clicking on the name and choosing *Rename*.**

Other design parameters

Other design parameters are set according to the design code recommendations or the default setting predefined in AXISVM.

Design parameters of masonry wall

If design parameters have not been changed, *Auto* is displayed in the first line of the list, otherwise *Custom* is displayed.

Side view of a storey

The *Masonry walls* dialog shows the side of view of the selected masonry wall strip with the connecting slabs.

Coordinate system



The side view of the wall and the internal forces of the strip are displayed in the local coordinate system of the virtual strip. The same system is used in the *Masonry walls* and the *Design check of masonry walls* dialogs.

Closing the *Masonry walls* dialog, the main window displays the vertical section of the wall with its joints (see **6.9.1.5**).

Settings

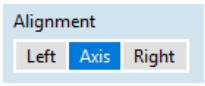
The following settings are assigned to the entire virtual strip:

Settings of global design parameters



The global settings of the masonry wall strips are available by clicking on the *Settings of global design parameters* icon.

Alignment



If the selected virtual strip contains storeys with different wall thickness values, the user can adjust the reference plane of the walls. By default, the program matches the mid-planes (*Axis*) of the walls (like in case of internal walls).
In case of facade walls or in other situations, the walls can be arranged to the *Left* or *Right* (based on the local coordinate system of the virtual strip), matching the left or right sides of the wall. In these cases, the effect of an additional bending moment has to be taken into account (see next section).
The side view displays the axis of the virtual strip and the axis of the wall with the largest wall thickness as a dashed green line. The dashed yellow line shows the axis of the wall on the selected storey.
If all walls in the strip have the same thickness the *Alignment* option is disabled.

Design parameters of masonry wall



Clicking on the *Design parameters of masonry walls* icon, the *design parameters* of the *storeys* can be seen summarized in a table. The icon is only active if the model already contains at least one (masonry) virtual strip that has storey(s).

For details, see **6.9.1.6 Results.**

Design calculations



Clicking on the *Design calculations* icon, the software generates detailed *report* of the results based on the active virtual strip and the load combination/load case selected in the model. The icon is only active if the model already contains at least one (masonry) virtual strip that has storey(s).

A short summary of design calculations is available, which can be printed on its own or together with the detailed calculations.

For details, see **6.9.1.6 Results.**

Design results



Clicking on the *Design results* icon, the *design results* of the *virtual strips* can be seen summarized in a table. The icon is only active if the model already contains at least one (masonry) virtual strip that has storey(s).

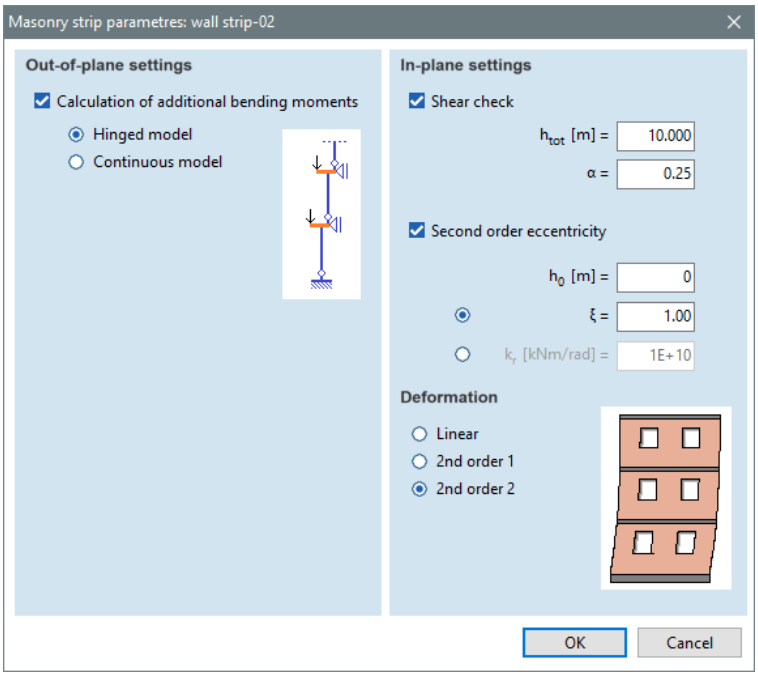
For details, see **6.9.1.6 Results.**

6.9.1.2 Parameters of a masonry wall strip

Global parameters of a masonry wall strip



Click on *Settings of global design parameters* icon to edit the parameters assigned to the entire masonry strip.

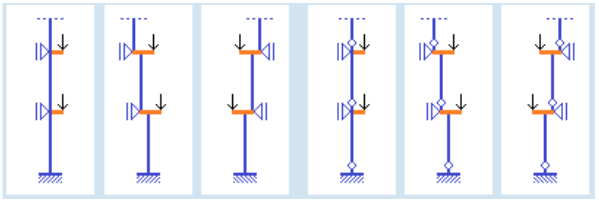


Out-of-plane settings

Calculation of additional bending moments

Settings of the masonry wall strip (alignment of walls, masonry joint settings, stress distribution and the slab reaction force) may introduce additional eccentricities so bending moments has to be taken into account. If the model does not include these effects the masonry module offers two different models to approximate and calculate them: *Hinged model* or *Continuous model*. For details see **6.9.1.7 Principles of checking.** The design bending moment is the sum of the moment of the virtual strip and the results of the above models.

The diagram next to the selected model symbolizes the eccentric forces of the joints and the various wall thicknesses. The figure also depends on the reference plane of the walls (*Left/Axis/Right*) in case of various wall thickness.



If *Calculation of additional bending moments* is unchecked, no additional bending moments are calculated.

In-plane settings

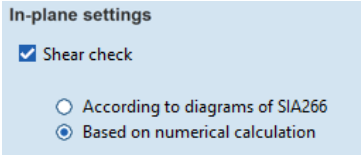
These settings are related to the in-plane checks:

Shear check

Selecting *Shear check* the module performs both type *A* and *B* checks as detailed in chapter **6.8.** If this feature is not activated, only checking type *A* will be performed.

SIA design code

In the case of SIA standard, the following options are available to determine the shear capacity:



The above methods are detailed in chapter **6.9.1.10 Design check according to SIA**.

Initial eccentricity

The program only calculates the initial eccentricity if this option is selected. The initial eccentricity is calculated by the general regulation of *EN 1996-1-1 section 5.3.* supposing that the structure is inclined at a certain angle to the vertical and considering the above parameters.

Second-order eccentricity

Check *Second-order eccentricity* to consider this effect. The condition of necessity is detailed in *section 5.4* in *EN 1996-1-1*. The program only calculates the second-order eccentricity if this option is selected.

Calculating design eccentricities

The initial and second-order eccentricities in the plane of the wall are calculated using the following parameters

h_{tot} Total height of the wall measured from the top of the foundation

α Ratio of the vertical loads stabilized by the wall strip ($0 < \alpha \leq 1$).

Additional parameters for second-order eccentricity:

h_0 Base point

ξ Magnification factor for the rotational stiffness of the restraint of the structural element being considered

k_r rotational stiffness of the restraint

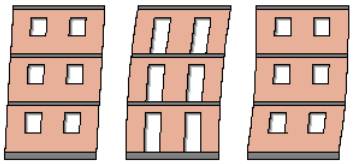
Only one of the previous two parameters can be given.

☞ **In case of seismic type combination, the program does not automatically calculate second-order eccentricity. This can be taken into account using the plastic stability index available in the table of Seismic sensitivity of storeys and following the related regulations of the standard *EN 1998-1-1*.**

Deformation

In the design cross-sections, the second-order eccentricity is approximated supposing one of the following (typical) deformation shape.

- Linear
- 2nd order 1
- 2nd order 2



6.9.1.3 Check domain as a masonry wall

Check domain as a masonry wall



Check domain as a masonry wall is a simplified check of a single storey wall without existing virtual strips. The virtual strip is automatically created during the process. A simple masonry region has to be selected and a reference point for the virtual strip.

The created virtual strip is always 1 m wide, but this value can be changed any time editing the settings of *virtual strips (*see **2.16.19 Virtual beams**). If the entered starting point is near to the edge of the domain, the axis of the virtual strip is automatically shifted to ensure that the entire strip falls within the domain.

Finishing the selection, the *Design check of masonry walls* window opens, where the design parameters and joints can be set and the results can be checked.

The masonry wall strip defined by the function of *Check domain as a masonry wall* can also be managed by the *Design parameters* function like the other virtual strips.

Note: using this function the software assumes that calculation of additional bending moment (see later) is not needed and the shear check is not marked. To change these settings, click on the automatically created virtual strip and make changes through the *Masonry walls* dialog.

6.9.1.4 Design check of masonry walls, definition of design- and joint parameters

Design check of masonry wall

Click on the *Check* button of the *Masonry walls* dialog to display the *Design check of masonry wall* dialog where the design parameters of the wall storeys can be specified and design check is performed.

Set the first dropdown list on the top toolbar to the desired storey of the wall and set the load case or combination.

The left side of the window displays the *Vertical section* or the *Side view* of the selected masonry storey. (If shear check is not active, only the section of the wall can be seen.) Dimensions and joints are shown in detail including the connecting slab, thermal insulation, etc.

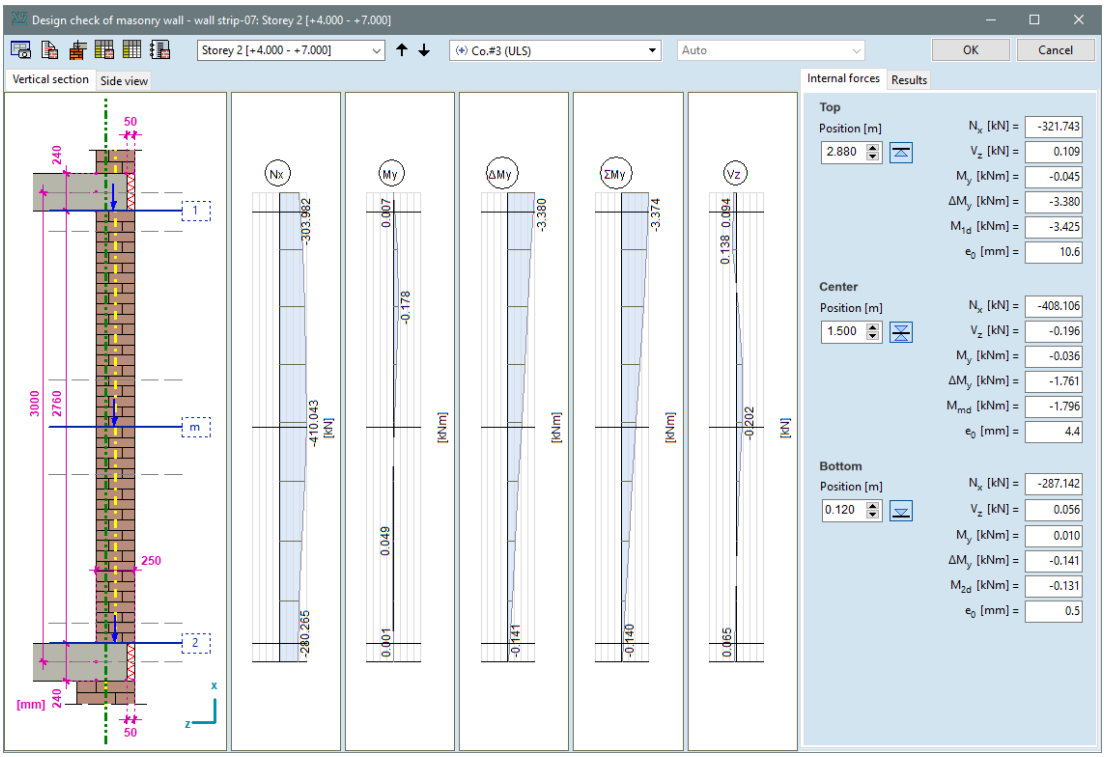
In the *Side view* of the wall, the connecting wall domains are also visible in a pale colour, which is a help to specify the design sections (openings) for shear design.

The coordinate system in the right/left corner (depending on the selected view) shows the coordinate system of the virtual strip.

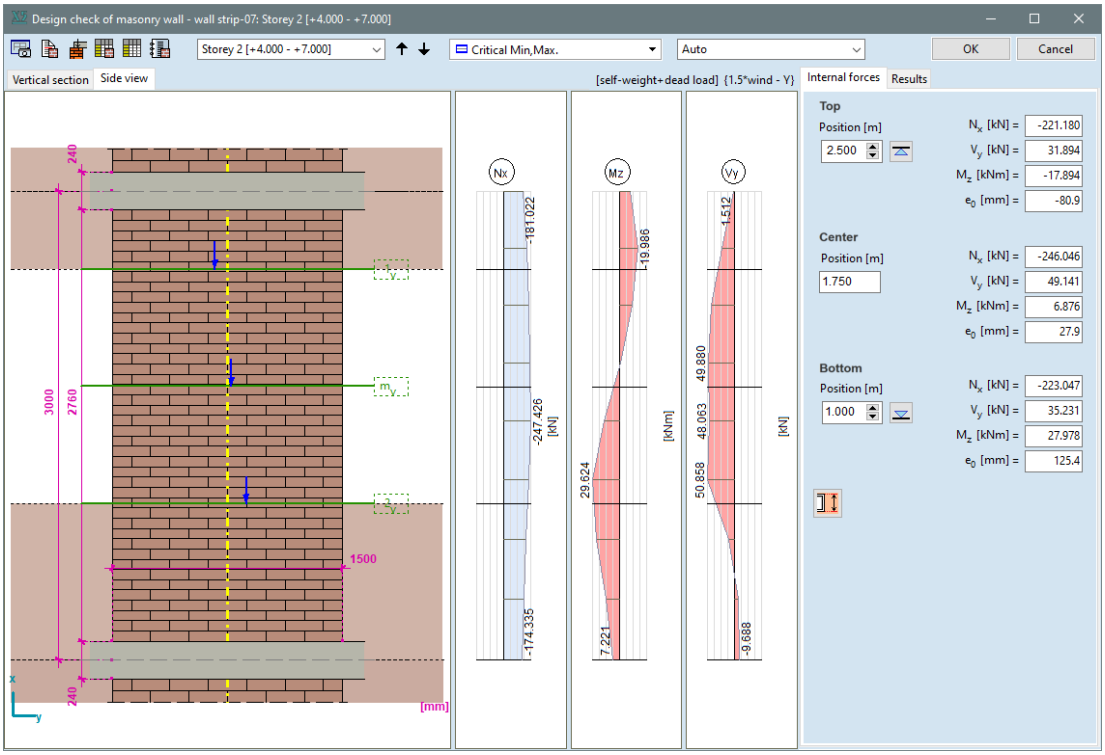
The green axis shows the axis of the virtual strip. If the wall thickness is not the same on all storeys, this axis is identical with the axis of the thickest wall. The yellow axis is the axis of the storey. If wall thickness is not the same on all storeys the yellow axis is not necessarily coincide with the green axis.

The material colour used is the same as the colour of masonry elements. Filling pattern cannot be changed and it does not follow the type of masonry elements.

Vertical section



Side view



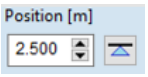
Active storey

☞ **If the mouse is moved over the masonry wall, the cursor changes to a hand symbol. Double-click on the wall to get to the *Design parameters of masonry wall* dialog.**

☞ **If the mouse is moved over the top or bottom joint of the masonry wall, the cursor changes to a hand symbol. Double clicks to open the *Masonry joints* dialog.**

Reference cross-section

Position of reference cross-sections can be entered at the *Top / Center / Bottom*. Internal forces of the virtual strip are read in these positions for design check.



Reference cross-sections

1, m, 2

If shear check is not selected, only 3 reference cross-sections can be set (with labels ***1, m, 2***). They can be specified when the *Vertical section* of the wall is active. These sections belong to the check type *A* (see before).

Thick, blue lines show the positions of reference cross-sections. These lines can be dragged with the mouse between the limits (black dashed lines).

The position of reference cross-sections is automatically adjusted to the edge of concrete slabs at the top and the bottom if concrete slabs are present according to the joint settings. Click on the small button right to the edit field to restore the automatic position.

☞ **In case of SIA 266 standard, the position of the middle cross-section cannot be set.**

Reference cross-sections

1_v, m_v, 2_v

(position of the openings)

If shear check is activated three more reference cross-sections can be set in the *Side view of* the wall. These are the positions where design check B) are performed (complex stability and shear check). The effects of openings and connecting domains can be considered with these positions (shear design is performed on the wall section between the lower and upper sections). These positions are independent of sections *1*, *m*, *2* and are shown in green with signs ***1_v, m_v, 2_v.*** Only the top and bottom positions can be set, the middle reference line is always halfway between them.

Setting the position of openings

Default positions of sections *1_v*, *m_v*, *2_v* are determined based on the connecting domains. These can be modified by dragging the reference lines or they can be set directly in the model after clicking on the *Position of opening* icon. The icon is only available on the *Internal forces* panel (if *Side view* is selected).



Further information to the reference cross-sections

Using the three edit boxes on *Internal forces* tab, the positions can be given by exact values. These values are accepted only if they are between the limits (dashed lines).

The position of arrows above the thick blue lines represents the eccentricity of the compression force in that specific cross-section. The colour of arrows is blue but if the resistance of the cross-section is not adequate according to the design check the colour changes to red. If the resistance cannot be calculated due to an error, the arrow is not shown.

☞ **If positions of reference cross-sections change both considered internal forces and design results will change.**

Internal forces

In the *Design check of masonry wall* window, the following internal force diagrams are shown:

Vertical section

- N_x Axial force (design axial force)
- M_y Bending moment about local *y* axis
- DM_y Additional bending moment due to eccentric effects (calculated only *if Calculation of additional bending moments* option is checked on *Masonry walls* window)
- SM_y Design bending moment (the sum of bending moment of the virtual strip and additional bending moment)
- V_z Shear force of the virtual strip perpendicular to the wall plane

Side view

- N_x Axial force (design axial force)
- M_z Bending moment about local *z* axis
- V_y Shear force of the virtual strip within the wall plane

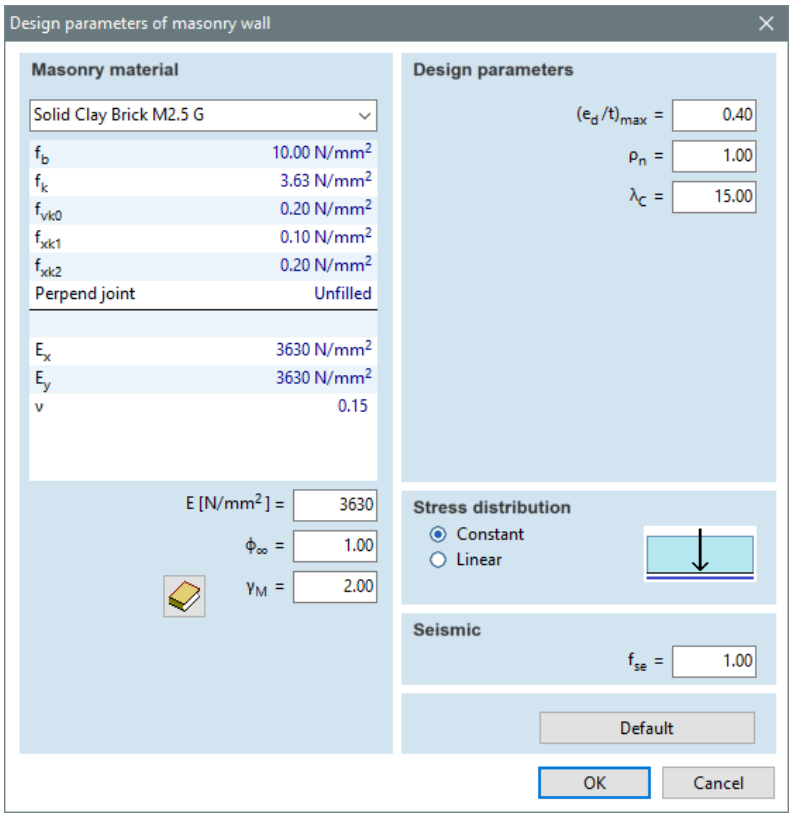
Thin black lines indicate the position of reference cross-sections on the internal force diagrams.

Results panel

The design results are summarized in a table on the *Results* tab. For further information see **6.9.1.6 Results**.

Design parameters of masonry wall

Design parameters of the selected storey can be specified in the *Design parameters of masonry wall* dialog:



Masonry material

After the definition of storeys, the module automatically sets the material of the masonry wall based on the material of the involved domains. This default material can be changed here.

Editable material parameters

The following parameters can be edited. (Changes in this window do not change the parameters of the domain material. These values are used only in the design check of this particular storey of the selected masonry wall.):

Eurocode

- *E* Young's modulus
- φ_∞ Final value of creep coefficient
- γ_M Partial safety factor of material strength

NTC

- γ_M Partial safety factor of material strength

SIA 266

- E_{xk} Young's modulus
- γ_M Partial safety factor of material strength

☞ **As it is explained in the previous section, the masonry material is selected automatically after the definition of storeys at the first time. In case of orthotropic materials, the orientation of local axes needs to be revised.**

☞ **Any changes in this window related to the material of the masonry wall does not imply the change of the material of the domains connected to the virtual strip. The material of the domains needs to be changed by the user if necessary.**

Design code specific parameters

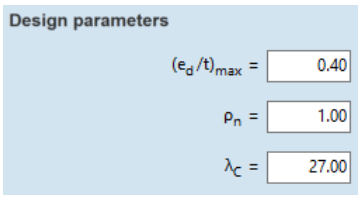
Eurocode 6

- $(e_d/t)_{max}$ – the limit value of the relative (out-of-plane or in-plane) eccentricity of axial force:

If the relative eccentricity is higher than this limit value in any cross-section, then the design check fails and the resistance of the cross-section is considered inadequate.

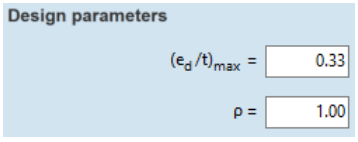
In case of top and bottom sections, the relative eccentricity is calculated based on the effective thickness of the wall and relative to its axis.

- ρ_n reduction factor considering buckling, default value is 1.0,
- λ_C value of the slenderness ratio up to which eccentricities due to creep can be neglected.



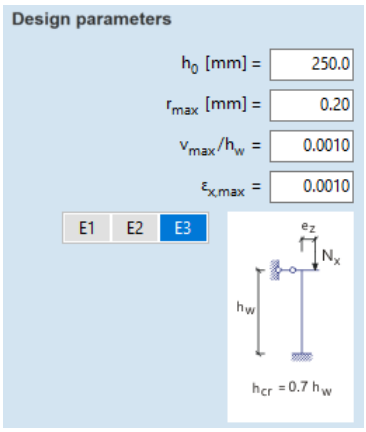
NTC

- $(e_d/t)_{max}$ – the limit value of relative eccentricity,
- ρ reduction factor considering buckling, default value is 1.0.



SIA 266

- h_0 is the sum of the height of masonry element and the height of mortar layer,
- r_{max} is the maximum acceptable crack width,
- v_{max}/h_w is the allowed relative displacement considering reference sections 1_v and 2_v (where h_w is the distance between the two reference sections). For maximum values see *Table 8* in the standard.
- $\varepsilon_{x,max}$ maximum permissible value according to *Table 8* in the standard,
- boundary conditions *(E1 / E2 / E3)*.



Stress distribution

In case of one-side slabs, constant and linear stress distribution can be considered under the slab reaction force. The stress distribution set has a significant effect on the calculated additional bending moment.



Additional bending moment is calculated as the product of slab reaction force and eccentricity based on the position of resultant of stress distribution. The slab reaction calculated from the difference between the axial forces of subsequent storeys.

☞ **Stress distribution can be selected only if *Calculation of additional bending moments* option is checked on *Masonry walls* window.**

☞ **In case of the topmost storey, the total axial force at the top is considered to be slab reaction force. For multi-storey structures, it is highly recommended to use virtual strips that span all of the storeys if the wall is continuous, otherwise, the software cannot calculate the additional bending moments properly.**

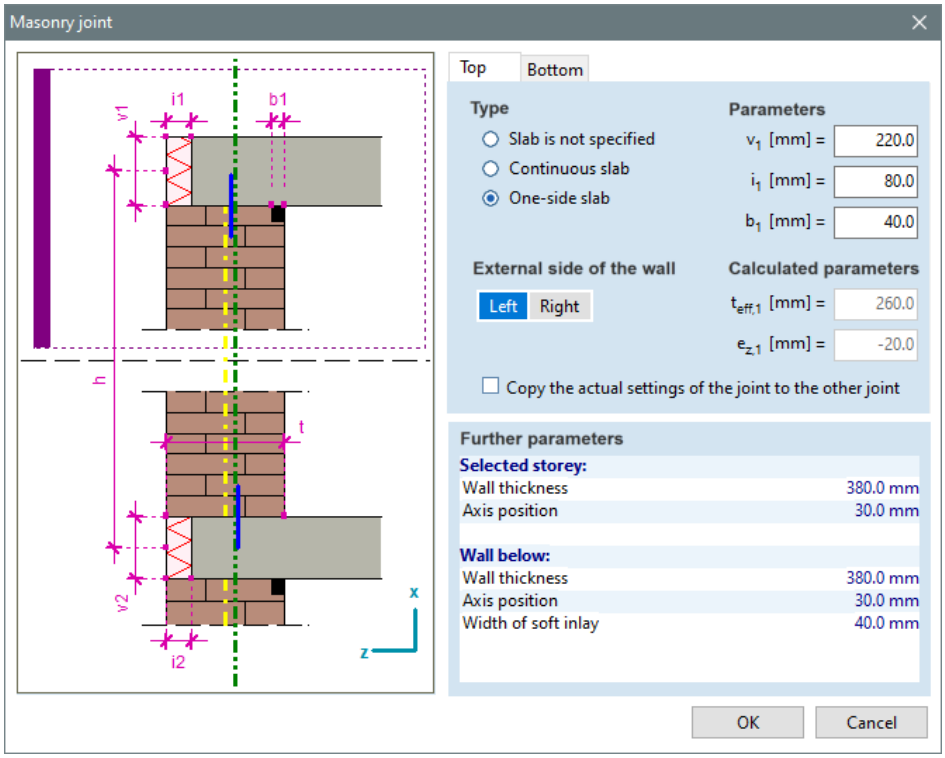
☞ **If *SIA 266* design code is selected, only constant stress distribution can be considered.**

Seismic

For further details about the coefficient of seismic forces see **4.10.25 Seismic loads – SE1 module.**

Joint settings

The parameters of joints can be specified in *Masonry joint* window:



The two drawings show the top and bottom masonry joints. Click on the top or bottom one to change the active joint. The dashed purple frame shows which joint is active. Parameters of the active joint can be edited on the right.

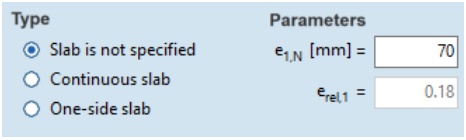
☞ **If the mouse is moved over the top or bottom joint of masonry wall in the *Design check of masonry wall* window, the cursor changes to a hand symbol. Double click to open the *Masonry joints* dialog.**

Type of the slab

Possible cases are: *Slab is not specified, Continuous slab, One-side slab.* Further parameters depend on the selected case. The lower indices indicate the joint position (*1* = top; *2* = bottom).

Slab is not specified

Slab is not specified option can be used if the user does not want to define the parameters of the slab.



Optional parameters:

$e_{1,N}$ is the eccentricity of the slab reaction (it is originated from the 'jump' in the axial forces or the difference between the axial forces of subsequent storeys.). (This parameter is only available for top joints). The sign of the eccentricity is selected according to the coordinate system of the virtual strip.

$e_{rel,1}$ is the relative eccentricity based on the wall thickness (calculated).

☞ **The calculated relative eccentricity is revised relative to the limit set in *Design parameters* without the consideration of the (other) internal forces that reduces the eccentricity of the slab reaction.**

In the verification, at the top section, the effective wall thickness is identical with the wall thickness of the storey. This rule is also valid for the bottom section, but if the thickness of the wall below the investigated storey is lower than the wall thickness of the actual storey, then only the lower wall thickness is taken into account in the calculation.

Continuous slab

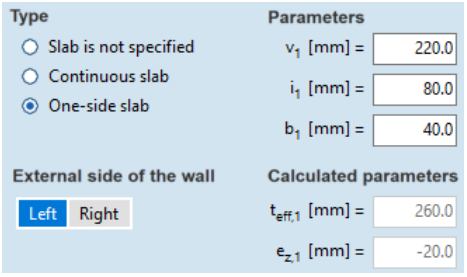
In case of *Continuous slab*, it is assumed that the slab reaction loads are centric. The storey height is calculated with the consideration of slab thickness.



Optional parameters:

- *v* is the thickness of the slab.

One-side slab



The eccentricity of slab reaction is calculated with the consideration of joint parameters (e.g., thickness of thermal insulation) and the type of stress distribution as described above. The storey height is calculated with the consideration of slab thickness.

Optional parameters:

- *v* is thickness of the slab,
- *i* is thickness of thermal insulation,
- *b* is the width of soft inlay (only for top joint)
- *External side of the wall (Left/Right),* in agreement with the local coordinate system of the selected virtual strip,

If *Calculation of additional bending moments* option is checked on *Masonry walls* window, the software shows the defined stress distribution on the drawing.

Calculated parameters

The software calculates and shows the following parameters:

- t_{eff} if the effective thickness (considering the thickness of thermal insulation and the width of soft inlay)
- e_z is the difference between the centre of effective area and the centre of the wall.

Copy the actual settings of the joint

Using *Copy of actual settings of the joint to the other joint* option, the defined parameters can be copied to the opposite joint of the storey.

Further parameters

The *Further parameters* list shows relevant parameters that cannot be read from the drawings.

Toolbar

Save to Drawings Library



Saves the drawing to the Drawing Library.

Design parameters of masonry walls



see **6.9.1.6 Results**

Design results



see **6.9.1.6 Results**

Design calculations



see **6.9.1.6 Results**

A short summary of design calculations is available, which can be printed on its own or together with the detailed calculations.

Switch between storeys



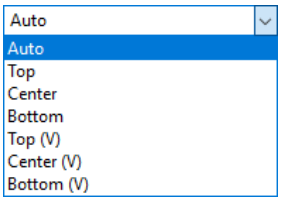
The investigated storey can be selected from the list below. The arrows beside the list can be used to switch one storey up/down.



Load case / combination

In the drop-down list of load combinations, the user can select the load case to be displayed.

Interpreting of the results in case of Critical combinations or Envelope



Selecting *Critical combination* or *Envelope*, the results displayed belong to the load case or combination resulting in the maximum utilization based on the results of all cross-sections. This is the default mode of operation, called *Auto*.
There may be cases when the maximum utilization of each reference cross-section comes from different combination which can cause misunderstanding if the results are not analysed in depth.
To display the maximum utilization of a certain reference section, select a reference cross-section from the drop-down list.

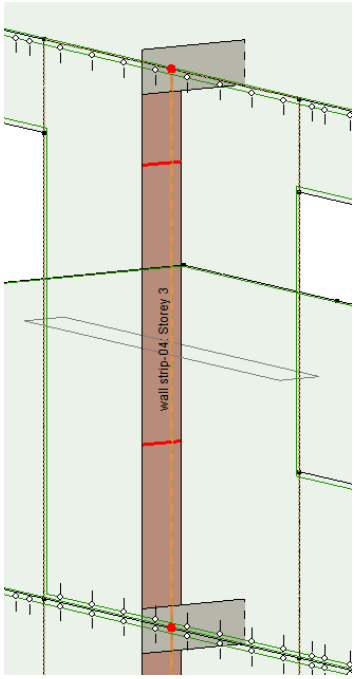
The combination resulting in the maximum utilization is displayed above the internal force diagrams.

☞ **Note: if considering different load combinations, a continuous internal force diagram cannot be plotted and only the results of related load combinations can be compared and interpreted together.**

The overall results of the sections can be compared and checked simultaneously in the *Result table*.

☞ **The design calculation report uses the *Reference position* or *Auto* function selected here. *Auto* function is used if the design calculation report was created from the *Masonry walls* dialog.**

6.9.1.5 Display of masonry walls in the model



Based on the defined or automatically assigned parameters, the software shows the vertical section of the wall. It can help to control the joint parameters and to interpret a coordinate system of the strip.

Red circles on the axis of virtual strip indicate the boundaries of the defined storeys.

Thick red lines on the sections show the position of the upper and lower reference cross-sections set for shear design.

6.9.1.6 Results

Design results of masonry walls are available in many ways.

Control results

The result summary of the investigated storey is listed on the *Results* tab of *Design check of masonry wall* window. If any error occurs during the design check, the module shows the source of the error.

The contents of the Results tab depend on the current design code. In case of *SIA* and *NTC* the design and interpolated curves are also presented.

Without shear check

When shear check is not requested, the result panel appear as follows:

Eurocode

Internal forces | Results

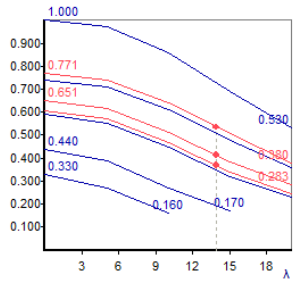
Result summary	
Critical section	Bottom
Utilization	0.479
	passed!
Top	
N_{1d}	-29.39 kN
M_{1d}	0.33 kNm
$e_0 = e_{1d} + e_{he}$	-11.3 mm
Δz	-40.0 mm
$e_{0,1}$	-51.3 mm
e_{init}	±6.2 mm
e_{min}	±11.0 mm
e_1	-57.5 mm
e_1/t	0.261
Φ_1	0.477
N_{Rd1}	192.65 kN
Utilization	0.153
	passed!
Center	
N_{md}	-85.70 kN
M_{md}	2.22 kNm
$e_0 = e_{md} + e_{he}$	-25.9 mm
e_{init}	±6.2 mm
e_m	-32.1 mm
e_k	±0 mm
e_{min}	±15.0 mm
e_{mk}	-32.1 mm
e_{mk}/t	0.107
Φ_m	0.731
N_{Rdm}	402.40 kN
Utilization	0.213
	passed!
Bottom	
N_{2d}	-106.68 kN
M_{2d}	0.33 kNm
$e_0 = e_{2d} + e_{he}$	-3.1 mm
Δz	-40.0 mm
$e_{0,2}$	-43.1 mm
e_{init}	±6.2 mm
e_{min}	±11.0 mm
e_2	-49.3 mm
e_2/t	0.224

NTC

Internal forces | Results

Result summary	
Critical section	Bottom
Utilization	0.195
	passed!
Top	
N_d	-30.079 kN
M_d	2.501 kNm
e_0	-83.2 mm
Δz	60.0 mm
$e_{0,1}$	-23.2 mm
e_a	±17.5 mm
e_{min}	±17.5 mm
e_1	-40.7 mm
e_1/t	0.156
Φ	0.368
N_r	175.790 kN
Utilization	0.171
	passed!
Center	
N_d	-35.380 kN
M_d	0.677 kNm
e_0	-19.1 mm
$e_a/2$	±8.8 mm
e_{min}	±17.5 mm
e_2	-27.9 mm
e_2/t	0.073

Φ Reduction factor

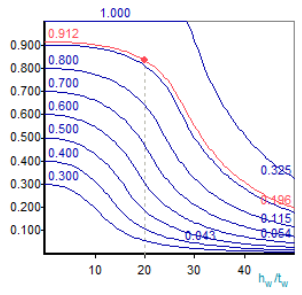


SIA 266

Internal forces | Results

Result summary	
Combination	ULS
Critical section	Top
Utilization	0.547
	passed!
Top	
N_{xd}	-319.173 kN
M_{zd}	-2.805 kNm
e_z	8.8 mm
e_z/t_w	0.044
$Max(e_z/t_w)$	0.272
Stress limitation	
e_0	8.8 mm
e_0/a	0.044
$Max(e_0/a)$	0.375
t_{nom}	182.4 mm
$N_{Rd,nom}$	638.474 kN
Utilization	0.500
Stability check	
h_w/t_w	20.000

Design curves for k_N coefficient

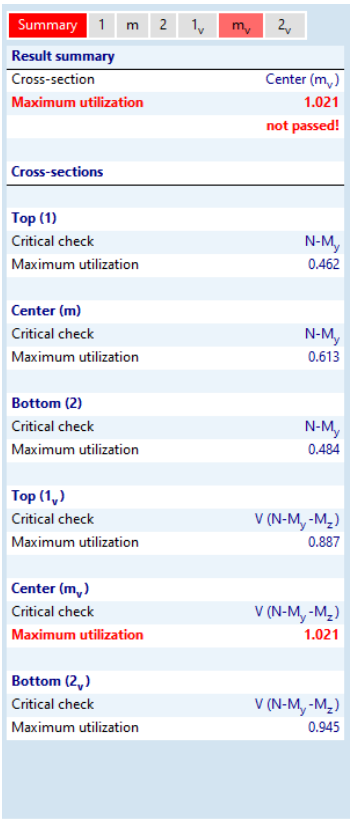


Results in case of shear check

In case of shear check, the user can choose between the result types. Summary or detailed results of different verifications can be displayed.

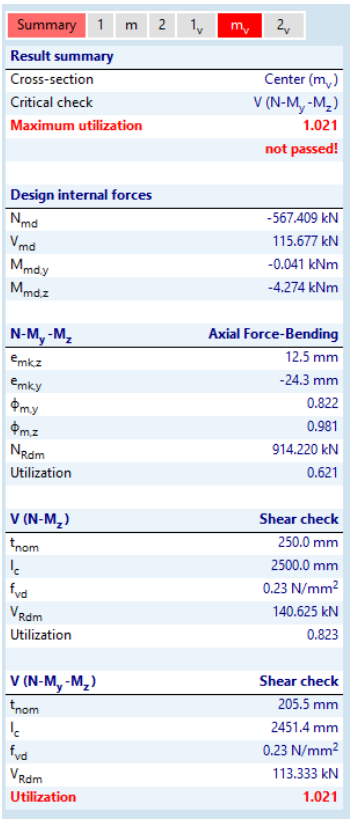
The horizontal symbol list at the top summarize the design results in reference cross-sections *1, m, 2* and 1_v, m_v, 2_v (see **6.9.1.4 Design check of masonry walls, definition of design- and joint parameters**). The background of verified sections is grey, failed sections are red. The selected position is blue if the section is verified otherwise it remains red. The font color of the selected position is always white.

In case of *Eurocode*, the following options are available:



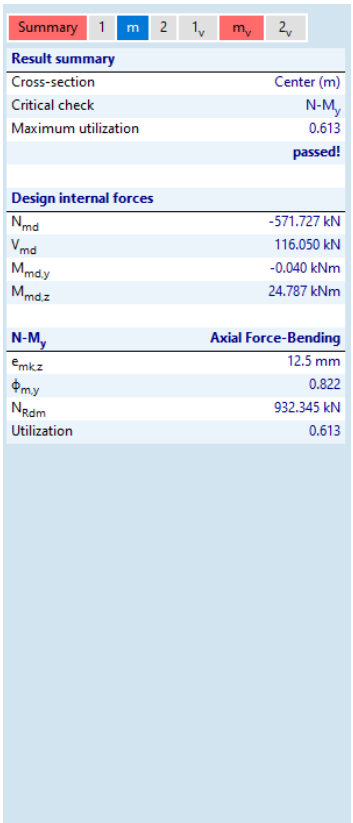
Summary	1	m	2	1_v	m_v	2_v

Result summary	
Cross-section	Center (m_v)
Maximum utilization	**1.021**
	not passed!
Cross-sections	
Top (1)	
Critical check	N-M_y
Maximum utilization	0.462
Center (m)	
Critical check	N-M_y
Maximum utilization	0.613
Bottom (2)	
Critical check	N-M_y
Maximum utilization	0.484
Top (1_v)	
Critical check	V (N-M_y-M_z)
Maximum utilization	0.887
Center (m_v)	
Critical check	V (N-M_y-M_z)
Maximum utilization	**1.021**
Bottom (2_v)	
Critical check	V (N-M_y-M_z)
Maximum utilization	0.945



Summary	1	m	2	1_v	m_v	2_v

Result summary	
Cross-section	Center (m_v)
Critical check	V (N-M_y-M_z)
Maximum utilization	**1.021**
	not passed!
Design internal forces	
N_{md}	-567.409 kN
V_{md}	115.677 kN
$M_{md,y}$	-0.041 kNm
$M_{md,z}$	-4.274 kNm
N-M_y-M_z	**Axial Force-Bending**
$e_{mk,z}$	12.5 mm
$e_{mk,y}$	-24.3 mm
$\phi_{m,y}$	0.822
$\phi_{m,z}$	0.981
N_{Rdm}	914.220 kN
Utilization	0.621
V (N-M_z)	**Shear check**
t_{nom}	250.0 mm
l_c	2500.0 mm
f_{vd}	0.23 N/mm²
V_{Rdm}	140.625 kN
Utilization	0.823
V (N-M_y-M_z)	**Shear check**
t_{nom}	205.5 mm
l_c	2451.4 mm
f_{vd}	0.23 N/mm²
V_{Rdm}	113.333 kN
Utilization	**1.021**

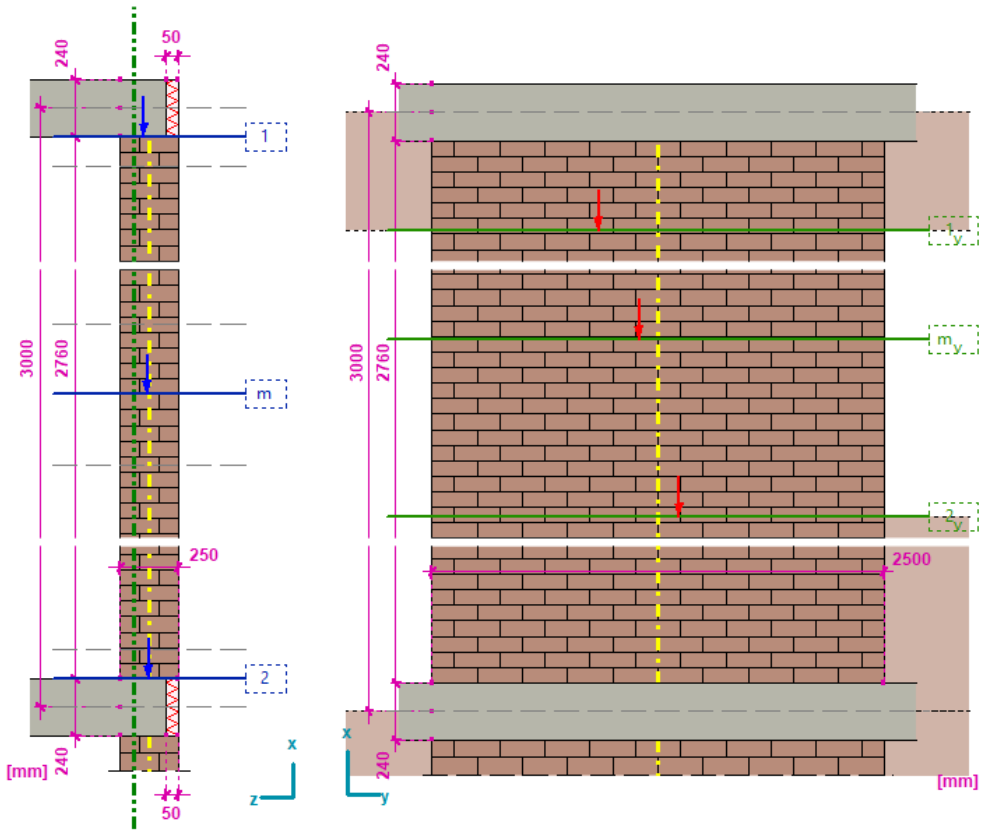


Summary	1	m	2	1_v	m_v	2_v

Result summary	
Cross-section	Center (m)
Critical check	N-M_y
Maximum utilization	0.613
	passed!
Design internal forces	
N_{md}	-571.727 kN
V_{md}	116.050 kN
$M_{md,y}$	-0.040 kNm
$M_{md,z}$	24.787 kNm
N-M_y	**Axial Force-Bending**
$e_{mk,z}$	12.5 mm
$\phi_{m,y}$	0.822
N_{Rdm}	932.345 kN
Utilization	0.613

Results displayed on the Vertical section or on the Side view

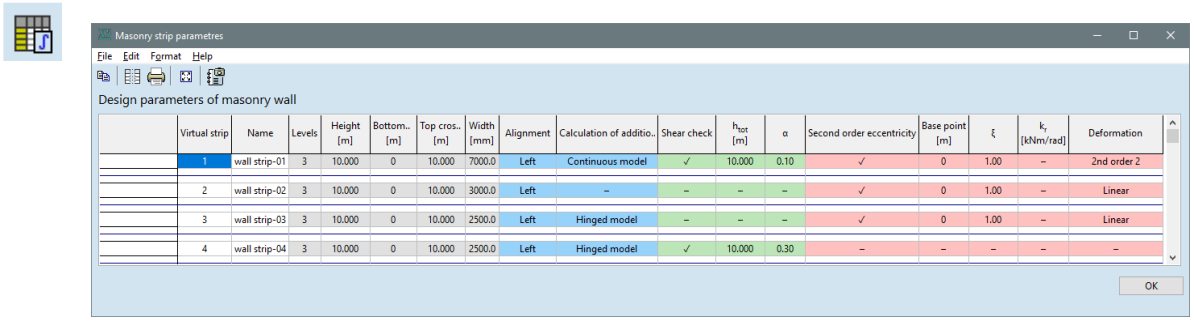
In case of each reference cross-section, on its reference line, the program indicates the point of action of the vertical force based on the internal forces (the eccentricity does not contain the initial eccentricity/ minimum eccentricity).
If the resistance of the wall section is adequate then the colour of the arrow is blue, otherwise, it is red.
If the vertical resistance cannot be calculated then the arrow is not displayed.



Tables

Table of global design parameters

Global design parameters table is available from the *Design check of masonry wall* dialog and displays the design parameters assigned to masonry wall strips.



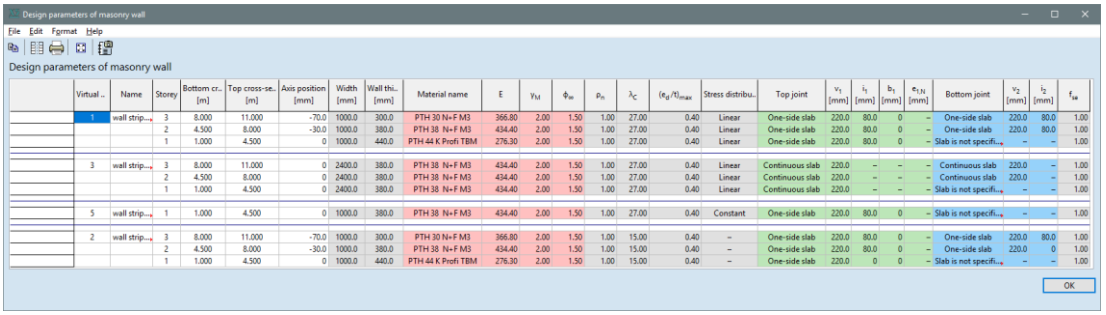
	Virtual strip	Name	Levels	Height [m]	Bottom.. [m]	Top cros.. [m]	Width [mm]	Alignment	Calculation of additio..	Shear check	h_{tot} [m]	α	Second order eccentricity	Base point [m]	ξ	k_r [kNm/rad]	Deformation
	1	wall strip-01	3	10.000	0	10.000	7000.0	Left	Continuous model	✓	10.000	0.10	✓	0	1.00	–	2nd order 2
	2	wall strip-02	3	10.000	0	10.000	3000.0	Left	–	–	–	–	✓	0	1.00	–	Linear
	3	wall strip-03	3	10.000	0	10.000	2500.0	Left	Hinged model	–	–	–	✓	0	1.00	–	Linear
	4	wall strip-04	3	10.000	0	10.000	2500.0	Left	Hinged model	✓	10.000	0.30	–	–	–	–	–

The table can be printed and saved into the documentation.

Design parameters of masonry walls

Design parameters of masonry wall table is available from both the *Design check of masonry wall* and *Masonry walls* dialogs. This table shows the defined or automatically assigned design parameters.

If the table is opened from *Masonry walls* window, it summarizes the parameters of every masonry walls in the model, otherwise only the investigated virtual strip is listed with its storeys.



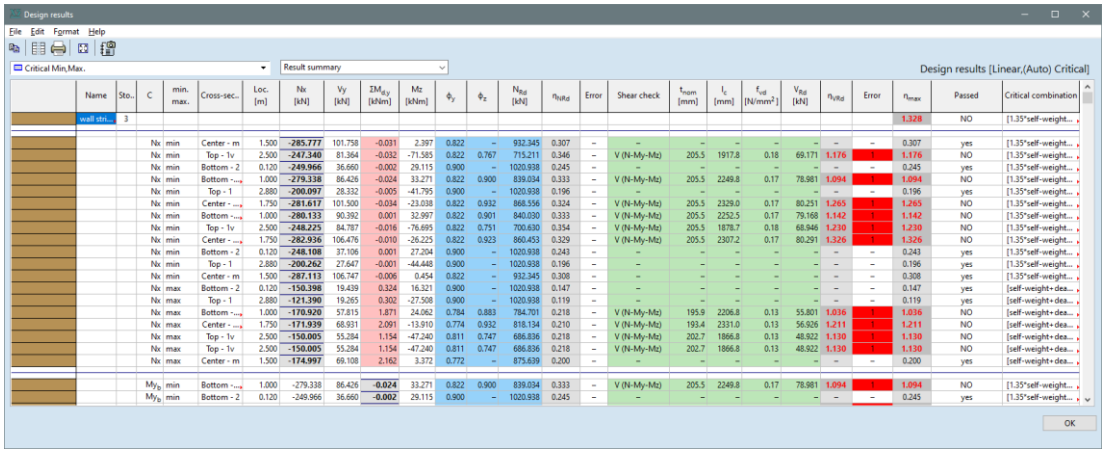
	Virtual ..	Name	Storey	Bottom cr.. [m]	Top cross-se.. [m]	Axis position [mm]	Width [mm]	Wall thi.. [mm]	Material name	E	γ_M	ϕ_∞	ρ_n	λ_C	$(e_d/t)_{max}$	Stress distribu..	Top joint	v_1 [mm]	i_1 [mm]	b_1 [mm]	$e_{1,N}$ [mm]	Bottom joint	v_2 [mm]	i_2 [mm]	f_{se}
	1	wall strip-...	3	8.000	11.000	-70.0	1000.0	300.0	PTH 30 N+F M3	366.80	2.00	1.50	1.00	27.00	0.40	Linear	One-side slab	220.0	80.0	0	–	One-side slab	220.0	80.0	1.00
			2	4.500	8.000	-30.0	1000.0	380.0	PTH 38 N+F M3	434.40	2.00	1.50	1.00	27.00	0.40	Linear	One-side slab	220.0	80.0	0	–	One-side slab	220.0	80.0	1.00
			1	1.000	4.500	0	1000.0	440.0	PTH 44 K Profi TBM	276.30	2.00	1.50	1.00	27.00	0.40	Linear	One-side slab	220.0	80.0	0	–	Slab is not specifi...	–	–	1.00
	3	wall strip-...	3	8.000	11.000	0	2400.0	380.0	PTH 38 N+F M3	434.40	2.00	1.50	1.00	27.00	0.40	Linear	Continuous slab	220.0	–	–	–	Continuous slab	220.0	–	1.00
			2	4.500	8.000	0	2400.0	380.0	PTH 38 N+F M3	434.40	2.00	1.50	1.00	27.00	0.40	Linear	Continuous slab	220.0	–	–	–	Continuous slab	220.0	–	1.00
			1	1.000	4.500	0	2400.0	380.0	PTH 38 N+F M3	434.40	2.00	1.50	1.00	27.00	0.40	Linear	Continuous slab	220.0	–	–	–	Slab is not specifi...	–	–	1.00
	5	wall strip-...	1	1.000	4.500	0	1000.0	380.0	PTH 38 N+F M3	434.40	2.00	1.50	1.00	27.00	0.40	Constant	One-side slab	220.0	80.0	0	–	Slab is not specifi...	–	–	1.00
	2	wall strip-...	3	8.000	11.000	-70.0	1000.0	300.0	PTH 30 N+F M3	366.80	2.00	1.50	1.00	15.00	0.40	–	One-side slab	220.0	80.0	0	–	One-side slab	220.0	80.0	1.00
			2	4.500	8.000	-30.0	1000.0	380.0	PTH 38 N+F M3	434.40	2.00	1.50	1.00	15.00	0.40	–	One-side slab	220.0	80.0	0	–	One-side slab	220.0	0	1.00
			1	1.000	4.500	0	1000.0	440.0	PTH 44 K Profi TBM	276.30	2.00	1.50	1.00	15.00	0.40	–	One-side slab	220.0	0	0	–	Slab is not specifi...	–	–	1.00

The table can be printed and saved into the documentation.

Design results

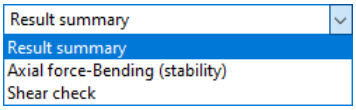


Design results table is available both from the *Design check of masonry wall* and *Masonry walls* dialogs. If the table is opened from *Masonry walls* window, load case/combination, envelope or automatic critical load combinations can be selected from a list-box which is considered in the design check. After modifying selection in this list-box, the window automatically performs the design checks that can take only a few seconds. Without shear check the table appears as follows:



The table can be printed and saved into the documentation.
If shear check is requested, the table is built up differently because of the significant number of checking. Result type can be selected from the drop-down list above the table.

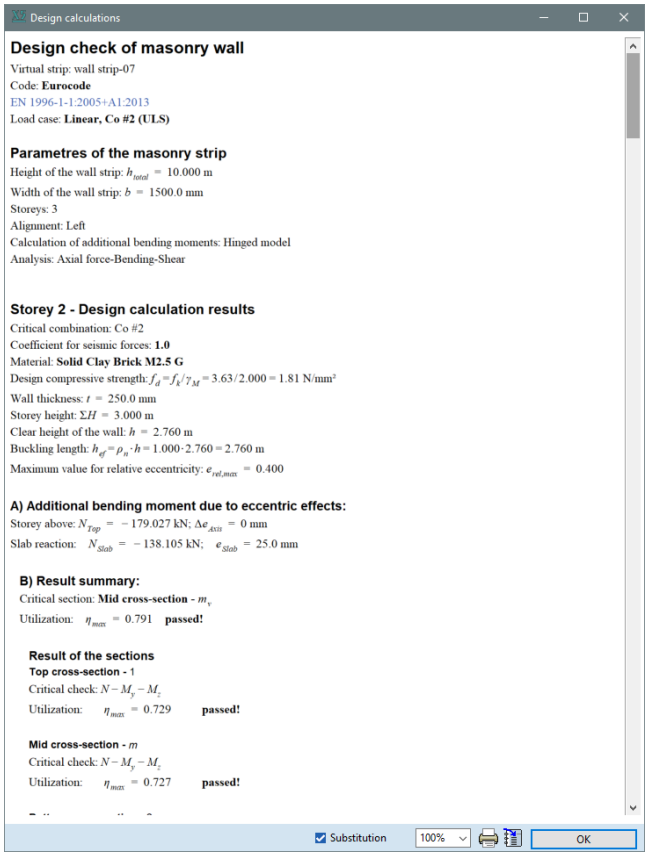
In case of Eurocode the following types of result are available (*Result summary*, *Axial force - Bending (stability)*, *Shear check*).



Design calculations

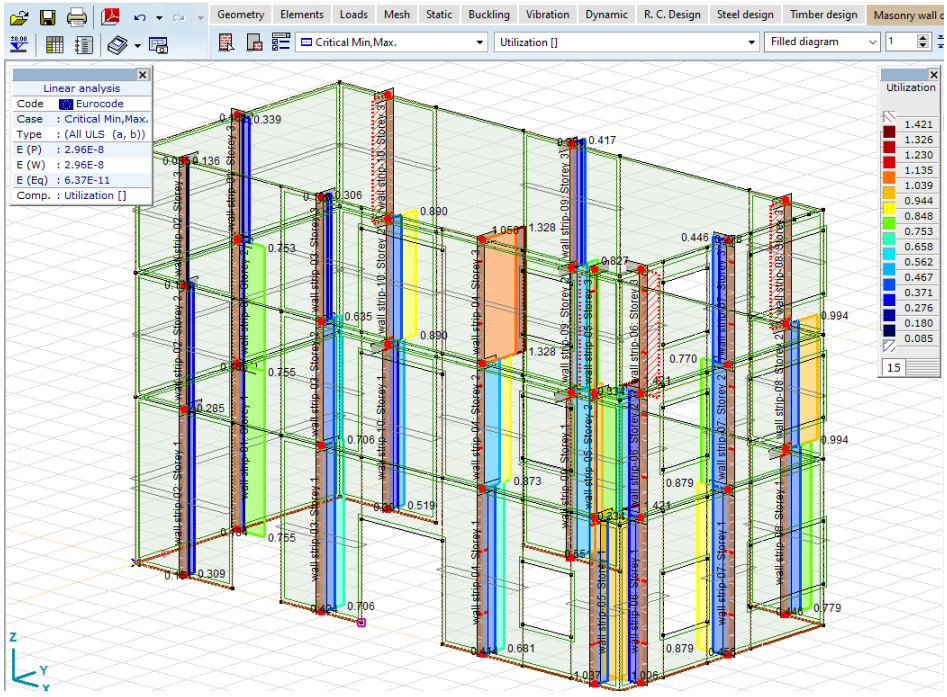


Click on *Design calculations* button to see the design details. The calculation can be saved into the report. *Design calculations* is available via *Design check of masonry wall* and *Masonry walls* windows, as well.
If it is opened from *Masonry walls* window, it summarizes the design details for the selected storeys of the actual masonry wall strip.
If it is opened from *Design check of masonry wall* window, only the design results of the investigated storey are summarized.
If shear check is requested, the design calculation contains more calculation units and it appears in more complex form. Clicking on the *Design calculation* icon, the user may specify the elements to be included in the documentation:
If the position of top (*1 - 1v*), middle (*m - mv*), or bottom (*2 - 2v*) reference section coincide, only the results of the critical sections are detailed.



Masonry wall utilization result component

If masonry wall parameters are assigned to virtual strips in the model and static (linear/non-linear) or dynamic results are available, a new result component becomes available on the *Masonry wall design* tab related to design check results.



If the utilization cannot be calculated at any storey, the program displays a red dashed line on the negative side of the diagram indicating the error.

Results in the Table Browser

Design results are also available in the Table Browser (see **2.9 Table Browser**), where the results of every reinforced concrete wall are available in summary tables.

6.9.1.7 Principles of checking

Checking

The software can perform two types of analysis with the following restrictions:

A. Design check of masonry walls subjected to mainly vertical loads.
In this checking only the out-of-plane eccentricities can be taken into account, shear check is not included. The verification is based on the reference cross-sections signed *1*, *m*, *2*. (Only the eccentricities perpendicular to the plane of the wall are taken into account, irrespective of the width of the wall strip, two-directional eccentricity cannot be analysed.)

B. Design check of masonry bracing walls, which contains a complex stability analysis considering out-of-plane and in-plane eccentricity of the axial force and an in-plane shear check as well.
This verification is concerned to the reference cross-sections signed 1_v, m_v, 2_v, but executed only if shear check is requested.

- The tensile strength of masonry perpendicular to bed joints is zero.
- Only single-leaf walls can be examined with constant wall thickness along a storey.
- The additional bending moments due to the eccentric effects (alignment of the walls in case of various wall thickness, masonry joint settings, stress distribution under the slab reaction) of the settings can only be calculated by approximation models if the results of the model do not contain them. The changes in the stiffness due to the eccentric effects cannot be taken into account.

Calculation of design internal forces

The internal forces of a virtual strip are calculated by the integration of the internal forces of the domain considering the total width of the strip.

The design axial, shear force and in-plane bending moment are equal to the identical internal force of the virtual strip (N_x, V_y and M_z).

The out-of-plane design bending moment is the sum of the moment of the virtual strip M_y and the additional bending moment calculated by the approximate models.

The available approximate models are the followings:

Hinged model:

This is a simple beam model with bottom hinges. The walls are supported laterally on each storey.

Continuous model:

The additional bending moment calculated using a continuous beam model which involves all the walls. The walls are supported laterally on each storey, the bottom joint of the lower storey is supposed to be fixed.

☞ **In these models, only the linear stiffness properties of walls are considered, and the stiffness supposed to be constant for a storey. The various stiffness of connecting slabs and lateral supports cannot be taken into account.**

☞ **In SIA standard only the *Hinged model* is available.**

Standard specifications

The specific verification rules of the available standards see 6.9.1.8…6.9.1.10.

6.9.1.8 Design check according to Eurocode 6

Naming conventions

During the design check 6 different reference cross-section are distinguished:

- the reference sections *1*, *m*, *2* are identical to the design sections explained in chapter *6.1* in *Eurocode* (top, middle and bottom sections);
- sections 1_v, m_v, 2_v are the top, middle and bottom sections and used for shear check.

Stability check

In case of reference sections signed with *1*, *m* and *2*, only the out-of-plane eccentricities of the axial force are taken into account in the analysis. At sections 1_v, m_v, 2_v both eccentricities (out-of-plane and in-plane) are considered. The two cases are discussed together below.

General principles

Considering the general principles of the standard (chapter *6.1.*) the program executes the following verifications:

The resistance of the wall section is adequate if the next criterion

$$N_{Ed} \leq N_{Rd}$$

and the detailed rules are satisfied.

The following formula is applied to determine the vertical resistance of a wall:

$$\mathrm{N}_{Rd} = \Phi_y \cdot \Phi_z \cdot t \cdot b \cdot f_d$$

where Φ_y and Φ_z are the reduction factors related to *y* and *z* direction. (In case of reference cross-sections *1*, *m* and *2*, the capacity reduction factor Φ_z = 1 because the in-plane eccentricity is not taken into account).

At the top and bottom sections, the capacity reduction factor is calculated by the effective wall thickness - t_{eff} (see masonry joint settings).

In case of shear check, if the top 1_v or bottom 2_v cross-section is adjusted to a corner of internal opening and the distance between the reference cross-section (1_v or 2_v) and the top/bottom of the wall is greater than the width of the wall, the section is considered as 'middle' section during the verification. Reference cross-section m_v is always considered as 'middle' section.

The design compressive strength is: $f_d = f_k/\gamma_M$

If the cross-sectional area of a wall strip is less than 0,01 m^2, the design compressive strength should be lowered: $\mathrm{f}_{d,red} = f_d \cdot (0{,}7 + 3A)$, where *A* is the loaded horizontal gross cross-sectional area of the wall, expressed in square metres.

In the module, only the cross-sectional area of the virtual strip can be considered, irrespective the total width of the concerned domain.

Checking at the top or bottom of the wall

The design eccentricities, the reduction factors and the vertical resistance of the sections are calculated as follows:

Out-of-plane eccentricity

The design eccentricity is calculated by the next equation:

$$e_{i,z} = \frac{M_{id,y}}{N_{id}} + e_{he,z} + e_{init,z} \geq 0{,}05 \cdot t$$

where $e_{i,z}$ is the eccentricity at the top (i=1) or the bottom (i=2) of the wall.

The program does not separate the eccentricities depending on the load type (vertical/horizontal) and indicates the sum of them in the calculation as follows. The eccentricity based on the internal forces calculated to the axis of the wall:

$$e_{0,z} = \frac{M_{id,y}}{N_{id}} + e_{he,z}$$

In the verification, the program applies lowered wall thickness (t_{eff}), if not the whole section is working (due to the masonry joint settings). In this case, the eccentricity is calculated to the axis of the effective wall thickness:

$$e_{0,i,z} = e_{i,z} - \Delta z$$

where Δz is the difference between the axis of the wall and effective wall thickness.

(If the wall thickness *t* is not reduced due to the joint settings, then $t_{eff} = t$ and $e_{0,i,z} = e_{0,z}$.)

Considering the rules of the standard and the markings above, the design eccentricity comes from the following expression:

$$e_{i,z} = e_{0,i,z} + e_{init,z} \geq 0{,}05 \cdot t_{eff}$$

The initial eccentricity is measured in the unfavourable direction relative to the axis of the effective thickness, the minimum eccentricity is calculated by t_{eff}.

The $\Phi_{i,y}$ capacity reduction factor is calculated as follows:

$$\Phi_{i,y} = 1 - \frac{2 \cdot e_{i,z}}{t_{eff}}$$

In-plane eccentricity

The design eccentricity is calculated by the next formula:

$$e_{i,y} = e_{0,i,y} + e_{init,y} + e_{\Delta 2,y}$$

where $e_{i,y}$ the in-plane eccentricity of the axial force in the *y* direction.

The initial eccentricity $e_{init,y}$ is calculated by following the rules of chapter 5.3 in EN 1996-1-1 5.3 considering initial incline of the building and the global design parameters of the masonry strip.

The second-order eccentricity $e_{\Delta 2,y}$ is calculated using *Annex B* of *EN 1996-1-1*.

These additional eccentricities are taken into account in the unfavourable direction.

The capacity reduction factor $\Phi_{i,z}$ is calculated as follows:

$$\Phi_{i,z} = 1 - \frac{2 \cdot e_{i,y}}{b}$$

where *b* is the total width of the wall (strip).

In case of seismic type combination, the program does not automatically calculate second-order eccentricity. This can be taken into account using the plastic stability index available in the table of *Seismic sensitivity of storeys* and following the related regulations of the standard *EN 1998-1-1*.

Vertical resistance

Considering the total wall width *b*, the vertical resistance is the following:

$$\mathrm{N}_{Rd,i} = \Phi_{i,y} \cdot \Phi_{i,z} \cdot t_{eff} \cdot b \cdot f_d$$

Checking in the middle of the wall height

The design eccentricities, the reduction factors and the vertical resistance of the middle sections are calculated as follows:

Out-of-plane eccentricity

The design eccentricity is calculated by the next expression:

$$e_{m,z} = \frac{M_{md,z}}{N_{md}} + e_{hm,z} \pm e_{init,z}$$

The program does not separate the eccentricities depending on the load type (vertical/horizontal) and indicates the sum of them in the calculation as follows. The eccentricity based on the internal forces calculated to the axis of the wall:

$$e_{0,m,z} = \frac{M_{md,y}}{N_{md}} + e_{hm,z}$$

and the eccentricity due to loads:

$$e_{m,z} = e_{0,m,z} \pm e_{init,z}$$

finally, the design eccentricity comes from the following expression:

$$e_{mk,z} = e_{m,z} + e_{k,z} \geq 0{,}05 \cdot t$$

where $e_{k,z}$ is the eccentricity due to creep, but it only has to be taken into account if the slenderness of the wall exceeds the limit of λ_C:

$$\frac{h_{ef}}{t_{ef}} > \lambda_c$$

in this case

$$e_{k,z} = 0{,}002 \cdot \phi_\infty \cdot \frac{h_{ef}}{t_{ef}} \sqrt{t \cdot e_{m,z}}$$

λ_C is specified in the National Annex (in the program the parameter λ_C can be modified if necessary).

In ***NEN-EN 1996-1-1*** (***Dutch code***) the following criterion is considered when calculating $e_{mk,z}$:

$$e_{mk,z} = max\left\{e_{m,z} + e_{k,z};\ 0{,}05 \cdot t;\ \frac{h_{ef}}{300};\ 10\ \mathrm{mm}\right\}$$

The factor of $\Phi_{m,y}$ is calculated by Annex *G* (*G1…G4*) in *EN 1996-1-1*.

In-plane eccentricity

The design eccentricity is calculated by the next formula:

$$e_{m,y} = e_{0,m,y} + e_{init,y} + e_{\Delta 2,y}$$

where $e_{m,y}$ the in-plane eccentricity of the axial force in the *y* direction.

The initial eccentricity $e_{init,y}$ is calculated by following the rules of chapter 5.3 in EN 1996-1-1 5.3 considering initial incline and the set global design parameters of the masonry strip.

The second-order eccentricity $e_{\Delta 2,y}$ is calculated using *Annex B* of *EN 1996-1-1*.
These additional eccentricities are taken into account in the unfavourable direction.

The capacity reduction factor $\Phi_{m,z}$ is calculated as follows:

$$\Phi_{m,z} = 1 - \frac{2 \cdot e_{m,y}}{b}$$

where *b* is the total width of the wall (strip).

In case of seismic type combination, the program does not automatically calculate second-order eccentricity. This can be taken into account using the plastic stability index available in the table of *Seismic sensitivity of storeys* and following the related regulations of the standard *EN 1998-1-1*.

Vertical resistance

Considering the total wall width *b*, the vertical resistance is the following:

$$N_{Rd,i} = \Phi_{m,y} \cdot \Phi_{m,z} \cdot t \cdot b \cdot f_d$$

Design and detailing rules

The following rules are checked:

- slenderness of the wall: $\lambda \leq 27$.
- minimum wall thickness according to the National Annex. It is examined, only if a specific value is given in the National Annex. In cases, when the minimum thickness depends on the type of the wall, the program applies only the minimum of the given values.
- minimum area of wall: 0,04 m^2.
- maximum relative eccentricity (it is not clearly regulated in *EN 1996-1-1*). In the program, the limit can be set freely (see *Design parameters*). The default setting is: $e_{rel,max}$ = 0,40 (only one value can be set, and it is taken into account in all three sections).
  Checking
  - at the top (i=1) or bottom (i=2):
    $$e_{rel,i} = \frac{e_i}{t_{eff}} \leq e_{rel,max}$$
  - in the middle of the wall height:
    $$e_{rel,m} = \frac{e_{mk}}{t} \leq e_{rel,max}$$

  The relative eccentricity is checked in both direction (*y* and *z*).

Shear check

Shear capacity is calculated in sections 1_v, m_v, 2_v at each storey considering the following principles.

In *EN 1996-1-1* standard, the general formula (*chapter 6.2, 6.13)* for shear check does not provide clear instructions to handle the out-of-plane eccentricity of the axial force. For this reason, two different shear checks are executed ('*linear*' and '*interaction*' check), of which the lower shear capacity is considered as critical. In the design calculation, only the result of the critical check is documented.

Linear shear check

In case of *Linear shear check*, the capacity is calculated by the formula detailed in chapter *6.2* in *EN-1996-1-1*, but the out-of-plane eccentricity of the axial force is neglected (supposed to be zero)

$$V_{Rd,i} = f_{vd} \cdot t \cdot l_c$$

$V_{Rd,i}$ is the shear resistant of the top (1_v), middle (m_v) or bottom (2_v) reference cross-section (index i indicates the position of the section *1, m, 2*),

l_c is the length of the compressed part of the wall assuming linear stress distribution.

f_{vd} is the design value of the shear strength of masonry based on the average of the vertical stresses over the compressed part of the wall.

Interaction shear check

In case of interaction shear check, eccentricities are considered in both directions and the length of the compressed part of the section is calculated assuming constant stress distribution:

$$V_{Rd,i} = f_{vd} \cdot t_{nom} \cdot l_c$$

$V_{Rd,i}$ is the shear resistant of top (1_v), middle (m_v) or bottom (2_v) reference cross-section (index i indicates the position of the section *1, m, 2*),

t_{nom} is the width of the compressed part of the wall considering the out-of-plane eccentricity,

l_c is the length of the compressed zone calculated by in-plane eccentricity.

Based on the results of the stability test:

$$l_c = \Phi_{i,z} \cdot b$$

$$t_{nom} = \Phi_{i,y} \cdot t$$

f_{vd} is the design value of the shear strength of masonry based on the average of the vertical stresses over the compressed part of the wall.

☞ **When calculating the resistance by the above methods, if the reference cross-section 1_v or 2_v are considered as *'top'* or *'bottom'* section in the verification (the distance between the top/bottom level and the position of the reference cross-section lower than the width of the wall), then lowered wall thickness t_{eff} is taken into account in the calculation.**

Notes to the result of the shear check

Usually, there is only slight difference between the results of above methods but it depends on many parameters. In case of the interaction check, the area of the compressed zone is lower, but the design shear strength may be greater due to the greater average vertical stresses. However, the design shear strength is limited by the maximum shear strength according to the *National Annexes*.

Typically, the interaction check gives the critical (lower) capacity.

Error messages

The following error messages appears in the results:

Displayed error message:	*Interpretation of the error:*
Not passed!	The resistance of the wall/section is not adequate.
The force is out of the cross-section	The point of action of the vertical force is out of the cross-section.
Relative eccentricity is too high.	In the cross-section the relative eccentricity exceeds the limit (see *Design parameters*).
The cross-section is in tension.	The cross-section is in tension, the vertical resistance cannot be calculated.
Axial compression force is zero.	In the cross-section the axial compression force is negligibly small, less than 0,1 kN.
Slenderness is too high.	The slenderness of the wall exceeds the limit according to the standard.
The wall thickness is insufficient.	The wall thickness is less than the limit according to the National Annex.
The wall section is insufficient.	The cross-sectional area of the wall (cross-section of the virtual strip) is less than the limit according to the standard.

6.9.1.9 Design check according to NTC

General principles

Following the general principles of the NTC standard, a wall has to be checked at three sections (top, mid-height of the wall, bottom) similar to *Eurocode 6*.

In the standard, it is supposed that the wall has hinges at the top and bottom. The user has to build such a model with the right settings that follows the guidelines of the standard.

Signs

During the design check 6 different reference cross-section are distinguished:

- the reference sections signed as *1*, *m*, *2*-es are identical to the design sections explained in chapter *6.1* in *Eurocode* (top, middle and bottom sections);
- sections signed as 1_v, m_v, 2_v are the top, middle and bottom sections and used for shear check.

Stability check

In case of reference sections signed with *1*, *m* and *2*, only the out-of-plane eccentricities of the axial force are taken into account in the analysis. At sections 1_v, m_v, 2_v both eccentricities (out-of-plane and in-plane) are considered. The two cases are discussed together below.

The resistance of the wall section is adequate if the following criterion

$$N_d \leq N_r$$

and the detailed rules are satisfied.

The program uses the following formula to determine the vertical resistance of a wall:

$$\mathrm{N}_r = f_{d,rid} \cdot t \cdot b$$

and

$$\mathrm{f}_{d,rid} = \Phi_y \cdot \Phi_z \cdot f_d$$

where Φ_y and Φ_z are the reduction factors related to *y* and *z* direction. (In case of reference cross-sections *1*, *m* and *2*, the capacity reduction factor Φ_z=1 because the in-plane eccentricity is not taken into account).

At the top and bottom sections, the reduction factor is calculated by the effective wall thickness (see masonry joint settings).

The design compressive strength *is* $f_d = f_k/\gamma_M$

Checking at the top or bottom of the wall

The design eccentricities, the capacity reduction factors and the vertical resistance of the sections are calculated as follows:

Out-of-plane eccentricity

In *NTC*, the design eccentricity is calculated by the following equation:

$$e_{1,z} = |e_{s,z}| + e_{a,z}$$

where $e_{s,z}$ is the eccentricity of vertical loads in the local *z* direction.

The program does not separate the eccentricities depending on the load type (vertical/horizontal) and indicates the sum of them as $e_{0,z}$.

In the verification, the program applies lowered wall thickness (t_{eff}), if not the whole section is working (due to the masonry joint settings). In this case, the eccentricity is calculated to the axis of the effective wall thickness:

$$e_{0,1,z} = e_{0,z} - \Delta z$$

where Δz is the difference between the axes of the wall and effective wall thickness.

(If the wall thickness *t* is not reduced due to the joint settings, then $t_{eff} = t$ and $e_{0,i,z} = e_{0,z}$.)

Finally, the design eccentricity comes from the following expression:

$$e_{1,z} = e_{0,1,z} + e_{a,z}$$

Where $e_{a,z} = h/200$ (*h* is the storey height).

The initial eccentricity $e_{a,z}$ is measured in the unfavourable direction relative to the axis of the effective thickness (t_{eff}).

Based on the initial parameters, the factor of $\Phi_{i,z}$ is calculated by linear interpolation using the table given in the standard (see below).

The relative eccentricity: $e_{rel,1,z} = e_{1,z}/t_{eff}$, where t_{eff} is the effective wall thickness.

The eccentricity coefficient *m* is calculated by the following: $m = 6 \cdot e_{rel,1,z}$,

and the slenderness: $\lambda = h_0/t$, where $h_0 = \varrho \cdot h$.

In-plane eccentricity

The in-plane eccentricity and the capacity reduction factor are calculated according to the principles of *EN 1996-1-1* standard (see details above).

Vertical resistance

Considering the total wall width *b*, the vertical resistance is the following:

$$N_r = \Phi_{1,y} \cdot \Phi_{1,z} \cdot f_d \cdot t_{eff} \cdot b$$

Checking in the middle of the wall height

The design eccentricities, the capacity reduction factors and the vertical resistance of the middle sections are calculated as follows:

Out-of-plane eccentricity

The design eccentricity is calculated by the following expression:

$$e_{2,z} = \frac{e_{1,z}}{2} + e_{v,z}$$

where $e_{2,z}$ is the design eccentricity at the top in the local *z* direction, $e_{v,z}$ eccentricity is calculated from horizontal loads.

The program does not separate the eccentricities depending on the load type (vertical/horizontal) and indicates the sum of them as $e_{0,z}$.

The design eccentricity comes from the following expression:

$$e_{2,z} = e_{0,z} + \frac{e_{a,z}}{2} \geq e_{a,z}$$

Where $e_{a,z} = h/200$

In the above formula, half of the $e_{a,z}$ is taken in the calculation considering the hinged model recommended in the standard.

If the user uses hinges for the domains and also applies the model called *Hinged model* to determine the additional bending moments, then this formula gives the same result as the standard. The applied models have to be considered by the users.

Based on the initial parameters, the factor of $\Phi_{2,z}$ is calculated by linear interpolation using the table given in the standard (see below).

The relative eccentricity: $e_{rel,2,z} = \frac{e_{2,z}}{t}$, where *t* is the total wall thickness.

The eccentricity coefficient *m* is calculated by the following: $m = 6 \cdot e_{rel,2,z}$,

and the slenderness: $\lambda = \frac{h_0}{t}$, where $h_0 = \varrho \cdot h$.

In-plane eccentricity

The in-plane eccentricity and the capacity reduction factor are calculated according to the principles of *EN 1996-1-1* standard (see details above).

Vertical resistance

Considering the total wall width *b*, the vertical resistance is the following:

$$N_r = \Phi_{2,y} \cdot \Phi_{2,z} \cdot f_d \cdot t_{eff} \cdot b$$

Interpolation table for the determination of Φ_y factor

The following table (given in the standard, Tab. 4.5.III) is used to determine the factor of Φ_y.

Slenderness - λ	*m = 6e/t*				
	0	*0,5*	*1,0*	*1,5*	*2,0*
0	1,00	0,74	0,59	0,44	0,33
5	0,97	0,71	0,55	0,39	0,27
10	0,86	0,61	0,45	0,27	0,16
15	0,69	0,48	0,32	0,17	-
20	0,53	0,36	0,23	-	-

Shear check

Shear resistance is calculated according to the principles of *EN 1996-1-1* standard (see details above). Design shear strength is determined according to *NTC* standard.

Design and detailing rules

The following rules are checked:

- slenderness of the wall: $\lambda \leq 20$.
- minimum wall thickness: t ≥ 150 mm (only this value is considered, the minimum value depending on the type of the unit is not checked).
- minimum area of wall: 0,04 m^2 (applying the rule given *Eurocode 6*).
- maximum relative eccentricity (the default value is 0.33, but it can be modified).
  Checking
  - at the top or bottom:

$$e_{rel,1} = \frac{e_1}{t_{eff}} \leq 0{,}33$$

  - in the middle of the wall height:

$$e_{rel,2} = \frac{e_2}{t} \leq 0{,}33$$

  The relative eccentricity is checked in both direction (*y* and *z*).

Error messages

The following error messages appears in the results:

Displayed error message:	*Interpretation of the error:*
Not passed!	The resistance of the wall/section is not adequate.
The force is out of the cross-section.	The point of action of the vertical force is out of the cross-section.
Relative eccentricity is too high.	In the cross-section the relative eccentricity exceeds the limit (see *Design parameters*).
The cross-section is in tension.	The cross-section is in tension, the vertical resistance cannot be calculated.
Axial compression force is zero.	In the cross-section the axial compression force is negligibly small, less than 0,1 kN.
Slenderness is too high.	The slenderness of the wall exceeds the limit according to the standard.
The wall thickness is insufficient.	The wall thickness is less than the limit according to the National Annex.
The wall section is insufficient.	The cross-sectional area of the wall (cross-section of the virtual strip) is less, than the limit according to the standard.
Φ cannot be calculated!	Φ cannot be evaluated based on the initial parameters of interpolation.

6.9.1.10 Design check according to SIA

General principles

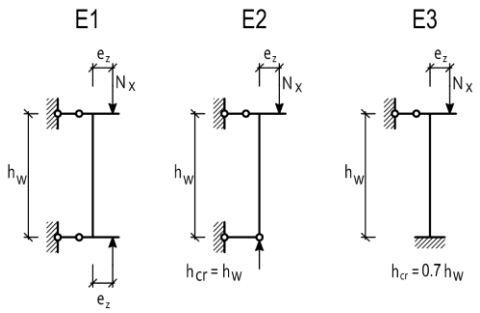
The principles of *SIA 266* design code calculation method differ significantly from the procedure described in *Eurocode 6*. Based on the standard, the wall has to be checked in ultimate and serviceability limit state (ULS and SLS) as well.

The program performs ULS or SLS verification depending on the type of combination. If the type is not known, both checking is performed, but only the result of the critical state is displayed.

The design code contains constraints on the static model, so the user should take care of the correct choice of the static model. For the assumptions, see details below.

Calculating the additional bending moment only the *Hinged model* is available, and only constant stress distribution can be applied.

The applied static model (*E1*, *E2* and *E3*) has to be selected as the program cannot determine it from the model.



Signs of design cross-sections

Four different design cross-sections are distinguished in the verification

- cross-sections with a sign of *1 (top)* or *2 (bottom)*,
- cross-sections with a sign of *1v (top)* or *2v (bottom) for shear check*.

ULS verification

In ULS verification, the wall is checked in two sections of the wall: at the top and the bottom of the wall. Four different analyses are performed:

- stress limitation at cross-sections *1* and 1_v,
- stability check considering the internal forces of the virtual strip at the reference position of sections *1* and 2_v,
- stress limitation at cross-sections *2* and 2_v,
- shear check at the top cross-section (1_v).

The verifications of cross-sections 1_v and 2_v are performed only if shear check is requested.

At design cross-sections *1* and *2* only the out-of-plane eccentricity of the axial force is taken into account, but at sections 1_v and 2_v the in-plane eccentricity is also considered.

Stress limitation

Firstly, the relative eccentricity is checked according to the standard: (e_z/t_w) is limited by the formula *(6)* and Figure 4 ($e_z/t_w \leq 0{,}375$). On the other hand, the program calculates the load-bearing capacity considering the compressed zone of the wall and it is compared to the design axial force.

The compressed width of the section considering out-of-plane eccentricity:

$$t_{nom} = t - 2 \cdot e_{0,z}$$

In the verification, the program applies lowered wall thickness (t_{eff}), if not the whole section is working (due to the masonry joint settings).

$e_{0,z}$ is the eccentricity is calculated to the axis of the effective wall thickness:

$$N_{Rd,nom} = l_c \cdot t_{nom} \cdot f_{xd}$$

where l_c is equal to l_w when checking cross-sections *1* and *2*. In case of cross-sections 1_v and 2_v the compressed length of the wall is ($l=l_2$) lowered due to the in-plane bending moment:

$$l_2 = l_w - \frac{2 \cdot M_{z,i,d}}{N_{xd}}$$

Stability check

In case of cross-section *1*, the program follows the regulations of the standard (*SIA 266)* described in chapter *4.3.1.3* (11):

$$N_{xd} \leq k_N \cdot l_w \cdot t_w \cdot f_{xd}$$

In case of cross-section 2_v, formula (17, in chapter *4.3.3.1*) is used considering in-plane bending moment:

$$N_{xd} \leq k_N \cdot l_2 \cdot t_w \cdot f_{xd}$$

where

- N_{xd} is the design vertical force at the top of the wall,
- l_w width of the wall (in the program it is equal to the width of the virtual strip)
- t_w wall thickness,
- f_{xd} is the design compressive strength, $f_{xd} = f_{xk}/\gamma_M$
- l_2 length of the compressed part of the wall assuming constant stress distribution:

$$l_2 = l_w - \frac{2 \cdot M_{z2d}}{N_{xd}}$$

The factor of k_N is calculated by linear interpolation based on the set static model (*E1*, *E2* and *E3*) using the design curves given in the standard (*SIA 266, Figure 5*).

The interpolation is determined by the relative slenderness of the wall (h_w / t_w or h_{cr} / t_w) and the relative eccentricity (e_z / t_w) calculated at the top of the wall.

In case of eccentricity e_z, the software assumes the followings:

- the eccentricity is originated of the eccentricity of slab reaction. The resultant of the slab reaction force is in the axis of the effective wall thickness (lowered wall thickness considering masonry joint settings).
- the eccentricity of the vertical force from the wall above is supposed to be zero at the investigated top section ($e_z \approx 0$).

The user should consider whether these approximations are appropriate based on the built model and settings.

In the standard, the relative eccentricity (e_z/t_w) is limited by the formula (*6*) and the Figure 4 ($e_z/t_w \le 0{,}375$). But in the program, this value is limited to 0.35, because of the limits of the design curve for interpolation.

$$\frac{e_z}{t_w} \le min\left\{\frac{1}{2}\cdot\left(1-\frac{N_{xd}}{l_w\cdot t_w\cdot f_{xd}}\right);0{,}35\right\}$$

If this regulation is not satisfied then the vertical resistance of the wall cannot be calculated.

☞ **The standard does not contain a similar checking method for the mid-height section as *Eurocode 6*, so the wall is not checked for the internal forces in the middle section.**

Shear check

The shear capacity is determined by the method set in the parameters of the masonry strip:

According to diagrams of SIA 266:

The program follows the rules of section *4.3.3.1* (*SIA 266*) and uses formula (18):

$$V_d \le k_V\cdot l_1\cdot t_{nom}\cdot f_{yd}$$

where

V_d is the design shear force in reference cross-section *1v*,

l_1 is the length of the compressed part considering in-plane bending moment:

$$l_1 = l_w - \frac{2\cdot M_{z1d}}{N_{xd}}$$

t_{nom} is the width of the compressed zone due to the out-of-plane bending:

$$t_{nom} = t - 2\cdot e_{0,y}$$

(In the verification, the program applies lowered wall thickness - t_{eff}, if not the whole section is working due to the masonry joint settings).

f_{yd} is the design compressive strength in the direction parallel to the bed joints:

$$f_{yd} = \frac{f_{yk}}{\gamma_M}$$

The factor of k_V is calculated by linear interpolation based on the design curves given in the standard (*SIA 266, Figure 9*). h_w is the height difference of cross-sections 1_v and 2_v. The software makes also interpolation between the 3 given curves depending on the ratio of f_{xd}/f_{yd} in the range of 0.30...0.75.

Based on numerical calculation:

Considering the basic principles and boundary conditions of section *4.3.2.1 (SIA 266)*, the program solves the satisfying equation system by iterative method to find the possible stress field, which results in the maximum shear capacity.

Error messages

The following error messages appears in the results:

Displayed error message:	*Interpretation of the error:*
Not passed!	The resistance of the wall/section is not adequate.
The force is out of the cross-section.	The point of action of the vertical force is out of the cross-section.
Relative eccentricity is too high.	In the cross-section the relative eccentricity exceeds the limit (see Design parameters).
The cross-section is in tension.	The cross-section is in tension, the vertical resistance cannot be calculated.
Axial compression force is zero.	In the cross-section the axial compression force is negligibly small, less than 0,1 kN.
Slenderness is too high.	The slenderness of the wall exceeds the limits of the design curves for interpolation ($h_w/t_w \le 50$ or $h_{cr}/t_w \le 50$), the factor of k_N cannot be calculated.

SLS verification

The program follows the regulations of the standard (*SIA 266)* described in chapter *4.4.1.5* (21):

The maximum crack width can be calculated by the next formula:

$$r = k_r \cdot \frac{N_x \cdot h_0}{E_{xk} \cdot l_w \cdot t_w}$$

where

N_x is the vertical force on the top of the wall,
h_0 is the total height of the masonry unit and mortar,
E_{xk} moduli of elasticity,
l_w width of the wall (in the program it is equal to the width of the virtual strip),
t_w wall thickness,

The factor of k_r is calculated by linear interpolation based on the set static model (*E1*, *E2* and *E3*) using the design curves given in the standard (*SIA 266, Figure 12*).

The interpolation is determined by the relative slenderness of the wall (h_w/h_E or h_{cr}/h_E) and the relative eccentricity (e_y/t_w) calculated at the top of the wall.

The parameter of h_E is calculated as follows:

$$h_E = \pi \cdot \sqrt{\frac{E_{xk} \cdot l_w \cdot t_w^3}{12 \cdot N_x}}$$

The wall is adequate for the SLS verifications, if $r \leq r_{max}$, where r_{max} is the maximum crack width according to *4.2.2.2*. The default value of the maximum crack width is 0,2 mm, but it can be set among the design parameters.

In case of shear check

If shear check is requested, then the SLS verification is extended by the followings according to *4.4.2.1* in *SIA 266*.

The relative displacement – the value of '*v*' according to formula (22) - between the top (1_v) and bottom (2_v) sections is calculated by the relative deformation of the virtual strip.

The maximum strain $\varepsilon_{x,max}$ is determined by the formula (23):

$$\varepsilon_{x.max} = \frac{6 \cdot (M_{z1} + V \cdot h_w)}{E_{xk} \cdot l_w^2 \cdot t_w} - \frac{N_x}{E_{xk} \cdot l_w \cdot t_w}$$

The relative displacement and the maximum strain are always determined by the top (1_v) and bottom (2_v) sections defined for the shear check.

The above results are compared to the limits set among the design parameters of a storey.

Error messages

The following error messages appears in the results:

Displayed error message:	*Interpretation of the error:*
Not passed!	The resistance of the wall/section is not adequate.
Relative eccentricity is too high.	In the cross-section the relative eccentricity exceeds the limit of 0.35.
The cross-section is in tension.	The cross-section is in tension, the vertical resistance cannot be calculated.
Axial compression force is zero.	In the cross-section the axial compression force is negligibly small, less than 0,1 kN.
kr cannot be calculated!	The factor of *kr* cannot be evaluated based on the initial parameters of interpolation (h_w/h_E or h_{cr}/h_E and e_z/t_w)

Design and detailing rules

The following rules are checked during the verification:

- the slenderness of the wall: $\lambda \leq 50$ (limit of the design curves of k_N factor, $\lambda = h_w/t_w$ or $\lambda = h_{cr}/t_w$).
- maximum relative eccentricity (limited in the design curves for the interpolation):
  $e_{rel,max} = 0{,}35$.
  The relative eccentricity is checked only at the top cross-section:
  $$e_{rel,i} = \frac{e_z}{t} \leq e_{rel,max}$$
  where *t* is the effective cross-section.

7. Programming AXISVM – COM module

AXISVM COM server

AXISVM, like many other Windows application, supports Microsoft COM technology making its operations available for external programs. Programs implementing a COM server register their COM classes in the Windows Registry providing interface information.

Any external program can get these descriptions, read object properties or call the functions provided through the interface. A program can launch AXISVM, build models, run calculations and get the results through the AXISVM COM server. This is the best way to

- build and analyse parametric models,
- finding solutions with iterative methods or
- build specific design extension modules.

DLL modules placed in the *Plugins* folder of AXISVM are automatically included in the *Plugins* menu imitating the subfolder structure of the *Plugins* folder. The AXISVM COM server specification and programming examples are downloadable from the AXISVM website, **www.axisvm.eu.**

Programming AXISVM via Python is supported by the PyAXISVM package that requires at least Python 3.6 version (**https://github.com/AxisVM/pyaxisvm**).

8. AXISVM Viewer and Viewer Expert

AXISVM Viewer

AXISVM Viewer is a freely downloadable version of the program for viewing models without making changes. It can be downloaded from **my.axisvm.eu** after registration, see the next chapter.

Printing of drawings, tables or reports is not available.

This program allows a detailed presentation of a model in an environment where AXISVM has not been installed.

If you do not want others to use your work as a basis for their models, but you would like to let them see, the model can be saved in an AXISVM Viewer (*.AXV) file format (see *File/Export*). The market version cannot read AXV files but the Viewer can. This format guarantees that your work will be protected.

AXISVM Viewer Expert

Owners of the AXISVM market version can buy the Viewer Expert version which lets the user print diagrams, tables and reports or place temporary dimension lines and text boxes. No changes can be saved.

9. MYAXISVM – AXISVM Trial, Student, Viewer and Light version request

MYAXISVM

To download and request a trial/student/viewer version, you need to register a MYAXISVM account at **https://my.axisvm.eu/.**

You will receive an approval in email. Once downloaded and installed, the program can be used by entering the email address and password you provided when registering the account.

If the *Save Password* option is enabled, the program will automatically log you into the appropriate account on subsequent launches.



Trial version

To explore the capabilities of the software, you can download a trial version of AxisVM, which includes all design modules and does not limit the number of elements you can use. The trial version can be used for 30 days from the first launch.

Student version

University or college students can download a full-featured, unrestricted software package to do complex homeworks, study research topics and write a thesis (BSc, MSc). The downloaded version can be used for 6 months from the date of registration and can be used for the entire period of study (on a maximum of two computers) by extending the license, provided that the student has a valid student status.

Viewer version

Starting with X8, the Viewer can also be downloaded via the MyAXISVM interface. This free version can be used without sending a license request or time limit. More about the Viewer **8 AXISVM Viewer and Viewer Expert**

Light version

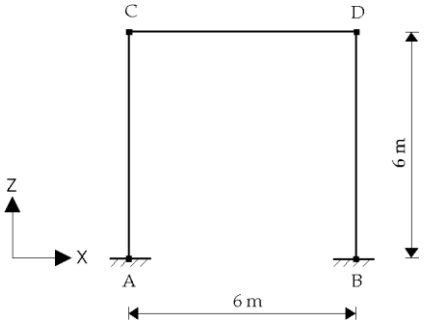
This free version can be used without sending a license request or time limit. It does not require a license file or hardware key, but the number of finite elements that can be used is limited to 40 beams or trusses, 150 surface elements, 50 load cases and 10 vibration shapes.

10. Examples

10.1 Linear static analysis of a steel plane frame

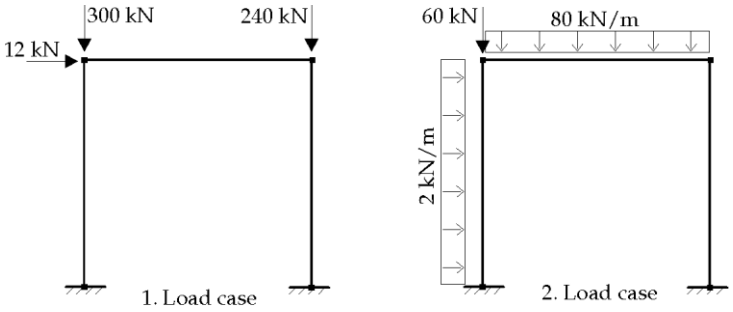
Input data **AK-ST-I.axs**

Geometry:



Material: Steel
Cross-section: I 240

Loads:



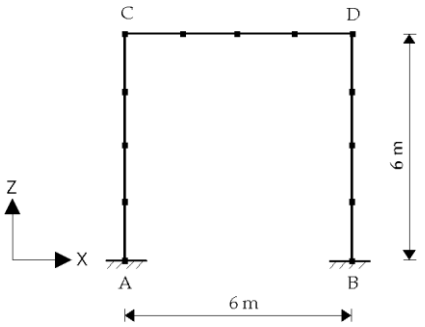
Results **AK-ST-I.axe**

Component		Analytic	AXISVM
1 Lc.	$u_x^{(C)}$ [mm]	17.51	17.51
	$M_y^{(A)}$ [kNm]	-20.52	-20.52
2 Lc.	$u_x^{(C)}$ [mm]	7.91	7.91
	$M_y^{(A)}$ [kNm]	63.09	63.09

10.2 Geometric nonlinear static analysis of a steel plane frame

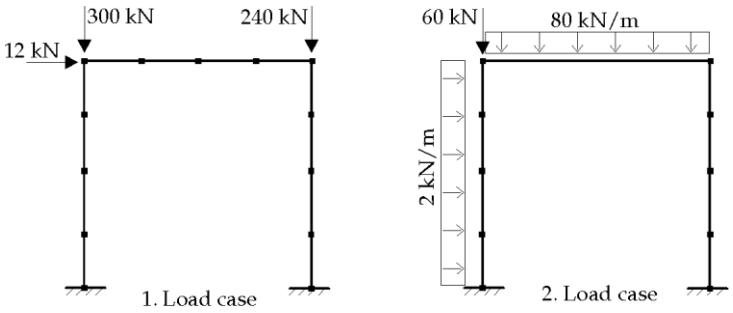
Input data **AK-ST-II.axs**

Geometry:



Material: Steel
Cross-section: I 240

Loads:



Results **AK-ST-II.axe**

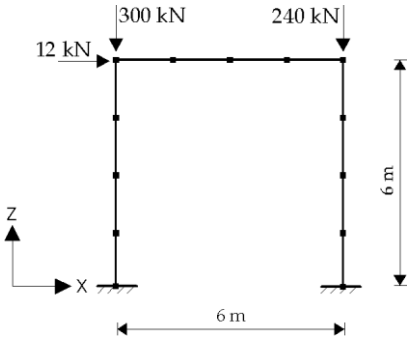
Component		With Stability Functions	AXISVM
1 Lc.	$u_x^{(C)}$ [mm]	20.72	20.58
	$M_y^{(A)}$ [kNm]	-23.47	-23.41
2 Lc.	$u_x^{(C)}$ [mm]	9.26	9.22
	$M_y^{(A)}$ [kNm]	66.13	66.25

Verify The equilibrium must be verified taking into account the deflections.

10.3 Buckling analysis of a steel plane frame

Input data **AK-KI.axs**

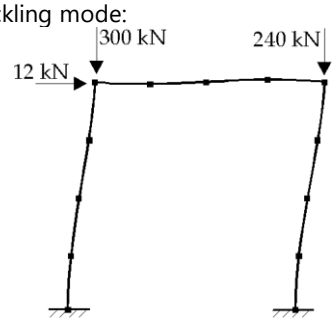
Geometry and loads:



Material: Steel
Cross-section: I 240

Results **AK-KI.axe**

Buckling mode:

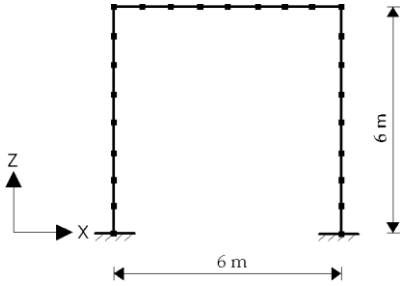


Critical load parameter	Cosmos/M™	AXISVM
n_{cr}	6.632	6.633

10.4 Vibration analysis (I-Order) of a steel plane frame

Input data **AK-RZ-I.axs**

Geometry:



Material: Steel
Cross-section: I 240

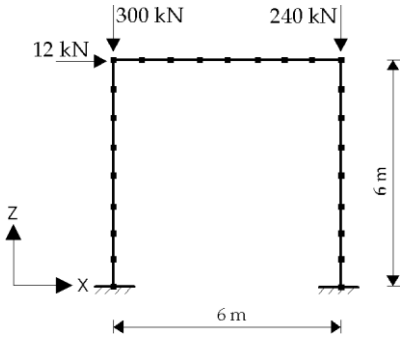
Results **AK-RZ-I.axe**

Mode	Frequency [Hz]	
	Cosmos/M™	AXISVM
1	6.957	6.957
2	27.353	27.353
3	44.692	44.692
4	48.094	48.094
5	95.714	95.714
6	118.544	118.544

10.5 Vibration analysis (II-Order) of a steel plane frame

Input data **AK-RZ-II.axs**

Geometry and loads:



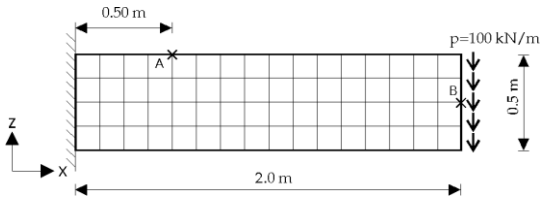
Material: Steel
Cross-section: I 240

Results **AK-RZ-II.axe**

Mode	Frequency [Hz]	
	Cosmos/M™	AXISVM
1	0.514	0.514
2	11.427	11.426
3	12.768	12.766
4	17.146	17.145
5	27.112	27.109
6	39.461	39.456

10.6 Linear static analysis of a reinforced concrete cantilever

Input data **VT1-ST-I.axs**



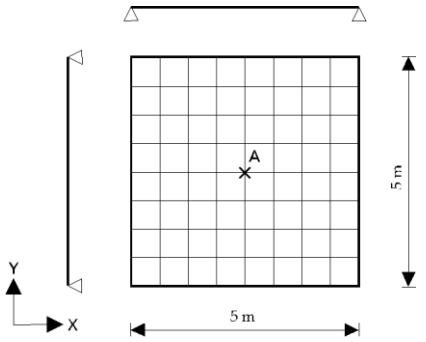
E=880 kN/cm^2
ν=0
t=0.10 m
p=100 kN/m
Mesh: 4x16

Results **VT1-ST-I.axe**

Component	Beam theory (shear deformations included)	AXISVM
$u_z^{(B)}$ [mm]	15.09	15.09
$m_x^{(A)}$ [kN/m]	1800.00	1799.86

10.7 Linear static analysis of a simply supported reinforced concrete plate

Input data **VL1-ST-I.axs**

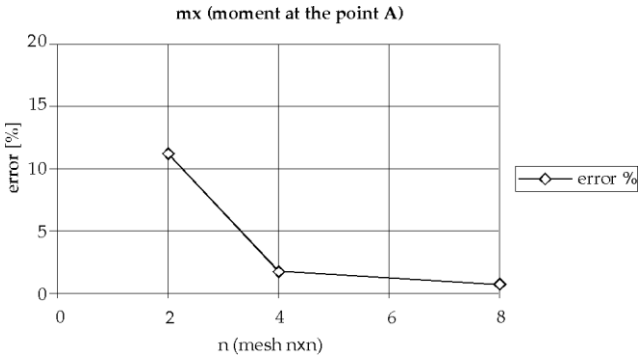


E=880 kN/cm^2
ν=0
t=0.15 m
p=50 kN/m^2
Mesh: 8x8

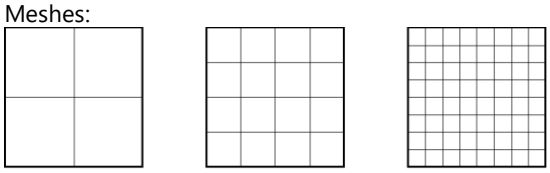
Results

Component	Analytic (shear deformations not included)	AXISVM (shear deformations included)
$w^{(A)}$ [mm]	51.46	51.46
$m_x^{(A)}$ [kNm/m]	46.11	46.31

Convergence analysis



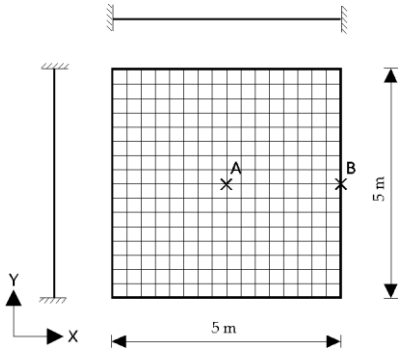
Meshes:



10.8 Linear static analysis of a clamped reinforced concrete plate

Input data

VL2-ST-I.axs



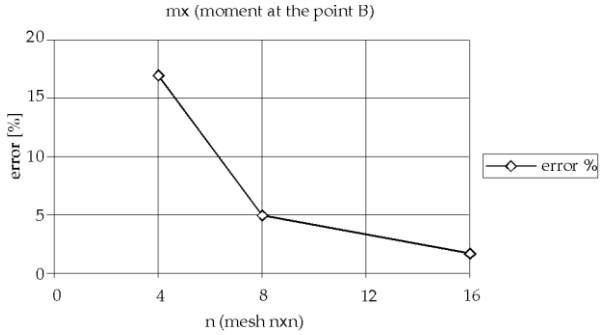
E=880 kN/cm2
ν=0
t=0.15 m
p=50 kN/m2
Mesh: 16x16

Results

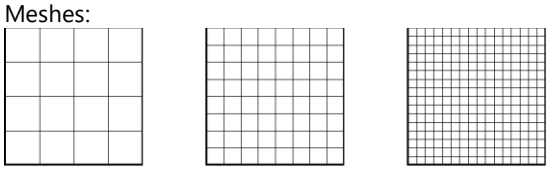
VL2-ST-I.axe

Component	Analytic (shear deformations not included)	AXISVM (shear deformations included)
$e_z^{(A)}$ [mm]	16.00	16.18
$m_x^{(A)}$ [kNm/m]	22.01	22.15
$m_x^{(B)}$ [kNm/m]	64.43	63.25
$q_x^{(B)}$ [kN/m]	111.61	109.35

Convergence analysis



Meshes:



n=4 n=8 n=16

11. References

1. **Bathe, K. J., Wilson, E. L.,** *Numerical Methods in Finite Element Analysis*, Prentice Hall, New Jersey, 1976
2. **Bojtár I., Vörös G.**, *A végeselem-módszer alkalmazása lemez- és héjszerkezetekre*, Műszaki Könyvkiadó, Budapest, 1986
3. **Chen, W. F., Lui, E. M.**, *Structural Stability*, Elsevier Science Publishing Co., Inc., New York, 1987
4. **Hughes, T. J. R.**, *The Finite Element Method*, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1987
5. **Owen D. R. J., Hinton E.**, *Finite Elements in Plasticity*, Pineridge Press Limited, Swansea, 1980
6. **Popper Gy., Csizmás F.**, *Numerikus módszerek mérnököknek*, Akadémiai Kiadó · Typot$_e$x, Budapest, 1993
7. **Przemieniecki, J. S.**, *Theory of Matrix Structural Analysis*, McGraw Hill Book Co., New York, 1968
8. **Weaver Jr., W., Johnston, P. R.**, *Finite Elements for Structural Analysis*, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1984
9. **Dr. Szalai Kálmán**, *Vasbetonszerkezetek, vasbeton-szilárdságtan*, Tankönyvkiadó, Budapest, 1990. 1998
10. **Dr. Kollár László:** *Vasbeton-szilárdságtan*, Műegyetemi Kiadó, 1995
11. **Dr. Kollár László:** *Vasbetonszerkezetek I., Vasbeton-szilárdságtan az Eurocode 2 szerint*, Műegyetemi Kiadó, 1997
12. **Dr. Bölcskei E., Dr. Dulácska E.:** *Statikusok könyve*, Műszaki Könyvkiadó, 1974
13. **Dr. Dulácska Endre:** *Kisokos, Segédlet tartószerkezetek tervezéséhez*, BME Építészmérnöki Kar, 1993
14. **Porteous, J., Kermani, A.,** *Structural Timber Design to Eurocode 5,* Blackwell Publishing, 2007
15. **Dulácska Endre, Joó Attila, Kollár László:** *Tartószerkezetek tervezése földrengési hatásokra,* Akadémiai Kiadó, 2008
16. **Pilkey, W. D.,** *Analysis and Design of Elastic Beams - Computational methods,* John Wiley & sons, Inc., 2002
17. **Navrátil, J.,** *Prestressed Concrete Structures,* Akademické Nakladatelství Cerm®, 2006
18. **Szepesházi Róbert:** *Geotechnikai tervezés (Tervezés Eurocode 7 és a kapcsolódó európai geotechnikai szabványok alapján),* Business Media Magyarország Kft., 2008
19. **Györgyi József:** *Dinamika*, Műegyetemi Kiadó, 2003
20. **Bojtár Imre, Gáspár Zsolt:** *Végeselemmódszer építőmérnököknek***,** Terc Kft., 2003
21. **Kollár Lajos:** *A mérnöki stabilitáselmélet különleges problémái,* Akadémiai Kiadó, 2006
22. **Cholnoky Tibor:** *Rugalmasságtan II. rész,* Tankönyvkiadó, 1969
23. **A. H. Varma.** *Structural Stability and Design,* Purdue University https://view.officeapps.live.com/op/view.aspx?src=https%3A%2F%2Fengineering.purdue.edu%2F~ah-varma%2FCE%2520579%2FSpring%25202005%2FCE%2520579_Lecture6_Jan31.ppt&wdOrigin=BROWSELINK
24. **G. Vörös.** *Free vibration of thin walled beams,* Periodica Polytechnica, Mechanical Engineering, 2004 https://www.researchgate.net/publication/228573555_Free_vibration_of_thin_walled_beams
25. **Wilson, E. L.,** *Three-Dimensional Static and Dynamic Analysis of Structures – A Physical Approach With Emphasis on Earthquake Engineering*, Computers and Structures Inc., Berkeley, 2002
26. **E. J. Sapountzakis.** *Bars under Torsional Loading: A Generalized Beam Theory Approach,* International Scholarly Research Notices, 2013 https://www.hindawi.com/journals/isrn/2013/916581/
27. **Paz,M., Leigh, W.,** *Structural Dynamics - Theory and Computation*, Fifth Edition, Springer, 2004
28. **Chopra, A. K.,** *Dynamics of Structures - Theory and Applications to Earthquake Engineering*, Third Edition, Pearson Prentice Hill, 2007
29. **Biggs, J. M.,** *Introduction to Structural Dynamics*, McGraw-Hill, 1964
30. **Weaver, W.,** *Jr., P. R. Johnston, Structural Dynamics by Finite Elements*, Prentice-Hall, 1987
31. **Bathe, K. J.,** *Finite Element Procedures,* Prentice-Hall, 1996
32. **Borst, R., Crisfield, M. A., Remmers, J. J. C., Verhoosel, C. V.,** *Non-Linear Finite Element Analysis of Solids and Structures*, Second Edition, John Wiley & Sons Ltd., 2012

33. **Willford, M.R., Young, P.** *A Design Guide for Footfall Induced Vibration of Structures, Concrete Society, 2006*
34. **Smith, A. L., Hicks, S. J., Devine, P. J.** *Design of Floors for Vibration: A New Approach, The Steel Construction Institute, Ascot, 2009*
35. https://en.wikipedia.org/wiki/Von_Mises_yield_criterion
36. https://en.wikipedia.org/wiki/Bresler_Pister_yield_criterion
37. https://en.wikipedia.org/wiki/Whittaker%E2%80%93Shannon_interpolation_formula
38. https://en.wikipedia.org/wiki/Nyquist%E2%80%93Shannon_sampling_theorem
39. **C. Basaglia, D. Camotim, N Silvestre**. *Torsion warping transmission at thin-walled frame joints: Kinematics, modelling and structural response, Journal of Constructional Steel Research (69), 2012*
40. **P. C. J. Hoogenboom, A. Borgart.** *Method for including restrained warping in traditional frame analyses,* ResearchGate, 2005 https://www.researchgate.net/publication/27347413_Method_for_including_restrained_warping_in_traditional_frame_analyses
41. **A. F. Hughes, D. C. Iles, A. S. Malik.** *Design of Steel Beams in Torsion in Accordance with Eurocodes and the UK National Annexes,* The Steel Construction Institute, 2011 https://www.steelconstruction.info/images/6/6f/Sci_p385.pdf
42. **Csellár Ödön, Dr. Halász Ottó, Réti Vilmos:** Vékonyfalú acélszerkezetek, Műszaki Könyvkiadó, 1965
43. **Vörös Gábor:** Merevített lemez és héj elemek méretezése, mechanikai vizsgálata; MTA doktori értekezés, 2008 http://real-d.mtak.hu/320/1/V%C3%B6r%C3%B6s_G%C3%A1bor.pdf
44. **E. Riks:** *An incremental approach to the solution of snapping and buckling problems,* International Journal of Solids and Structures, 15(7), pp. 524–551, 1979. https://www.sciencedirect.com/science/article/abs/pii/0020768379900817
45. **H.R. Ronagh, M.A. Bradford, M.M. Attard:** *Nonlinear analysis of thin-walled members of variable cross-section. Part I: Theory,* Computers and Structures, 77, pp. 285-299, 2000 https://www.sciencedirect.com/science/article/abs/pii/S0045794999002230
46. **Federal Emergency Management Agency:** *Prestandard and Commentary for the Seismic Rehabilitation of Buildings* https://www.nehrp.gov/pdf/fema356.pdf