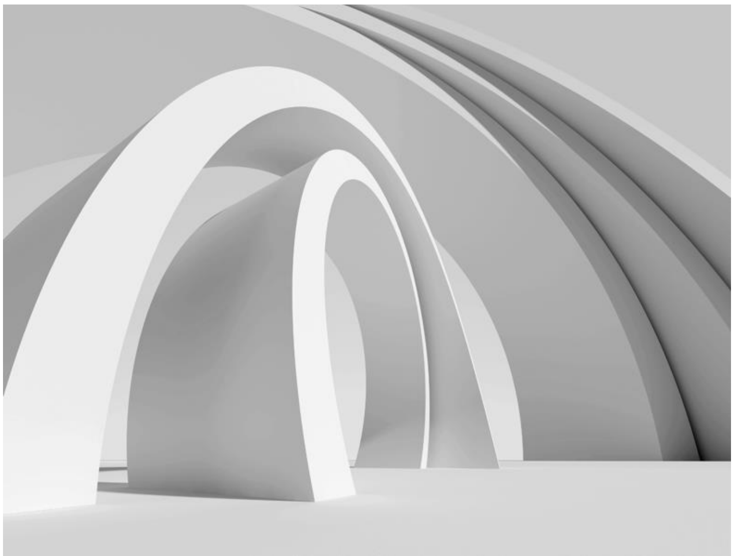


User's Manual

AXISVM 13

Finite Element Analysis & Design Program

Inter-CAD Kft. 2016

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.

C O N T E N T S

1. New features of Version 13

General

More efficient hardware graphics acceleration methods for faster rotation and zooming.	*3.3.11 Preferences / Colours*
New methods to rotate the model (rotating around the center of the bounding box of on-screen elements, rotating around a selected point), optional display of the rotation center symbol.	*2.16.2 Zoom icon bar*
Compressed AXS model file. The average compressed size is about 10% of the original.	*3.3.11 Preferences / Data integrity*
Keyboard shortcut editor	*3.3.12 Keyboard shortcuts*
Further color and pen width options for 3D object contours, steel and timber design members.	*3.3.11 Preferences / Graphic symbols*
Large icons, customizable scaling of dialog windows for high pixel density monitors	*3.3.11 Preferences / Dialog windows*
Faster way to increase / decrease label font size	*3.5.7 Changing label font size*
Property filter of the Selection toolbar displays cross-sections in alphabetical order.	*2.16.1 Selection*
New categories within logical parts. Logical parts can be created from steel or timber optimization groups, eccentricity groups (see Tapered and eccentric domains in the Elements chapter) and along selected gridlines.	*2.16.14 Parts 4.9.5 Domain 6.6.1.1 Steel cross-section optimization 6.7.1 Timber cross-section optimization – TD9 module*
Index and name of materials used in the model can be set to appear in bold in Table Browser	*4.9.1 Material*
Unused materials and references can be deleted in the Table Browser	*4.9.1 Material 4.9.20 References*
PS1: Tendon geometry can be imported from Clipboard	*4.10.26 Tensioning - PS1 module*

Editing

New option to keep the view unchanged in undo operations	*3.3.11 Preferences / Editing*
New parametric thick-walled cross-sections (haunched shapes, trapezoid, rounded rectangle)	*3.1.14.1 Cross-Section Editor*
Radius and arc length dimensioning for cross-sections	*3.1.14.1 Cross-Section Editor*
Layers can be locked to prevent editing.	*3.3.3 Layer Manager*
Enhanced intersecting function for lines allows selecting element types to intersect.	*4.8.8 Intersect*
Display option to show only walls and columns from the neighbouring story. Default column or wall height is set to the level height when drawing on a story.	*3.3.4 Stories*
Stories can be renamed.	*3.3.4 Stories*
Re-importing an IFC file opens a dialog to review changes (new, modified or deleted objects).	*3.1.6 Import*
Walls can be drawn directly along existing lines or lines on the background layer by simply clicking on the lines.	*4.9.3 Direct drawing of objects*
New REV module to import models from Autodesk Revit 2015 or 2016 into AxisVM and convert it to a structural model	*3.1.6 Import*

Elements

Parametric ribbed slabs	4.9.5.4 Parametric ribbed plates
One-way or two-way tapered and eccentric domains. Eccentricity groups to align top or bottom of domains.	4.9.5 Domain
Colour coding for domain eccentricity, eccentricity groups, surface supports.	2.16.5 Color coding
Modeling of XLAM (cross-laminated) timber panels (XLM)	4.9.5.5 XLAM domain
Virtual beams and strips to reduce domain surface forces to beam results	2.16.16 Virtual beams
List of cross-sections can be ordered by parameter values in the Cross-section Library import dialog	4.9.2 Cross-section

Loads

Referential point load on beam and rib elements	4.10.4 Concentrated load on beam
Automatically generated snow and wind load cases can be converted to regular (and editable) load cases.	4.10.1 Load cases, load groups
Snow loads: Function to select roof edges with overhanging snow.	4.10.13 Snow load – SWG module
Wind loads: Effects of protruding roofs (pressure on the underside of the overhang).	4.10.14 Wind load – SWG module
The enhanced load panel distributes point, line and surface loads over the elements under the load panel, load transfer can be limited to selected elements	4.10.5 Point load on domain or load panel 4.10.8 Domain / Load panel line load 4.10.9 Surface load 4.10.12 Load panels
Conversion of loads distributed from load panels to individual loads	4.10.12 Load panels
Colour coding of surface load intensities.	2.16.5 Color coding
If spectra are different in X and Y directions, different q_{dX} and q_{dY} can be specified	4.10.23 Seismic loads – SE1 module

Analysis

Plastic behaviour is followed using a discretized section instead of the Ilyushin model.	3.1.13 Material Library
Nonlinear analysis can take into account reinforcement both in surface and line elements (beams/columns). Pushover analysis can take into account reinforcement both in surface and line elements (beams/columns).	5.1 Static analysis

Results

The plane used to display beam / rib axial force and torsion moment diagrams can be selected (local x-z or local x-y plane)	2.16.19.3 Drawing
Table display options dialog automatically excludes result componens from finding extremes if their respective columns are hidden.	6.1.5 Result tables

Design

New tool to define actual reinforcement in line elements against uniaxial bending (beam) without entering the beam design.	*6.5.3 Beam reinforcement parameters (uniaxial bending)*
New tool to define actual reinforcement in line elements against biaxial bending (column) without entering the column design	*6.5.4 Actual reinforcement against biaxial bending (column)*
It is possible to edit and check actual beam reinforcement	*6.5.10.3 Checking actual beam reinforcement*
Beam reinforcement parameters and actual reinforcement can be defined without designing the beam, actual reinforcement can be taken into account in nonlinear analysis	*6.5.3 Beam reinforcement parameters*
Reinforced concrete column check calculates efficiencies	*6.5.9 Column reinforcement – RC2 module*
Table of domain reinforcement parameters	
Renumbering/renaming of steel and timber design members	*2.16.13 Renaming/renumbering*
Steel design, Flexural buckling: Different buckling length definition methods: Buckling factor, Buckling length, Auto (automatic buckling length calculation for double symmetric cross-sections). *Steel design, Lateral-torsional buckling: Lateral support conditions can be reviewed and controlled for automatic calculation of Mcr.*	*6.6.1 Steel beam design according to Eurocode 3 – SD1 module*
Footing design module allows to turn on/off and customize checks performed in the design process.	*6.5.12 Footing design – RC4 module*
Stress calculation for XLAM panels (XLM)	
Timber design calculations	*6.7 Timber beam design – TD1 module*
Timber design optimization (TD9)	*6.7.1 Timber cross-section optimization – TD9 module*

Reports

Multipage preview	*2.10.2 Report*
Company logo can be inserted into report header and placed on the cover page.	*3.3.11 Preferences / Report*
Nonlinear, dynamic, vibration and buckling results in report templates	*2.10.3.2 Editing a template*
New option in Table Browser to display cross-section name instead of (or beside) the index of cross-section	

This page is intentionally left blank.

2. How to use AxisVM

Welcome to AxisVM!

AxisVM is a finite-element program for 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.

Preprocessing

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 linearly 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

8 GB RAM
50 GB of free hard disk space
DVD drive
17"color monitor (or larger), at least 1280x1024 resolution
A dual or multi-core processor over 2 GHz
Windows7 / Windows8 operating system
Mouse or other pointing device
Windows compatible laser or inkjet printer

Minimal configuration

2 GB RAM
10 GB of free hard disk space
DVD drive
15"color monitor, at least 1024x768 resolution
Mouse
Windows XP with SP3

Supported operating systems

Windows 10, Windows 8, Windows 7, Windows Vista, Windows XP/ SP3
Windows Server 2008, Windows Server 2003/SP1
Both 32 and 64 bit operating systems are supported.

Memory access 64 bit and 32 bit versions

To reach more memory is very important as it speeds up the analysis considerably.
The native **64 bit version of AxisVM13** runs only on 64 bit operating systems. It has direct access to the physical memory so no further settings are required.
The **32 bit version of AxisVM13** runs on both 32 and 64 bit operating systems. It has direct access to the lower 2 GB of the physical memory.

2.2. Installation

Software Protection

The program is protected by a hardware key. Two types of key are available: parallel port (LPT) keys and USB keys.

☞ ***Plug the key only after installation is complete, because certain operating systems try to recognize the plugged device and this process may interfere with the driver installation.***

Non-network drivers will be automatically installed. If you encountered problems you can install this driver later from the DVD.
Run the Startup program and select *Reinstall driver*.

Standard Key

First install the program then plug the key into the computer.

Network Keys

If you have a network version 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.

AxisVM program with a network licence is shipped with an USB **Sentinel Super Pro** network dongle.

1. Insert the AxisVM DVD in the DVD-ROM drive of the AxisVM server. Run [DVD Drive]: \ Startup.exe. Select *Reinstall driver.* This type of network key requires at least a 7.1 driver. DVD contains the 7.6.6 version of the driver.
2. Connect the key to the parallel or USB port of one of the computers. This way you select the AxisVM server.

The installed network key server runs automatically after startup.

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 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.***

Installation

AxisVM runs on XP / Vista / Windows 7/Windows 8 operating systems.
Insert the AxisVM DVD into the DVD 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 on your AxisVM DVD. Select *AxisVM 13 Setup* and follow the instructions.
Installer will suggest installation of the 64 bit version on 64 bit operating systems but the 32 bit version is also available. 64 bit version cannot be installed on 32 bit operating systems.

Installation under Vista/Windows 7/Windows 8 Operating System:

- You need the latest Sentinel driver. You can download it from *www.axisvm.eu - Service/downloads – Latest release updates / Sentinel Driver 7.6.6.* (at the bottom of the left window)

Right click on the program icon with 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

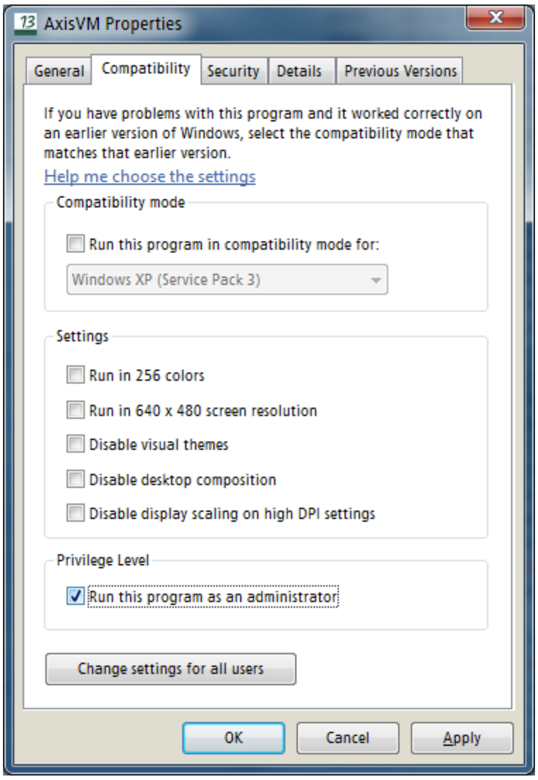
`C:\ AxisVM13`
and
`C:\ AxisVM13\Examples`

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\AxisVM1**3** folder (C:\Program Files (x86)\AxisVM1**3** under 64-bit operating systems). However in this case the *Run as administrator* property must be set for `AxisVM.exe`, `AxisVM_x64.exe` `\LTBeam\LTBeam.exe` and `\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\AxisVM1**3** folder (see *Permissions* under the *Security* tab).
On 64 bit operating systems the user can choose to install either the 32 bit or the 64 bit version of AxisVM. Installing the 64 bit version also copies the 32 bit version to the hard disk but no shortcut is created on the desktop for this file. If the **x64** module is not present in the configuration the 32 bit version will be launched instead.

On 32 bit operating systems only the 32 bit version is installed.

☞ ***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.***



On Windows 7, Windows 8 or Windows 10

- Click the right mouse button over the AxisVM shortcut on the Desktop.
- Select *Properties* from the popup menu.

Go to the *Compatibility* tab, find the *Privilege Level* group box and check *Run this program as an administrator*

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.***

False virus alarms

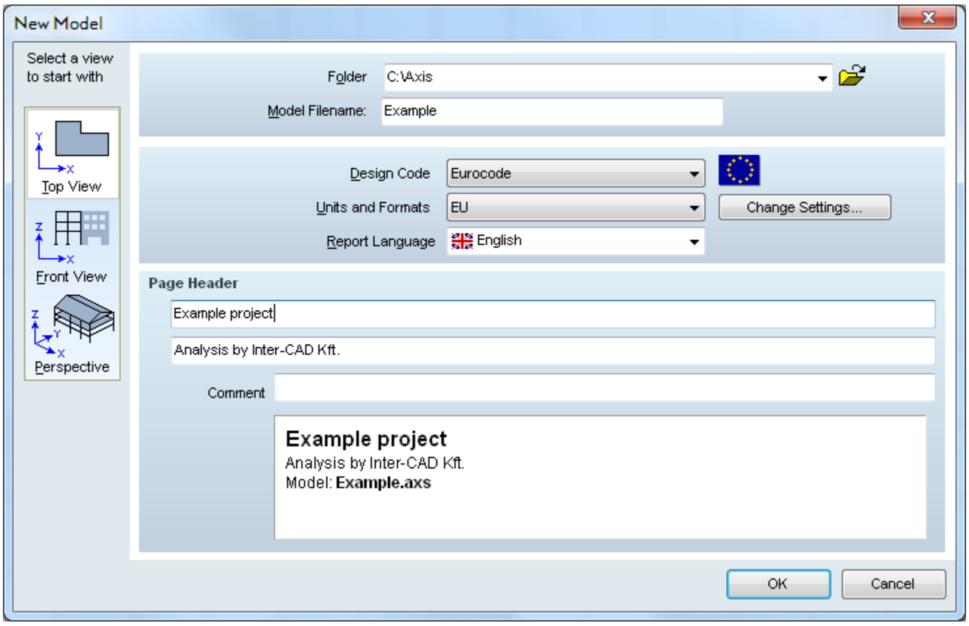
Certain antivirus products running on the PC can send a false alarm during installation. This is caused by heuristic algorithms searching for virus-like activities. These algorithms may detect the operation of the special protection system of AxisVM and send a false alarm. If this happens you can do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

- *If the antivirus product put AxisVM.exe into quarantine restore it*
- *Add AxisVM.exe to the exceptions (files not checked by the 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 AxisVM13 icon.
At startup a splash screen is displayed (see... *3.6.4 About*) 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 strartup* checkbox.
When setting up a new model the following dialog is displayed.



Running AxisVM in safe mode

Both AxisVM.exe (32 bit version) and AxisVM_x64.exe (64 bit version) can be started in safe mode entering `axisvm.exe /SAFE` or `axisvm_x64.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

Program and file types

	File type	File extension	can be opened by				
			Standard	Viewer	Academic	Trial	Light
	Standard	.axs/.axe	yes	yes	yes	yes	DEN
	Viewer	.axv/.axw	no	yes	No	no	no
made by	Academic	.axs/.axe	OWM	yes	yes	OWM	DEN
	Trial	.axs/.axe	yes	yes	yes	yes	DEN
	Light	.axd/.axr	yes	yes	yes	yes	yes

OWM open with WaterMarkt
DEN Depends on elements' number in the model (Light 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
	Support	Unlimited
	Gap	Unlimited
	Diaphragm	Unlimited
	Spring	Unlimited
	Rigid	Unlimited
	Link	Unlimited
Load cases		Unlimited
Load combinations		Unlimited
Frequencies (modal shapes)		99

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

2.3. Getting started

Step-by-step input schemes are presented in the Section 9.
See Example 1 of **Chapter** *10* with a step-by-step input scheme in *9.2 Plane frame model*

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.
In **Chapter** *9*, the step-by-step modeling of a few typical structures are presented.

The following types of structures are shown:

1. Plane truss girder
2. Plane frame
3. Plate structure
4. Membrane cantilever
5. Seismic analysis

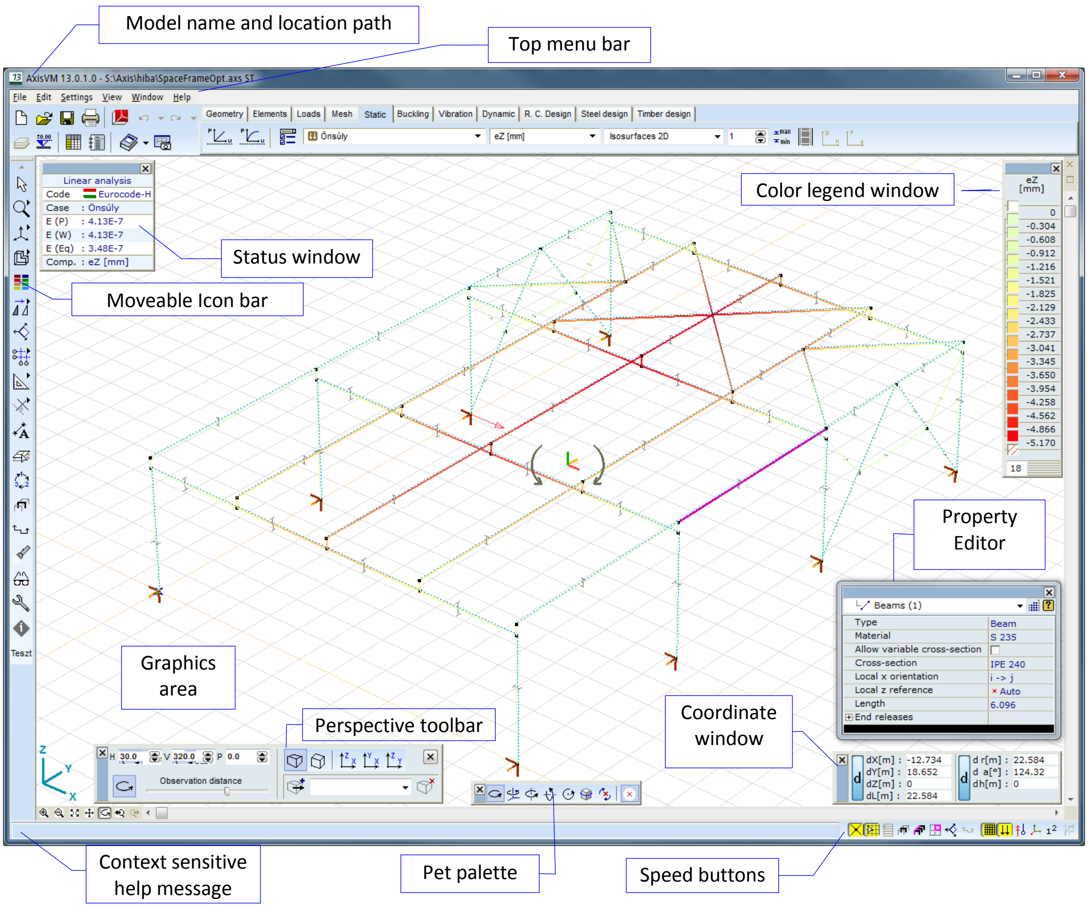
Understanding of these simple models will allow you to easily build more complex models.
It is recommended that you read the entire User's Manual at least once while exploring AxisVM.
In **Chapter** *1* you can find the timely, new features of the version.
Chapter *2* contains general information about 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

After you start AxisVM a screen similar to the following picture appears:



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, it can appear as a pick-box, crosshairs with pick-box, or pointer.

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 sub-menu will appear.

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

Icon bar The icons 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...** *4.8.17 Modify, transform*

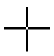
Speed buttons Speed buttons in the bottom right provide the fastest access to certain switches (parts, sections, symbols, numbering, workplanes, etc.)

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 are stored in two files: the input data in the **filename.axs** and the results in the **filename.axe** file.
AxisVM checks if AXS and AXE files belong to the same version of the model.

2.5. Using the cursor, the keyboard, the mouse

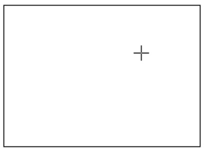
Unicode is a computing industry standard for the consistent encoding, representation and handling of text expressed in most of the world's writing systems. AxisVM 13 provides full Unicode support.
All windows appear according to the current Windows theme.

Graphics cursor

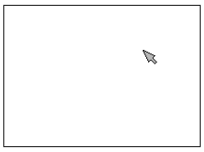


As you move your mouse, the graphics cursor symbol tracks the movement on the screen. To select an entity, an icon or menu item, move the cursor over it and click the left mouse button. The shape of the cursor will change accordingly (**see...** *4.7.1 Cursor identification*), and will appear on the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

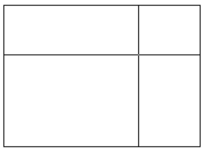
Crosshairs:



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 tool tip.

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
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	efficiency results and resistances
Timber Design	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 $n \cdot \Delta\alpha$, custom α or $\alpha + n \cdot 90°$.
[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 AxisVM13 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
  Stories ......................................... 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
    Teherkombinációk ............................... 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
    Stories ........................................ 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 story up .................................. PgUp
    One story 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

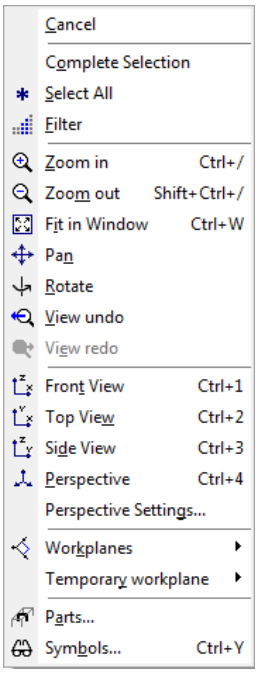
Scroll up	Zoom in
Scroll down	Zoom out
Wheel down + move	Pan (slow)
Wheel down + ALT + move	Rotate
Wheel down + CTRL + move	Pan (fast)

2.7. Quick Menu

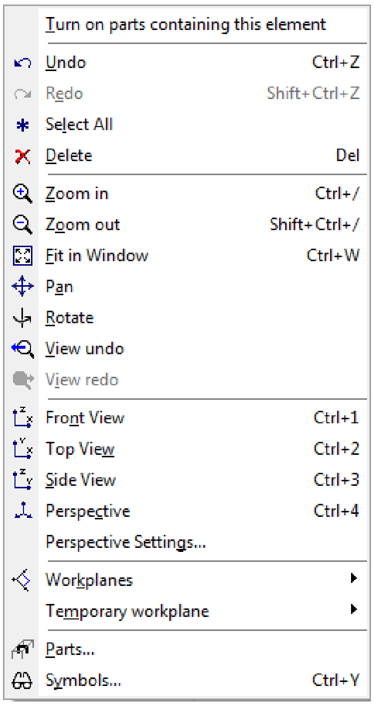
right button

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.

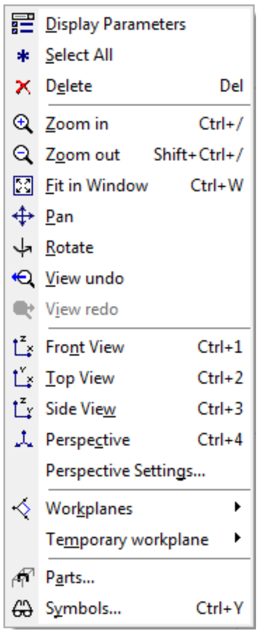
Selection



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.

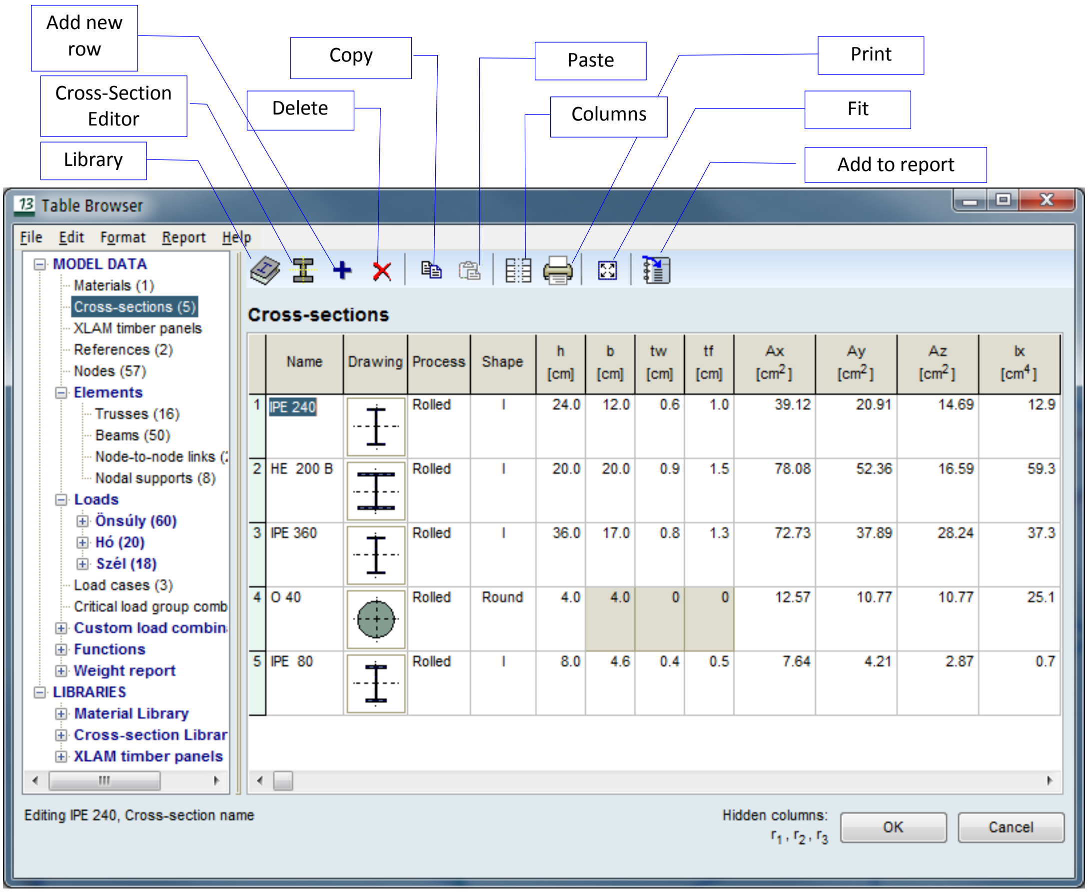
2.9. Table Browser

[F12]

AxisVM uses tables to display numerical information on the screen allowing changes in formatting. The tables operate in 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, input model data is displayed only. While working in the post-processor, 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.



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).

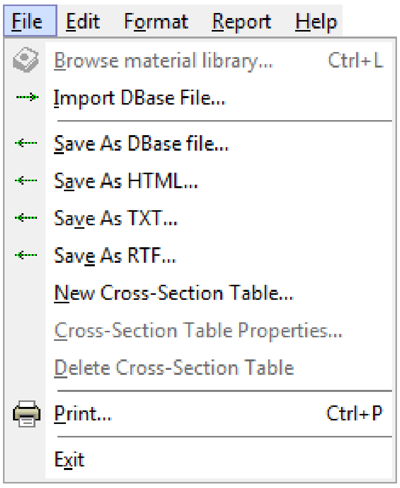
[Ctrl]+ [←] — Moves the focus to the previous (to the left) page of columns (only in tables where more columns can be displayed at the same time).

[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] / **right button** — Aborts the current editing in the edit box.

[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 Library **[Ctrl]+ [L]**

Loads cross-sectional or material data from a library. You can also save the current content of the table in a custom 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-SectionTable 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.

Print **[Ctrl + [P]**

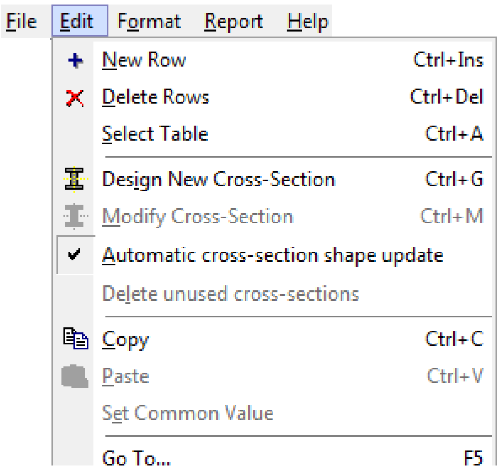
Prints all the information displayed in the table to the selected printer or to a file, with the page header and comment row previously set with the *File/Header* menu command.
Turning on ***Description of table columns*** an explanation of columns appears at the bottom of the table.

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.

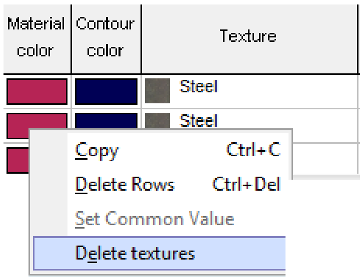
Delete Rows

[Ctrl]+ [Del]

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

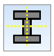
Delete textures

Available only if materials are listed. Removes texture from the selected materials. Available in the popup menu.

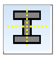


Select Table
[Ctrl]+ [A]

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 update

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

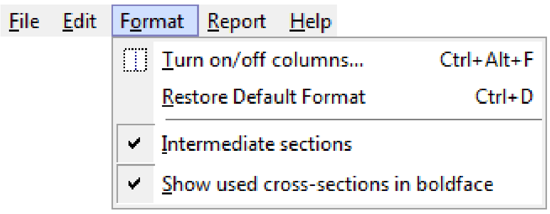
Sets a common value for the selected cells within a column.
Example: you can set the Z coordinate of all nodes to the same value making the model absolutely flat. Available from the *Table Browser Menu / Edit / Set Common Value.*
Also available in the popup menu.

Go to
[F5]

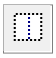
Jumps to a specified row in the table.

Format

During model building



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.

Format Defaults
[Ctrl]+ [D]

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

Order of load cases...

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

Intermediate sections

After dividing or meshing beams or ribs with variable cross-section AxisVM builds up intermediate cross-sections. This menu item is to turn on/off the display of intermediate cross-sections at the end of the list.

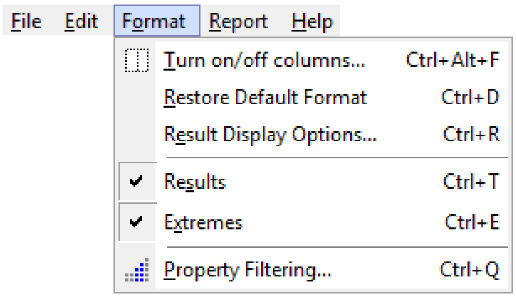
Show used cross-sections in boldface

After the *Delete unused cross-sections* command only the sections in bold will remain in the list.

The cross-section names which are signed by bold letter will remain in the table if the *Delete unused cross-sections* switch is turned on.

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

During result query



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… *6.1.5 Result tables*

Results On/Off **[Ctrl]+[T]**

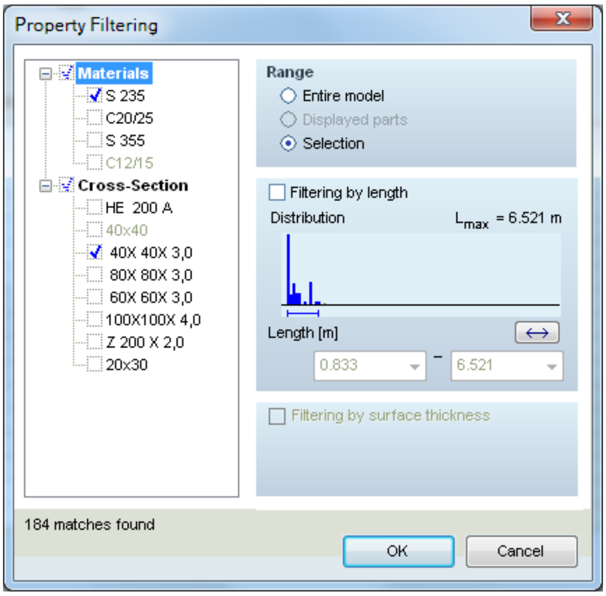
Display of results can be turned on / off.

Extremes On/Off **[Ctrl]+[E]**

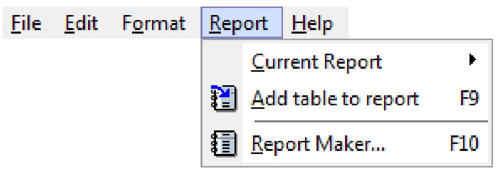
Display of extremes can be turned on / off.

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.***

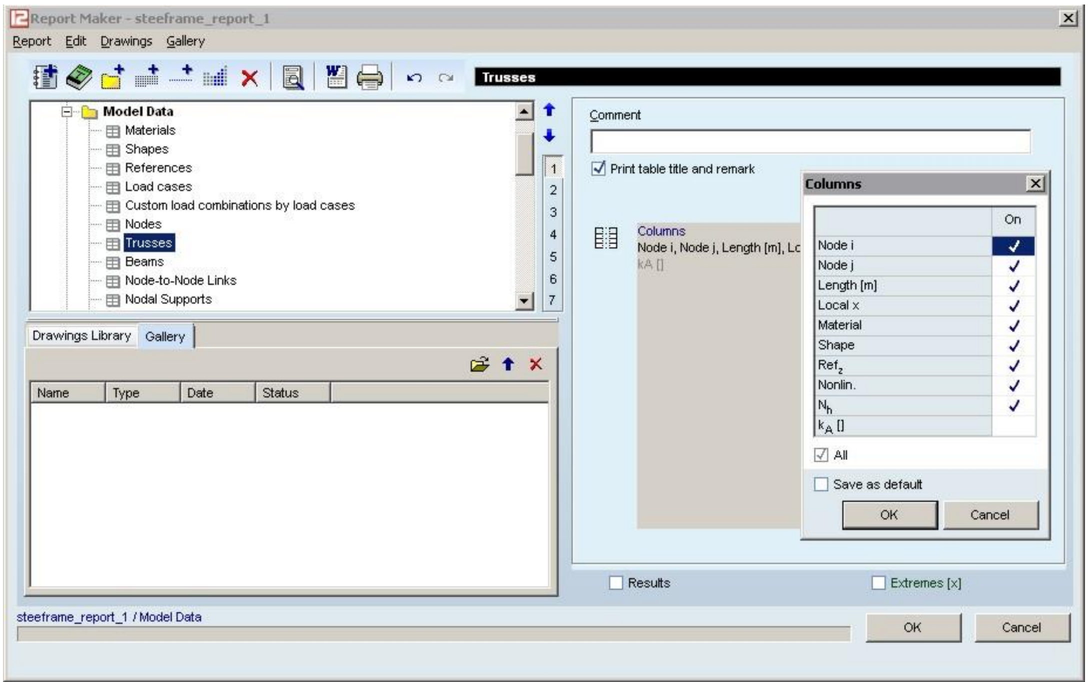
2.10. Report Maker

[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 are stored in the model file *(*.axs)* and can be printed or saved as a Rich Text Format (RTF) file. RTF files can be processed by other programs (e.g. Microsoft Word).

Tables exported from 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.

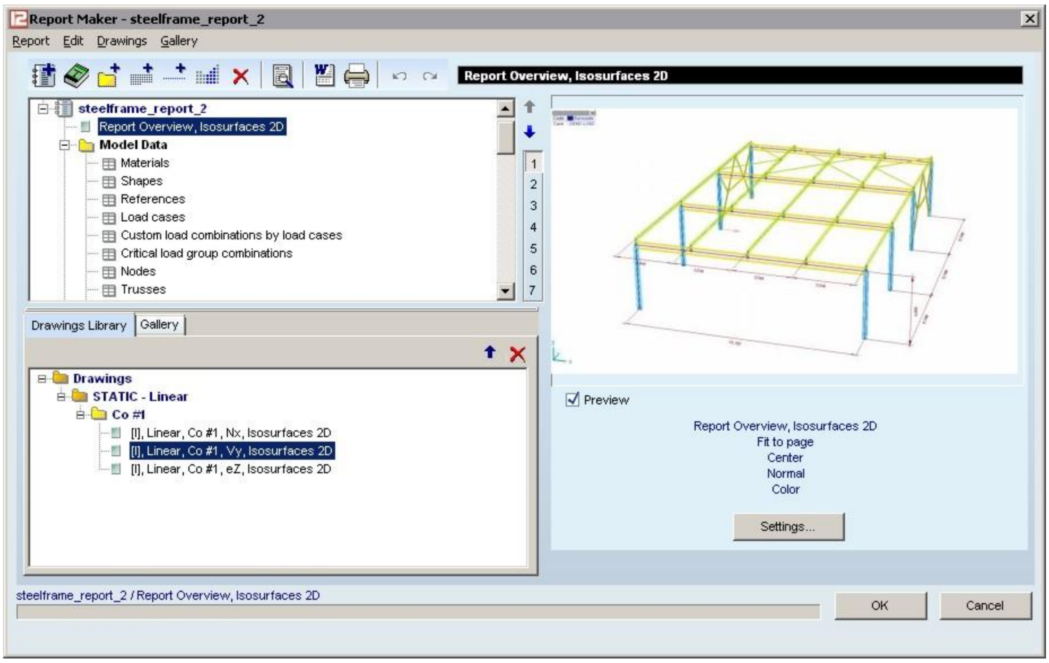


Folder If a folder is selected its name can be edited on the right.

Table If a table is selected, its comment text, column titles and other properties are shown. Display of title, comment and columns can be turned on and off.

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

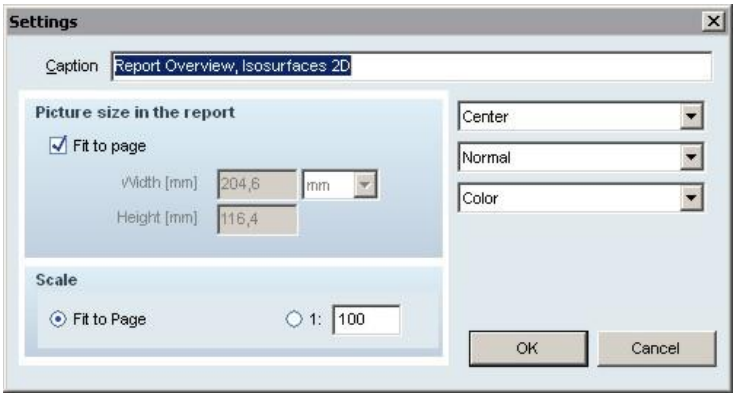
Picture or Drawing If a picture or drawing is selected it is shown on the right. Its size, alignment and caption can be set by clicking the *Settings* button.



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... *3.5.8 Drawings Library*, *3.5.9 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... *3.2.11 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*

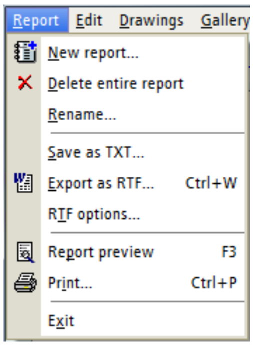
[Ctrl]+[Z]

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.

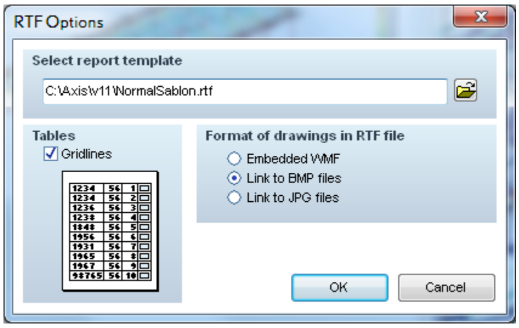
Delete entire report **[Del], [Ctrl]+[Del]** — Deletes the current report (i.e. the report which contains the selected item). Pictures used in the report are not deleted from Gallery.

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.

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 WMF: Drawings are embedded into the file. It improves portability but can result in huge file size.

Link to BMP, JPG: **This option** keeps the RTF file smaller as drawings are stored in external files. Drawings appear only if pictures are located in an *Images_modelname* subfolder relative to the folder of the RTF file.
Gridlines of exported tables can also be turned on/off.

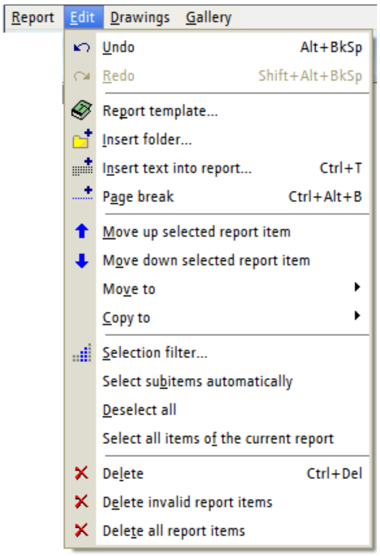
Report preview **[F3]** — Displays a print preview dialog. You can set the zoom factor between *10%* and *500%* (*Page Width* and *Full Page* is also an option). Click the buttons or use the keyboard to move backward and forward between pages (**[Home]** = first page **[⟵]** = previous page, **[⟶]** = next page, **[End]** = last page.
Report preview can display multiple pages. **[PgUp] [PgDn]** steps back and forward according to the number of pages displayed.

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

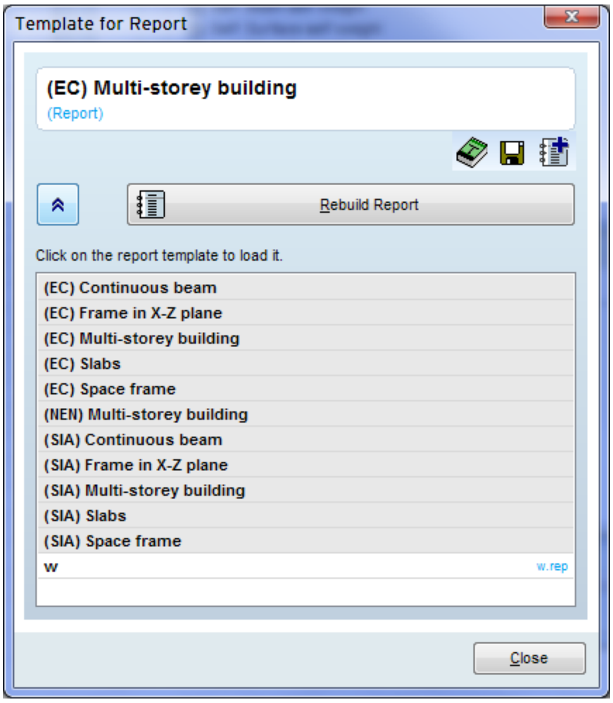
Deletes all items from the current report but does not delete the report itself.

2.10.3.1. Template-based reports

Report templates can be used to generate reports based on certain presets, filters and preferences. Generated reports consist of drawings and tables. Templates can be saved as files and reused to generate 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 (gray 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*

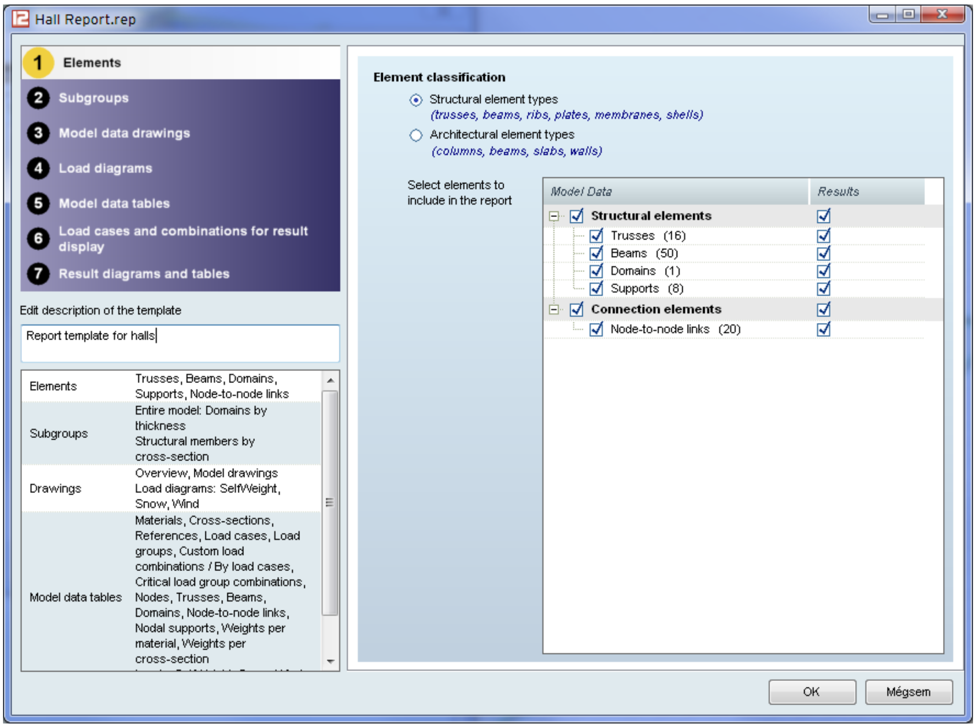
Save this template to a file
The current template can be saved to a file to reuse it in another model. Report templates have a **.rep* extension and are saved to the templates folder described above.

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

2.10.3.2. Editing a template

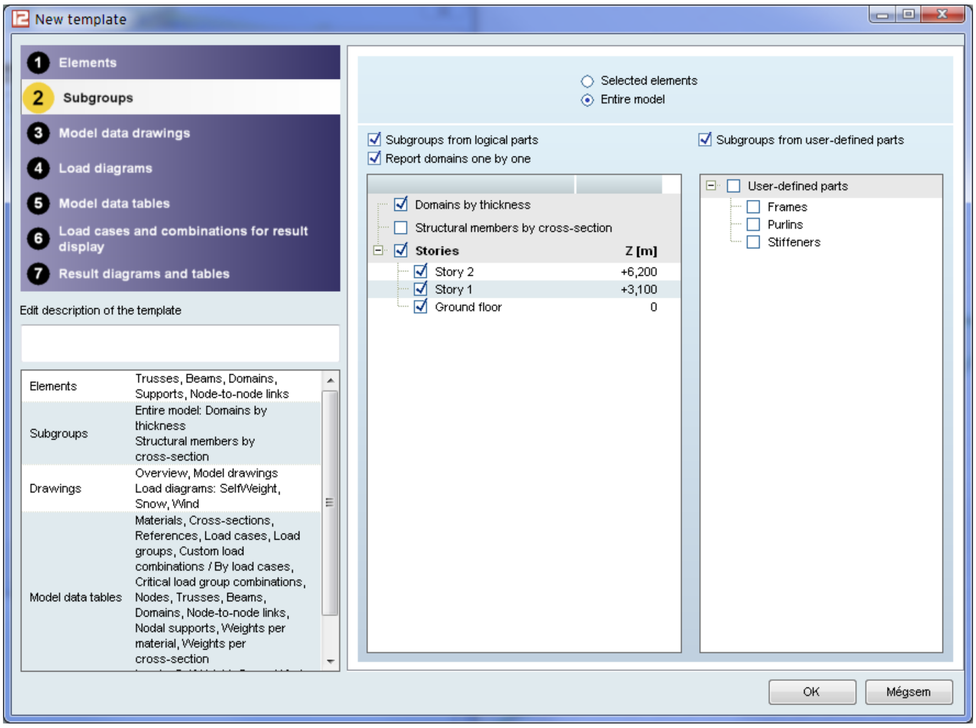
It is a complex task to create a report template. The steps of this process is listed on the left. Clicking on these items we can edit filter options and other settings. An edit box under the list allows entering a name for the template. The template administrator dialog lists templates by their names.

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.

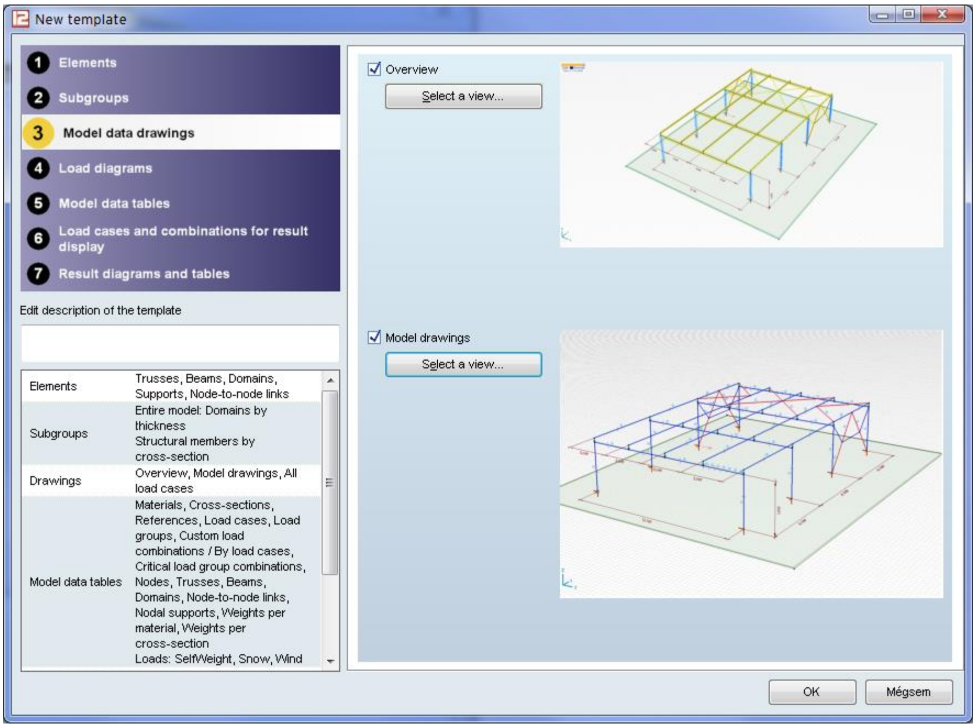
Subgroups



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 stories (only the selected stories will be reported).

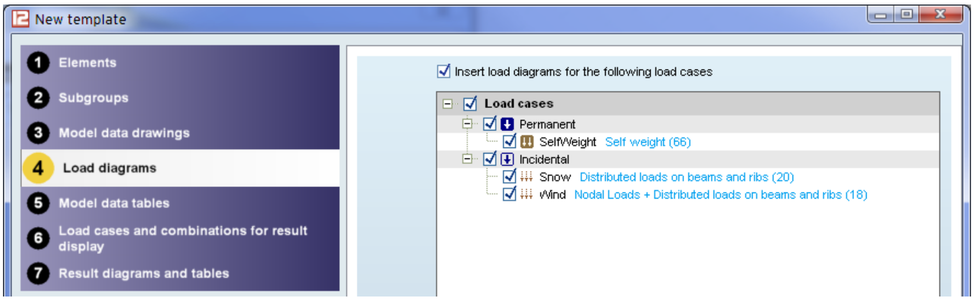
If *Report domains one by one* is checked each domain will be reported separately. Internal domains (being entirely within another domain) are reported with the outer domain even in this case.
If the *Entire model* is selected subgroups can be created also from user defined parts.

Model data drawings



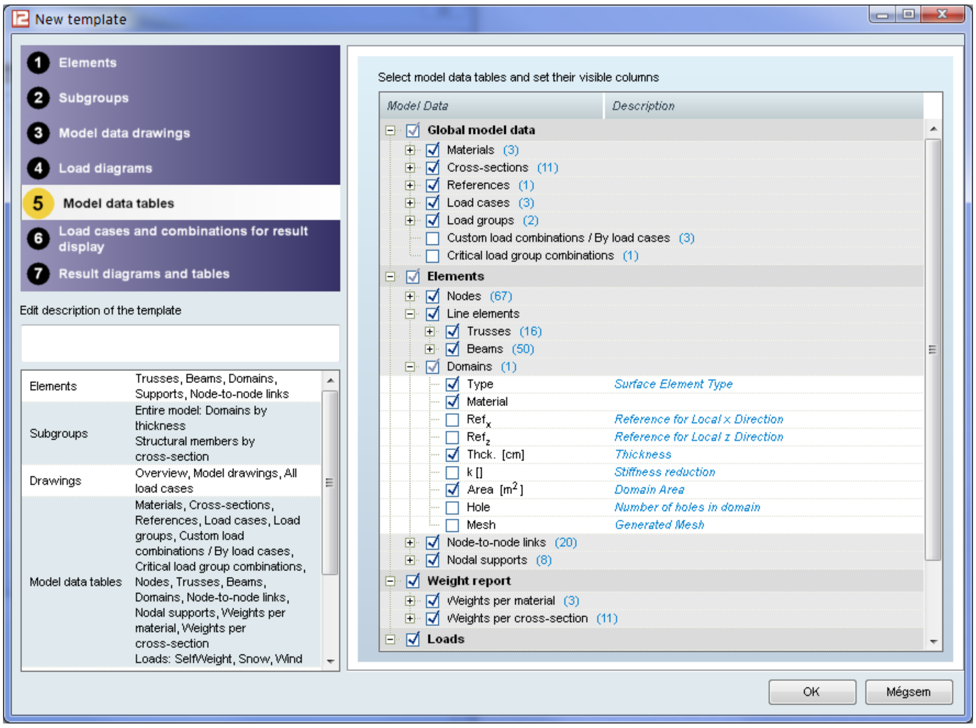
If *Overview* is checked a view of the model will be insterted at the beginning of the report. Click *Select a view...* to choose a view from the Drawings Library.
Check *Model drawings* if you want to include automatically created load and result diagrams. Click *Select a view...* to choose a view from the Drawings Library.
Generated diagrams will inherit all settings of the selected drawing (point of view, status of graphics symbols, numbering, labeling 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.

Load diagrams



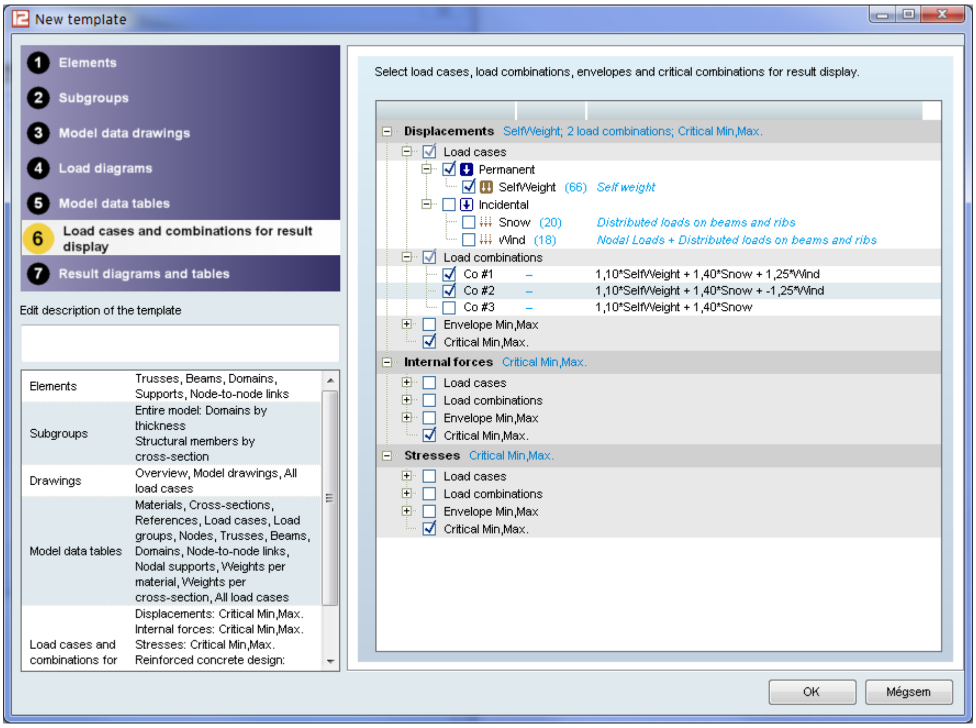
If *Insert load diagrams for the following load cases* is checked, select load cases to add their load diagrams to the report. Load diagrams will be generated from the view set for *Model drawings* in the previous step.

Model data tables



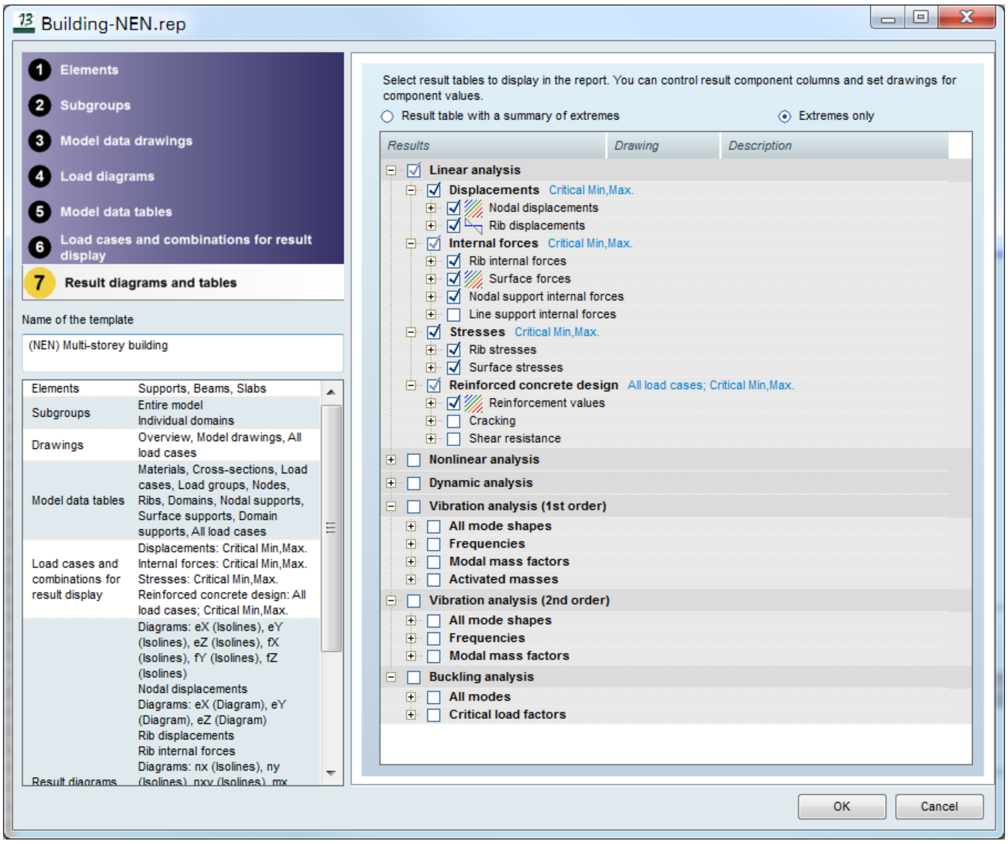
Select model data tables to add to the report. You can set the visible columns for tables to fine tune the report content.
Under *Elements* you will find only those elements you selected in the first step.
Under *Loads* you can select load cases to add their load data tables to the report.

Load cases and combinations for result display



AxisVM provides a huge amount 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

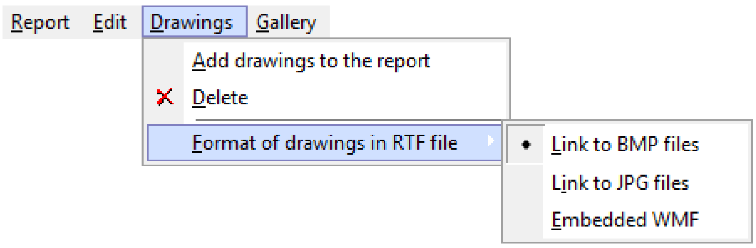


The last step is to select from the possible result tables and control the visibility of their columns. It is also possible to generate result diagrams for result components. Click in the *Drawing* column in a row of a result component. You can choose a drawing mode for that component from the dropdown 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.

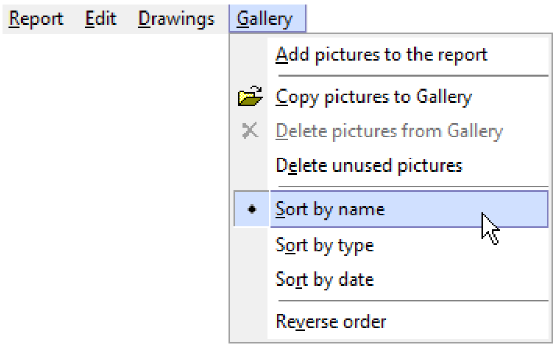
2.10.4. Drawings



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)* and Windows Metafiles *(.WMF, .EMF)* 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 Stories*

2.12. Layer Manager

See in detail... *3.3.3 Layer Manager*

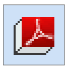
2.13. Drawings Library

See in detail... *3.5.8 Drawings Library*

2.14. Save to Drawings Library

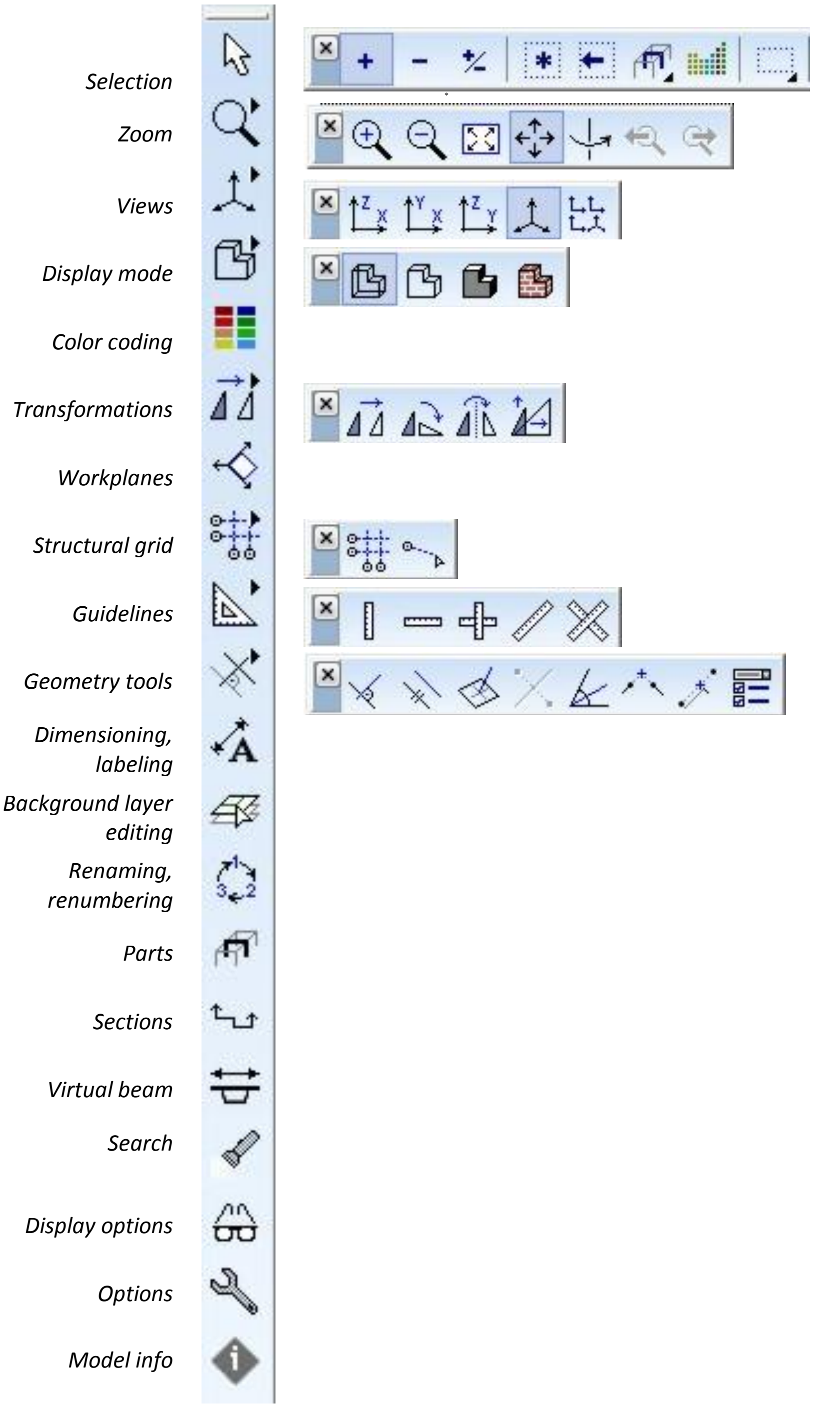
See in detail... *3.5.9 Save to Drawings Library*

2.15. Export current view as 3D PDF



Saves the current vew 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.

2.16. The Icon bar



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.

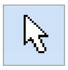
If the Icon bar is horizontal you can dock it at the top or at the bottom. You can change the position and the order of docked toolbars by dragging. In the Cross-Section Editor and in Beam and Column Reinforcement dialogs the Icon bar cannot be docked. Closing a floating Icon bar restores its original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

Dragging and docking of flyout toolbars

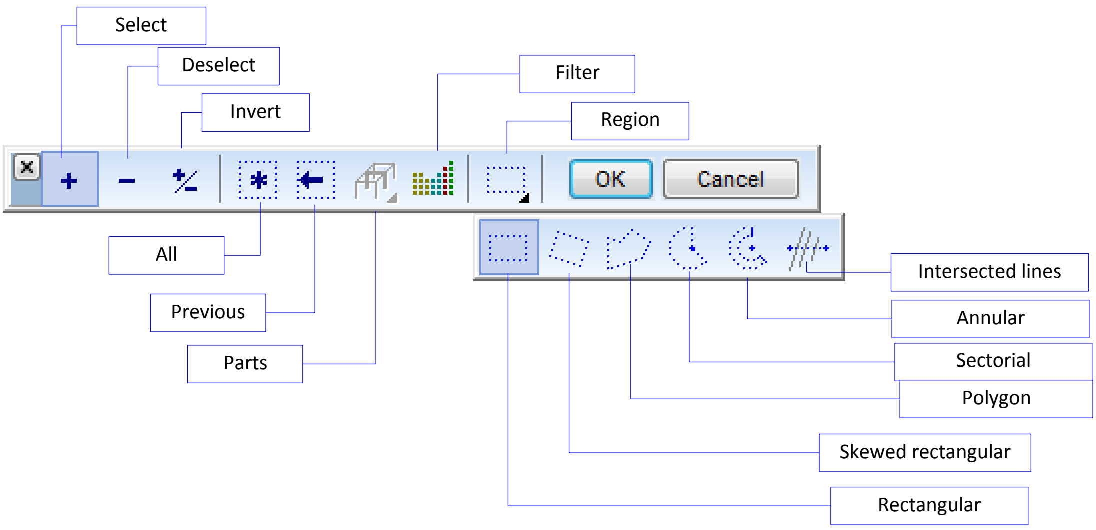
You can also separate flyout toolbars from the Icon bar by dragging their handle. Closing or dragging them back to the Icon bar restores their original position. Floating flyout toolbars can be docked at the top or at the bottom.

☞ ***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** *2.16.14 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.

Selection methods with selection frame:

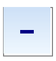
Dragging the selection frame from left to right selects elements entirely within the frame

Dragging the selection frame from right to left selects elements which are not entirely outside the frame

Select

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

Deselect

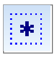


Removes the currently selected entities from the set of selected entities.

Invert

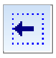
Inverts the currently selected entities' selection status.

All



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

Previous



Restores the previous selection set.

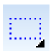
Selection of parts

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).

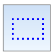
Method

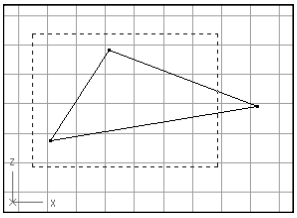


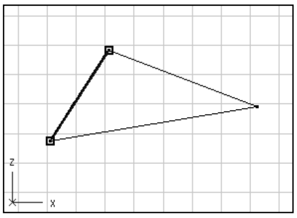
Selects entities using different methods (selection shapes). Rectangular, skewed rectangular, sectorial or ring selection shapes are available. In the followings examples of the application of various selection shapes are provided:

Selection: *Result:*

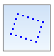
Rectangular

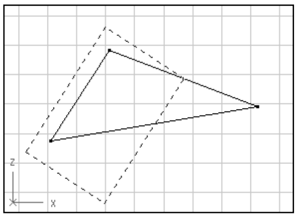


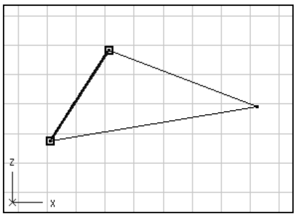




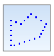
Skewed rectang.

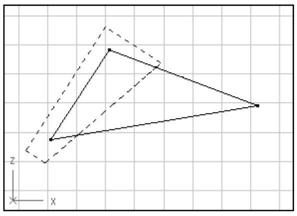


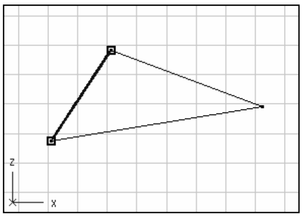




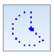
Polyline

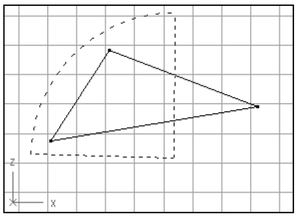


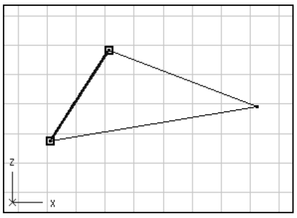




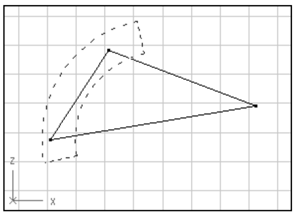
Sectorial

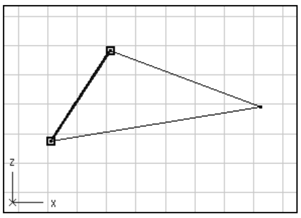




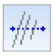


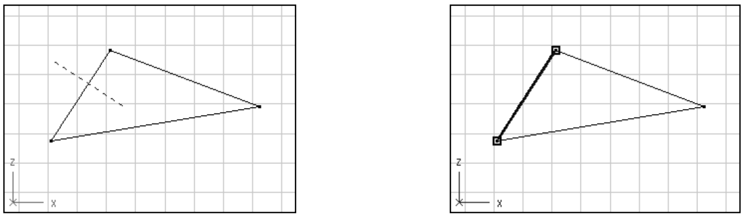
Annular





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.***

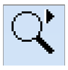
The selected nodes are marked with a surrounding magenta rectangle. Sometimes it is necessary to double-select nodes. In this case these nodes are marked with an additional blue rectangle surrounding them.

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

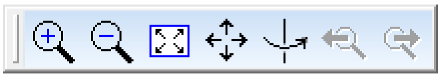
Double selections can be made by pressing and holding the **[Alt]** button while double clicking on the entities with the mouse.

☞ ***During the selection we can modify the apperiance of the structure, we can switch to another view or perspective.***

2.16.2. Zoom icon bar

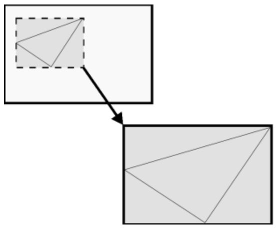


Displays the zoom icon bar.



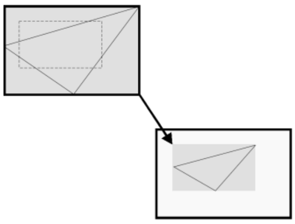
Zoom in

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.

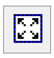


Zoom out

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 mouse while moving the mouse, until the desired position of the drawing is obtained on the screen.

Quick Drag:

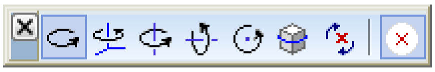
You can use the mid mouse button to drag the model drawing at any time (without the the *Pan* icon).

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.

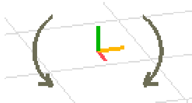
Rotation around the horizontal axis of the screen.

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.

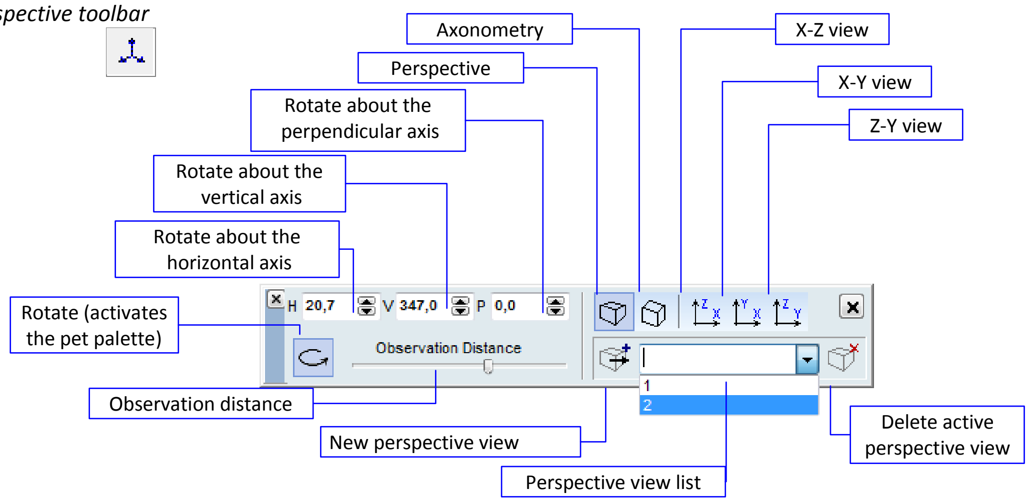
2.16.3. Views

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).

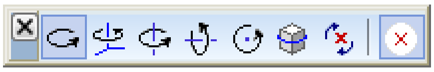
Perspective toolbar



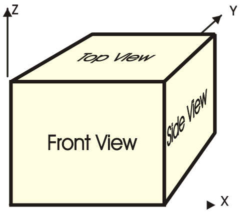
Sets the parameters of the perspective display. The proper view can be set by rotating the model drawing around the three axes, and by setting the observation distance. Rotation angles can be set with a precision of 0.1 degrees. You can assign a name to each setting that you want to save for later use. 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.

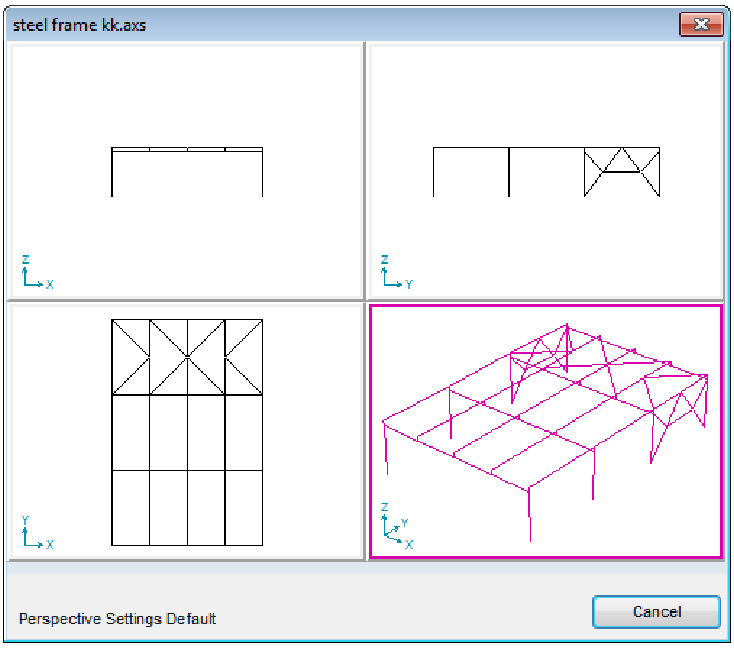
Observation distance Observation distance is the distance between the viewpoint and the centre of the encapsulating block of the model.

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

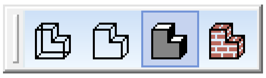


Views, perspective Displays three projection views and the perspective view of the model, and allows you select the view that you want to display. Click the view you want to select.

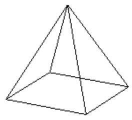




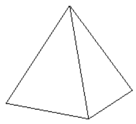
2.16.4. Display mode



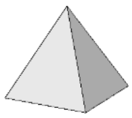
Wireframe:
Displays a wireframe model drawing. In this mode the axis of the line elements and the mid-plane of the surface elements are displayed.



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

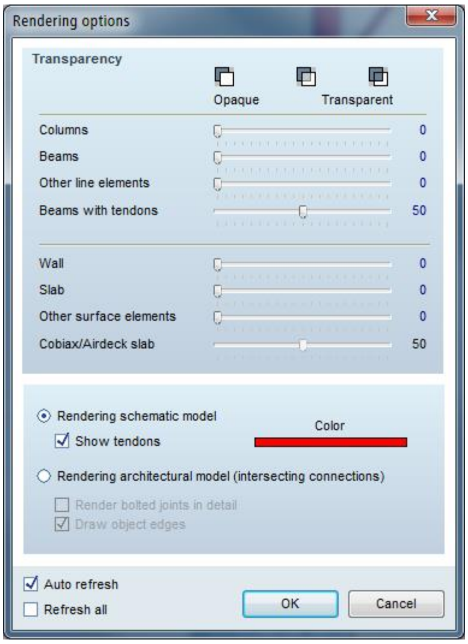


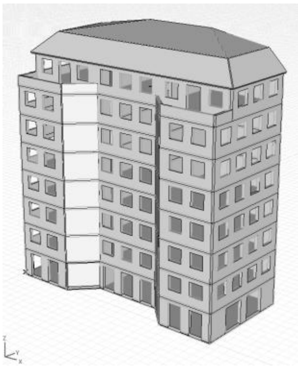
Rendered:
Displays a rendered model drawing. The line elements are displayed with their actual cross-section and the surface elements with their actual thickness. The elements colors are displayed corresponding to colors assigned to their materials. Rendered view is smoother and shows the details of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



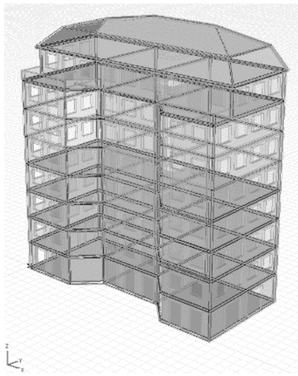
Transparency

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

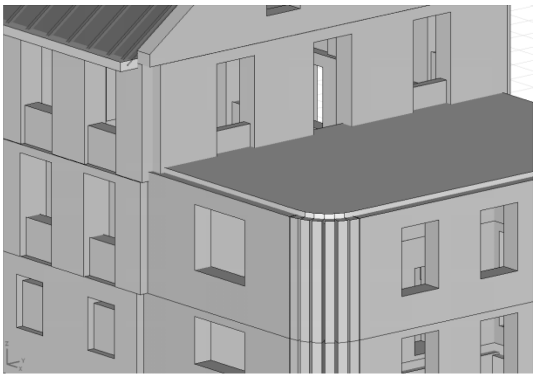
Turning on *Show tendons* a more realistic picture of tensioned beams is drawn. Tendon color can also be set here.

Rendering architectural model

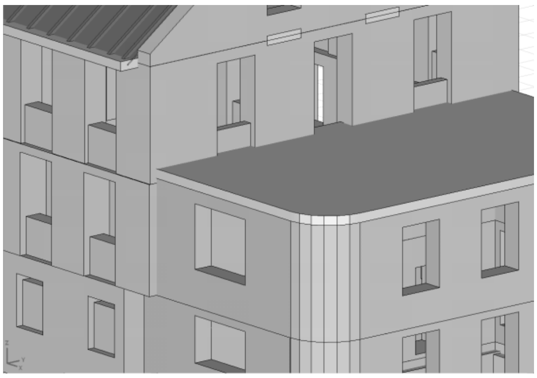
Instead of drawing the structural framework this rendering mode intersects connections getting closer to the final look of the model.

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

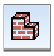
Draw object edges turns on/off object edges.



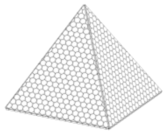
Schematic model

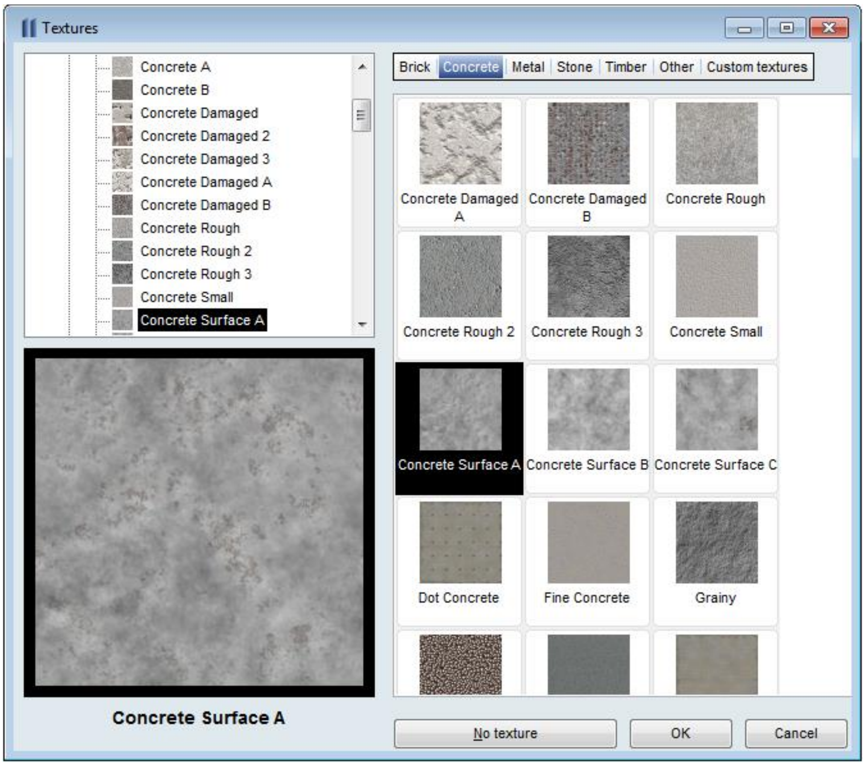


Architectural model



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.

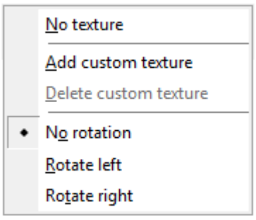




Branches of the tree view on the left and the horizontal list above the texture thumbnails show the material types (brick, concrete, metal, stone, timber, other). The last type (custom) is for the user-defined textures. Textures of the current type are displayed as thumbnails. The selected texture appears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

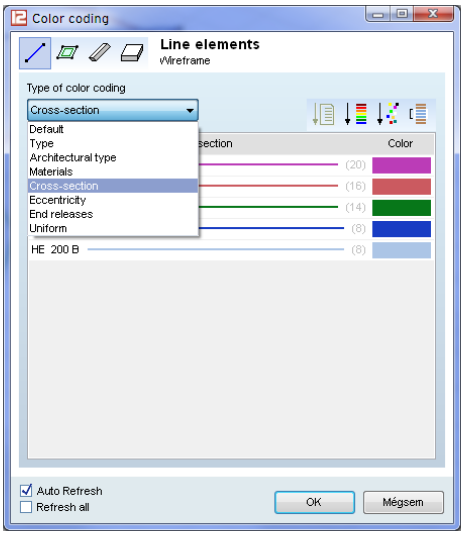
Delete custom texture Predefined textures cannot be deleted from the library, only the assigment can be removed. User-defined textures in the *Custom* category can be deleted.

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 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.

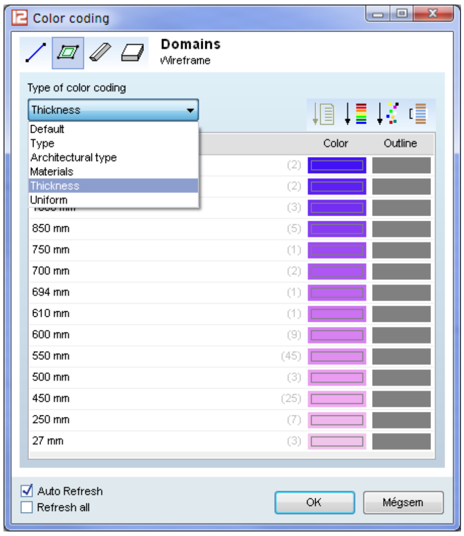
2.16.5. Color coding

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.

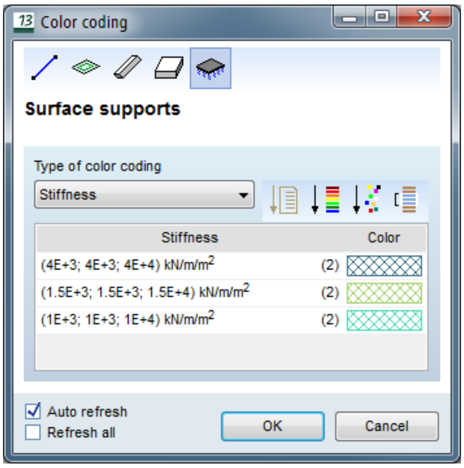
Line elements



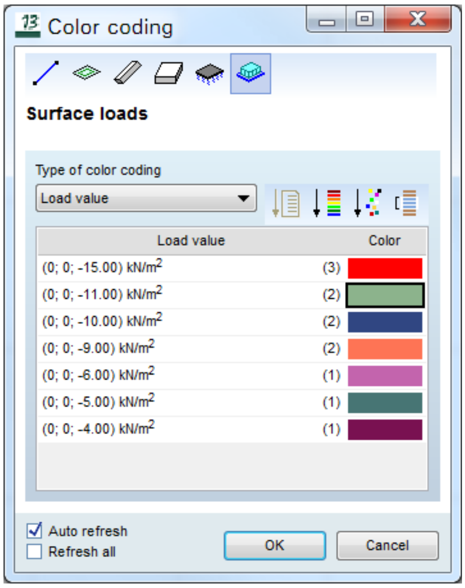
Domains



Surface supports



Surface loads

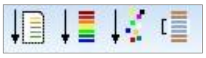


Type of color coding	*Default*	Uses default colors.
	Type	Finite element type (truss, beam, rib for line elements, shell, plate, membrane for domains and surface elements) determines the element color.
	Architectural type	Architectural type (column, beam and miscellaneous for line elements, slab, wall, 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

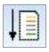
Load value Surface load colors are assigned by load intensity.
Areas where load polygons overlap are colored according to the sum of the individual loads.

Uniform Uniform color for all elements

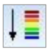
Setting colors



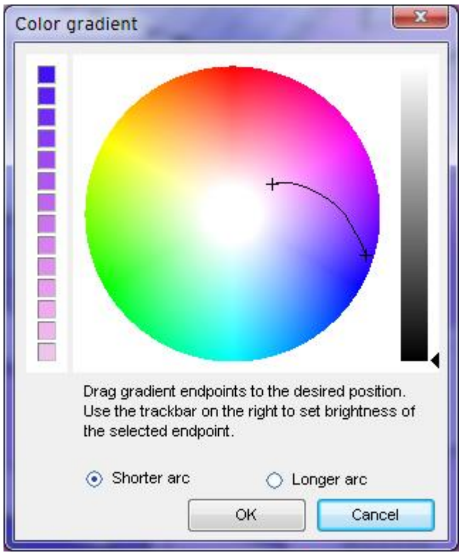
Click any color cell to change the color.
Toolbar buttons change more than one cell.



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 endpoints on the hue / saturation circle to the desired position. The program picks up the necessary number of colors between the endpoints.
Use the trackbar on the right to set brightness for the selected endpoint.
Shorter arc connects colors with the smallest possible hue changes. *Longer arc* goes around the hue circle in the other (longer) way.

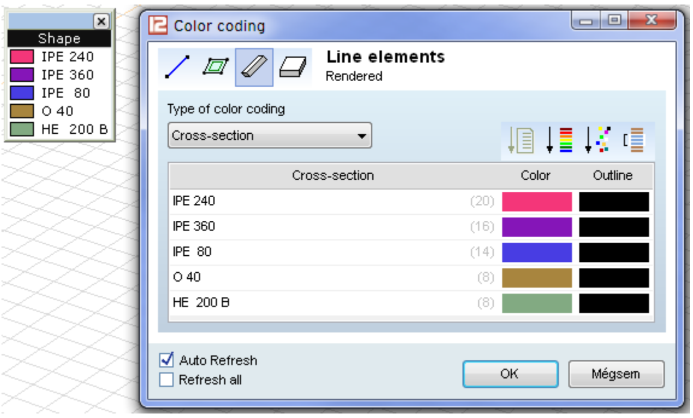


Random colors
Program selects random colors but ensures that colors are not too close.

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.***

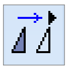
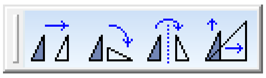
The current color coding is displayed as a separate info window.
You can turn on and off this window from the main menu *(Window / Color coding)*



Auto Refresh Views are automatically updated after changes.

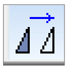
Refresh all Applies changes to all views. If unchecked only the active view is changed.

2.16.6. Geometric tranformations on objects

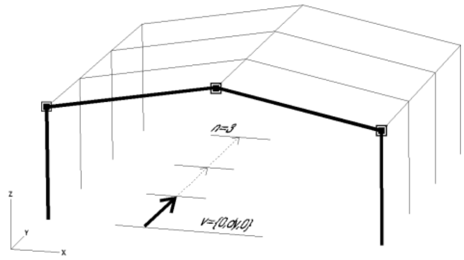



2.16.6.1. Translate

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*).



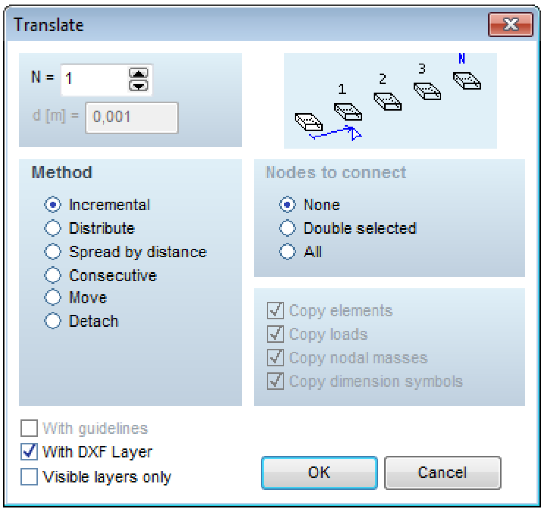
Translation options

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.

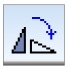
☞ ***If you have repetitive parts in your model, you should first create these (including the definition of finite elements, support conditions, loads, and dimension lines), and then make copies of them.***

You can use any existing point when you have to specify the translation vector.

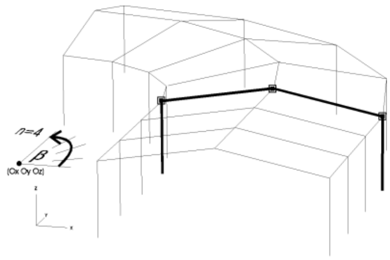
Selected loads can be copied or moved to another load case if load case is changed to the target load case during the operation.

2.16.6.2. Rotate

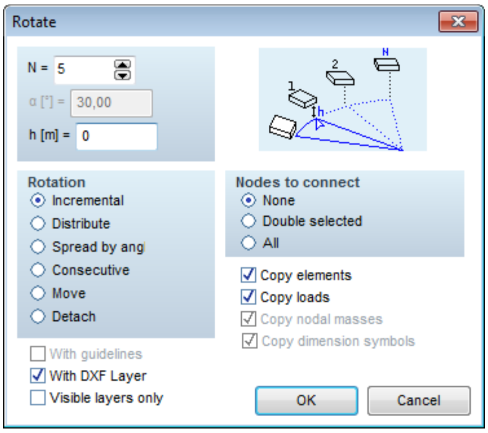
Rotation



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.

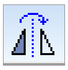
Nodes to connect **See...***2.16.6.1 Translate*

Switches **See...***2.16.6.1 Translate*

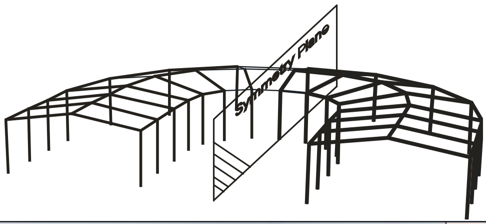
In perspective view, the centerpoint, start point and endpoint can be specified only using existing points or other identified 3D locations (i.e. a point on a line). In perspective view, cursor angle is determined by the global X and Y coordinates only.

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

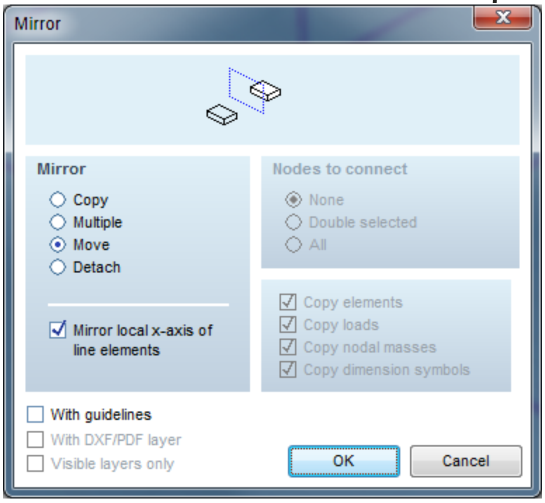
Copy: reflects a copy of the selected entities over the mirror plane.

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.



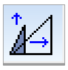
Nodes to connect **See...** *2.16.6.1 Translate*

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).

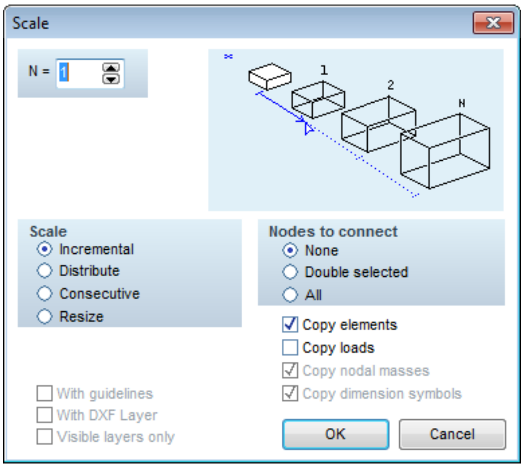
Scale options

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.

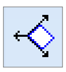
Resize: redefines the selected entities by scaling.



Nodes to connect **See...** *2.16.6.1 Translate*

Switches **See...** *2.16.6.1 Translate*

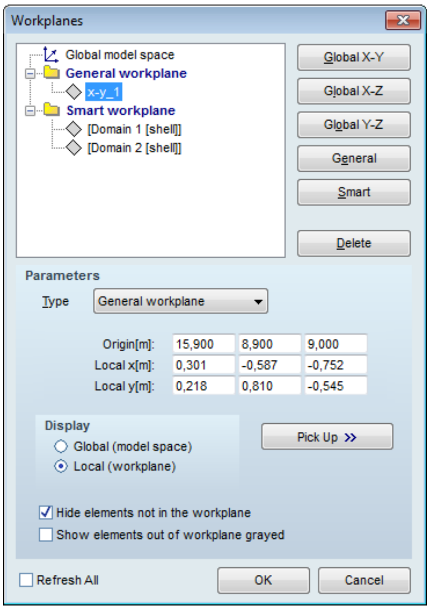
2.16.7. Workplanes



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

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

General workplanes

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

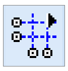
Deletes user defined workplanes.

Pick Up >>

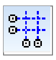
Lets you define workplane parameters (origin or axes) graphically.

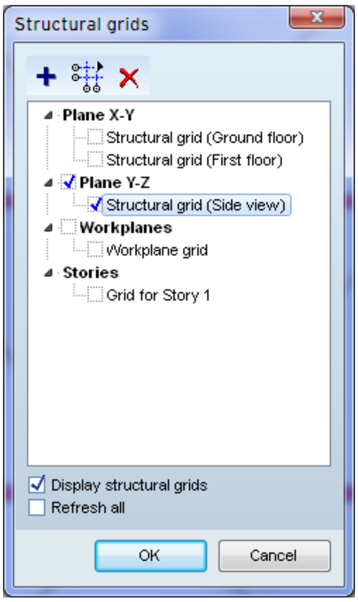
2.16.8. Structural grid

Two options are available: a structural grid or custom grid lines.



Structural grids





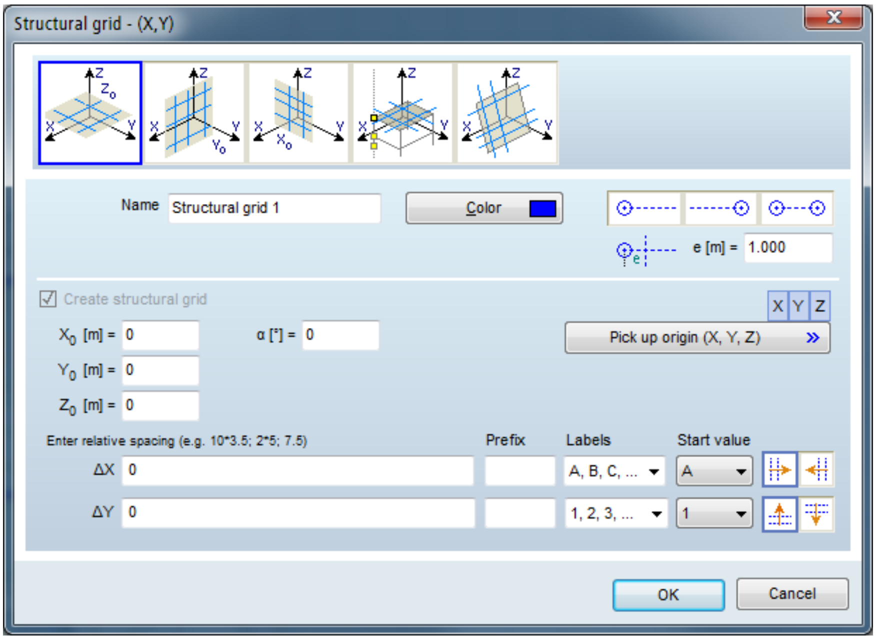
Structural grids are sets of coloured lines in a common plane, with a given length and a label helping the model building process. These sets of gridlines can be parallel to the global X-Y, X-Z or Y-Z planes, workplanes or stories.
Structural grids are displayed in a tree, organized by the grid plane.

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 stories can hide themselves if their workplane or story is not active.

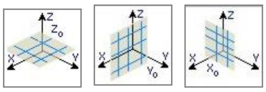
Refresh all
Update structural grids for all views.

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.

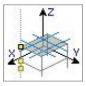
New structural grid



Grid plane

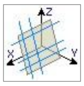


For grids parallel to a global plane the X_0, Y_0, or Z_0 distance between the grid plane and the global plane can be set



Story grid

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

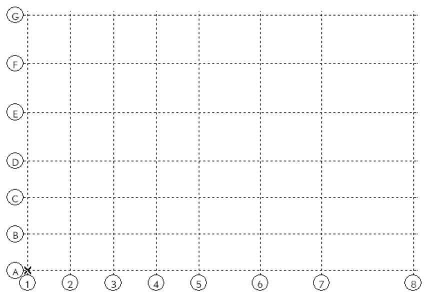
Color Click the button to change the color of the gridlines.

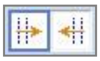
X_0/ Y_0/ Z_0 [m] Origin of the structural grid relative to the global origin.
The origin can be picked up by clicking on the *Pick up the origin* button and clicking on anywhere on the model. By activating or deactivating the *X, Y, Z* fields above the button the user can control which coordinates to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

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

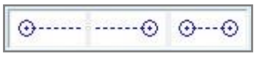
Create structural grid Grid spacings, prefixes, labels, directions can be defined.

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).

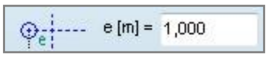


 Order of creating vertical grid lines

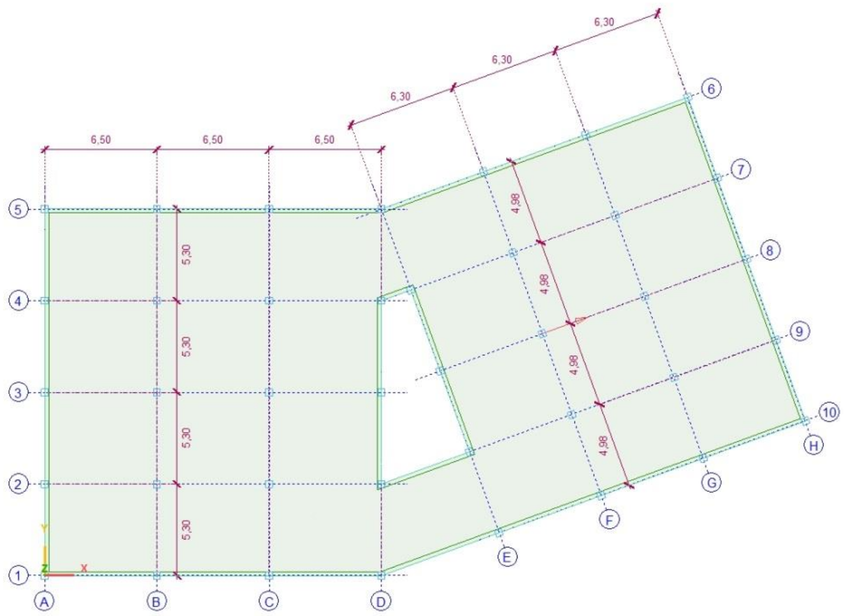
 Order of creating horizontal grid lines

 Labels can be positioned to the startpoint, endpoint or both.

e[m]



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



Modify structural grid

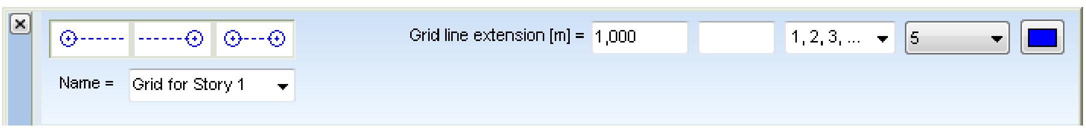
Name, labels, positions, color of a grid can be modified. If *Create structural grid* is checked the entire grid will be recreated with the new parameters. In this case all custom gridlines associated to the grid (see below) will be removed.

Delete structural grid

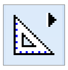
Selected structural grids will be deleted.
Pressing **Ctrl** or **Shift** during mouse clicks more than one grid can be selected.

Custom gridlines

Custom gridlines can also be defined by clicking on the startpoint and the endpoint. 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.

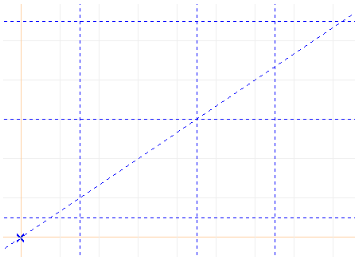


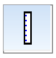
2.16.9. Guidelines



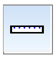
Helps in editing the geometry of the model. Guidelines can be defined in the global coordinate system. This way an arbitrary grid can be created, intersections can be determined and distances can be set. The cursor identifies the guidelines.
See... *4.7 Editing tools*

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.

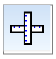




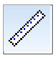
Places a vertical guideline at the current position of the cursor.



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.

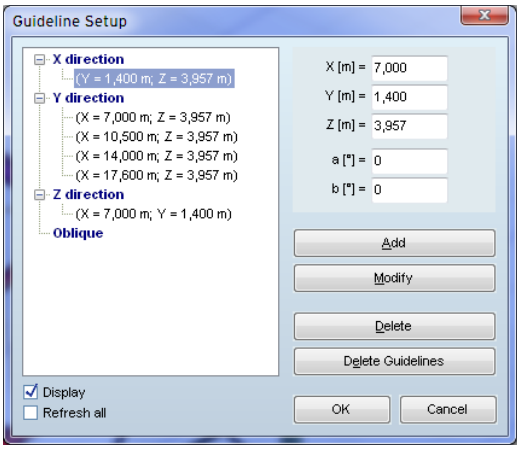


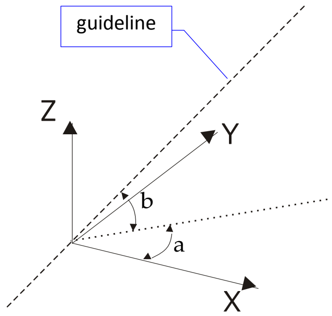
Places an oblique guideline at the current position of the cursor.

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:





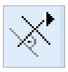
a: is the angle of the guideline's projection on the X-Y plane and the X axis.

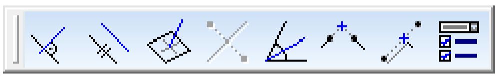
b: is the angle of the guideline and its projection on the X-Y plane.

Display Turns on / off the display of guidelines.

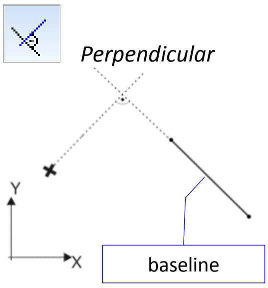
Refresh all If checked changes will be applied to all views otherwise only in the active view.

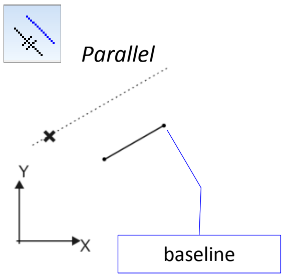
2.16.10. Geometry tools





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



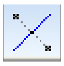


Begin to draw a line. Click the *Perpendicular* or *Parallel* icon then click an existing line or click two points to define the direction. The cursor will move perpendicular or parallel to this baseline.

Perpendicular to a plane

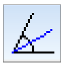
Begin to draw a line. Click the *Perpendicular to a plane* icon then click the domain defining the plane. The cursor will move perpendicular to the plane. The plane can also be defined by clicking three points.

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 startpont and endpoint of another line. Midpoint will determine the direction.

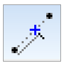


Bisector

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

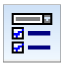
Point of intersection

Begin to draw a node or a line then click the icon, click the two lines or their start and endpoint. 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.

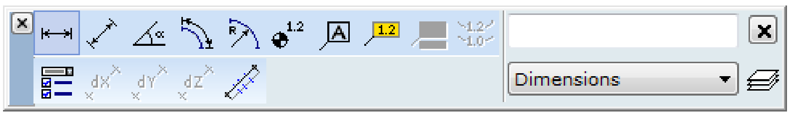


Point constraint operation

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.

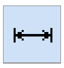
2.16.11. Dimension lines, symbols and labels

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.



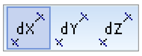
You can change the position of dimension lines or labels at any time by dragging them to their new position. If the dimension lines were associated with the model their position and dimension will be continuously updated as you modify the geometry of the model.

2.16.11.1. Orthogonal dimension lines



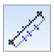
Associative orthogonal dimension lines or strings of dimension lines, parallel to the global X, Y, or Z axes can be assigned to the model by following the next steps:

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.



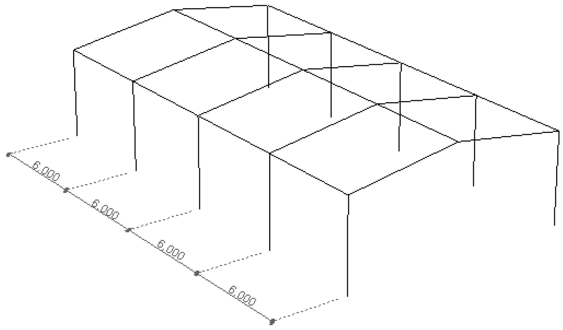
2. Move the mouse. The position of the dimension line depends on the direction in which you moved the mouse. There is one exception: when the segment is not parallel to any global plane and the editing is in the perspective view. In this case you have to select the direction dX, dY, or dZ from the toolbar.
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).



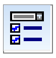
Smart dimension lines

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.

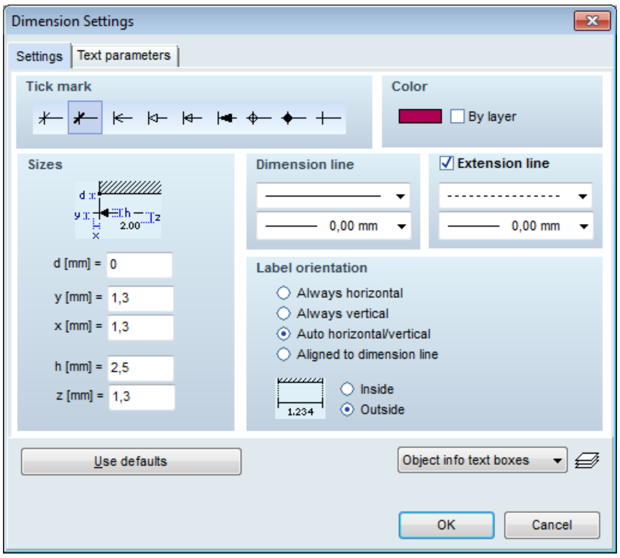


An example of smart dimension lines

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.

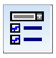
Apply font to all symbols Apply the same font to every dimension line.

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

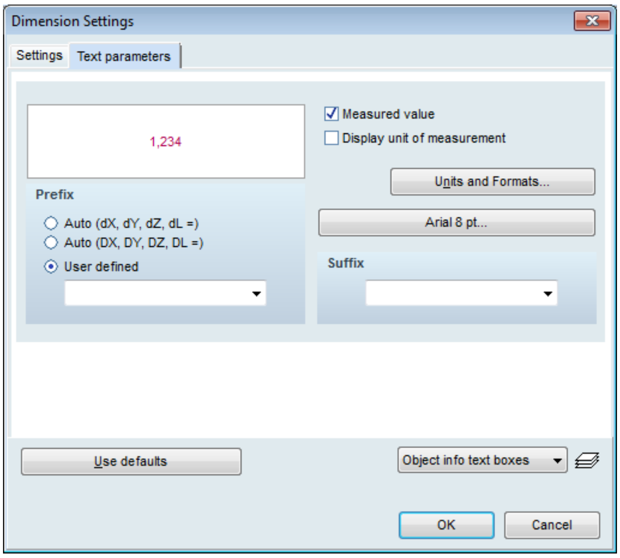
Apply to all dimension lines Applies the current setting to all existing orthogonal or aligned dimension lines to ensure a uniform look.

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.

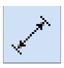
Display unit of measurement Display of the unit of measured value.

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

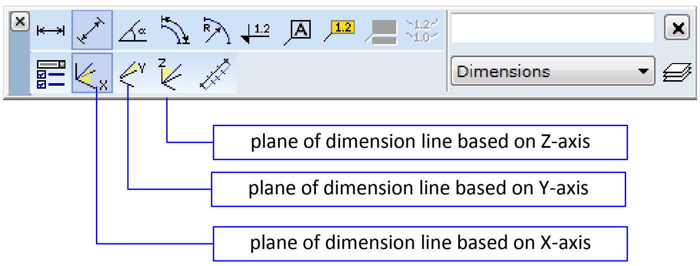
Prefix Sets the prefix used with the text on the dimension lines. You can choose from the following options:
Auto (dX, dY, dZ, dL = [depending on the direction])
Auto (DX, DY, DZ, DL = [depending on the direction])
User defined (this option will require you to enter the prefix).

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=.

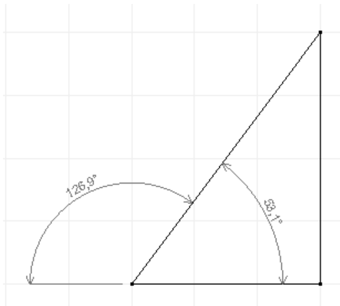
An example of associative dimension lines (orthogonal and aligned):

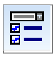
Before Scale command *After Scale command*

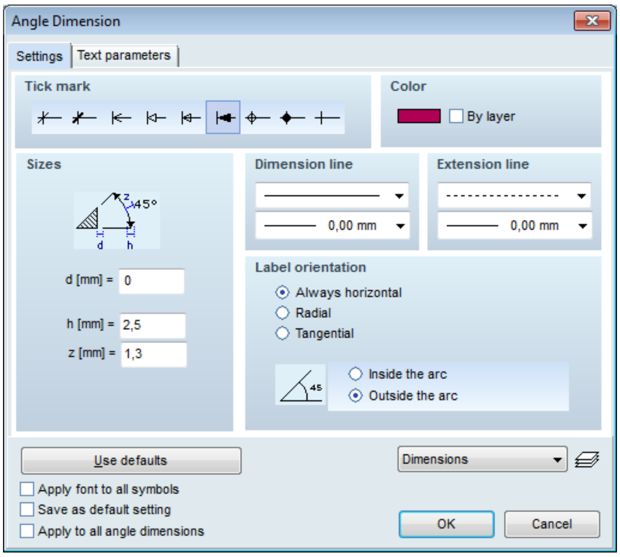
2.16.11.3. Angle dimension

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.

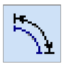




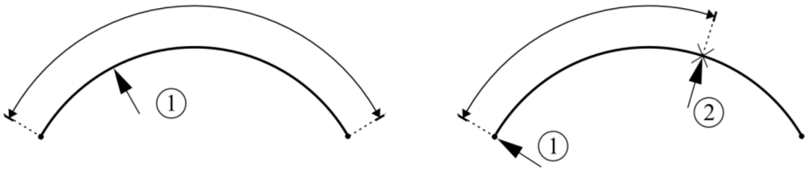


By clicking the *Text parameters / Units and formats…* button the angle number format can be set in the Dimensions section of the *Settings / Units and Formats* dialog box.

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 endpoint 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.

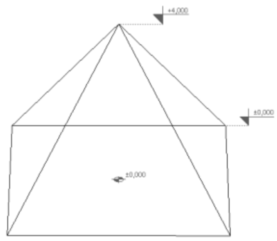
2.16.11.6. Level and elevation marks

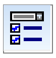
Creates associative level and elevation marks in your model.
By clicking the Units and formats button the number format can be set as the unit of Distance in the Geometry section of the *Settings / Units and Formats* dialog box. This is the unit and format used in the Coordinate Window.
See... *3.3.8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

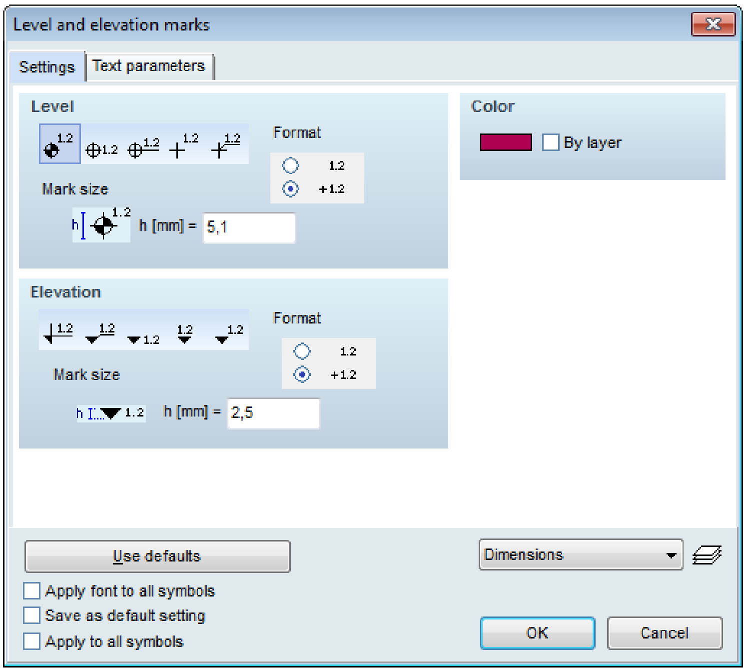
Elevation marks can be placed in front view, side view, or in perspective, by following the next steps:

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.





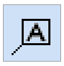
Level and elevation mark parameters.



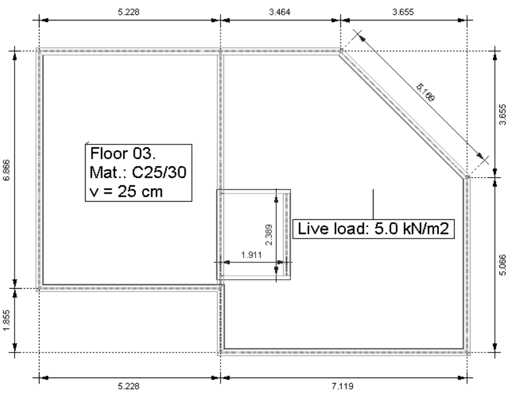
Level Selects the level mark symbol, and sets its size and format.

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

2.16.11.7. Text box

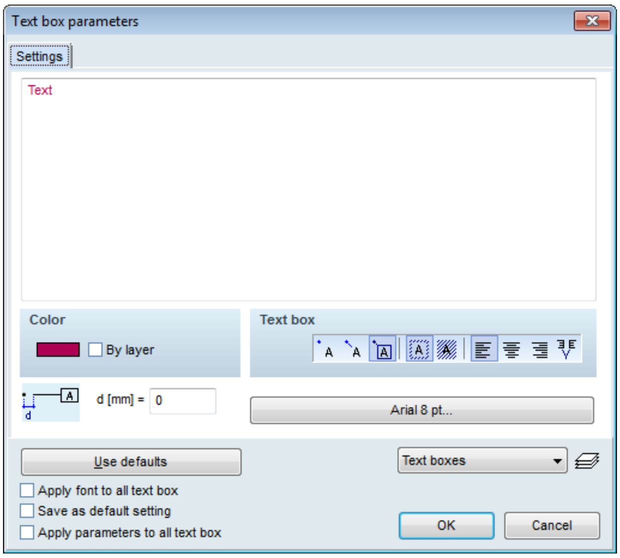


Creates an associative text box in your model. You can enter multiline text in a text box. The text will use the same text formatting within a text box.



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 tot 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.

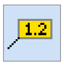
File reference A file reference is made of the -> characters and a file name. E. g.:
`->C:\MyModel\Reports\Details.doc`
If no full path is specified AxisVM starts from the folder of the model. So if our model is in *C:\MyModel* we can enter: **-> *Reports**Details.doc***
Clicking the text box starts 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.

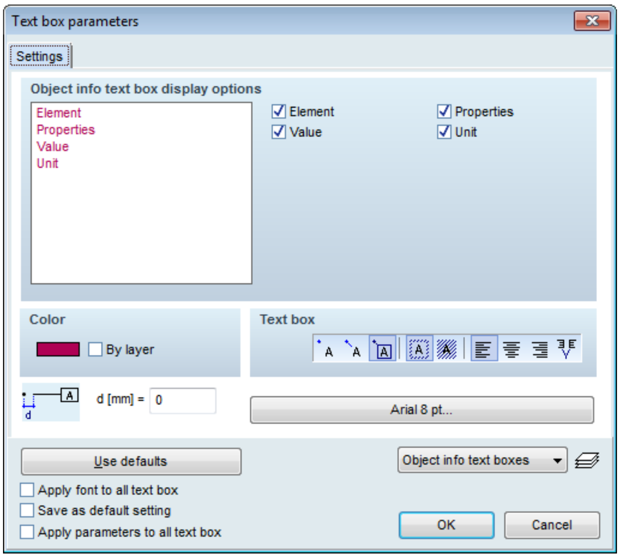
URL Supported protocols and link formats are:

`http://..., ftp://..., https://..., file://..., www. ...`
Clicking the text box 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:

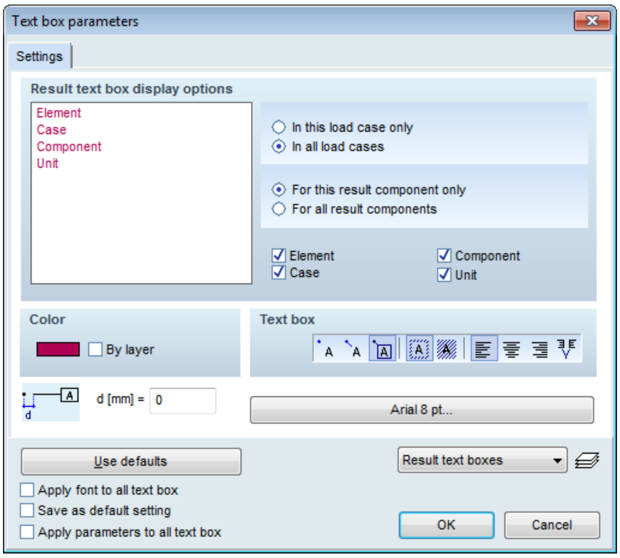




Result labels

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 case only

Result label is visible only in the load case in which it was created.

In all load cases

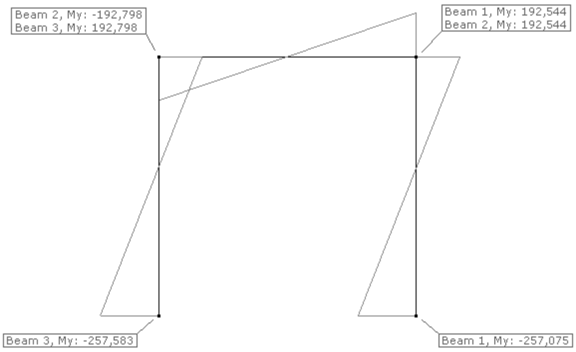
Result label remains visible regardless the load case. The actual values will be updated on changing the case.

For this result component only

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

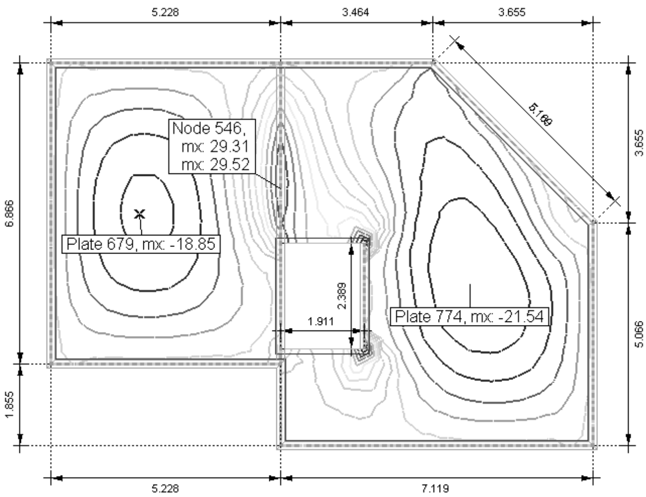
For all result components

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.

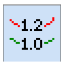
Apply parameters to all text box

After clicking the **OK** button parameters of all text boxes will be set to these values.

Layer Manager **[F11]**

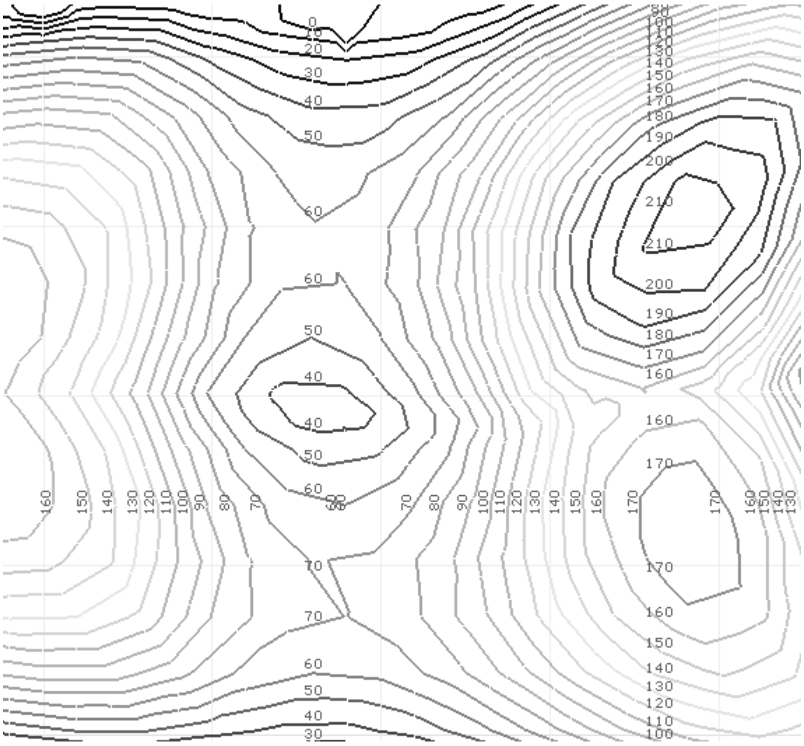
Lets you create new layers or modify existing ones.
This function is also available from the menu as *Settings / Layer Manager.*
See... *3.3.3 Layer Manager*

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



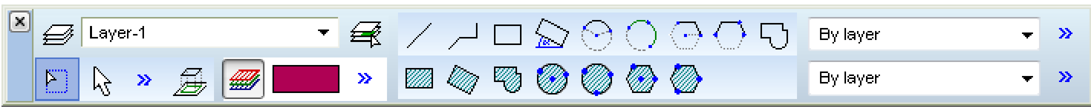
Automatic labeling of isolines can be activated in the color legend window. **See…** *2.18.4 Color legend window*

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

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.



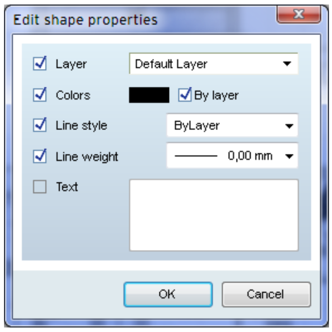
Layer Manager
Opens the Layer Manager. **See…** *3.3.3 Layer Manager*

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.
To use 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.

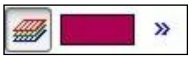
Pick up properties
Click on this button and to activate pick up. Clicking on a shape picks up all properties of that shape (i. e. all subsequent drawing functions will use these properties).

Convert 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.

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.

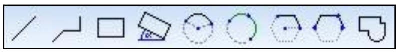
 Choose a color from a dialog.

 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 weights (bottom). Select the desired values.
These settings don't have any effect on filled shapes as they don't have an outline.

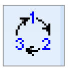
 Pick up line style or line weight of an existing shape



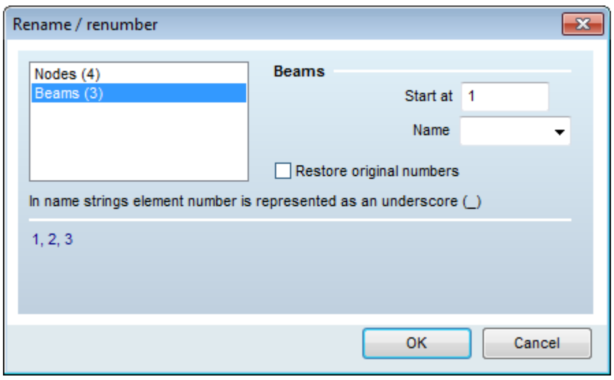
Toolbar for drawing lines and outlined shapes.

Toolbar for drawing filled shapes.

2.16.13. Renaming/renumbering



Nodes, trusses, beams, ribs and domains 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 the function icon on the Icon bar on the left.



The list on the left shows the number of selected nodes and elements. Choose what you want to rename / renumber.

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

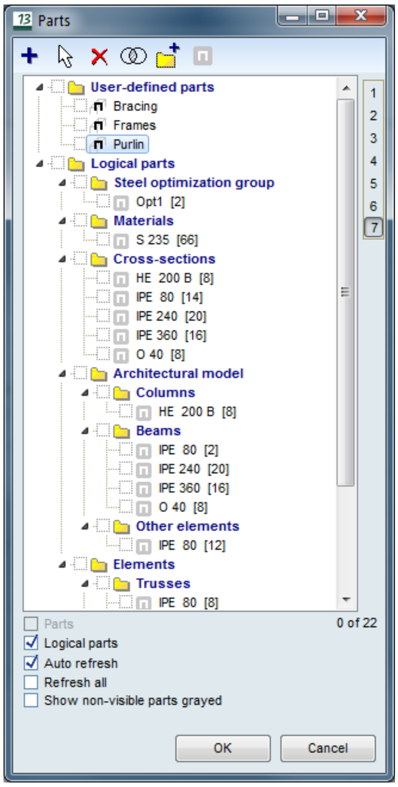
Name In name strings element number is represented as an underscore (_). For example: if starting number is 1, and the *Name* field contains *T_*, the names of the selected elements will be *T1, T2, T3, ...* If only one element is selected it is not necessary to include _ in the *Name*. Otherwise it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

Restore original numbers If *Restore original numbers* 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.

To turn on/off the display of numbers / names of elements open the Display options dialog (**see...** *2.16.18 Display options*) or use the speed button (**see...** *2.17 Speed Buttons*)

2.16.14. Parts

Lets you create sets of structural elements called parts. Working with parts makes the pre- and postprocessing 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, story 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.

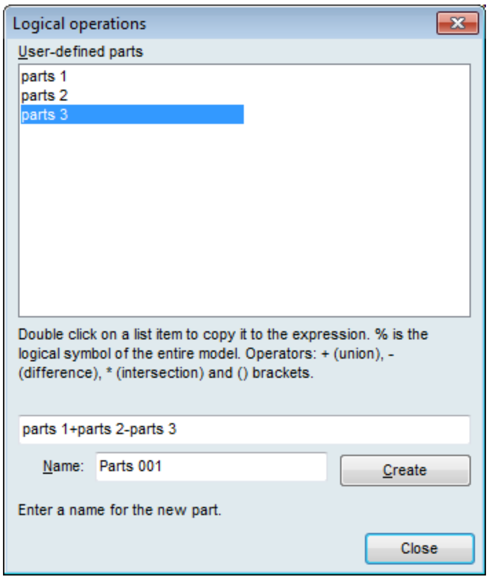
☞ ***If section segment result tables are selected only section segments within the active parts are listed.***

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").

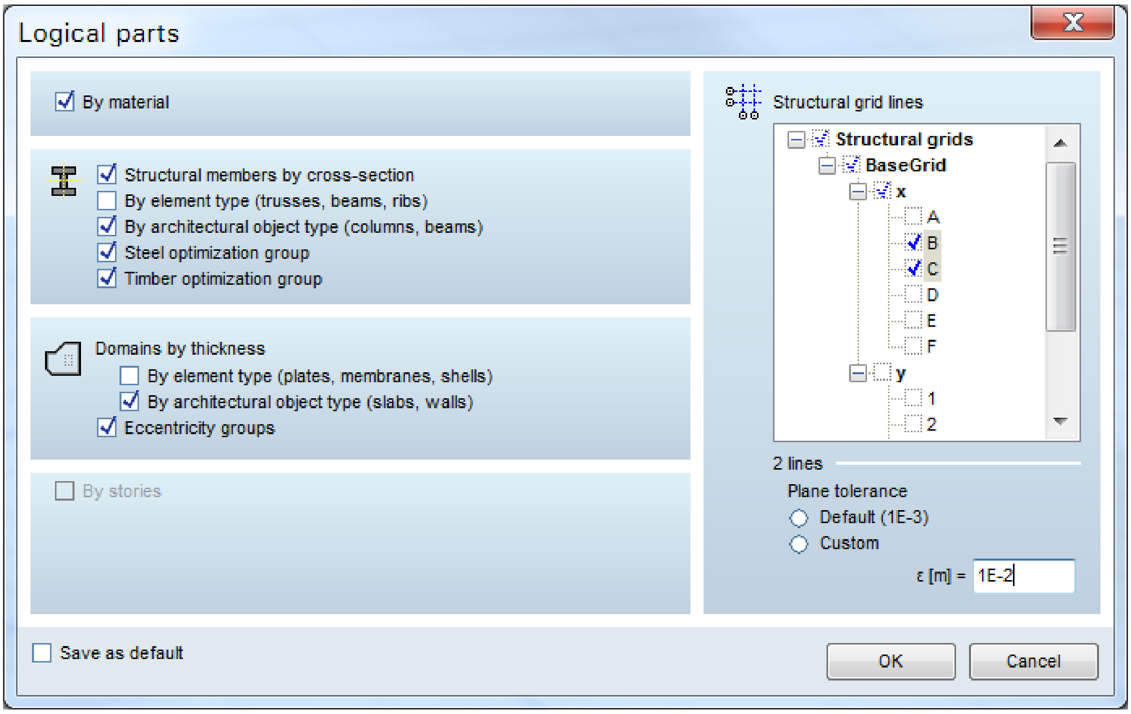


Creating new folders

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]** allows 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 stories defined logical parts can be created by stories.

Parts created from timber and steel optimization groups make it easy to change cross-section of an optimization group.

For eccentricity groups see *4.9.5 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:

All Turns on or off all the parts in the list.

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.

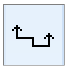
☞ ***When working on parts, only the data of the active parts will appear in the tables by default.***

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.

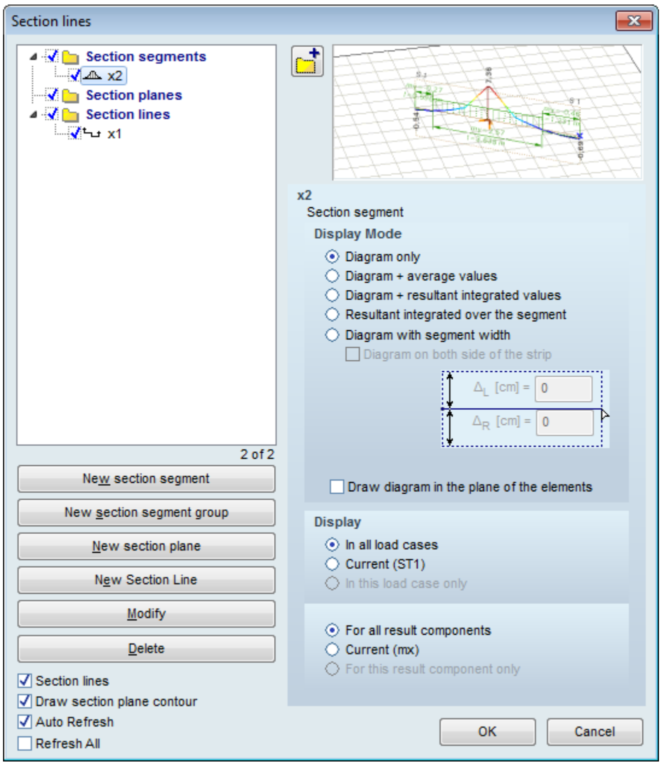
Show non-visible parts grayed
If it is on the entire modell wireframe is also displayed in gray to help identification of model parts.

2.16.15. 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).
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… *6.1.5.1 Section segment result tables*

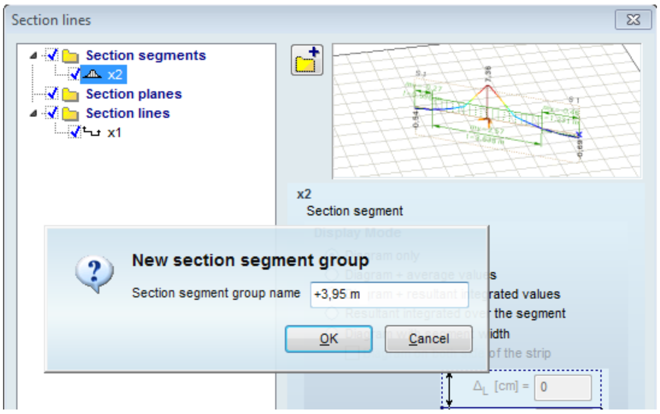


The dialog works similar to the *Parts* dialog.
Section lines, planes and segments can also be turned on and off using a speed button at the bottom toolbar.
If the result display mode is Section result diagrams are displayed only on section lines, planes and segments.
To reduce the complexity of drawings 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



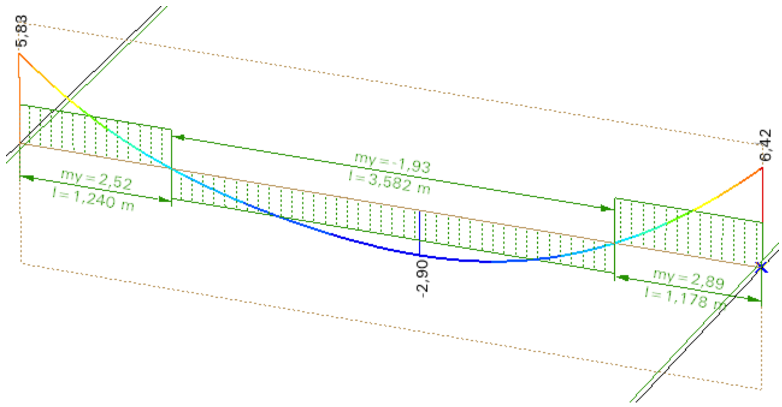
Section segment groups can be created to make it easier to turn on/off several section segments together.
Click *New section segment group*, enter a name for the group *(name)* and define any number of section segments. End definition by pressing **[Esc]**. Section segments will be numbered *(xx)* and get into the *name* folder as *name_xx.*

Creating new folders

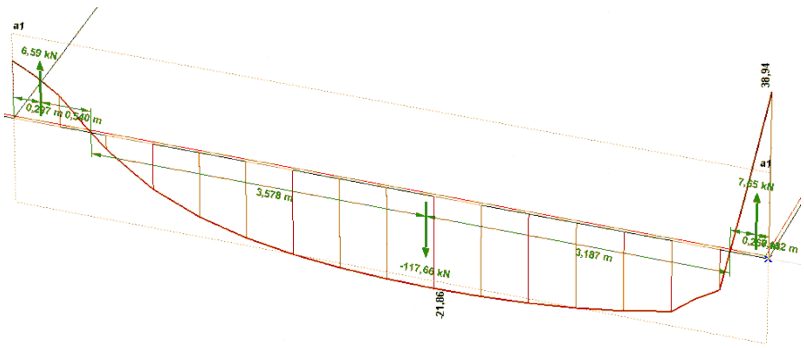
Creating folders offer a way of sorting sections. Segments can be moved and rearranged by dragging them to a new position within its own type group. **[Ctrl]** and **[Shift]** allows multiple selection in the usual way.
Turning folders on/off turns on/off the segments within the folder.

New section segment

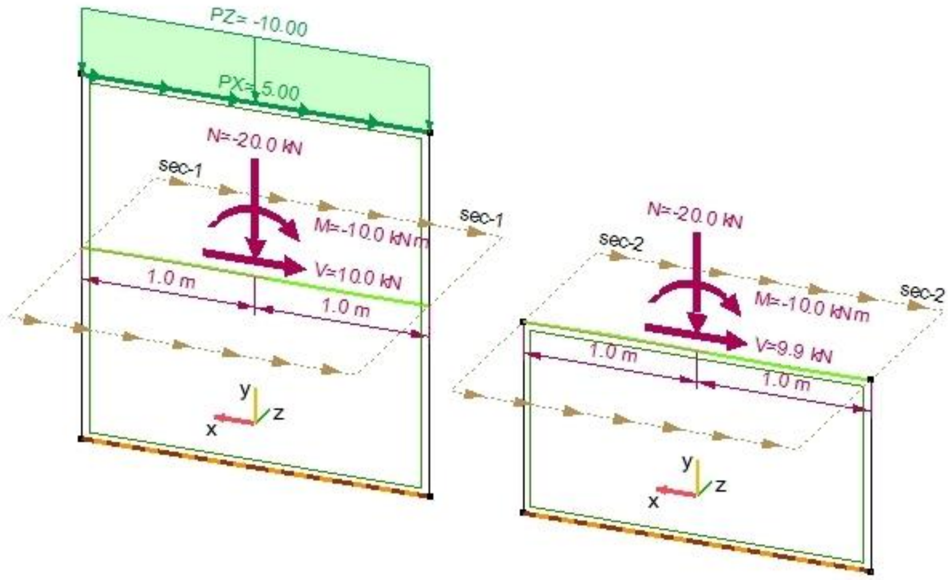
To define the segment enter two points of a domain or on domains in the same plane.
Setting the radio buttons you can control how the internal forces diagram will be displayed. Left or right segment width can also be specified.
Diagrams are usually displayed perpendicular to the element plane but checking 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.



Display of the average values

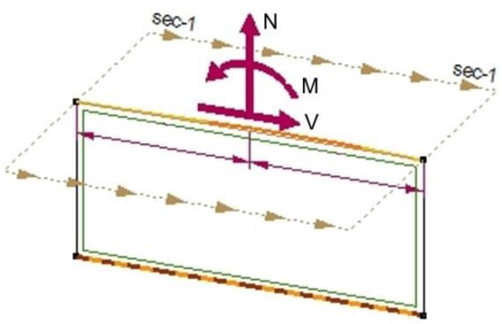


Display of the resultant integrated values



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 following picture shows the positive directions of the resultant components.

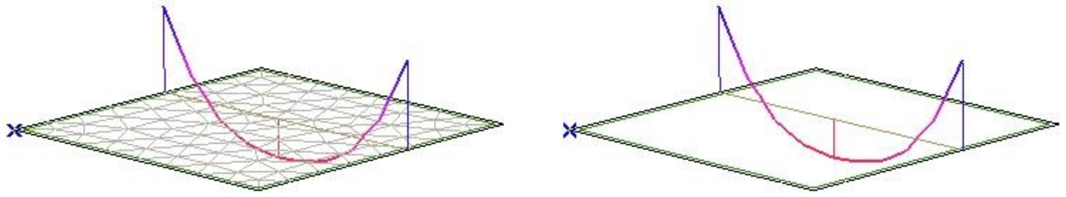


☞ ***The resultant integrated over the segment is displayed only for section segments prallel with the local x or y axis.***

☞ ***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 concomitant.***

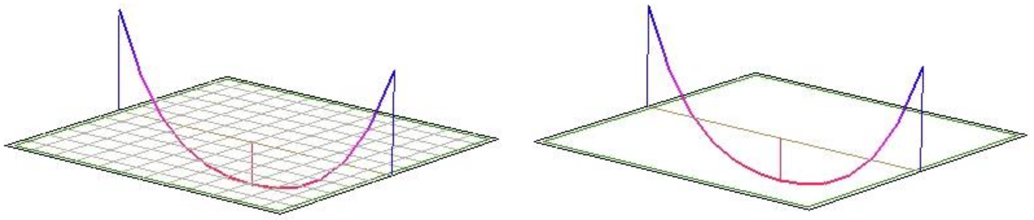
New section plane Click *New section plane* and assign a name to the section. This type of section is based on a plane. Click or enter two points to set the section plane. Then click OK in the Selection Icon Bar to save. In perspective view you have to click or enter three points to set the section plane. 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.***

2.16.16. 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.
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).
If active parts contain domains with virtual beams and a result component is selected from Virtual beam internal forces (*6.1.9 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.

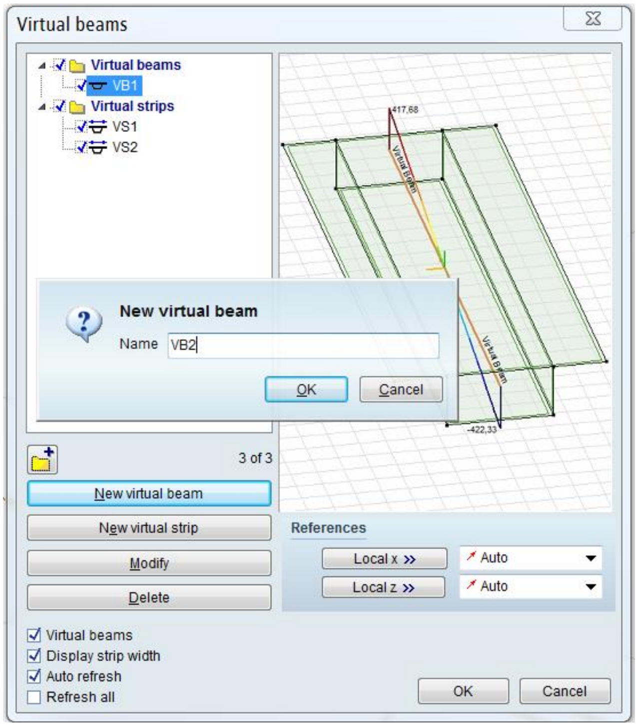


Table of virtual beam results can be displayed in the *Table Browser*.
See… *6.1.5.1 Section segment result tables*

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. By default it is parallel with the longest side of the bounding box of selected domains in the global space. The axis of the virtual beam (local *x*) and the local *z* direction can be set arbitrarily by selecting a reference vector. **See…** *4.9.20 References.*

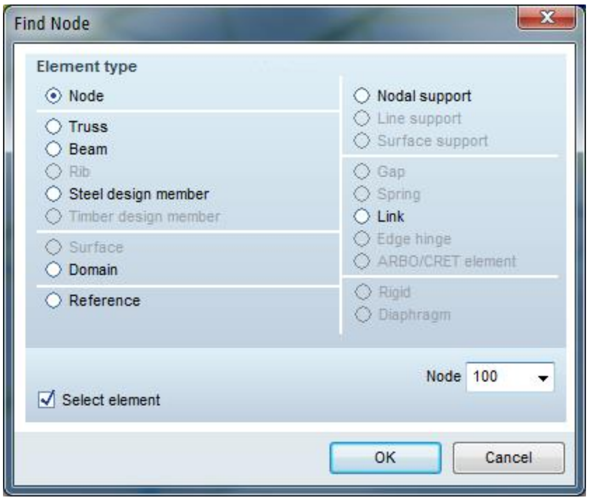
New virtual strip Enter a name then the startpoint and endpoint of the strip. These points define the axis of the virtual beam. If the two points are on different domains, domains must be in the same plane. The Δ_L and Δ_R strip width values of a new strip are set automatically to 0.5 m, but can be changed.

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

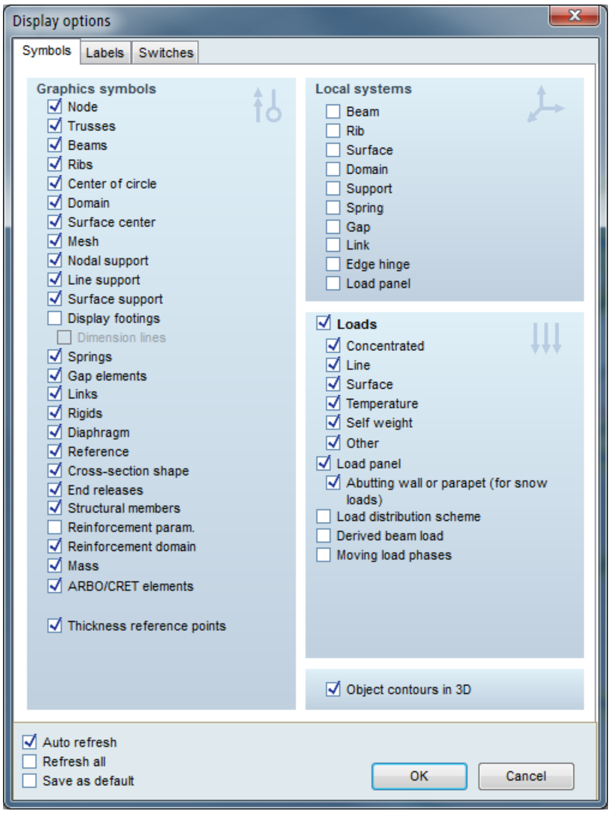
Creating new folders Creating folders offer a way of sorting virtual beams. Virtual beams can be rearranged by dragging them to a new position within its own type group. Turning folders on/off turns on/off all virtual beams within the folder. **[Ctrl]** and **[Shift]** allows multiple selection in the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

2.16.17. 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).

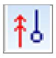


2.16.18. Display options



Symbols

Enables/disables the display of symbols.

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

Graphics Symbols

Mesh

Enables the display of the inner mesh lines.

When disabled the generated mesh lines are not displayed.

Node

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

Surface center

Enables the display of the center point (selection point) of the surface elements.

Color codes: plate = red, membrane = blue, shell = green.

Center of circle

Enables the display of centers of circles as a small cross.

Domain

Enables the 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.

Nodal support

Enables the display of the nodal supports.

Nodal supports appear as thick axes.
Color codes: axial displacement=yellow, axial rotation= orange.

Edge support

Enables the display of the edge supports.

Edge supports appear as a thick edge.
Color codes: axial displacement=yellow, axial rotation= orange.

Surface support

Enables the display of the surface supports.

Surface supports appear as a light brown hatch .

Footing

Footings designed on the R. C. Design tab appear with their calculated or specified shape and size.

Dimension lines

Enables the display of footing dimension lines.

Links

Enables the 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 endpoints.

Rigids

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

Diaphragm

Enabled the display of diaphragms as dashed thick purple lines.

Reference

Enables the display of the references.

Red vector, crosshairs or triangle.

Cross-section shape

Enables the 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.

End releases

Enables the 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.

Structural members

Enables the display of the structural elements.

An orange line along the member and the number of the member.

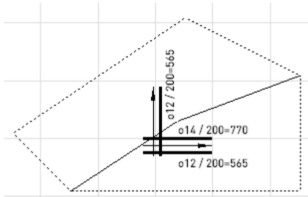
Reinforcement param.

Enables the display of brown stars at surface centers where reinforcement parameters are assigned.

Reinforcement domain

Enables the 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 the display of the symbol of the concentrated masses.

Double red circle.

Story center of gravity

Enables the display of center of gravity of each story. AxisVM converts loads of load cases used to calculate the vibration shapes for seimic analysis into masses then calculates the center of gravity for each story. The centers are displayed as black +s in black circles with a label Gm*i* where *i* is the level number.

Story shear center

Story shear center is determined form wall sections at the story level. The method to determine shear center of thin walled cross-sections is used.

Enables the display of shear center of each story. AxisVM calculates story shear centers by finding wall sections and using the same method as for thin-walled cross-sections.

The centers are displayed as red +s, with a label S*i*, where *i* is the level number.

ARBO-CRET elements

Aschwanden ARBO-CRET elements placed into the model.

A schematic drawing of the element is displayed.

COBIAX elements / AIRDECK elements

COBIAX / AIRDECK elements placed into the model.

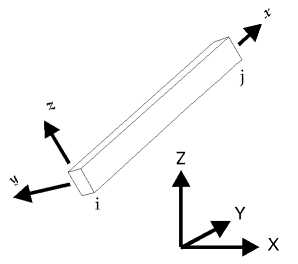
Void formers are displayed as circles in wireframe mode and balls in rendered view.

Thickness reference points

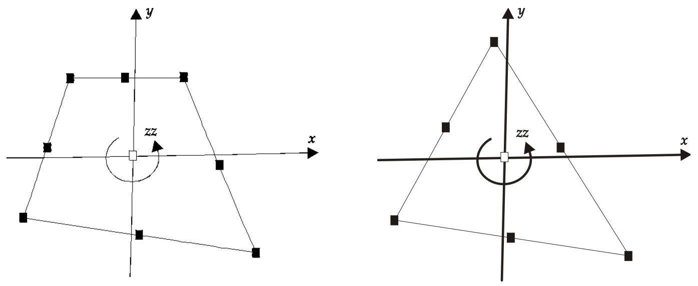
Reference points entered during definition of tapered domains

Orange crosses, thickness value labels, dashed lines connecting the reference points.

Local systems Enables the display of axes of the elements in the local coordinate system.

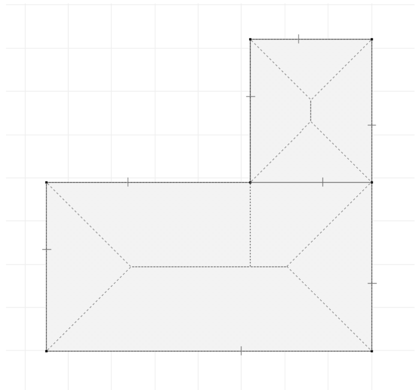


Beam element local coordinate system



Surface element local coordinate systems

Loads Display of load symbols can be set separately for each load type (concentrated, distributed along a line, distributed on surface, temperature, 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.* To display the derived beam loads check *Derived beam load.*



Load panel Load panel outlines are thick dashed turqouise lines. If load panels are attached to domains small rectangles appear along the outline showing the linkage.

Abutting wall or parapet (for snow loads) 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.

Derived beam load Displaying of derived beam loads

Moving load phases If this option is turned on all phases of moving loads are displayed in gray. If this option is turned off the moving load is displayed only in the position determined by the current load case.

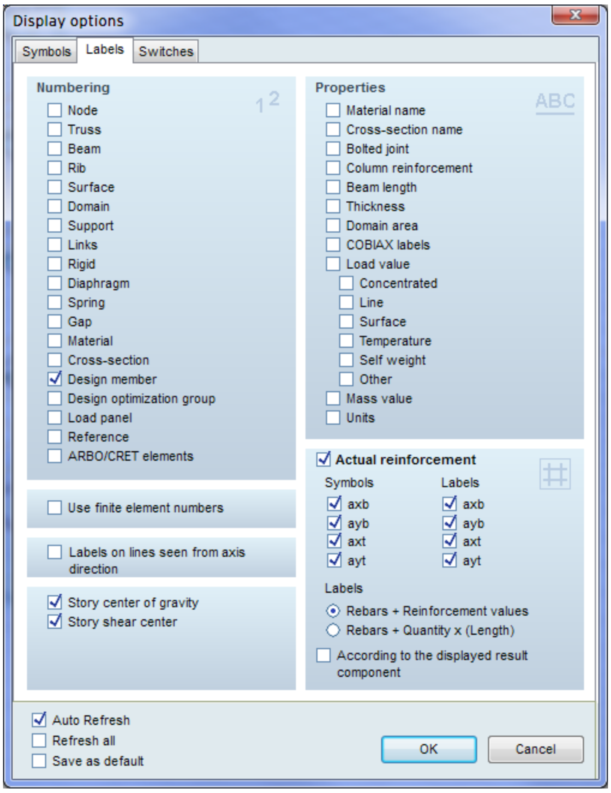
Object contours in 3D Displays static model with a 3D wireframe look (cross-section, thickness, eccentricity can be checked). Colors depend on architectural type (column, beam, wall, slab).

Auto Refresh If it is turned on any change in settings will make the active panel redrawn immediately.

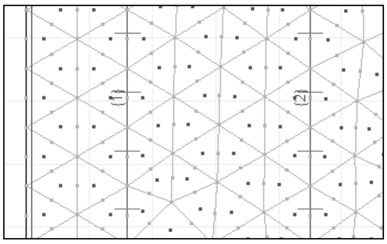
Refresh All Changes will affect all panels in multi-window mode.

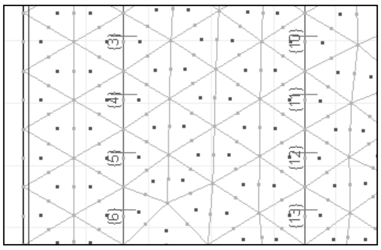
Save as default Saves the current symbol display option settings as default for new models.

Labels



Numbering





Displaying the number of nodes, elements, materials, cross-sections, references.

For meshed line elements checking *Use finite element numbers* displays the number of finite elements instead.

Use finite element numbers

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…** *3.2.13 Assemble structural members, 3.2.14 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).

Story center of gravity

If seismic stories are defined, their center of gravity can be displayed as a black circle on a black cross, with a label *Gm<index of story>*

Story shear center

If seismic stories are defined, their shear center can be displayed as a red x on a red cross, with a label *S<index of story>*

Properties

Enables the 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.

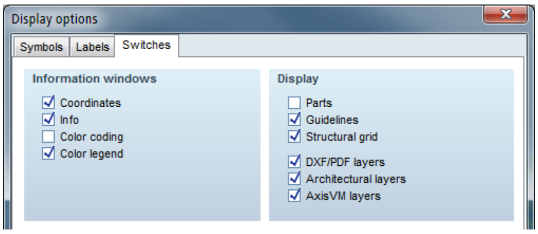
Actual reinforcement

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.

Switches

Information Windows

Coordinates **See...** *2.18.1 Coordinate window*
Info **See...** *2.18.2 Info window*
Color coding **See...** *2.18.3 Color coding*
Color legend **See...** *2.18.4 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.

2.16.19. Options

Allows the selection of the options for the settings of the grid, cursor, editing, drawing parameters, and design code.

2.16.19.1. Grid and cursor

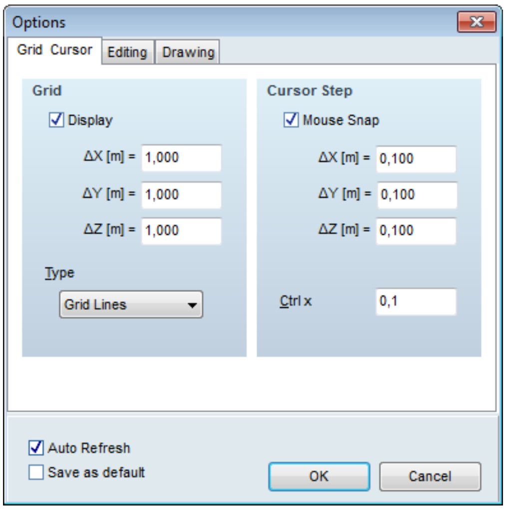
Grid The grid consists of a regular mesh of points or lines and helps you position the cursor to provide a visual reference. Depending on its *Type* the grid is displayed as:

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 to choose coordinates of an invisible dot mesh (not the grid).

You can set the cursor step parameters as follows:

Mouse Grid - 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.

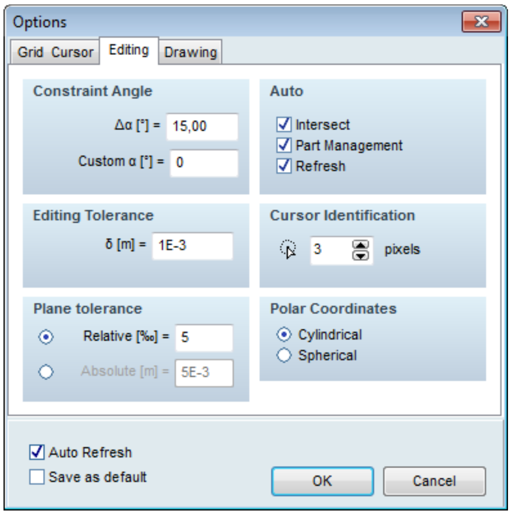
When using with constraints, the cursor step is applied in the constrained direction with the DX value. **See...** *4.7.4 Constrained cursor movements*

☞ ***If the grid step and the cursor step is set to the same value, nodes will be placed snapped to the grid.***

2.16.19.2. 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.

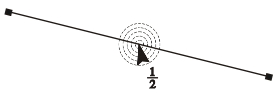
Refresh :

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

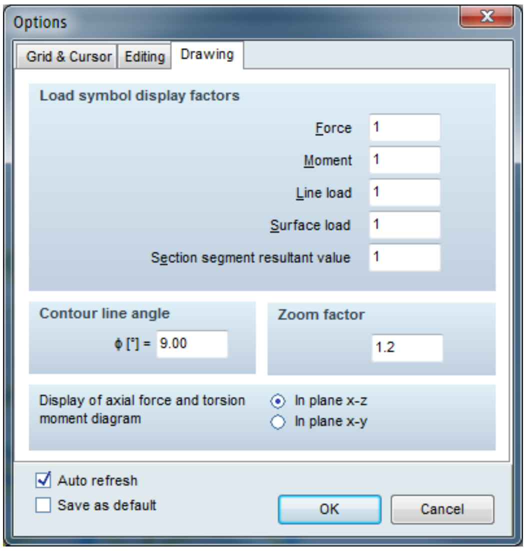
Nodes of domains and surfaces must be in plane. If a node of a domain or surface deviates from this plane more than the given value the element will be deleted. Plane tolerance can be specified in two ways:

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*

2.16.19.3. Drawing



Load symbol display factors

Sets the display size of the load symbols. This factor is applied when the checkbox in the Symbols icon / Graphics Symbols / Load is enabled. These values do not affect load values.

Force
Sets the display size of the symbol of concentrated force loads.

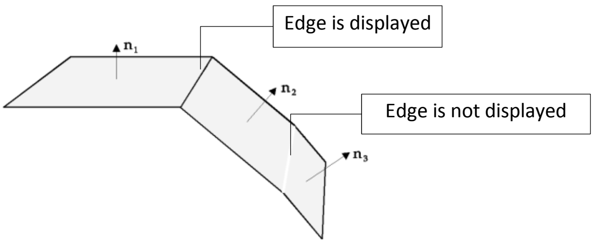
Moment
Sets the display size of the symbol of concentrated moment loads.

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.



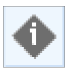
Zoom factor

Sets the scale of magnification/reduction of the zoom commands associated to the **[+]** and **[-]** keys.

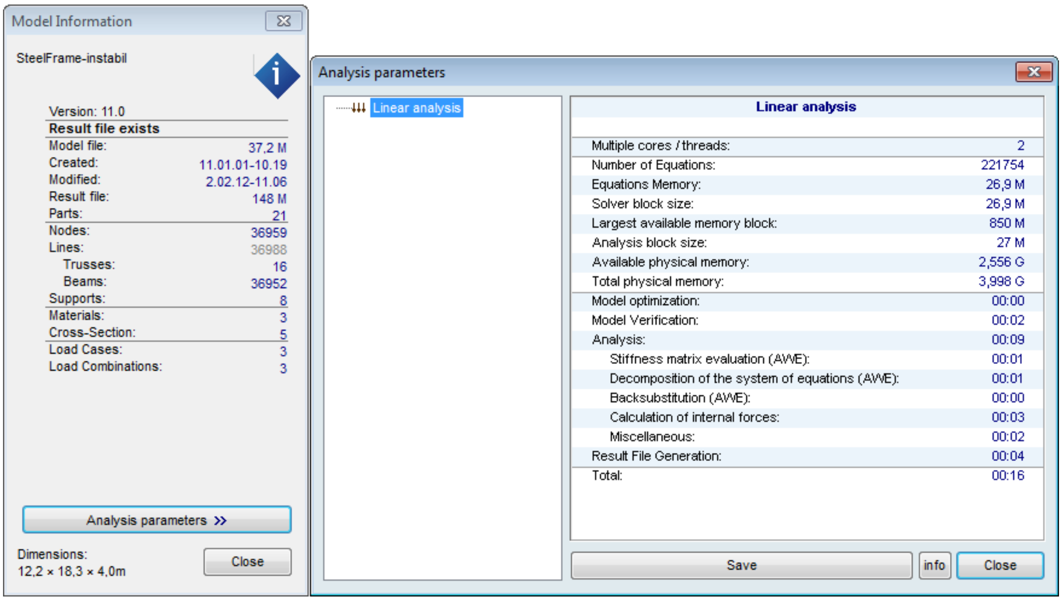
Display of axial force and torsion moment diagram

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.

2.16.20. Model info

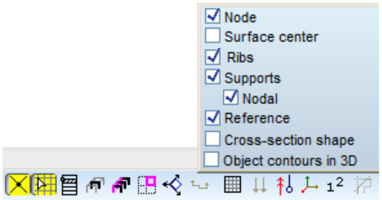


Shows the main parameters of the model.
Clicking the *Analysis parameters* button certain parameters of the latest analysis (memory usage, running time) can be studied. This information is available only if the model was analysed by Version 10 or later.



2.17. Speed Buttons

The quick switches toolbar allows you to change the display settings without entering the *Display option / Symbols* or *Options* dialog. The icons are located in the bottom right corner of the graphics area.



 Auto intersection

 Mouse snap

 Stories

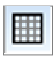
 Parts in tree view

 Display parts containing the selected elements

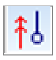
 Display selected element only

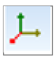
 Workplanes

 Section lines & planes & segments

 Display mesh

 Display load symbols

 Display symbols

 Display local systems

 Numbering

 Background layer

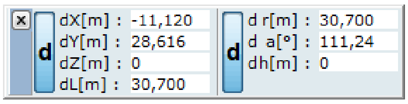
 Background layer detection

☞ ***Some of these settings are available also from Display and Options icons.***

2.18. Information windows

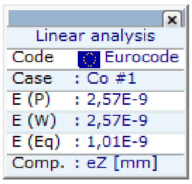
The information windows are situated in the graphics area. You can move these windows on the screen by clicking title bar, holding down the left mouse button, and dragging it to a new location on the screen.

2.18.1. Coordinate window



See... *4.4 Coordinate window*

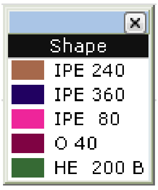
2.18.2. Info window



Shows information about the display of the results such as: active part(s), current perspective setting, type of analysis, current design code, current 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*

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.

2.18.3. Color coding



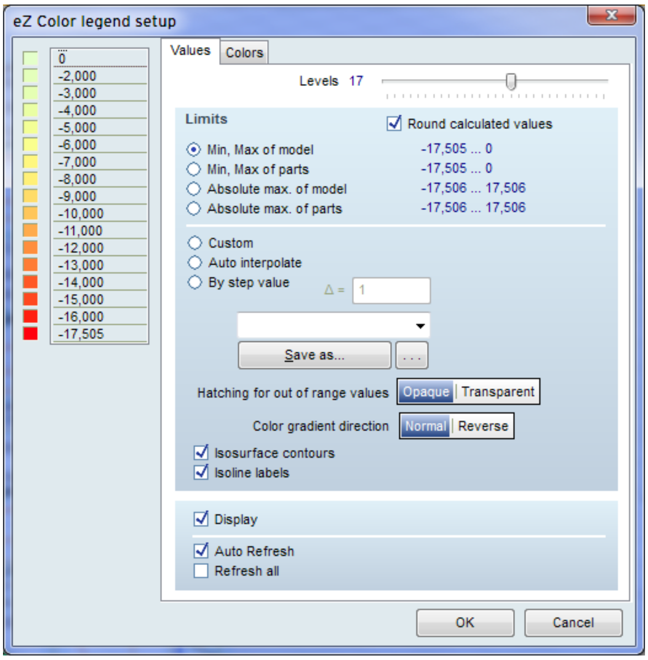
This info window appears after activating a color coding (**see…** *2.16.5 Color coding*) unless it is turned off in the Window menu (**see…** *3.5.2 Information Windows*).
The type of the coding is displayed within the black header line.

2.18.4. 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.

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.

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.

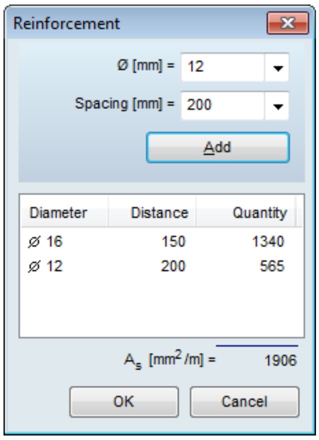
Isoline labels If checked, isolines are labeled automatically.

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.

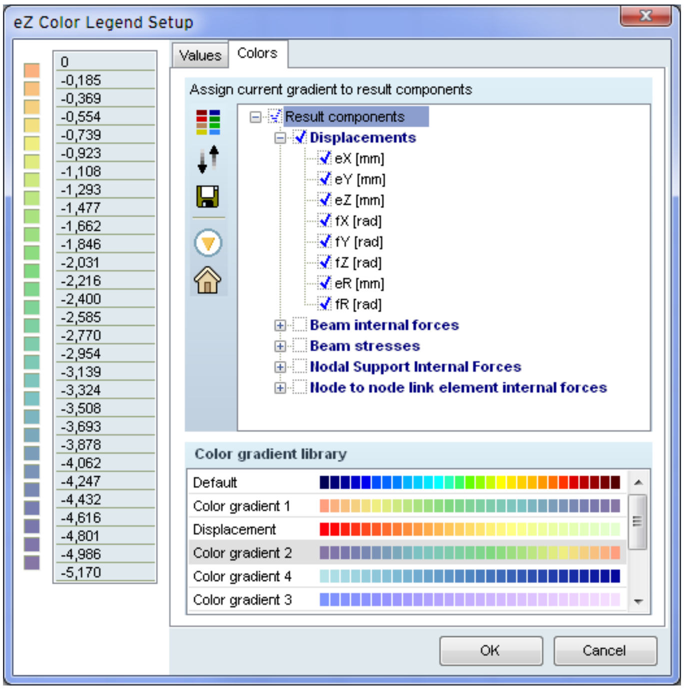
Calculate

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 form 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 endpoints to the desired position.

Revert color gradient
Swaps the start and end color of the gradient.

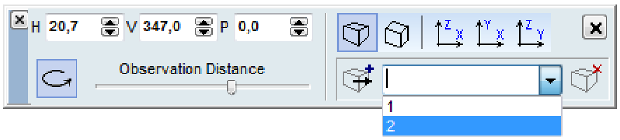
Save color gradient to the color gradient library
Gradients can be saved to a library for future use.

Save current color gradient assignments as default
Current result component assigments become the default setting for new models.

Restore default color gradient assignments
Sets the default color assigments for result components.

Clicking on an item of the *Color gradient library* applies the gradient.

2.18.5. Perspective window tool

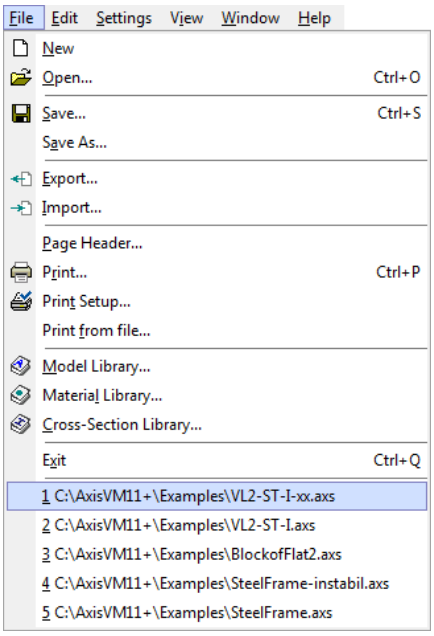


See... *2.16.3 Views*

This page is intentionally left blank.

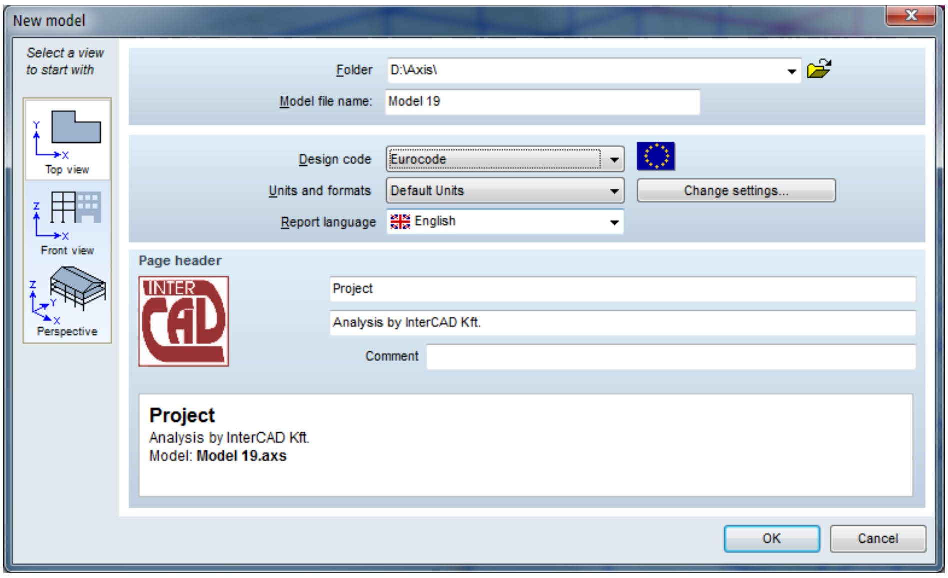
3. The Main Menu

3.1. File



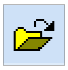
The menu commands are described below.

3.1.1. New model



Creates a new untitled 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...*

3.1.2. Open



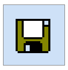
[Ctrl]+ [O]

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 in file names appearing as*** `Modelname.AXS` ***(input data), and*** `Modelname.AXE` ***(the results). Both file contains 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.***

The open dialog looks different on Windows XP and Vista / Windows 7 / Windows 8 operating systems.

3.1.3. 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.

3.1.4. 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.

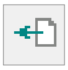
Converting models

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.

3.1.5. Export



DXF file

Export of this file type requires DXF module.

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, that lets you specify the units of measurement in the exported file.

Four different formats are available for DXF output.

- *AutoCAD 2004 DXF file*
- *AutoCAD 2000 DXF file*
- *AutoCAD R12 DXF file*
- *AutoCAD reinforcement design file*

It is recommended to use the AutoCAD 2004 format to avoid data loss. Earlier DXF formats support 256 palette colors and ASCII characters only.

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 (endpoints, material, cross section, reference) as a standard DSTV file. This file format is supported by several steel designer CAD software.

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.

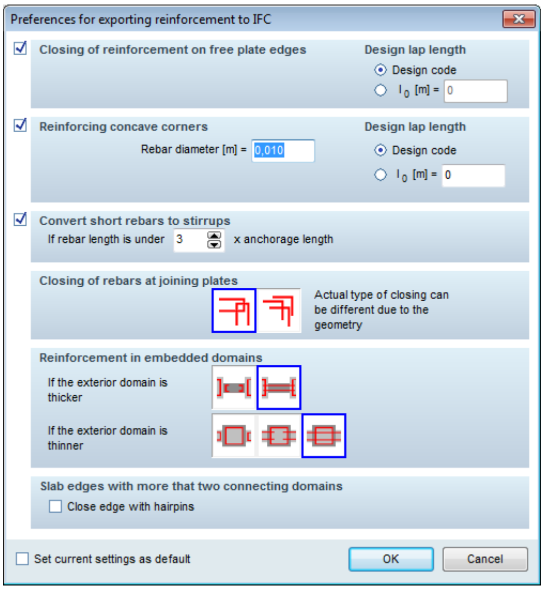
PianoCA file

Generates a *.pia interface file for PianoCA. It includes the data, supports, loads and the calculated results of the selected beam elements.

IFC 2x, 2x2, 2x3 file

Export of this file type requires 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.



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 lenghts 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 l_0. This option can be turned on or off.
- *reinforcing concave corners*: concave corners can be strenghtened by simple bars, whose lenghts 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 l_0. 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.
- 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

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.

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.

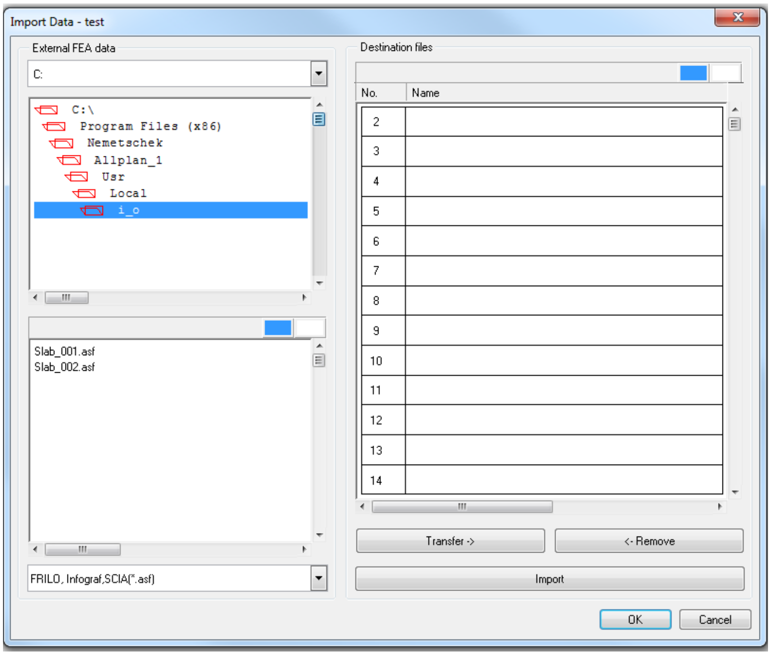
Nemetschek AllPlan

Export of this file type requires 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.

Import 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

For example:

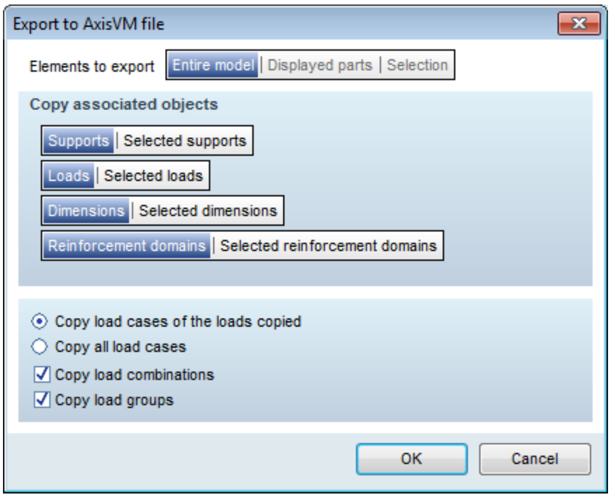
- Click on ***Create/Engineering/Bar Reinforcement/FEA reinforcement Colour Image*** menu item in Allplan menu
- Click on the desired asf file
- Choose layer (top/bottom) and type (compression/tension) of the imported reinforcement values
- Click on **OK**

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).

AxisVM Viewer Saves the model in AxisVM Viewer format (*.axv).
See… 7 *AxisVM Viewer and Viewer Expert*

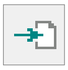
AXS file The following groups of elements can be exported: the entire structure, displayed parts or selected elements.
To select export options similar to those of the Copy options (**see…** *3.2.7 Copy / paste options*) click the *Settings* button of the *Export* dialog.



Export Selected Only Exports only the elements that are in the current selection set.

Coordinate units The coordinate units of the exported file can be selected here.
The default unit is meter [m].

3.1.6. Import

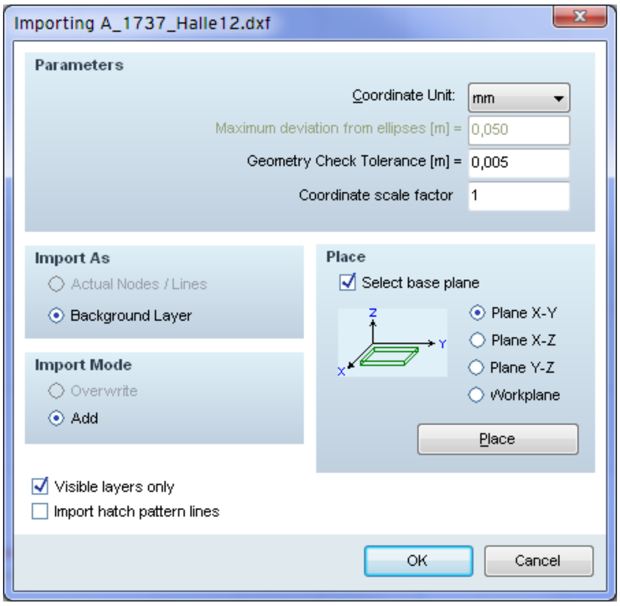


AutoCAD *.dxf

Import of this file type requires DXF module.

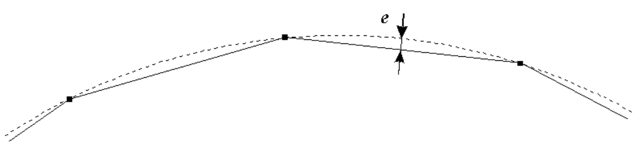
Imports a geometry mesh from a DXF file (drawing interchange file) exported in AutoCAD 12, 13, 14 and 2000 format into AxisVM. The layers of the imported file are loaded into the Layer Manager. **See**...*3.3.3 Layer Manager*
If the file date of the imported file has changed, the Layer Manager will ask 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.***



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

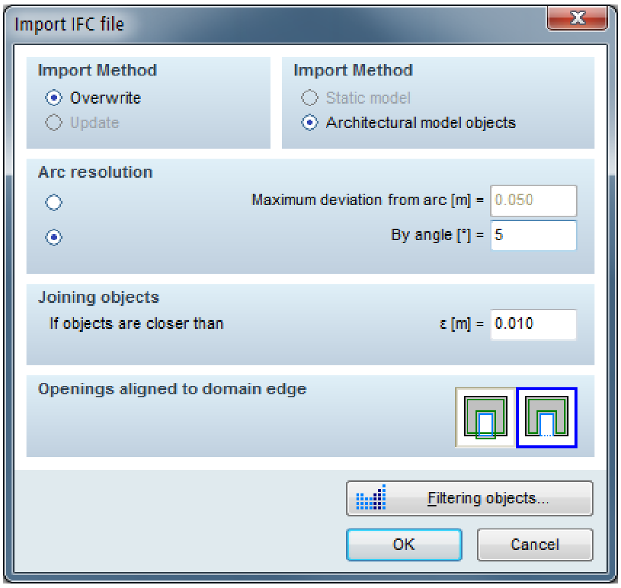
Hatching is represented by individual lines in a DXF file so in most cases it is not recommended to import them. If you need the hatching check this option.

IFC 2x, 2x2, 2x3, 2x4
*.ifc file

Import of this file type requires 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 Nemetscheck Allplan, Bocad and Xsteel. Programs.



Importing IFC files can extract the static model (if available) or the architectural objects overwriting or updating the existing information within the AxisVM model.

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.

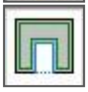
Architectural model objects

This option can overwrite or update existing architectural model information in the AxisVM model. AxisVM can read columns, beams, walls, slabs, roofs.

See… *4.9.21 Creating model framework from an architectural model*

Opening aligned to domain edge

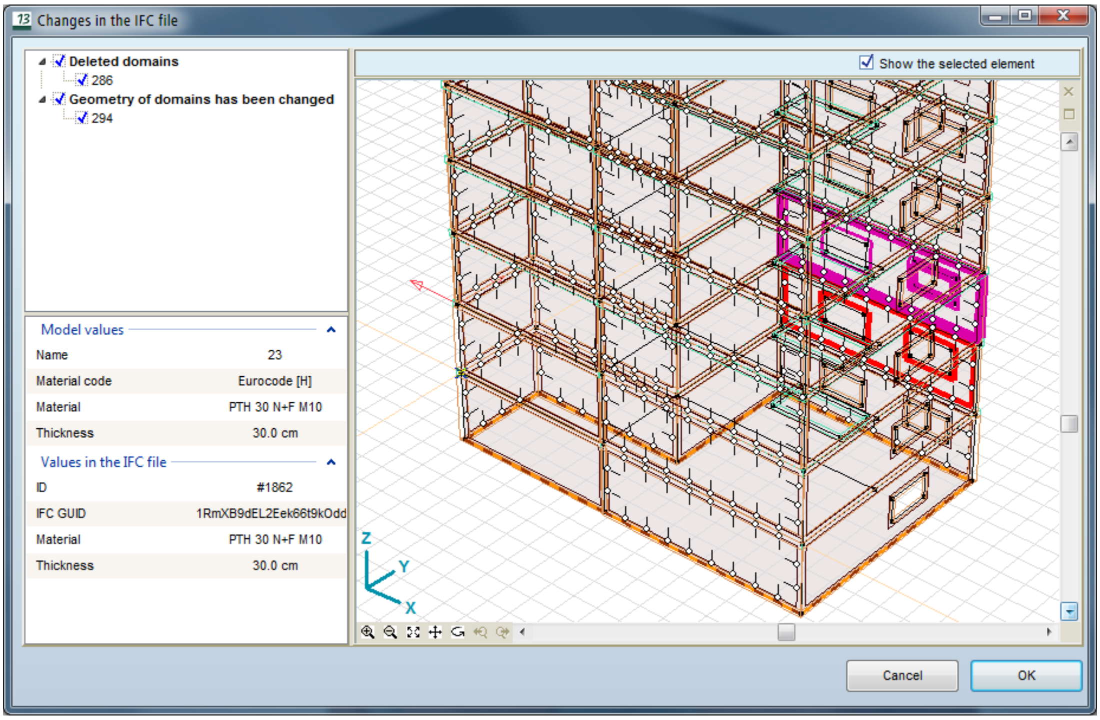
Import as opening



Adjust the domain outline

Filtering objects

This function lists new, modified and deleted obejcts 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.



When exporting a model from ADT (Architectural Desktop) turn off the automatic intersection of walls before creating the IFC file.

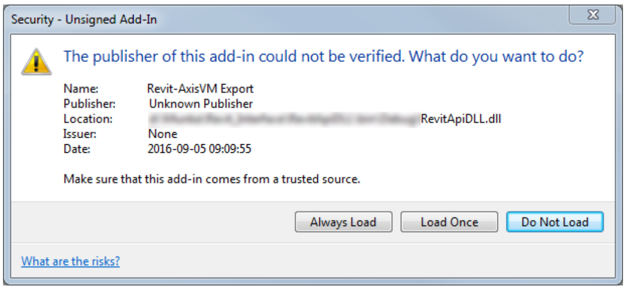
Import from Revit

AxisVM can import data through a direct connection between AxisVM and Autodesk Revit 2015 (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 extension file is created in folder *c:\Users\<UserName>\AppData\Roaming\Autodesk\Revit\Addins\yyyy*, where "yyyy" represents an integer value greater or equal to 2015. 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.**

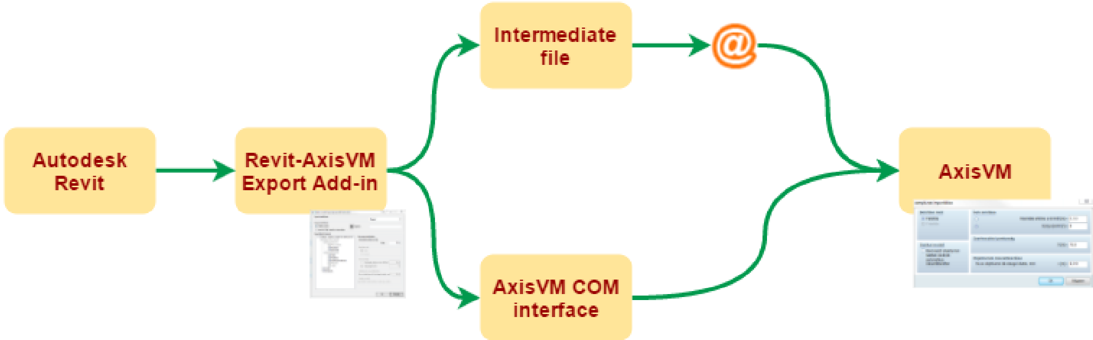
☞ ***Started from Revit2017 the popup window shown below will appear at the initialization phase of every add-in or third party executable.***



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.

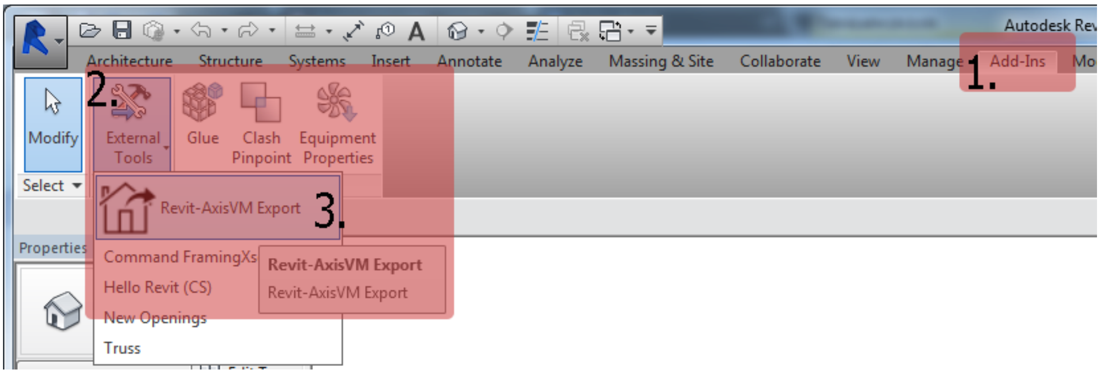
Using the REV module

The scheme of dataflow is



The process must be started **from Revit** as shown below:

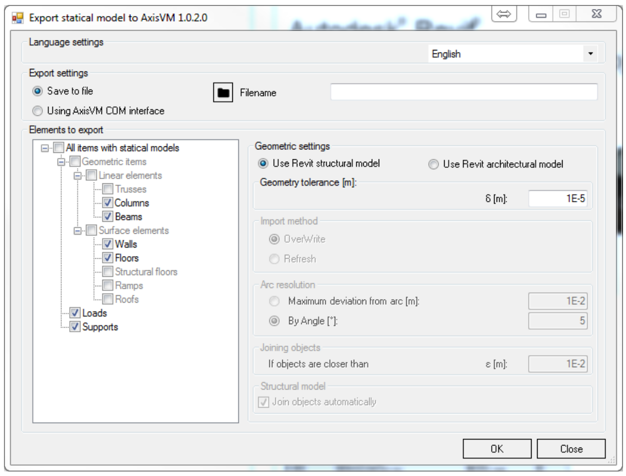
Add-Ins / External Tools / Revit-AxisVM Export



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.

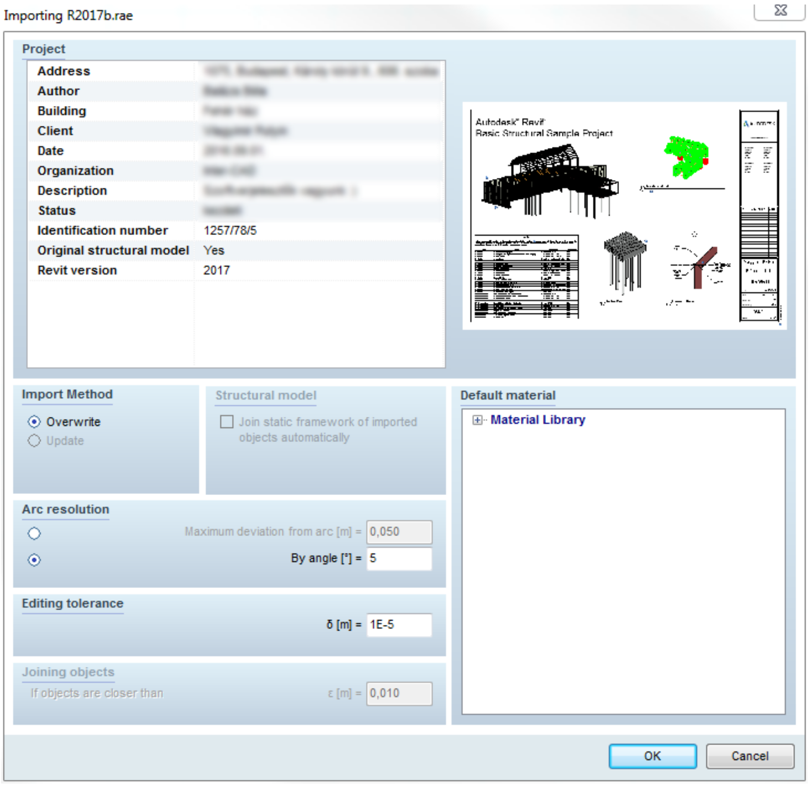


Import via an intermediate file

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. After a successful save operation, the file created can be imported into 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 require arcs to be converted into polylines. The conversion criterion can be set here.

Editing tolerance Tolerance used when calculating geometry.

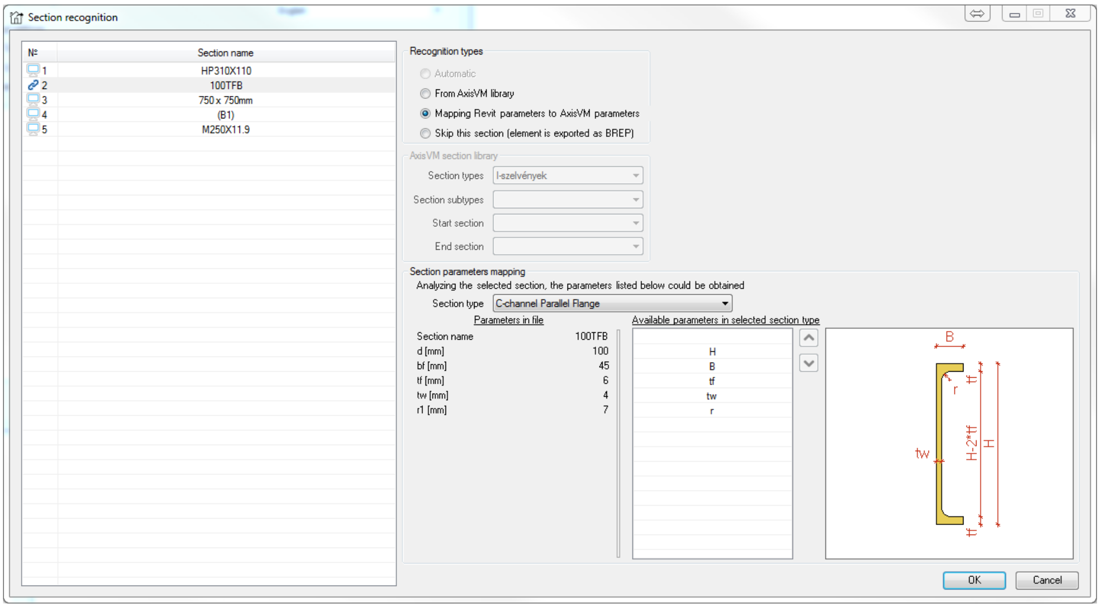
Joining objects To improve the efficiency of collision detection bounding boxes of objects are inflated a bit. This tolerance value can be set here. Active only if the checkbox on panel *Structural model* is checked.

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.

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.



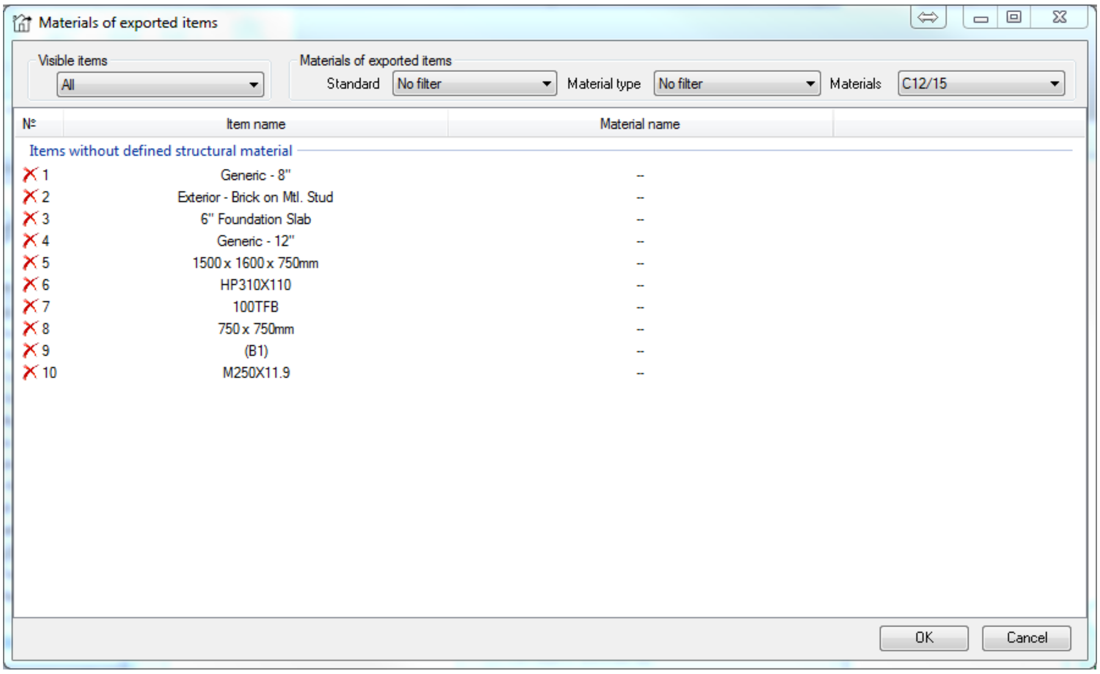
The list shows all sections in the model. Their icon shows whether the automatic recognition process succeeded or not.

There are three methods of cross-section identification:
Automatic The add-in queried section data from Revit succesfully.
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.

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.

☞ ***The add-in learns the parameter matching you defined so next time it will identify the section properties without user interaction.***

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.

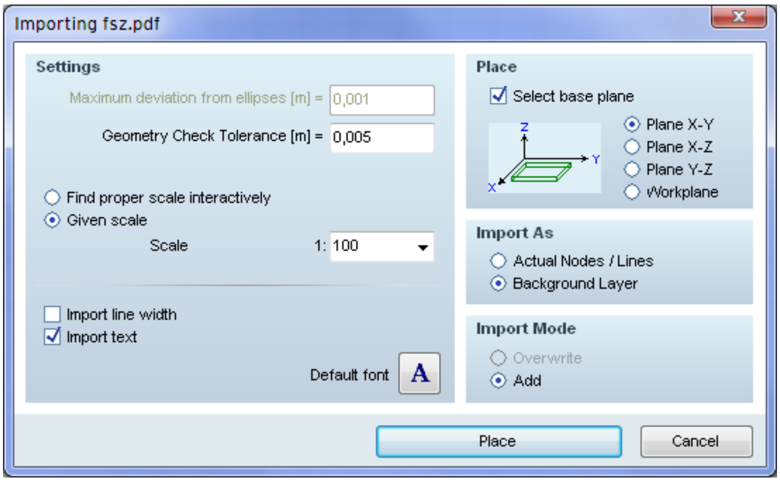


You can leave some elements unassigned. At the import phase, AxisVM will assign a default material to them.

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 are 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.***

*PDF *.pdf* 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.



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.

*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 *4.8.15 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.

*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.

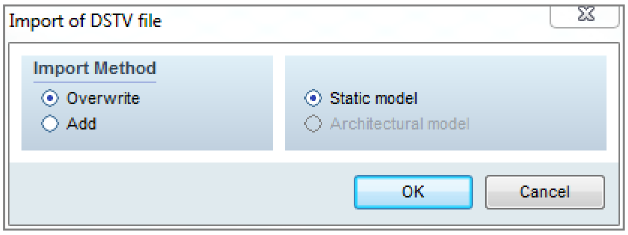
*Glaser -isb cad- *.geo file* Imports *.geo files exported by Glaser -isb cad- describing beam or surface models.

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.

After starting import, user is prompted the window below:



In as much as there are statical calculations embedded in the file and architectural modell too, the user can select between importing either the calcualtion or the architectural modell. When trying to import data onto an existing modell, the user can select between overwriting or just supplementing it.

3.1.7. Tekla Structures – AxisVM connection – TI module

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.

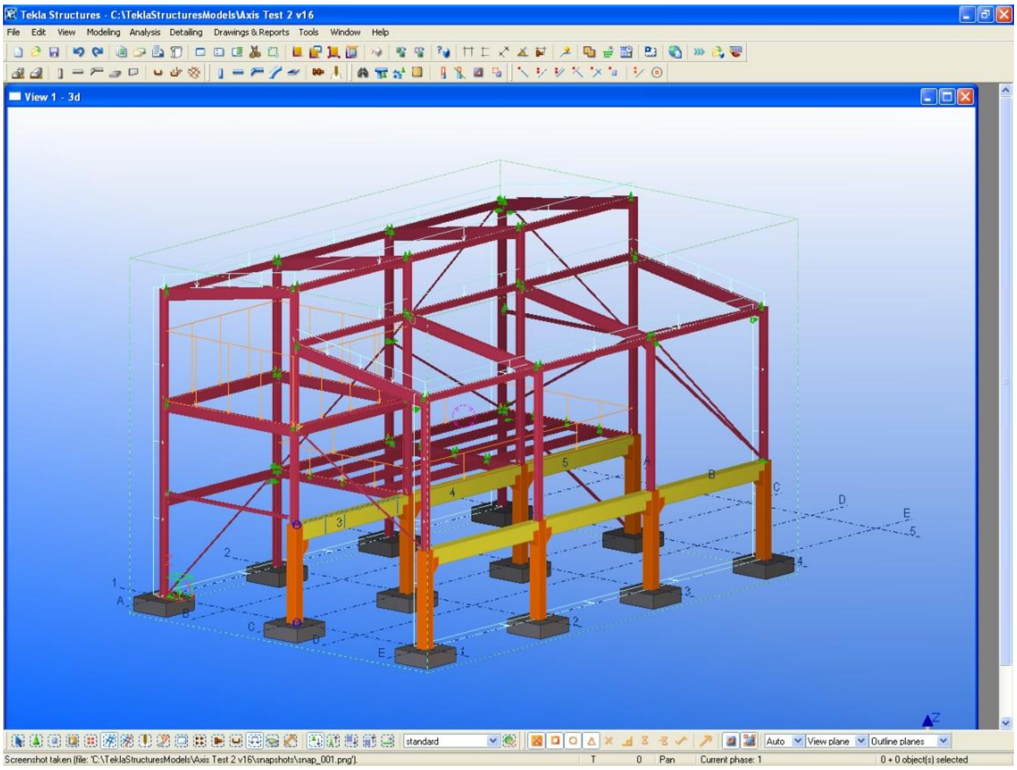
AxisVM setup automatically performs these registering operations, however if Tekla Structures is not installed the second registration cannot be completed. Therefore after installing Tekla Structures the registration has to be started again by running two batch files from the AxisVM program folder:

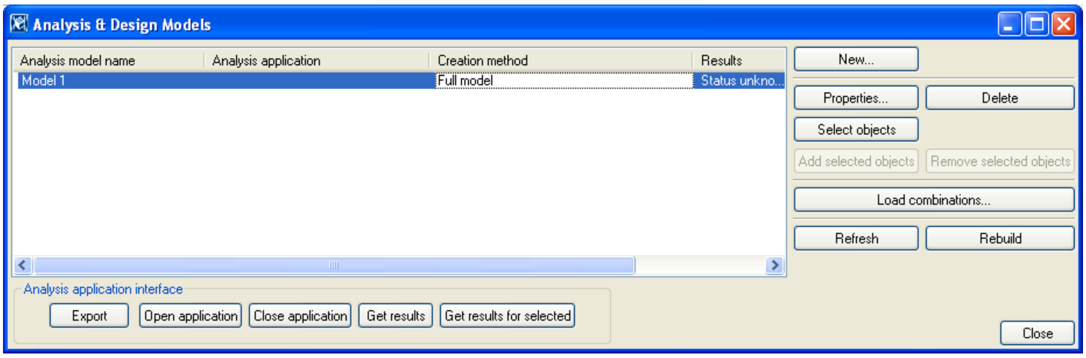
```
!REGISTER_AXISVM.BAT
!REGISTER_TEKLA.BAT
```

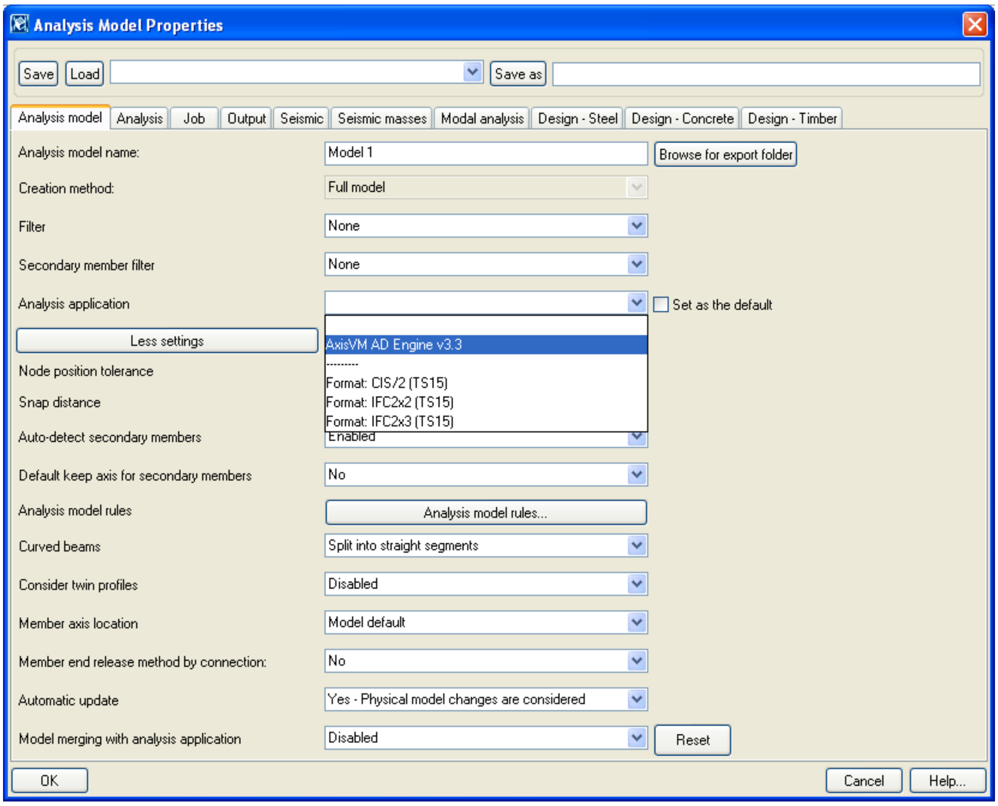
If connections fails any time it is recommended to run this registration again.

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.*

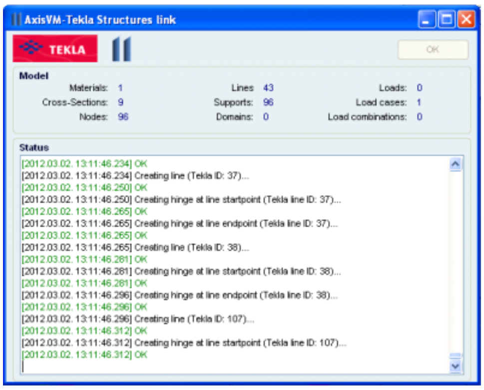




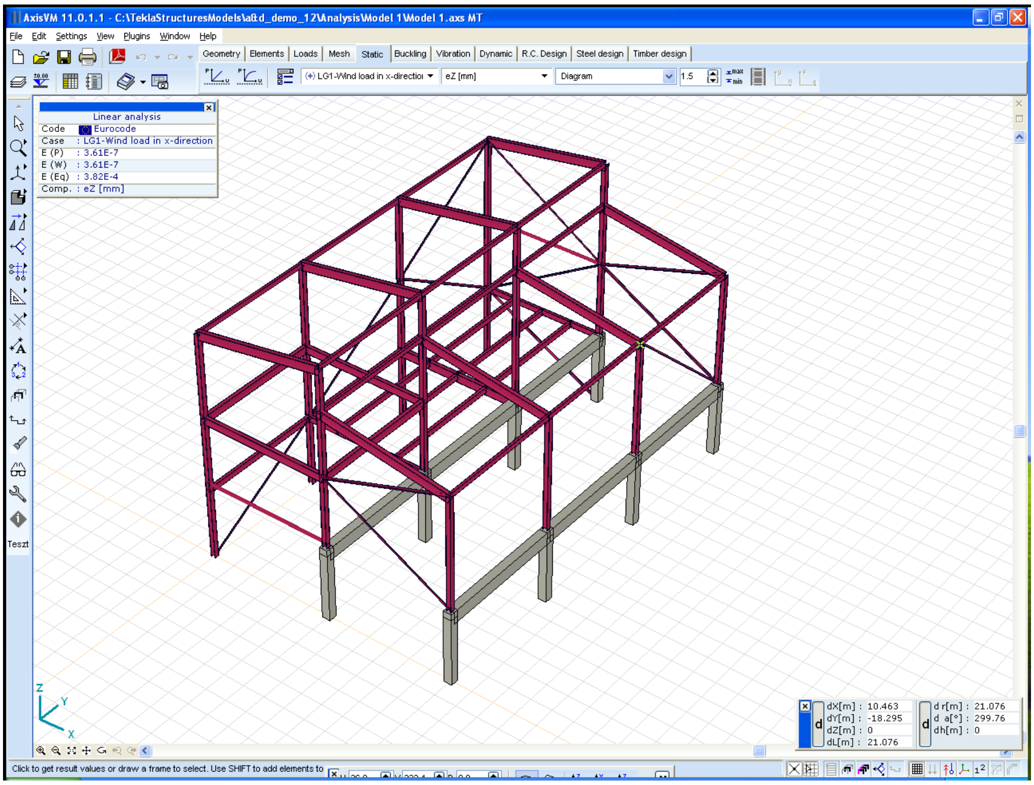


If AxisVM AD Engine does not appear in the dropdown list the registration was not successful and has to be repeated.

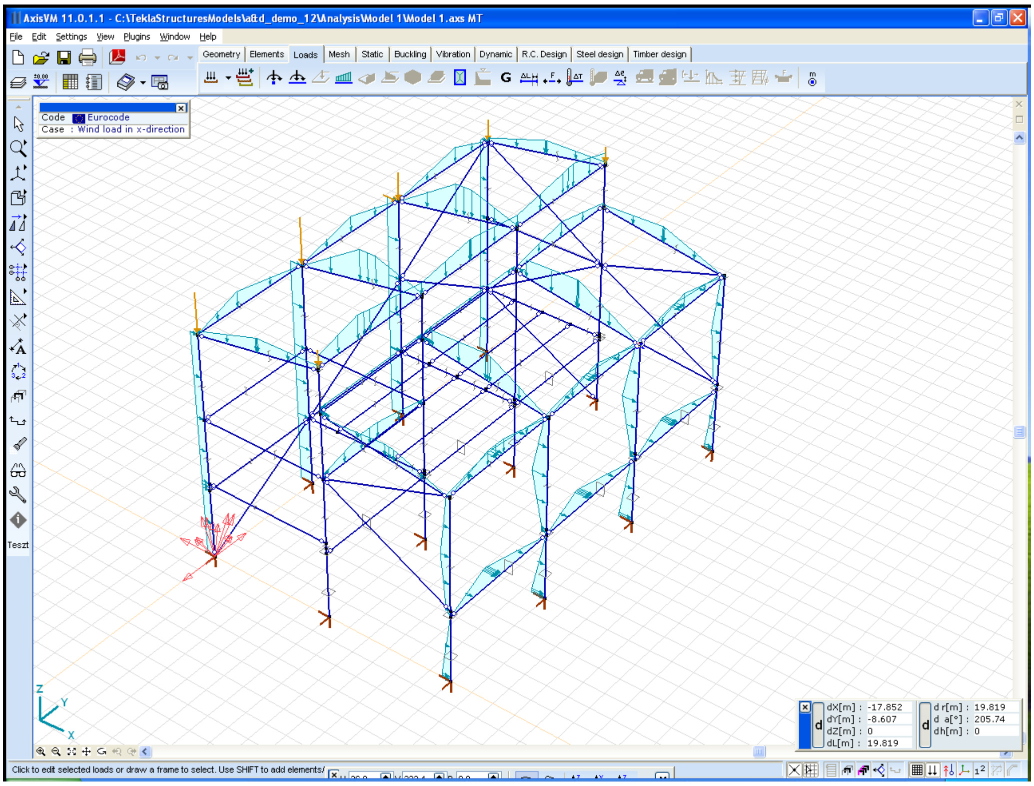
Getting back to the *Analysis & Design models* dialog click *Run* to start the transfer of the model. The process status is displayed in dialog. If the transfer is completed successfully click the OK button to see the model in AxisVM.



The model transferred to AxisVM:

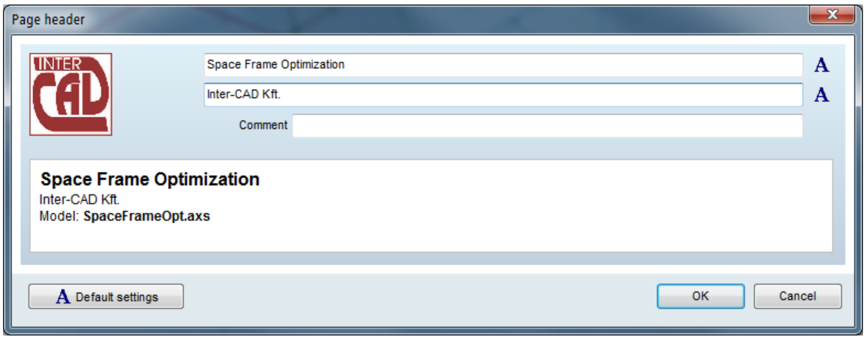


Loads and load cases specified in Tekla Structures are also converted.



3.1.8. 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...*

3.1.9. Print setup

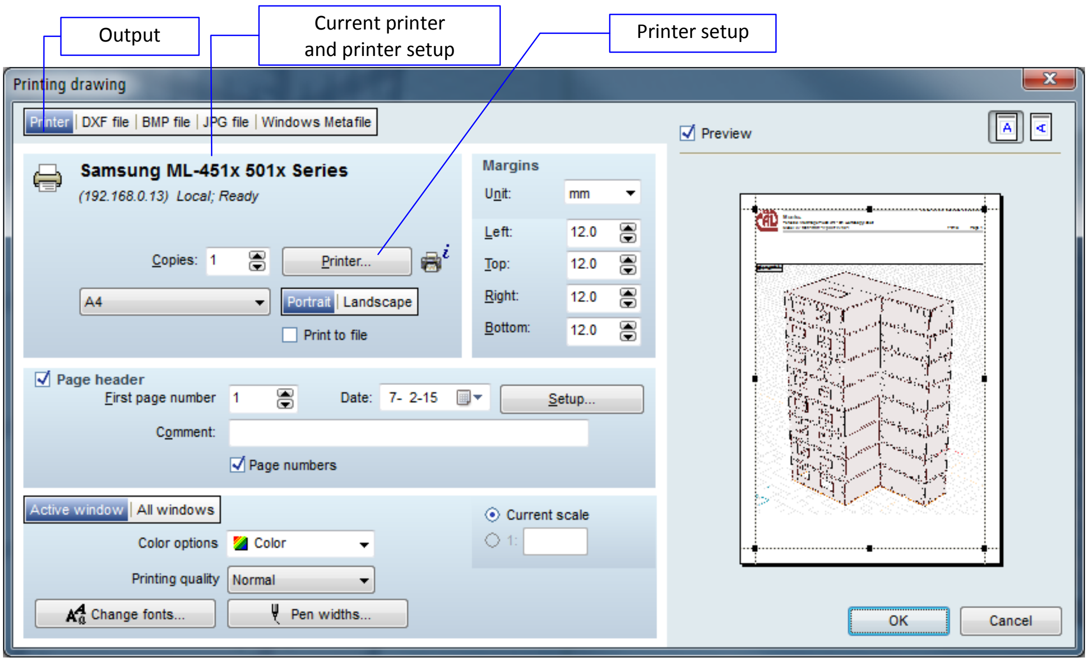
Allows setting the parameters of the default printer.
This is a standard Windows dialog therefore its language corresponds with the language of the installed operating system.

3.1.10. 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/plotter or to a graphics file (DXF, BMP 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)

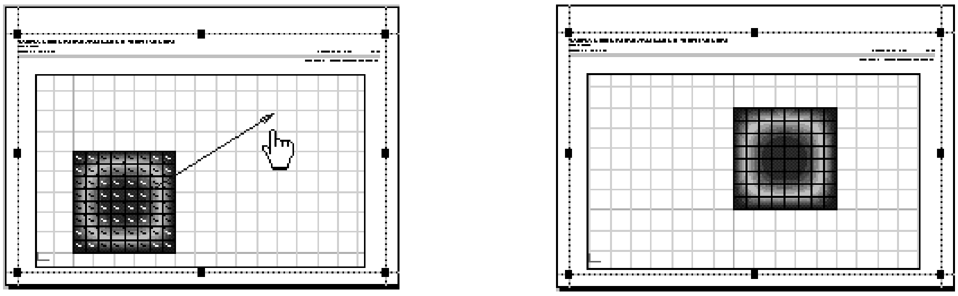
Lets you set the size and the units of the page margins. You can also drag margin lines within the preview area by their corner and midside handles.

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…** *3.1.8 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.

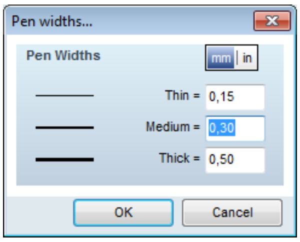
Paper size

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

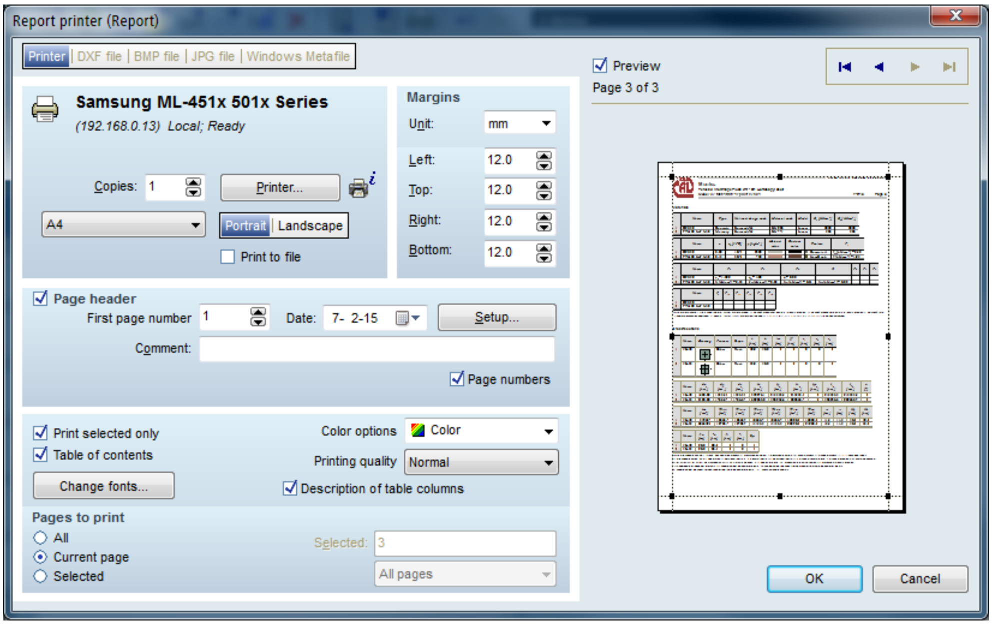
When *Print to File* is selected the printing is redirected to a file, `name.prn` that you can print anytime later.

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

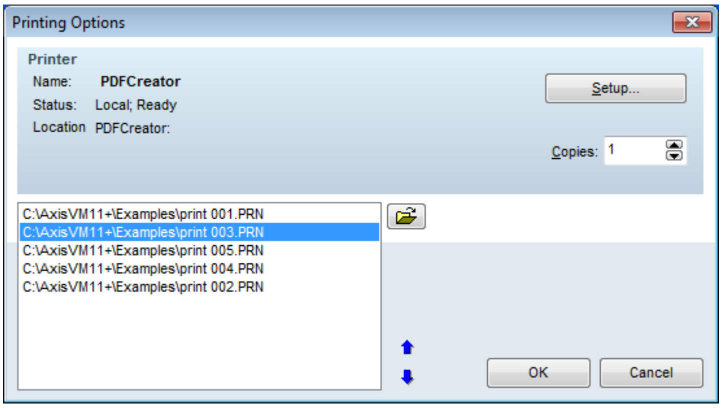
Printing table of contents at the beginning of the report is optional.

Description of table columns

If this option is checked a summary appears at the end of each table, describing the meaning of column headers.

3.1.11. Printing from file

You can print the prn file you created from the following window.

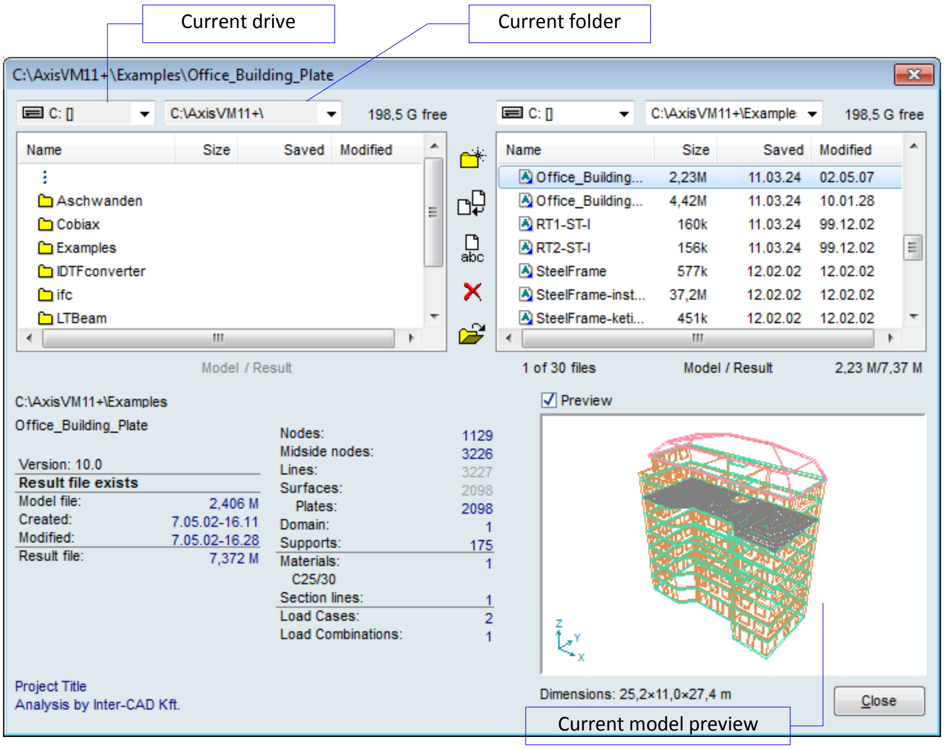


You can print more than one prn file at a time. You can set the printing order with the up/down arrows in the right of the file list box, or dragging the file names to a new position with the mouse.

3.1.12. Model Library

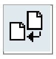
The *File/Model Library* command lets you preview, get information and manage your model files. As in *Open* and *Save as* dialog windows the standard file access dialog box items are displayed, but in the list box you can select multiple files.

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.

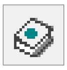
Preview

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.

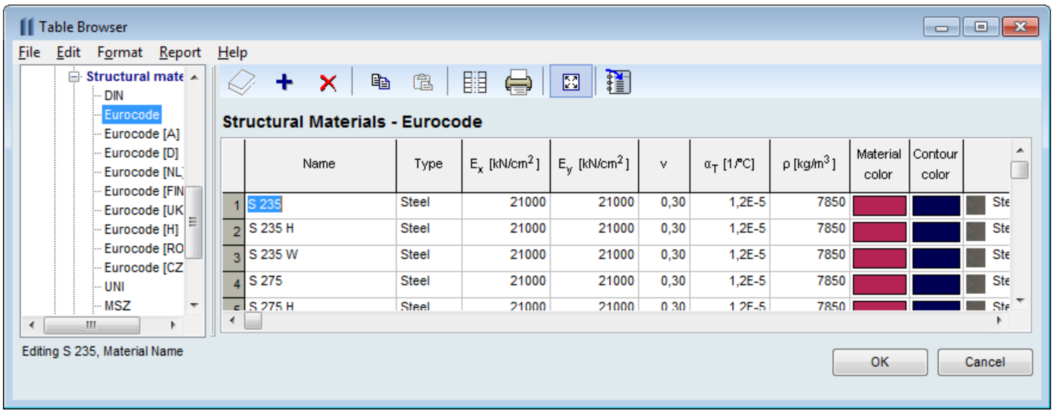
Close

Quits the Model Library.

3.1.13. Material Library



AxisVM provides a preloaded material library (that contains the most frequently used structural materials) and allows you to create material property sets that you can use over and over again in many different models. You must assign different names to each material property set.



The material library window can also be opened using the Table Browser icon and by selecting Libraries/Material Library. **See...** *4.9.8 Line elements, 4.9.21 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 MSz, Eurocode, DIN-1045, DIN-1045-1, NEN, SIA-162, a STAS 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

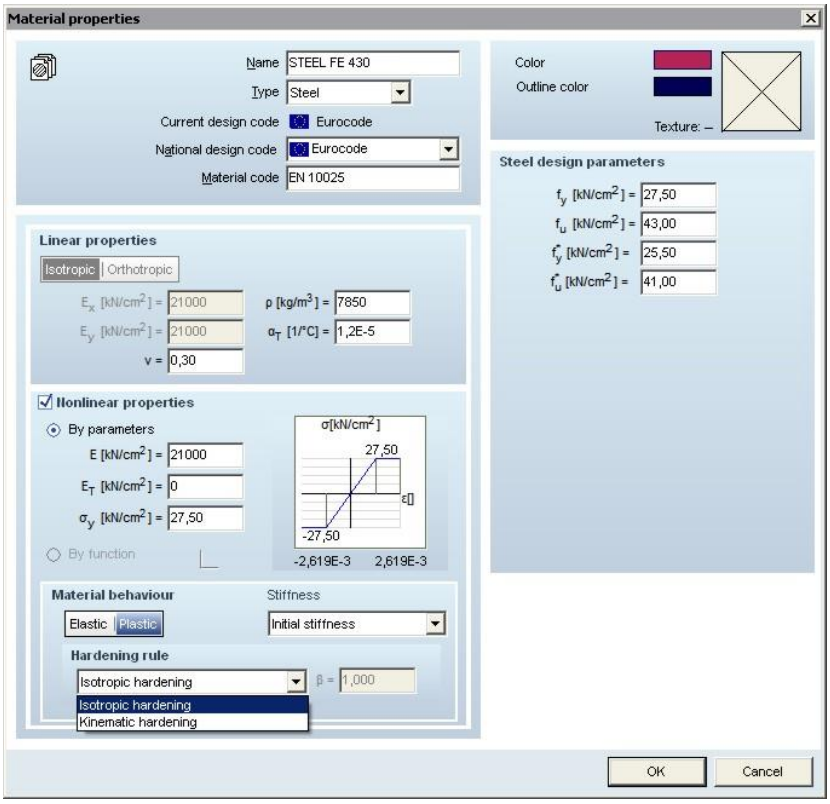
[Ctrl+Ins]

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, aluminum, 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

Calculation of further material properties

$$E_z = \max\{E_x, E_y\}; \quad \nu_{ij} = \begin{cases} \nu & \text{if } E_i \geq E_j \\ \nu \dfrac{E_i}{E_j} & \text{if } E_i < E_j \end{cases} ; \quad G_{ij} = \frac{E_i E_j}{E_i + E_j + 2\nu_{ij} E_j}$$

where $ij = \{xy, xz, yz\}$

In case of timber materials:

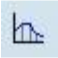
ρ is the air dry mass density (12% humidity) and, the modulus of elasticity E is based on bending test results. The effect of time (relaxation) is not taken into account.

Nonlinear properties

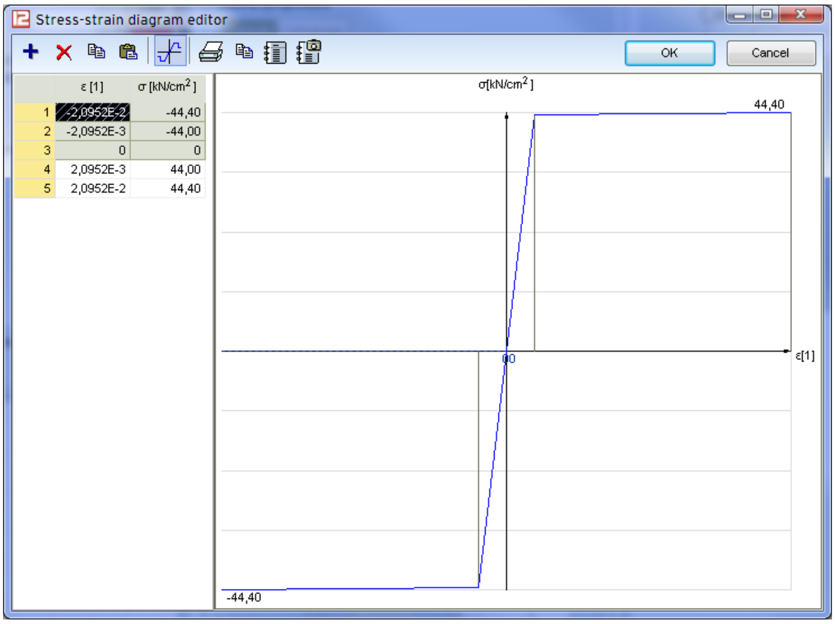
By parameters

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_y [kN/cm^2]	Yield stress

By function



Stress-strain diagram editor



Most of the toolbar functions are the same as in the time history diagram editor. **See…** *4.10.28 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.

Nonlinear material behaviour

The material model is only valid in the domain of small-strains.

Elastic Nonlinear elastic behaviour. The point representing the state of the material moves along the σ-ε curve when loads increase or decrease. No irreversible deformation.

Plastic Plastic behaviour. Increasing load moves the point representing the state of the material along the σ-ε curve, decreasing load moves it parallel to the initial section of the curve.

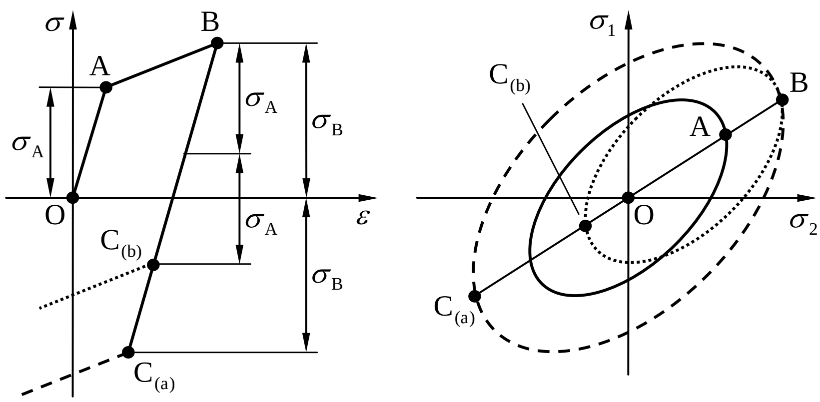
Initial stiffness

Initial stiffness The materially linear initial stiffness matrix is used for the evaluation of the global stiffness matrix.

Tangent stiffness The materially nonlinear tangent stiffness matrix is used for the evaluation of the global stiffness matrix.

Strain hardening

The strain hardening could be:
(a) Isotropic: $\beta = 1$ (b) Kinematic: $\beta = 0$

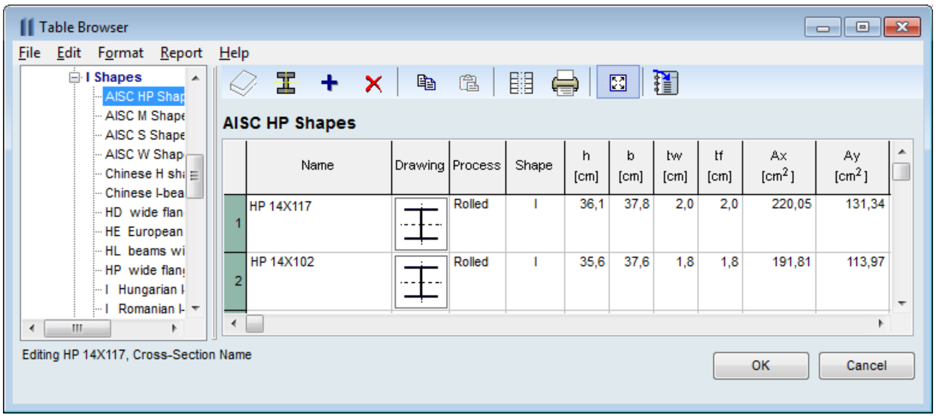


Design Parameters Design parameters depend on the material type and the design code.

Steel	EC, DIN 1045-1, 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)
	NEN	f_{yd}	Yield stress
		f_{yt}	Ultimate stress
		f_{yd}*	Yield stress (40mm < t < 100mm)
		f_{yt}*	Yield 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
	DIN 1045-1	f_{ck}	Characteristic compressive cylinder strength at 28 days
		$f_{ck,\ cube}$	Characteristic compressive cylinder strength of cube
		γ_c	Partial factor
		α	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
	NEN	f_{ck}'	Characteristic compressive cylinder strength at 28 days
		Φ	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

3.1.14. 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 library name, library file name and a cross-section type. 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:

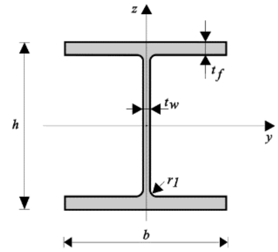
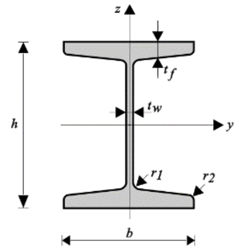
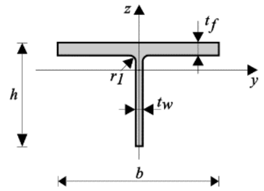
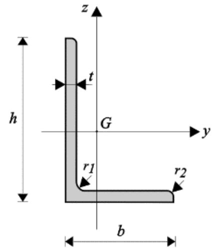
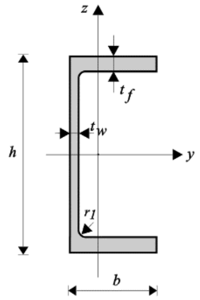
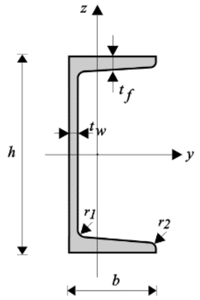
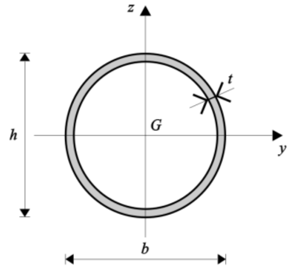
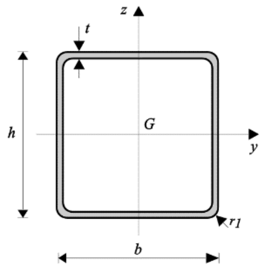
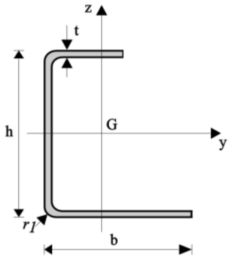
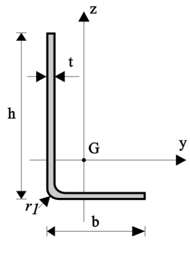
Name	
Process	*Rolled, welded, cold-formed, other.*
Shape	*I (H, W), U, L, Pipe, Round, Rectangle, C, Z, S, J, T, Box, Custom*

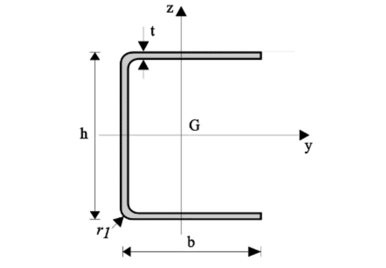
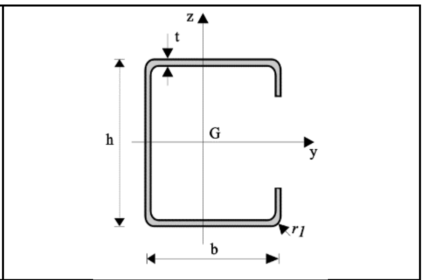
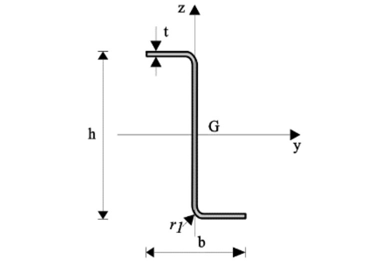
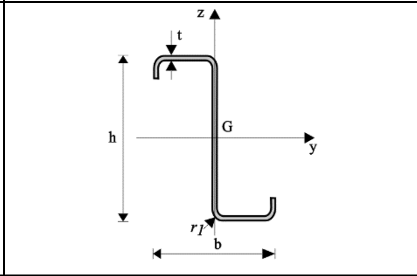
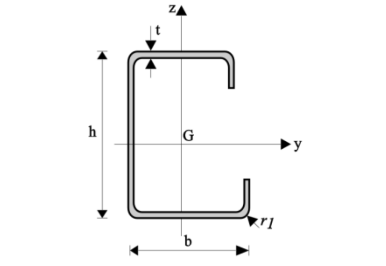
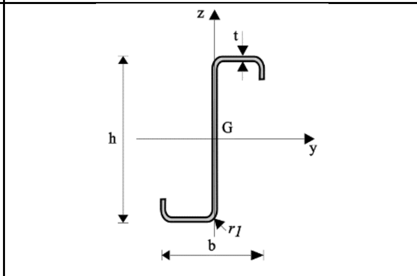
Cross-section properties When creating a new cross-section in the table all property values have to be entered.

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
$I\omega$	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
S.p.	Stress calculation points

The Cross-section Library contains different type of cross-sections:

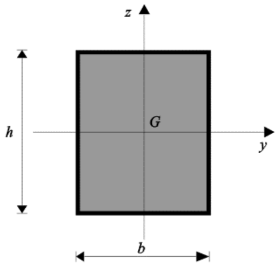
Steel cross-section

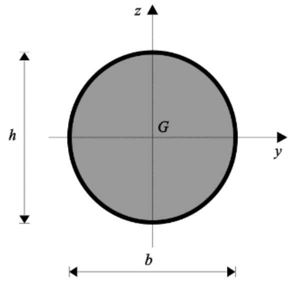
	
Hot rolled parallel flange I beam	*Hot rolled tapered flange I beam*
	
Hot rolled T shape	*Hot rolled angle*
	
Hot rolled parallel flange channel (U shape)	*Hot rolled tapered flange channel (U shape)*
	
Cold-formed pipe	*Cold-formed and hot rolled (RHS) box shape*
	
Cold-formed J shape	*Cold-formed 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)*

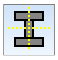
Concrete cross-sections

The concrete cross-sections are listed starting from the size 20x20 to size 80x80 cm in steps of 2 and 5 cm.





3.1.14.1. Cross-Section Editor



The Cross-section Editor allows you to edit thin and thick walled cross-sections. You can use parametric circular, rectangular, ring and polygonal shapes, or any shape listed in the cross-section libraries to edit composite cross-sections. The shapes used to build a new 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...** *4.9.8 Line elements*
The editor can be used when creating a native model from an architectural model through the IFC interface. **See...** *4.9.21 Creating model framework from an architectural model*

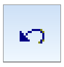
Editor Keys

See... *2.5 Using the cursor, the keyboard, the mouse*

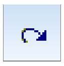
Toolbar Most important functions are available from the toolbar.

Prints the cross-section. **See...** *3.1.10 Print*

Adds the image of the cross-section to the Gallery.
See... *3.2.11 Saving drawings and design result tables*



Undoes the last operation.



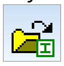
Redoes the operation which was undone.

Copies the image of the cross-section to the Clipboard.

From Cross-section Library

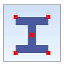
Loads a cross-section from the Cross-section Library. Only thick or thin-walled cross-sections are available depending on the cross-section editor tab position.

From DXF file



Contour of thick walled cross-sections can also be imported from a DXF file.

Stress-points

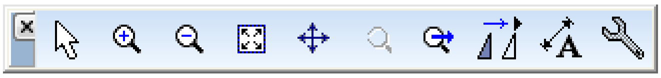


You can specify the points you want to calculate stresses for. The default stress-point is the center of gravity. You can specify up to 8 stress-points for each cross-section.
When applying a move command the stress-points can also be moved.
In case of thin-walled sections the shear flow is calculated only along the centerlines. 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 higest 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*.

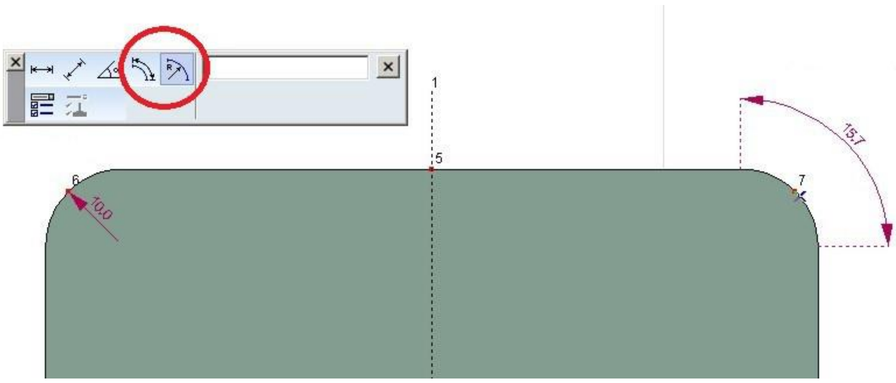
The only difference is that this Icon bar can be moved above the menus at the top or at the bottom but it is not dockable.



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

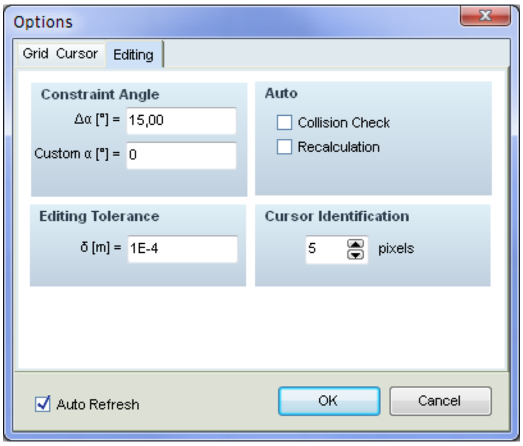


Dimensioning tools can be selected from the upper toolbar of the palette. These are orthogonal and aligned dimension lines, angle, radius and arc length dimensions.
The bottom toolbar contains the buttons for fine tuning dimension line properties and choosing the smart dimension line option which places multiple dimension lines between the start point and the end point.

Dimensioning

Select the type of the cross-section dimension line (orthogonal, aligned, angle) from the top toolbar.

The bottom toolbar contains the buttons for fine tuning dimension line properties and choosing the smart dimension line option.

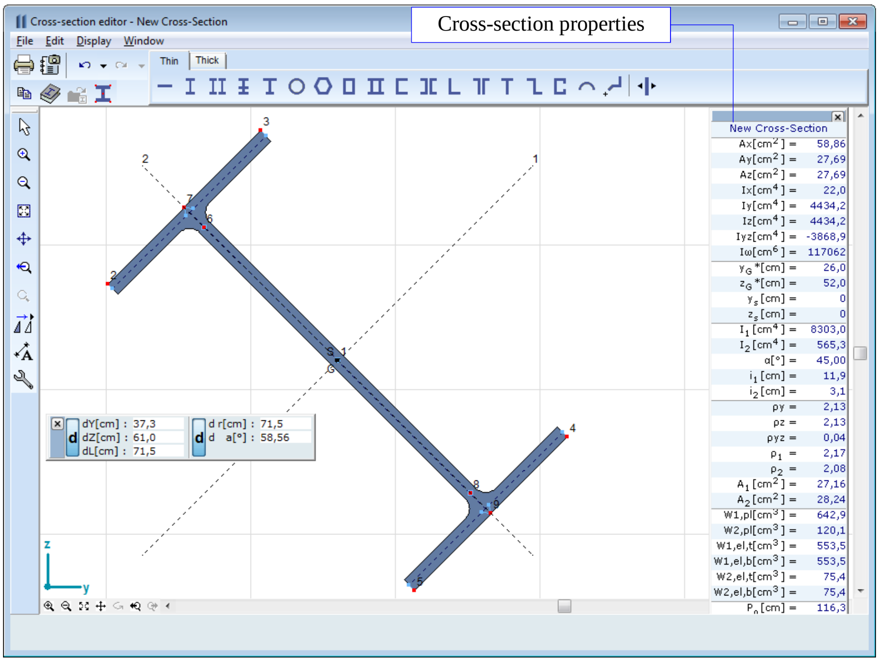


The Options dialog allows editing constraint angles and editing tolerance.

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.

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.).

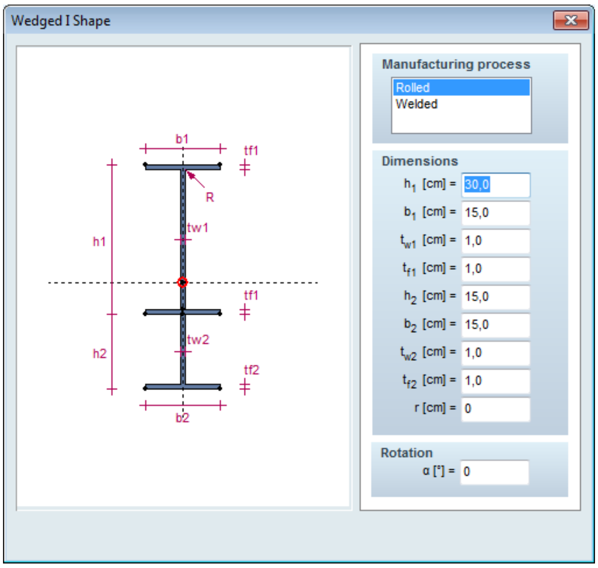
Rotation Lets you define a rotation by angle α. The default value is 0.

I shape

,

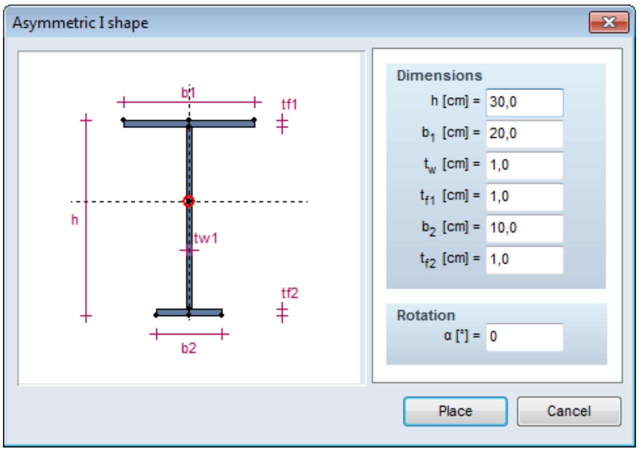
Wedged I shape

Definition of an I or wedged I shape by its height, width, web and flange thicknesses and a fillet radius.



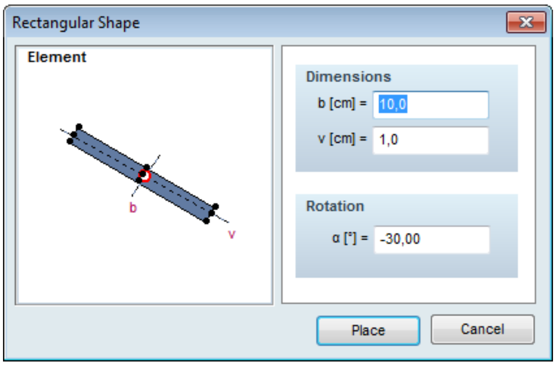
Asymmetric I shape

Definition of an asymmetric I shape by its height, width, web and upper / lower flange dimensions.

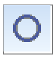


Rectangular

Definition of a rectangle by its parameters *b* (width), *v* (thickness), and α, with *b* > *v*.

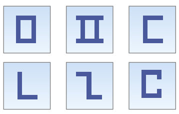


Pipe



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 shapes



Definition of cross-sections by height, width, thickness and in the case of rolled or bended cross-sections by the corner/fillet radius.

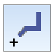
Double shapes

The base cross-section can be defined parametrically (width, height, web and flange thickness) or taken from the Cross-section Library.

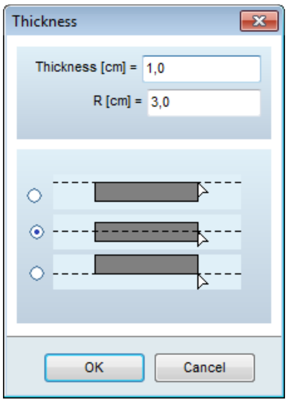
Special parameters for double shapes:

distance:	*a*
orientation :	facing or back-to-back (in case of 2U)

Polygonal Definition of a polygonal shape.



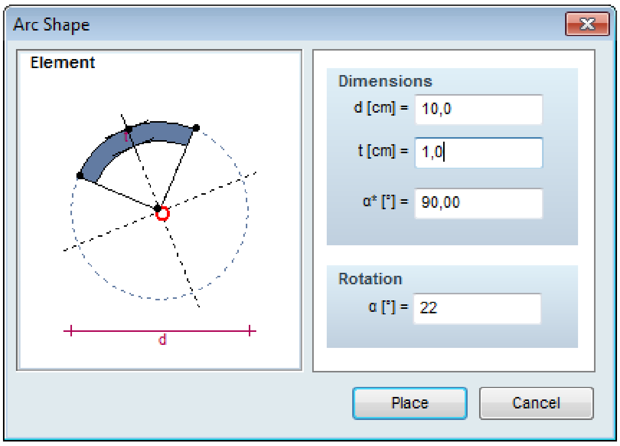
Before the definition the position of the control line of the segment can be selected:



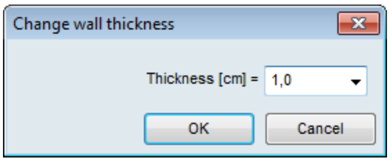
1. left side
2. center line
3. right side

R: Rounding (corner and fillet) radii

Arc shape Definition of an arc shape by its diameter, central angle and 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.

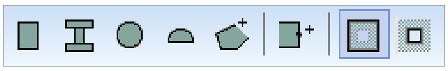
Delete Using the **[Del]** key you can invoke the Selection Icon Bar, and select the components you want to delete.
When deleting a component the stress-points will also be delete.

Stress-point Deletes the selected stress-points.

☞ ***You cannot delete the default stress-point (the center of gravity).***

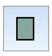
Options Lets you set the grid size, cursor step, and the zoom factors.

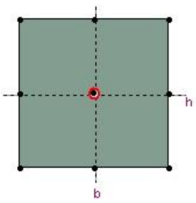
Thick-walled cross-sections



Parametric shapes:

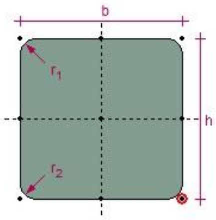
Rectangle





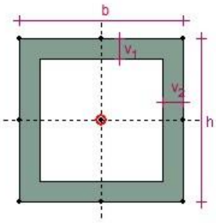
Rectangle with a width of *b* and a height of *h*.

Rounded rectangle



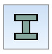
Rounded rectangle with a width of *b*, height of *h* and rounding radii *r1, r2.*

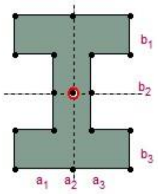
Hollow rectangle



Hollow rectangle with a width of *b*, height of *h* and *v1, v2* wall thickness.

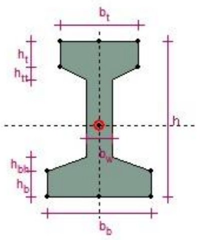
I shape





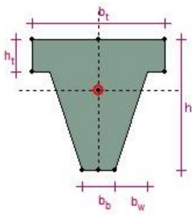
I shape defined by a_1, a_2, a_3, b_1, b_2, b_3 parameters.
One of the (a_1, a_3), (b_1, b_3) parameters can be set to zero to define T, U, L shapes

Haunched I shape



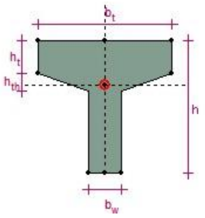
Haunched I shape defined by the b_w web thickness, b_t, b_b top and bottom width, *h* total height and h_t, h_{th}, h_b, h_{bh} parameters.

T shape with haunched web



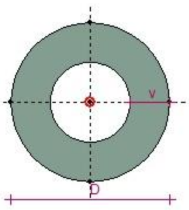
T shape with haunched web defined by b_w, b_b, b_t width, a *h* total height and h_t, flange thickness.

T shape with haunched flange



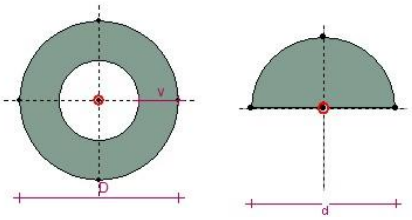
T shape with haunched web defined by b_w, web width b_t total width, *h* total height and h_t, h_{th} parameters

Hollow circle



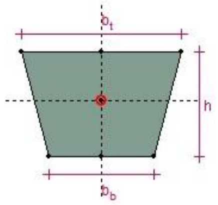
Hollow circle defined by the *D* outer diameter and the *v* wall thickness.

Round or semicircular shape



Round or semicircular 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.

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:

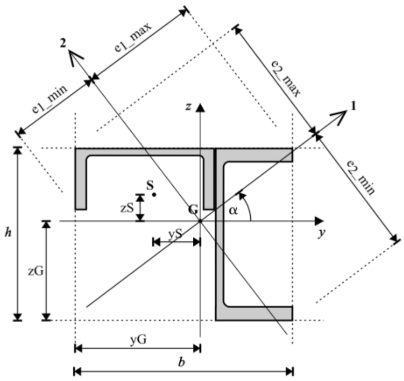
AxisVM calculates Ax, Iy, Iz, Iyz by integration, **Ay, Az, Ix, Iω**, ρ_y, ρ_z, ρ_{yz}, ρ_1, ρ_2, **A1, A2** by performing a finite element analysis of the cross-section.

☞ ***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 determinded.***

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 ***2***nd 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
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



$$I_1 = \frac{I_x + I_y}{2} + \sqrt{\left(\frac{I_x - I_y}{2}\right)^2 + I_{xy}^2}$$

$$I_2 = \frac{I_y + I_z}{2} + \sqrt{\left(\frac{I_y - I_z}{2}\right)^2 + I_{yz}^2}$$

$$\tan(2\alpha_n) = \frac{2n_{xy}}{n_x - n_y}$$

$-90° < \alpha \leq +90°$, 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 the cross-section was entered with nonzero for the shear area.

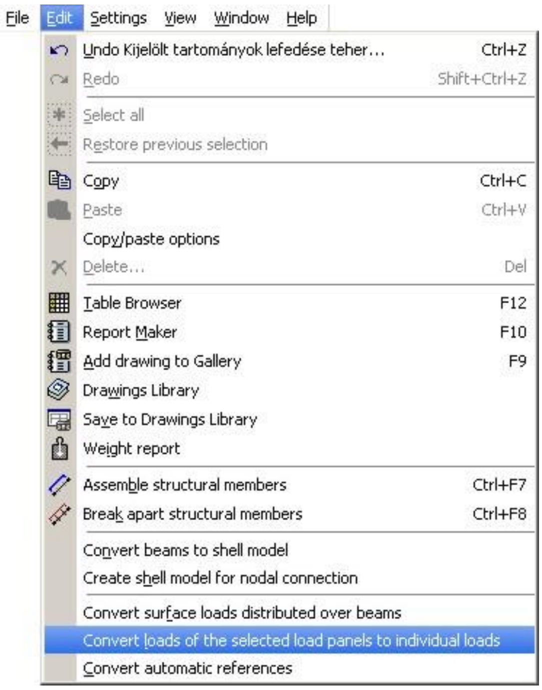
The shear areas are used by the rib elements, $A_y = 0$ and $A_z = 0$.
In the steel design module, the shear areas are calculated according to the corresponding design code, instead of using the values entered here.

$A_y = A_x/\rho_y \quad A_z = A_x/\rho_z$ where ρ = shear factor

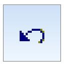
3.1.15. Exit

[Ctrl]+ [Q] Exits the program.

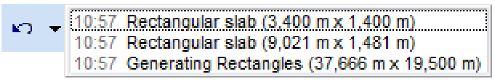
3.2. Edit



3.2.1. Undo

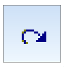


Alt+BkSp

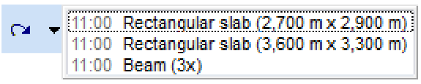


Undoes the effect of the previous commands. To undo a sequence of actions (more levels), click the down arrow next to the Undo icon, and then select the actions you want to undo based on the time or type of the commands.
You can set the number of undo/redo levels (maximum 99) in the Main menu *Settings / Preferences / Data integrity* dialog box.

3.2.2. Redo

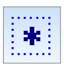


Shift+Alt+BkSp



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.

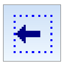
3.2.3. Select All



Num *

See... *2.16.1 Selection*

3.2.4. Restore previous selection



See... *2.16.1 Selection*

3.2.5. 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.

3.2.6. Paste

[Ctrl]+ [V]

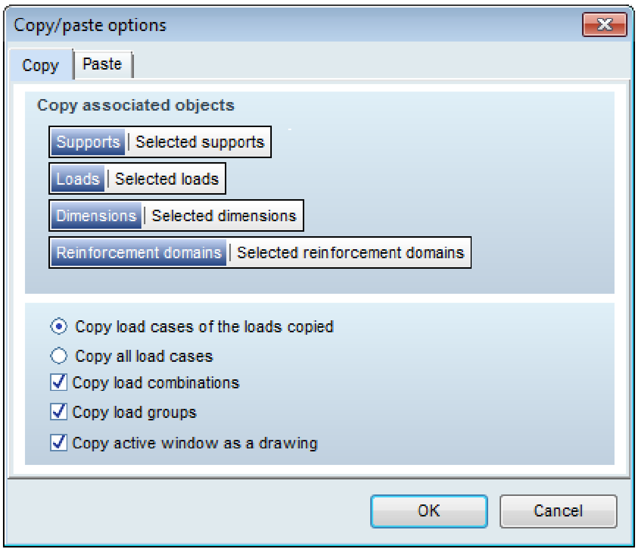
Pastes AxisVM elements from the Clipboard. For paste options see next chapter: Copy / paste options.

3.2.7. Copy / paste options

Copy options

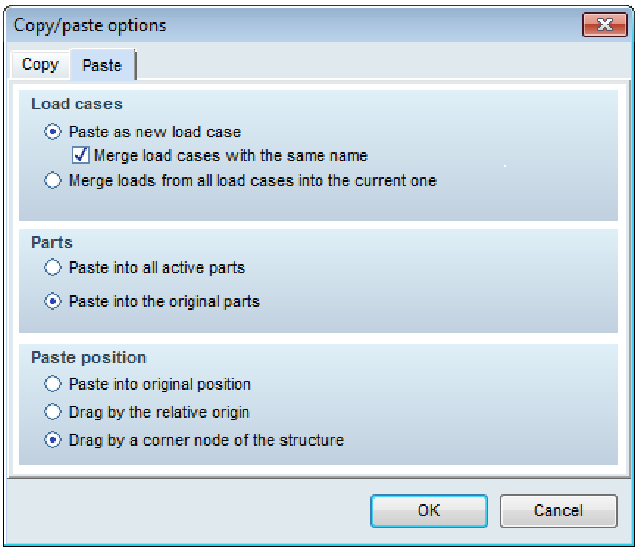
Selected elements are always copied to the Clipboard. User-defined parts containing the selected elements are also copied.
If domains, beams, ribs, trusses are copied certain associated objects (supports, loads, dimension lines, reinforcement domains) are also copied.

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.

3.2.8. Delete

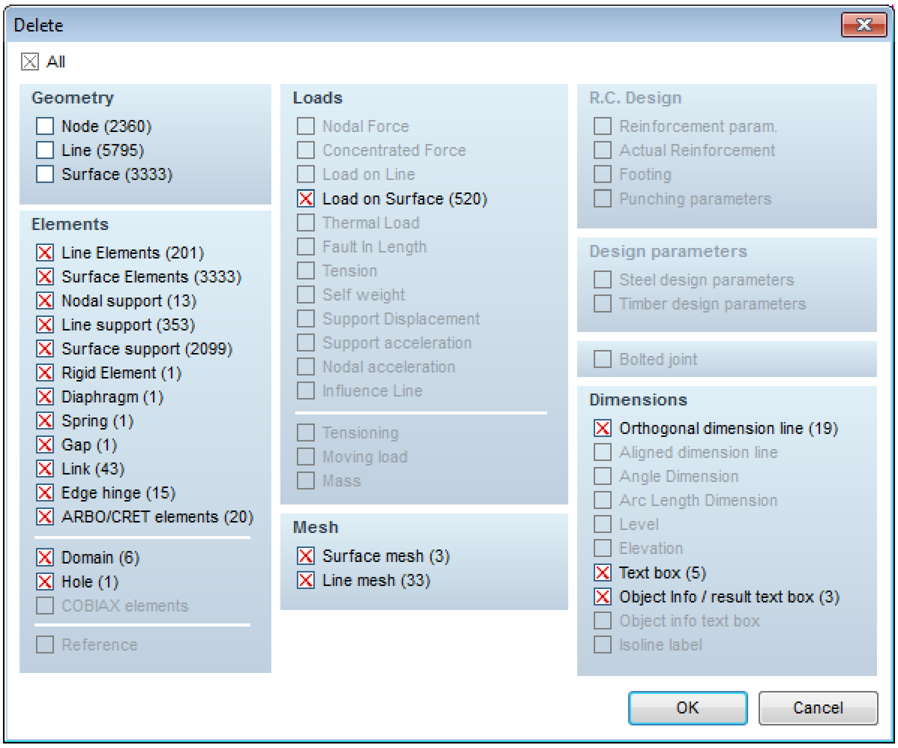
[Del]

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 for deletion. Deleting finite elements will not delete the respective geometric entity, but will delete the loads.

References Lets you select references for deletion. All finite elements that use the deleted references, and the associated loads will be deleted too.

Mesh Lets you remove mesh from domains.

R.C. Design Lets you select the reinforcement parameters attached to the selected elements for deletion. Footing parameters are also deleted.

Steel / Timber design 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.

3.2.9. Table Browser

[F12]

See... *2.9 Table Browser*

3.2.10. Report Maker

[F10]

See... *2.10 Report Maker*

3.2.11. 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.

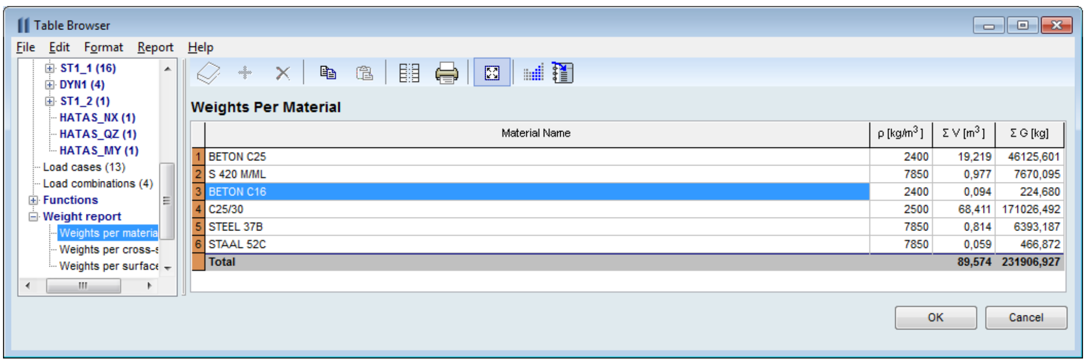
Windows metafiles *(.WMF, .EMF)* store a series of drawing commands so they can be scaled and printed in any size in the same quality. However if you choose hidden line removal or a rendered view drawn by OpenGL technology metafiles will contain only bitmaps.
To get a high resolution rendered view print 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.*

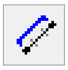
3.2.12. Weight Report

[F8]

The weight of the entire model, selected elements or details can be listed in tabular form per material, per cross-section or surface type.

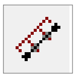


3.2.13. Assemble structural members


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.

3.2.14. Break apart structural members


Shift+B

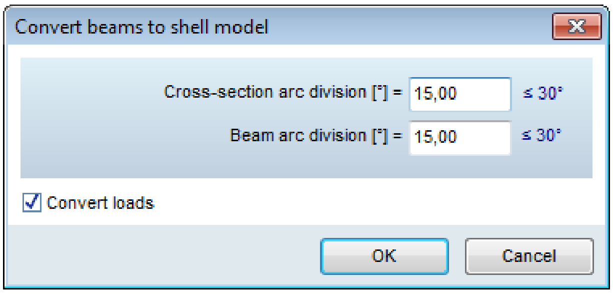
The *Break apart structural members* menu command breaks apart line elements created with the *Assemble structural members* command.

3.2.15. Convert surface loads distributed over beams

This menu item converts selected surface loads distributed over beams into individual distributed beam loads.

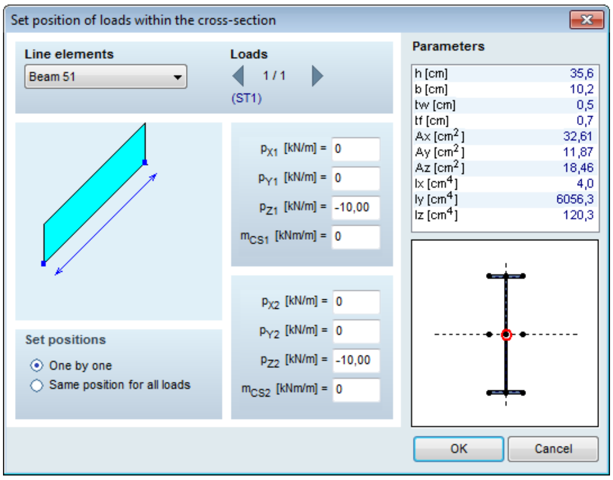
3.2.16. Convert beams to shell model

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.



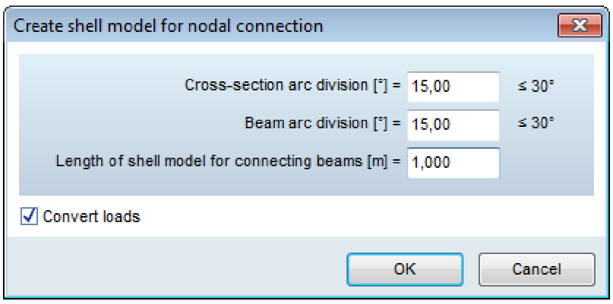
Convert loads

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.



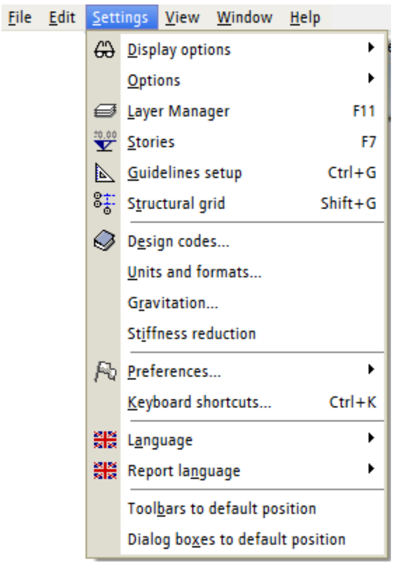
3.2.18. Convert loads of the selected load panels to individual loads

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.

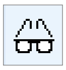
3.2.19. Convert automatic references

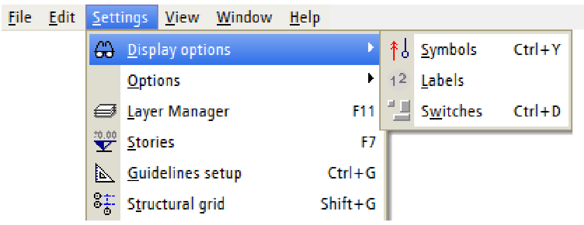
This menu item converts automatic references assigned to line or surface elements into reference vectors.

3.3. Settings



3.3.1. Display options



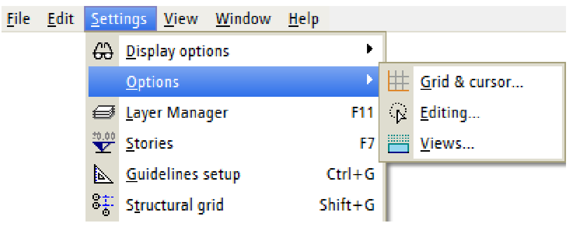


Symbols [Ctrl+Y] **See...** *2.16.18 Display options*

Labels **See...** *2.16.18 Display options*

Switches [Ctrl+D] **See...** *2.16.18 Display options*

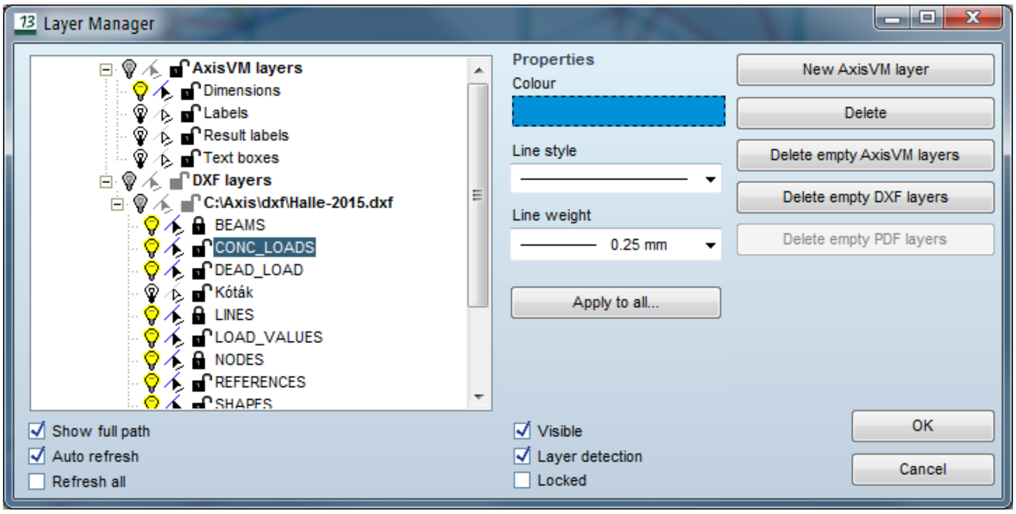
3.3.2. Options



See... *2.16.19 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 ont he 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.

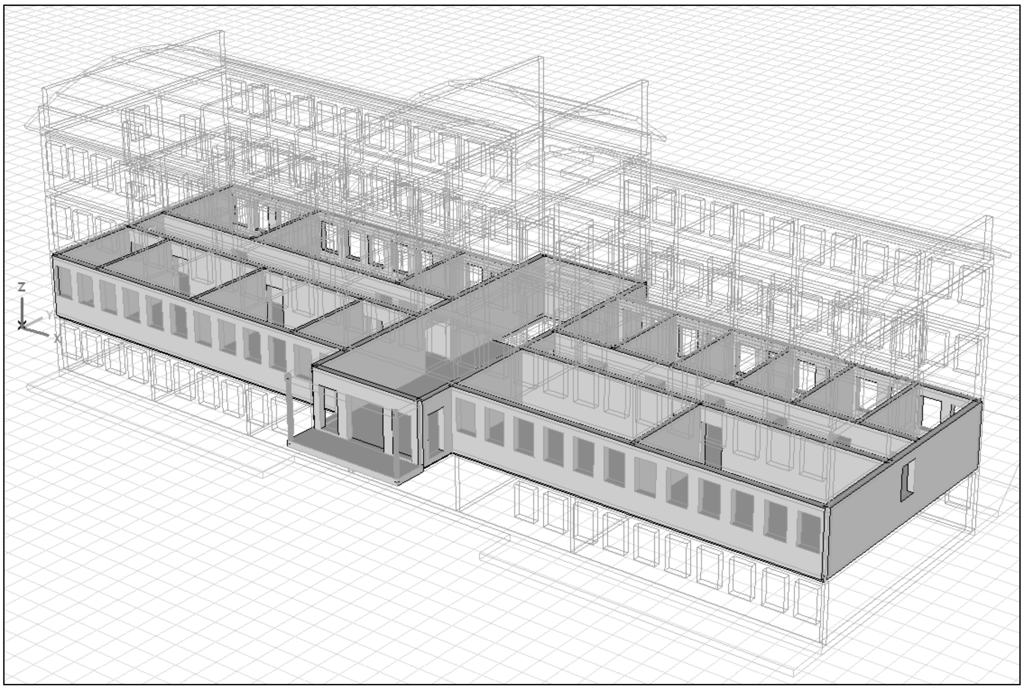
Show full path If activated, the tree view will display names of imported files with their full path.

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. Stories

[Ctrl + R]

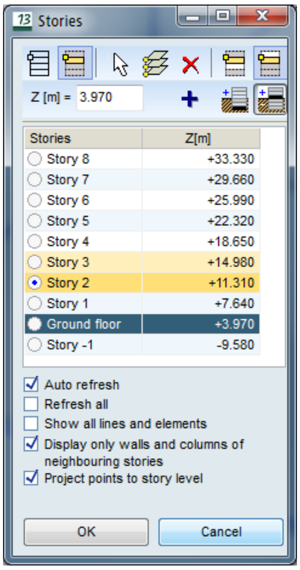


Stories are to make it easier to overview and edit the model. They can be defined before drawing the model or assigned to an existing structure.
A story is a workplane parallel to the global X-Y plane, with a given Z position. If a story is selected mouse movements will be projected to the plane of the story even if you find an element at a different Z position. Coordinates will always be projected to the story plane to help tracing objects at different levels.
Stories are always listed by decreasing Z position, having automatic names. Changing the report language makes story names change.
Elements are considered to be part of a story if their lowest Z coordinate is greater than or equal to the story level but less than the next story 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 story planes.
New elements will be linked to their story automatically.

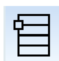
☞ ***Stories are logical parts of the model created for editing purposes and they do not affect the analysis results.***

If torsion effects has to be taken into account in seismic analysis seismic stories have to be defined separately in the seismic parameters dialog.

Stories can be managed in the following dialog.

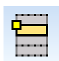


Turn off stories



If this button is down no stories are displayed. Windows will show the entire strucutre or the active parts. Stories can be added or deleted in this state as well.

Display current story



If this button is down and an active story is chosen the active story will be displayed. The active story 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 story 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 stories and not on the active story.
Active story is yellow. If a neighbouring story is also activated to help tracing objects on that story, it appears as light yellow.

☞ ***There can be only one active story. However display of neighbouring stories is also possible. Editing will be constrained to the active story.***

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 stories.

Enter a new story — Enter the Z coordinate into the edit field and click the + button. A new story will be added to the list.

Find — If you have an existing multi-story structure with slabs you can find and add Z coordinates of horizontal domains to the list with one click. If not all horizontal domains refer to a real story you can delete unnecessary stories later.

☞ ***Story position cannot be changed. Delete the story and define a new one.***

Delete — Deletes selected stories. Remaining stories will be renamed and story assignments of the elements will be updated automatically.

☞ ***Deleting a story does not delete any element.***

Display the story below the current story / *Display the story above the current story* — If any of these buttons is down elements of the story below/above the active story is also displayed to help tracing other objects.

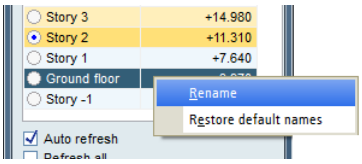
If this button is down elements of the story above the active story is also displayed to help tracing other objects.

☞ ***To display further stories open the Parts dialog instead where logical parts of any story can be turned on. Choosing a new active story overrides the parts settings.***

Define a new story

Numbering of stories — Numbering of stories can be controlled with these buttons. If the left one is down (*Numbering of stories from the bottom*) the lowest floor will be considered as ground floor and other stories will have a positive number. If *Signed numbering of stories* is selected the story closest to the zero level will be the ground floor. Underground stories will get a negative number, others will get positve numbers.

Renaming stories



Click the right mouse button over a list item to get to the popup menu. Stories 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 — Story settings will be applied on all windows.

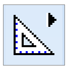
Show all lines and elements — If a story is activated all slabs, walls, columns and beams associated to the story will be displayed automatically. If other elements (geometry lines, springs, gap or link elements) on the story are also to be seen, check this option.

Display only walls and columns of neighbouring story — The story above/below the active story is usually displayed to find the supporting walls and columns. This options hides slabs and beams of the neighbouring story.

If a story is activated direct drawing of a column or a wall automatically starts with the story height.

Project points to story level — This feature is to project all detected coordinates to the base level of the story. It is useful when wall and column positions below the slab must be transferred the floor.

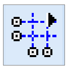
3.3.5. Guidelines



[Ctrl+G]

Guidelines
See... *2.16.9 Guidelines*

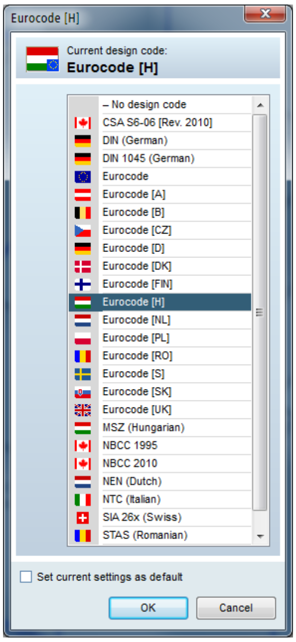
3.3.6. Structural Grid



[Shift+G]

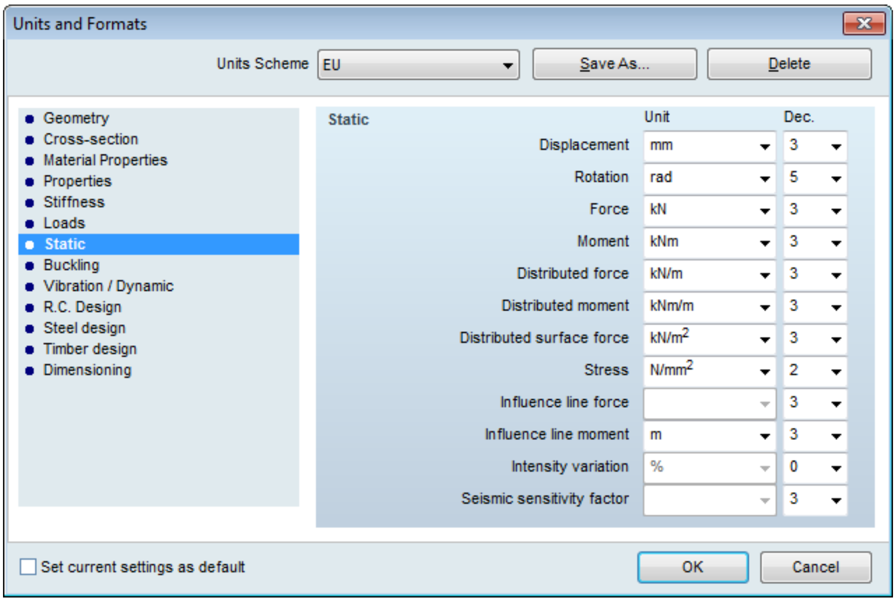
Structural Grid
See... *2.16.8 Structural grid*

3.3.7. Design codes



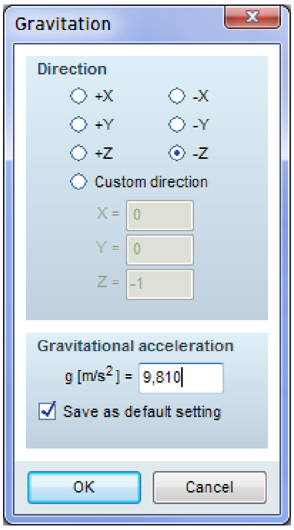
Sets the design code to be used in case of code specific tasks. Changing design code changes the method of calculating critical load combinations therefore all load group parameters but partial factors will be deleted. Seismic analysis parameters and seismic load cases will also be deleted. As material 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.

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.

3.3.9. Gravitation

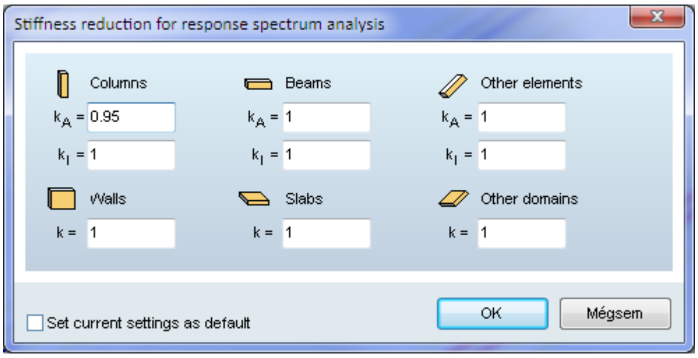


Lets you set the gravitational acceleration constant and the direction of gravitation as one of the global coordinate directions or a custom direction.
If *Custom direction* is selected the X, Y, Z components of the direction vector in the global coordinate system must be specified.
If *Save as default 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)* based on architectural element types (columns, beams, walls, slabs, other elements).

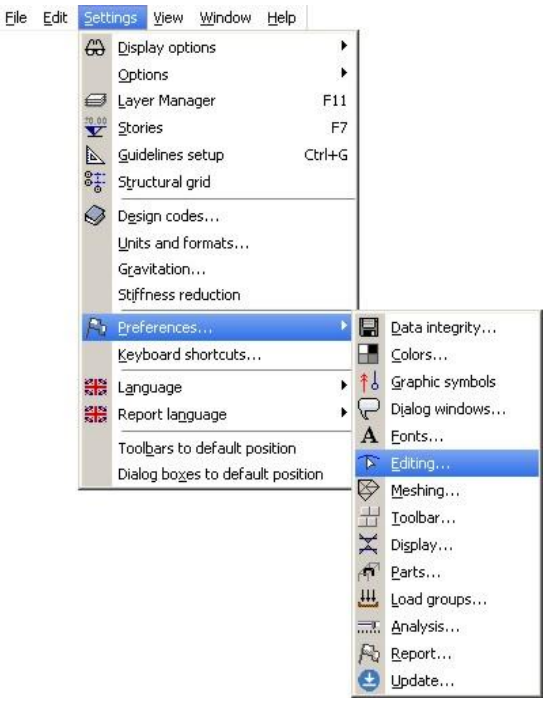
Setting stiffness reduction factors in itself does not change the static or dynamic results.



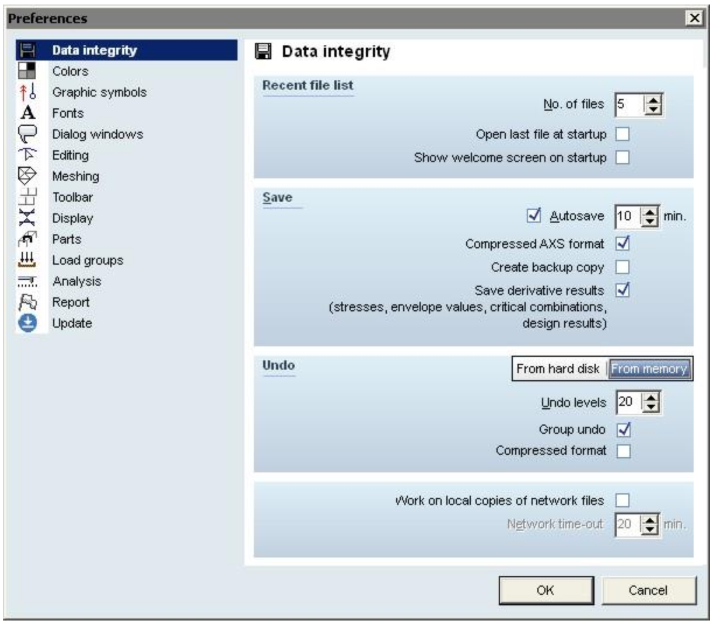
Vibration analysis lets the user apply reduced stiffness. If you choose reduced stiffness and you base the response spectrum analysis on vibration results calculated with reduced stiffness, the linear analysis will be automatically performed with reduced stiffness (for all load cases).

For surface elements the factor *k* reduces the element stiffness. For line elements separate factors can be set to reduce the cross-section area (k_A) and the area moment of inertia (k_I).
Factors can be set greather than 1, but a warning message appears.
Turning on *Set current settings as default* makes AxisVM store the values and set them for new models.
Stiffness reduction values can be labeled and appear in the table of domains.

3.3.11. Preferences



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 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`.

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).

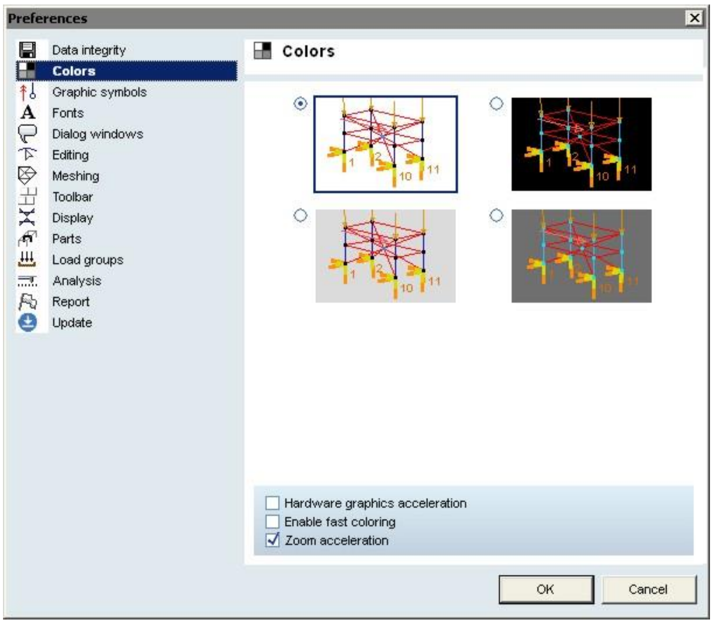
Work on local copies of network files

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.

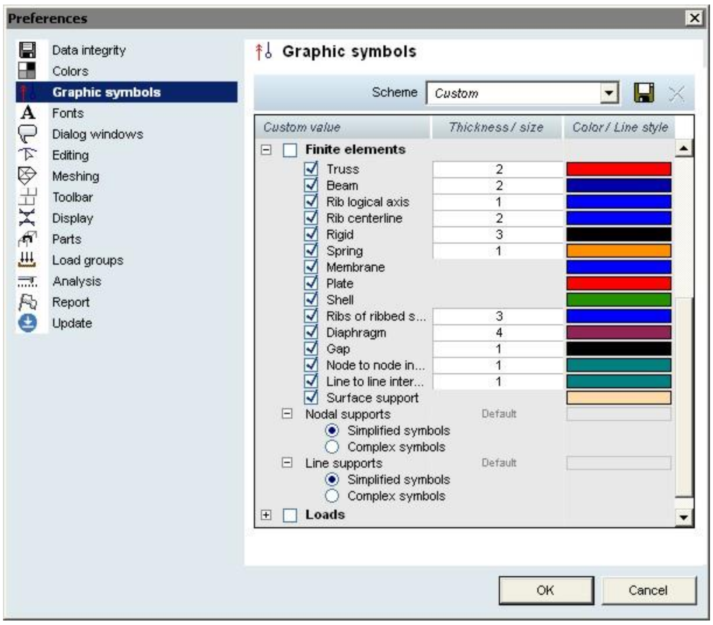
Colors



Lets you select graphics area background color (black, dark gray, light gray or white). Labels, numbers, symbols and elements will automatically change their colors to remain visible

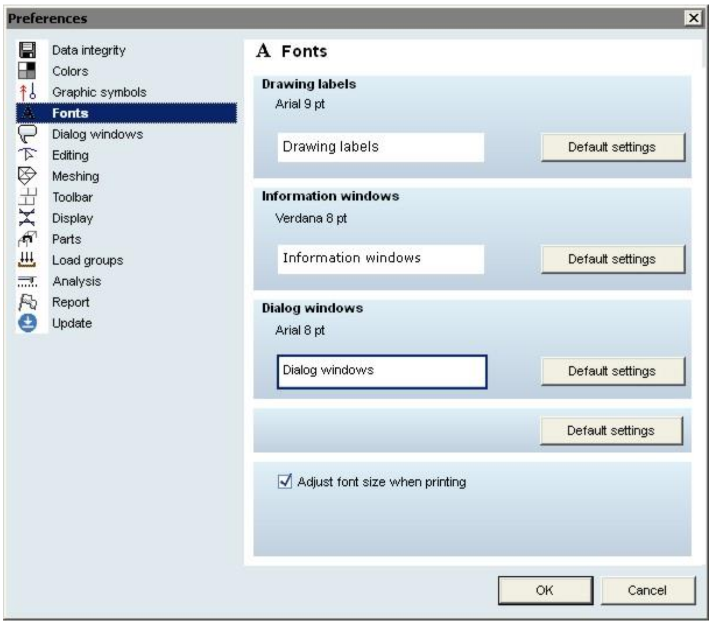
Turn on *Hardware graphics acceleration* for faster and smoother rotation of the model – if the video card and driver supports it.
Enable fast coloring activates a faster method of coloring objects in OpenGL.
If any of these acceleration options pose problems update the video driver or turn the function off.
Zoom acceleration hides labels and other non-scaled items during zooming.

Graphic symbols



Color and line thickness of graphic symbols can be customized.
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.

Fonts

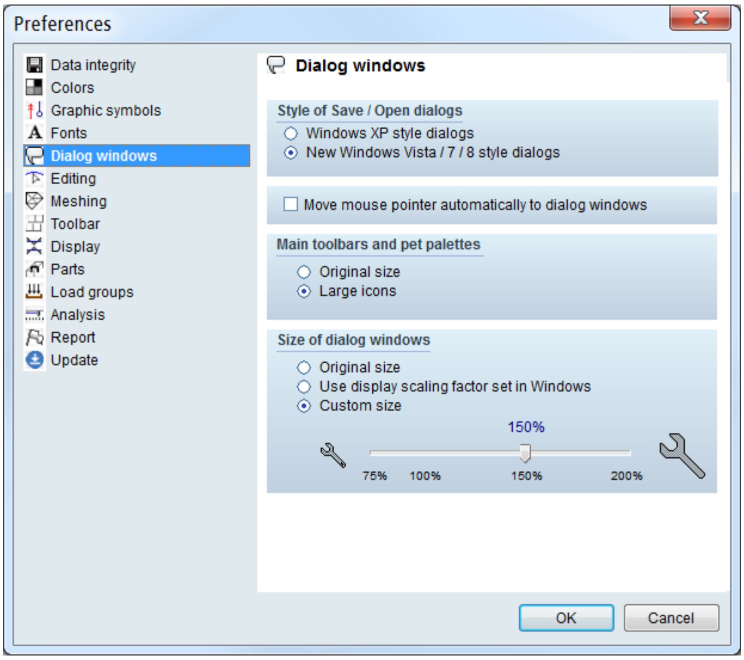


Lets you change the typeface and size of the fonts that are used when displaying your model and the floating palettes. Click white sample area to get to the font selection dialog.

Default settings can be restored by pressing the button on the right. The lowest *Default settings* button sets the default for all fonts.

Due to different resolution of the screen and the printer the ratio of label size and drawing is different on the two devices. Check / uncheck *Adjust font size when printing* if you are not satisfied with the result.

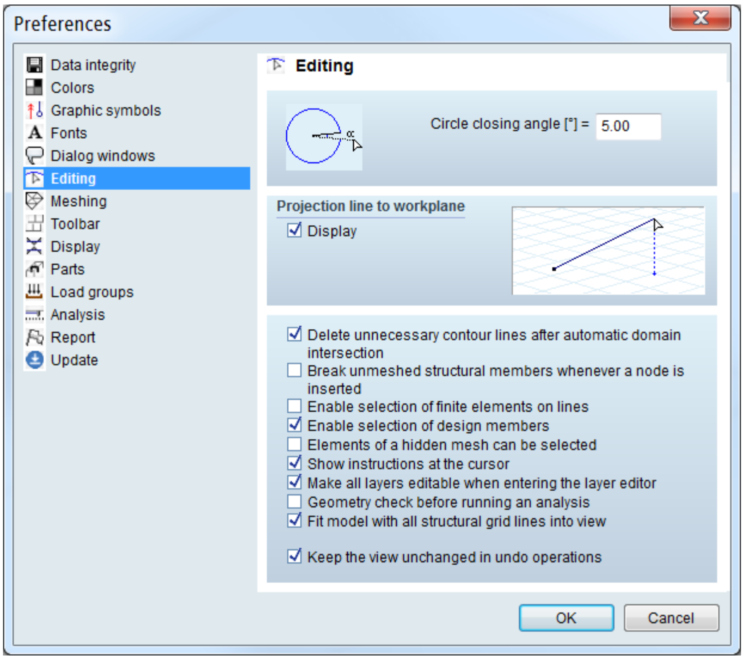
Dialog windows



If the operating system is Vista or later you can set the *Style of Save / Open dialogs* used in AxisVM. Under Windows XP only the first option is available.
If the second option is chosen AxisVM file preview requires a successful registration of a preview library (this DLL is part of the AxisVM package). Installing AxisVM with administrative rights automatically registers this library. Without administrative rights this registration fails making the preview unavailable. The preview library file can be registered later by running `!REGISTER_PreviewLib.BAT` from the AxisVM program folder.
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.

When working on high pixel density monitors it is recommended to enlarge toolbars, palettes, dialog windows and cursors. *Main toolbars and pet palettes* offers 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.

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.

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 *2.17 Speed Buttons*.

Geometry check before running an analysis

If activated a geometry check is automatically performed before analysis.
See… *4.8.15 Geometry check*

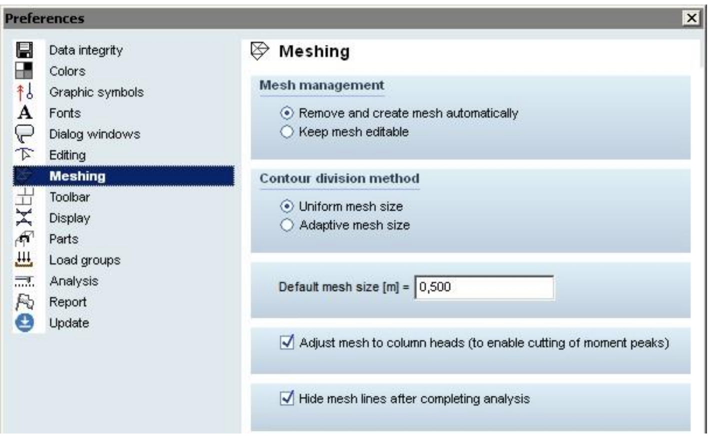
Fit model with structural gridlines into view

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.

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

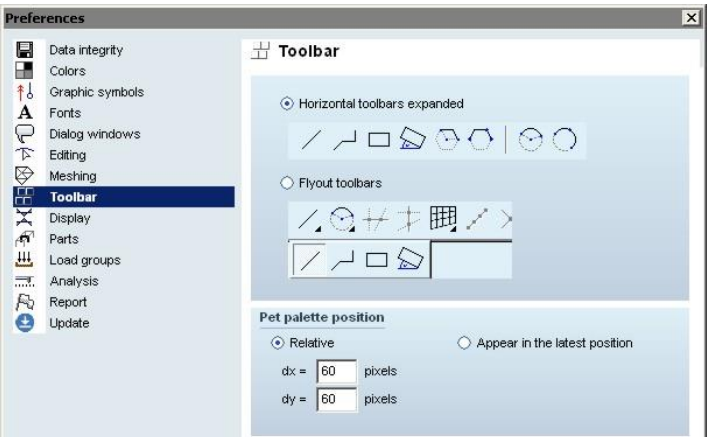
Takes the shape of the domain into consideration and creates a better mesh by increasing mesh density wherever it is necessary.

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...** *4.11.1.2 Meshing of domains.*

Hide mesh lines after completing analysis Checking *Hide mesh lines after completing analysis* automatically turns off mesh lines after completing analysis.

Toolbar



Displaying toolbar If *Horizontal toolbars expanded* is chosen, all icon appears in a row. 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.

Pet palette position Pet palette position can be:

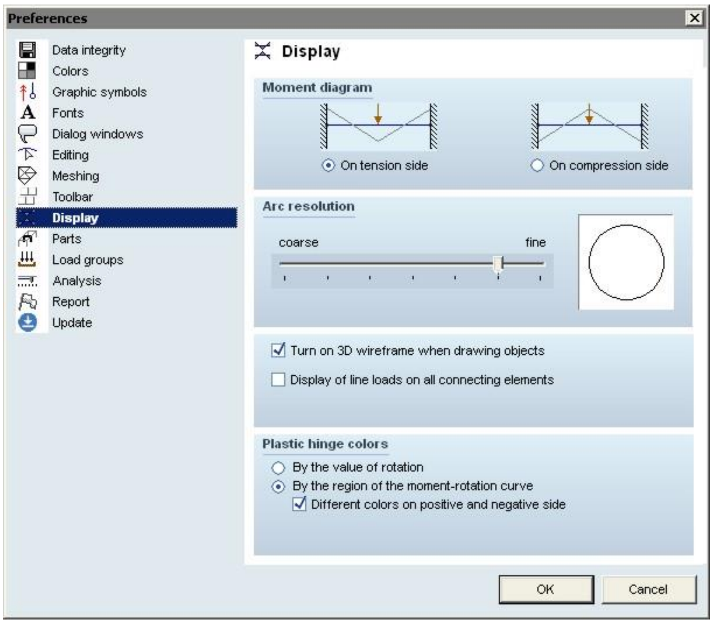
Relative

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.

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.

Switches

Turn on 3D wireframe when drawing models

Displays 3D wireframe of objects while drawing (**see…** *4.9.3 Direct drawing of objects*) even if the active view is not in rendered mode.

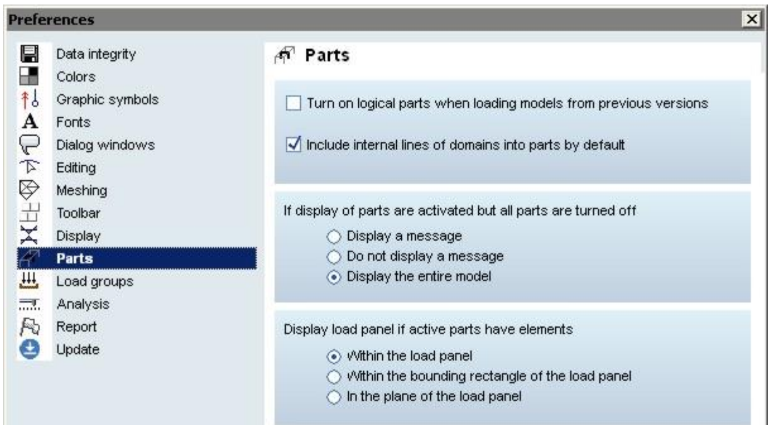
Display of line loads on all connecting elements

If an edge load is applied where a wall and two plates meet and parts are turned on (**see…** *2.16.14 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.

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.

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.

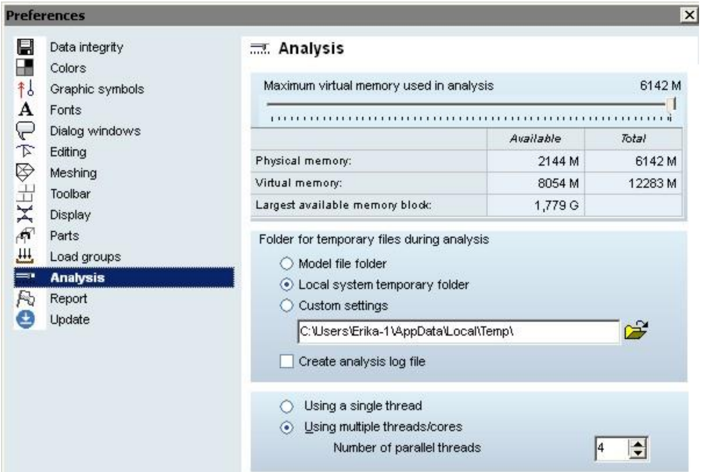
Include internal lines of domains into parts by default

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.

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.
Description of combinations can be extended to show the combination type (ULS or SLS types).

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.

Enable extended memory access (AWE)

If more than 4 GB of memory is installed with a 32 bit operating system, this option makes it possible to get more memory for analysis. If this option is disabled it means that memory pages are not locked.
See... *2.1 Hardware requirements* for details

Using a single thread / Using multiple threads

Using multiple threads/cores makes AxisVM run analysis on multiple threads. To make the most of this option it is recommended to use a processor with HT-Hyperthread or DualCore technology.
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.

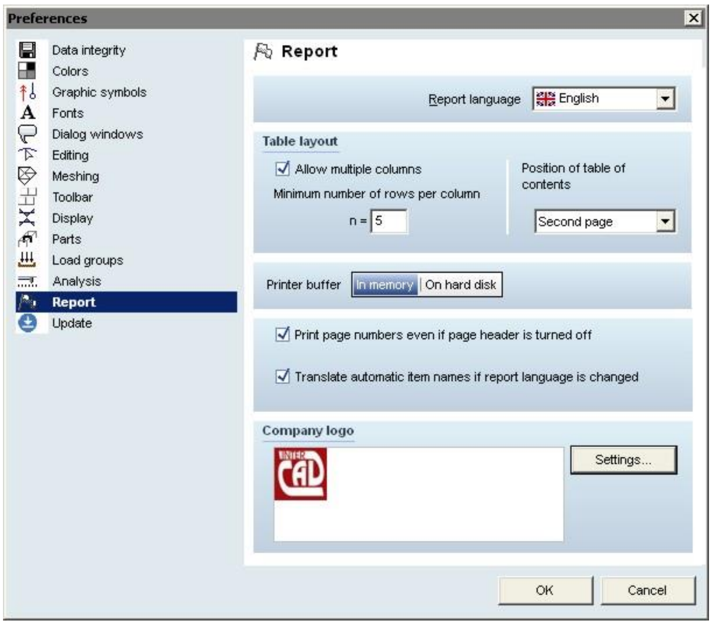
Folder for temporary files during analysis

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`.

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.

Report



Report language Depending on your configuration you can select from the following languages: *English, German, French, Italian, Spanish, Dutch, Hungarian, Russian, Portugese, Romanian, Serbian.*

Table layout If *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.

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.

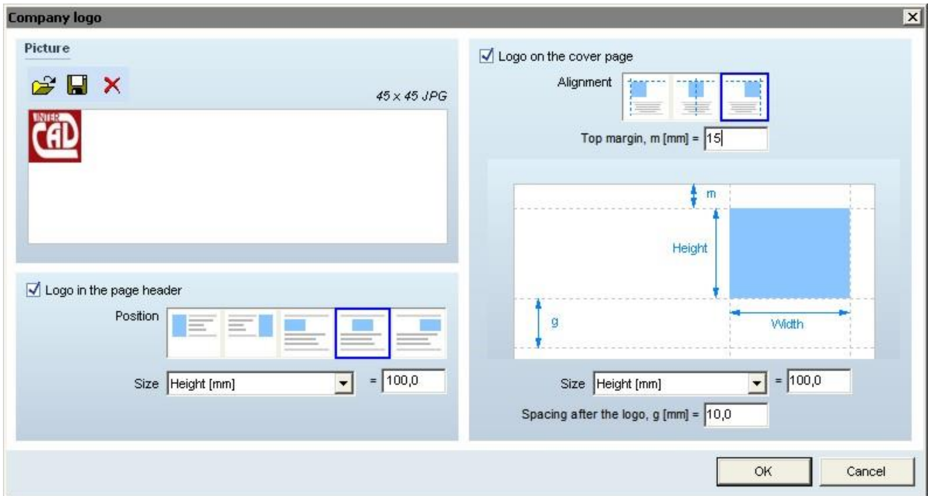
Print page numbers even if page header is turned off

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.

Company logo

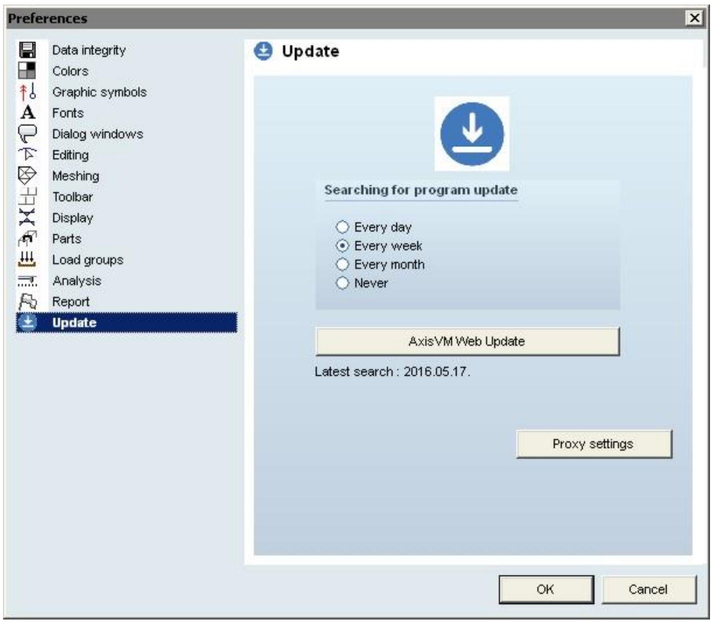


A company logo can be loaded, saved, deleted using the three toolbar buttons. This image will appear in the page header of printed drawings, tables, reports and/or on the cover page of the report according to the settings for position, size, margins.

These settings have no effect on the RTF output as that is based on a separate RTF template file.

See… *2.10.2 Report*

Update

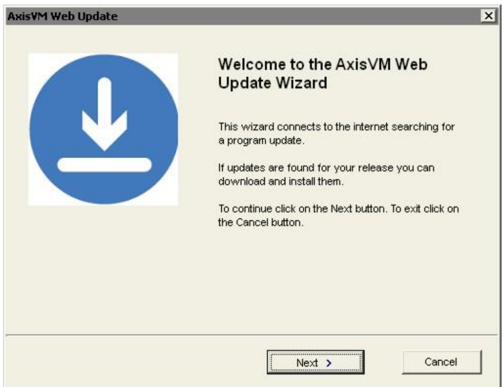


Searching for program update

AxisVM checks regularly if there is an update available on the web. The frequency of update checks can be controlled. If *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 has 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.



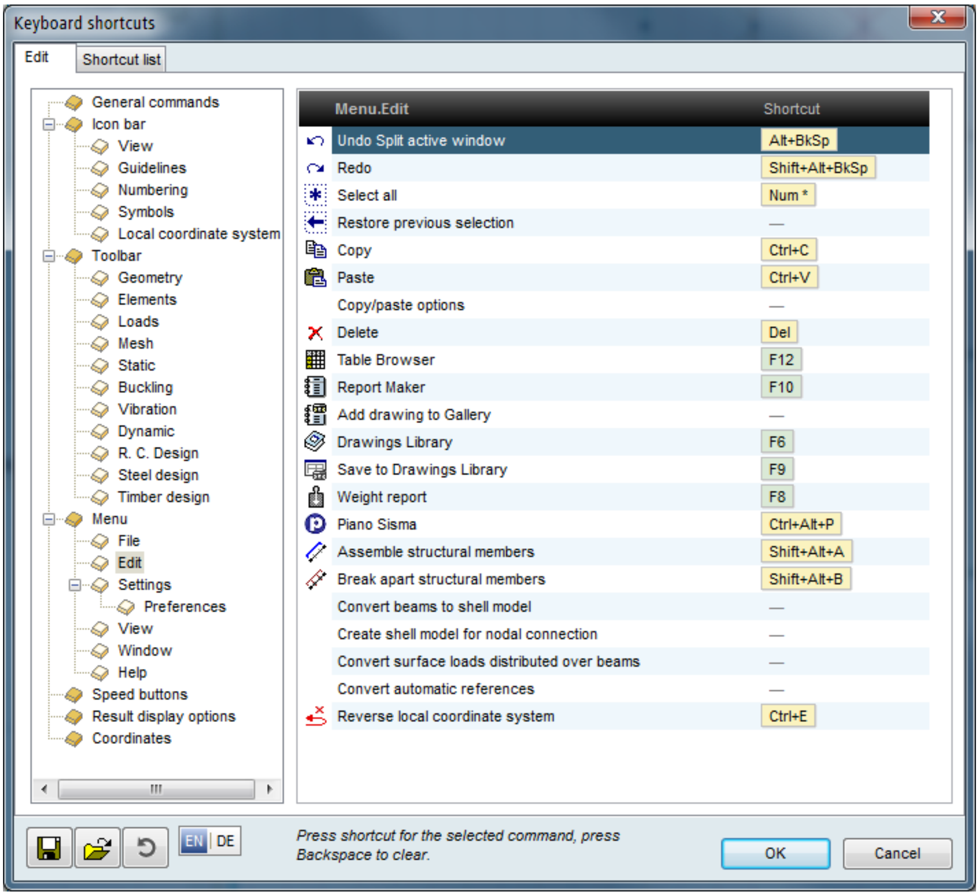
Proxy settings

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 menu items, toolbar buttons can be customized.

Edit



The tree on the left side displays the available command groups. Click a command in the list on the right side, then press the desired shortcut. **Backspace** clears the assigment.

If the desired shortcut is already in use a *Conflicting shortcuts* warning is displayed.

The message shows the command using the shortcut.

If *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.

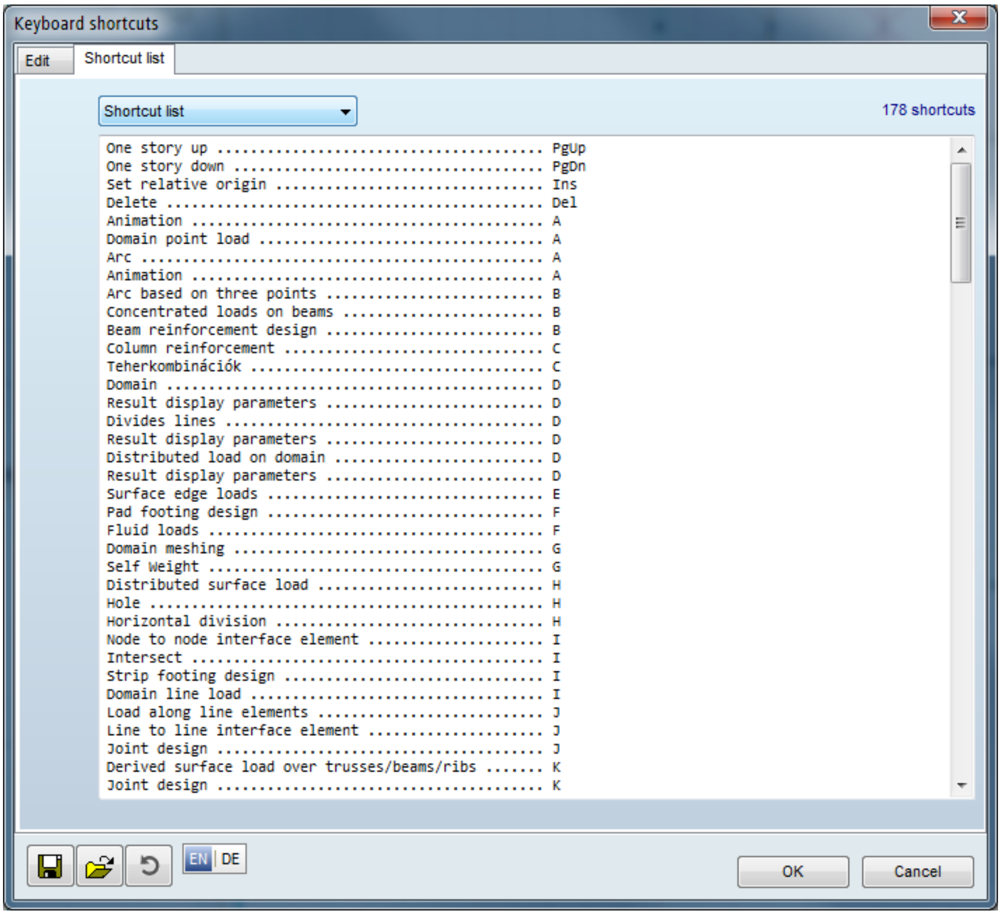
Loads a previously saved shortcut configuration from a *.*axsc* file.

By default shortcuts are stored in the `C:\Users\UserName\AppData\Roaming\AxisVM\13\Shortcuts` folder. The default shortcut configuration of AxisVM 12 can be loaded from `AxisVM12Default.axsc` the same for AxisVM 13 is `AxisVM13Default.axsc`.

Resets the shortcut configuration to its default.

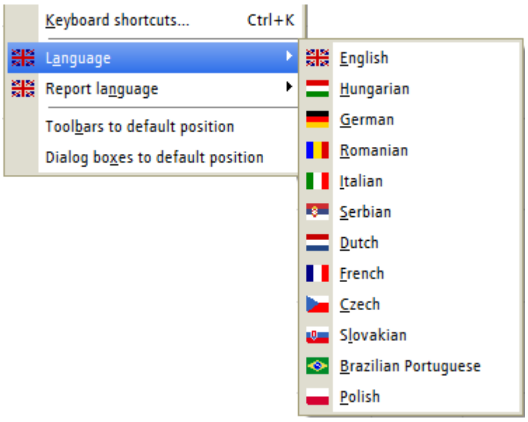
A switch to display English or German key names in the shortcut descriptions.

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



If program configuration includes the **DM** module this menu item allows the user to change the program language (used in menus and dialogs).

3.3.14. Report Language

If program configuration includes the **DM** module this menu item allows the user to change the report language (used when displaying printable drawings, tables and reports).

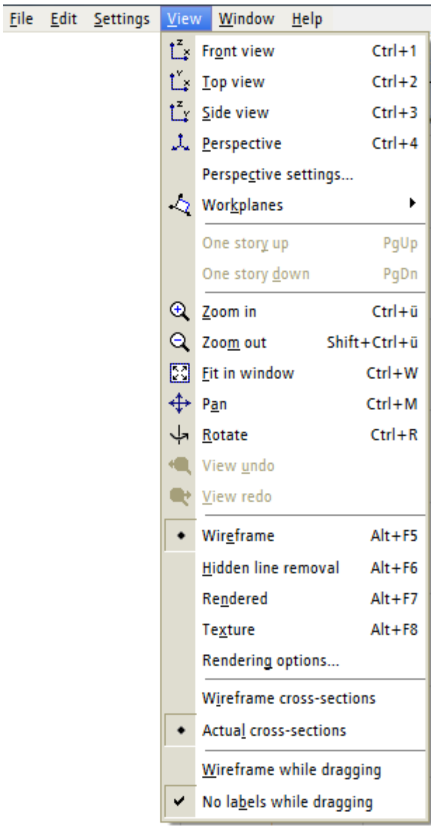
3.3.15. Toolbars to default position

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.

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.

3.4. 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 story up / One story down

If displaying of a story is activated this is the fastest way to go one story up or down.

Zoom in **[Ctrl + /], [+]** **See...** *2.16.2 Zoom*

Zoom out **[Ctrl + Shift+ /], [–]** **See...** *2.16.2 Zoom*

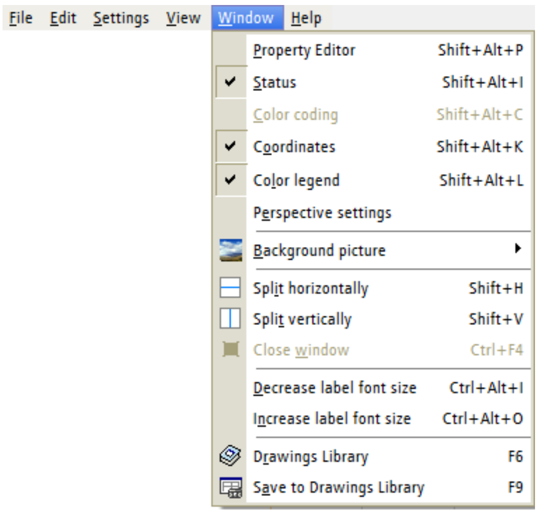
Zoom to fit **[Ctrl + W]** **See...** *2.16.2 Zoom*

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*
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. Window



3.5.1. 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 substracts 8 from the actual value

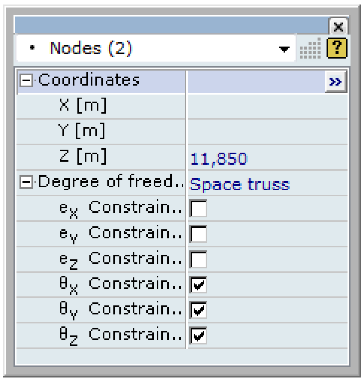
Negative numbers within operation have to be in brackets.
In these expressions # substitutes the actual value (For instance *#/3* divide it by 3). When entering a value of nodal coordinates, load values, surface thicknesses you can refer to global coordinates as *X, Y, Z* or *x, y, z.* In case of certain 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.

Selecting the cross-section property of a rib element you can select all rib elements with this cross-section then change their cross-section property.

3.5.2. Information Windows

Lets you set the display of the Status, Color coding, Coordinate, and Color legend windows to on or off. **See...** *2.18 Information*

3.5.3. 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.

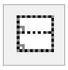
Picture in the active window can be turned on and off by clicking *Display* or by **[Ctrl+Alt+B].**

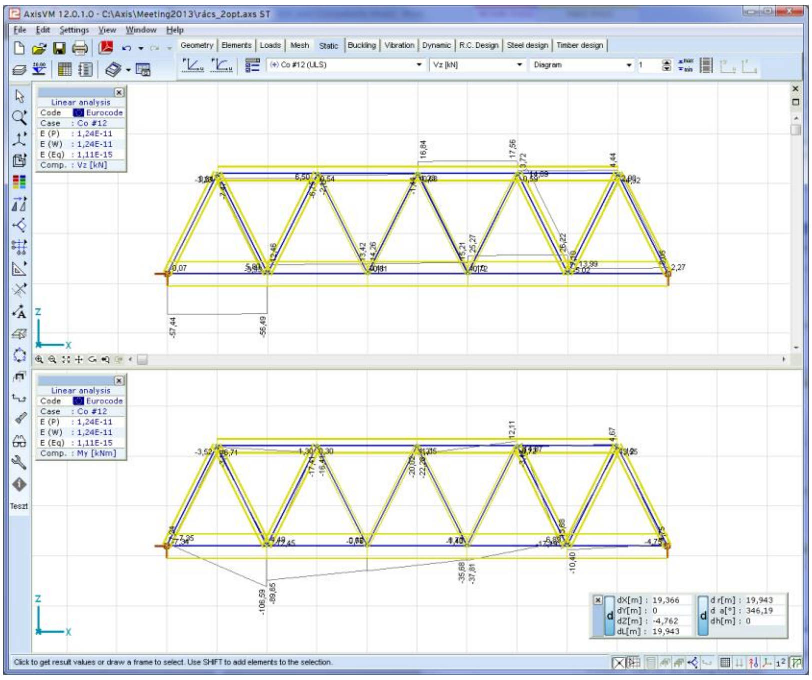
Save Background Picture saves the picture in the active window into a file. If the aspect of the picture differs from the window aspect *Shift Background Picture* makes it possible to drag the background to a new position. *Remove Background Picture* removes the picture in the active window.

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.

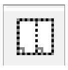
3.5.4. Split Horizontally

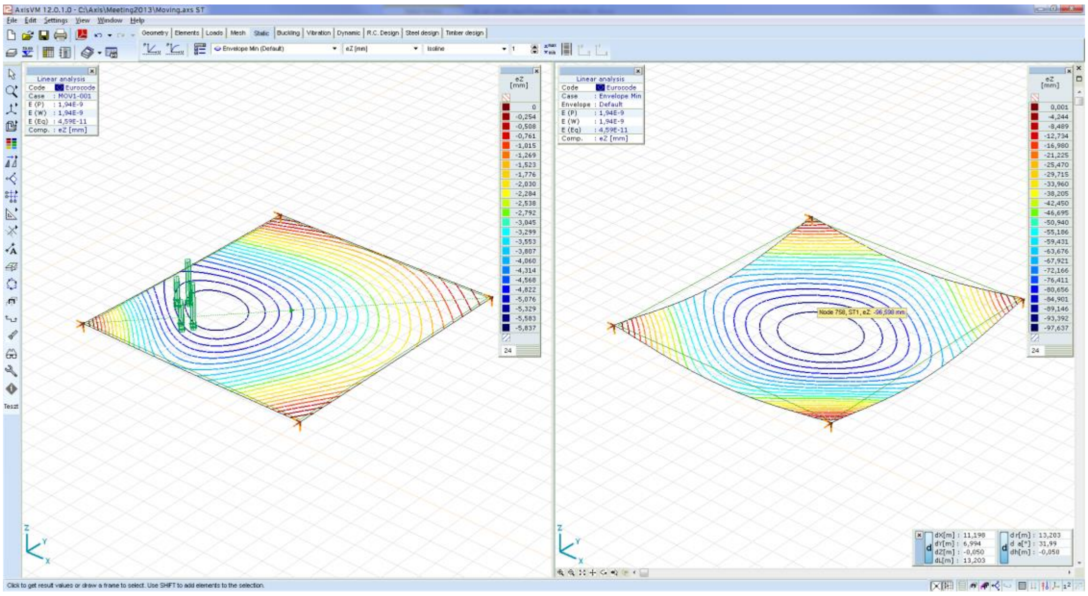




Splits the graphics window horizontally into two parts. Clicking into one of them makes that window active. The display settings of each window can be set independently. Different load case or combination can be selected for each window.
You can maximize or minimize or restore the graphics windows by using the buttons at the top-right of the windows.

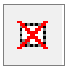
3.5.5. Split Vertically





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.

3.5.6. Close Window



Closes the current graphics window.

3.5.7. Changing label font size

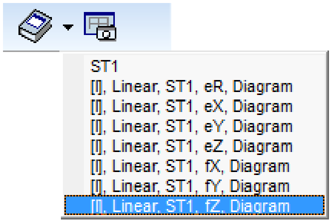
Decrease / Increase label font size

These two menu items is to change the font size for all labels on diagrams.

3.5.8. Drawings Library

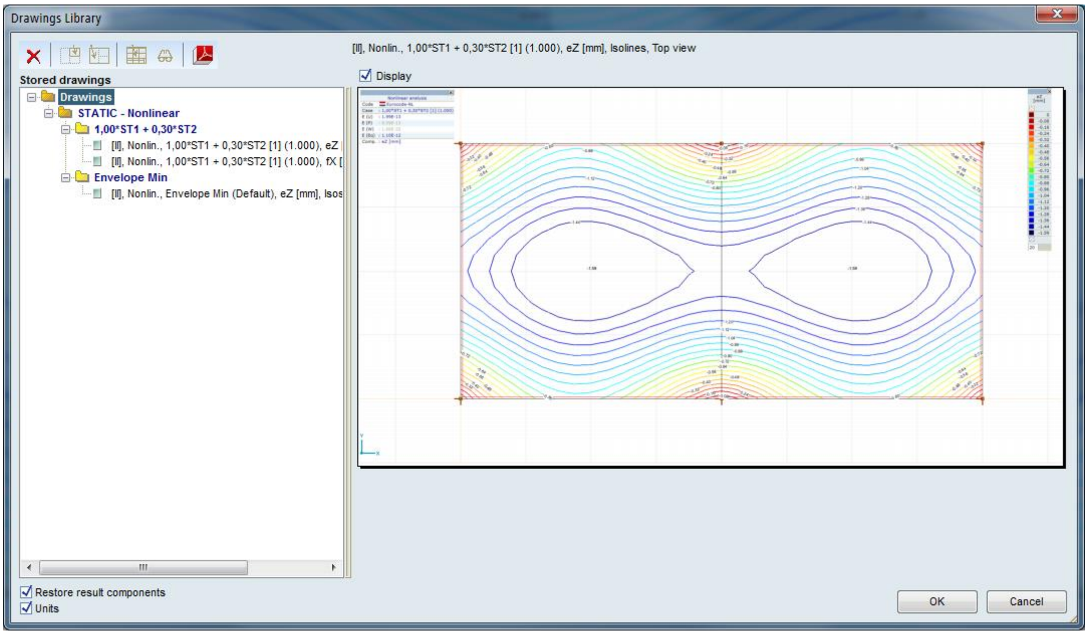
The Drawings Library contains drawings saved in the program. Drawings are not saved pictures but instructions how to draw a view of the model or parts of it including multi-window settings. Drawings can be reloaded to restore saved view and display settings. Including drawings into a report makes it 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.



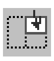
Clicking the arrow beside the tool button an existing drawing can be selected from a pop-up list, restoring its view and display settings.

After clicking the Drawings Library tool button a dialog appears.

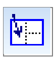


This dialog is to overview, maintain and reload saved drawings.

Deletes a drawing from the Drawings Library



Loads a chosen drawing to the active window.
(available in multi-window mode only)



Loads a chosen drawing to the window.

Displays a diagram dialog
This button is enabled if a beam result or design diagram is selected (see *6.1.7 Truss/beam internal* or *6.6.1. Steel beam design according to Eurocode 3* for examples). It displays the respective dialog letting the user make changes. After closing the dialog the drawing can be updated or the changed diagram can be saved as a new library item.

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.

Exports Drawing Library items as a 3D PDF file.
See… *3.5.8.1 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).

Units

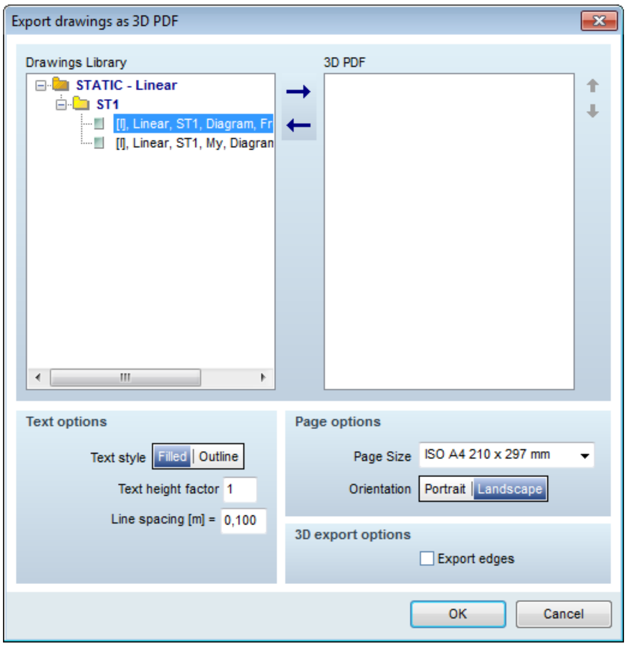
This checkbox controls whether generated names of diagrams contain the units (like eZ*[mm]*).

OK Saves the changes and loads the selected drawing.

Cancel Does not save changes.

3.5.8.1. 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.

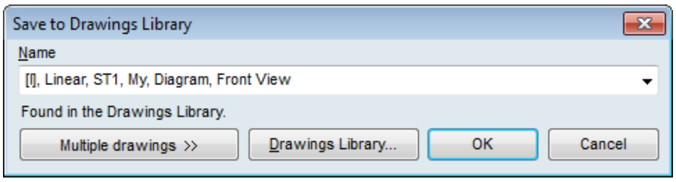


Text options Text size and appearance can be controlled in the *Text options* group.

Page options Size and orientation of the PDF document pages can be set in the *Page options* group.

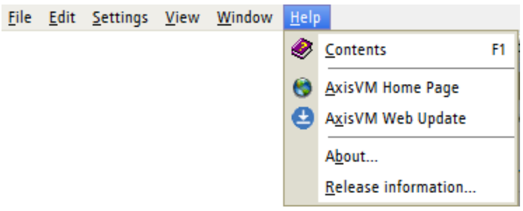
3D export options Sometimes drawing all edges makes the view a bit blurred. So export of edges can be turned on / off.

3.5.9. 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.
Clicking the *Drawings Library* button displays the *Drawings Library* dialog.

3.6. Help



Lets you use the online help of AxisVM. To get context-sensitive help information about the operations related to a dialog box press **[F1]**.

3.6.1. Contents

[F1]

Opens the table of contents of the help, and allows access to the topics you are interested in.

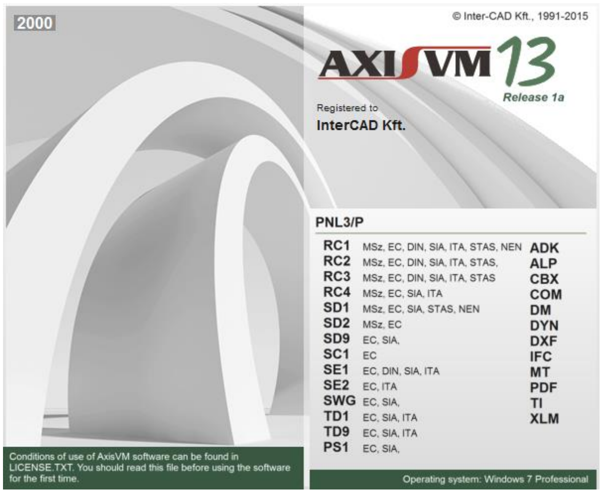
3.6.2. AxisVM Home Page

Visits AxisVM Home Page using the default Internet browser (*http://www.axisvm.eu*)

3.6.3. AxisVM Update

Launches the AxisVM Web Update Wizard. **See...** *3.3.11* Preferences

3.6.4. About

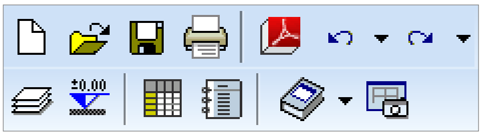


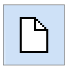
This window displays version and release number, configuration, serial number and time limit of your AxisVM version.
Available modules are in black, others are in gray.

3.6.5. Release information...

Latest release information and history of fixes and new developments.

3.7. Main toolbar

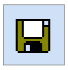




See... *3.1.1 New model*

[Ctrl+O]

See... *3.1.2 Open*

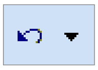


[Ctrl+S]

See... *3.1.3 Save*

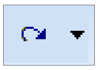
[Ctrl+P]

See... *3.1.10 Print*



[Ctrl]+[Z]

See... *3.2.1 Undo*



[Shift]+[Ctrl]+[Z]

See... *3.2.2 Redo*

[F11]

See... *3.3.3 Layer Manager*

[F7]

See... *3.3.4 Stories*

[F12]

See... *2.9 Table Browser*

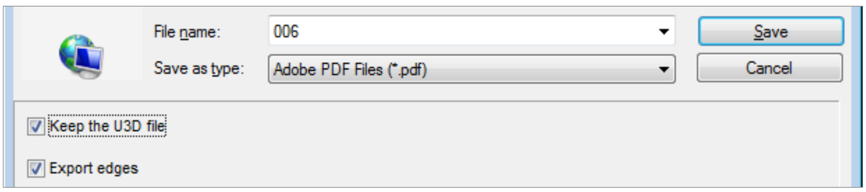
[F10]

See... *2.10 Report Maker*

See in detail... *3.5.8 Drawings Library*

See in detail... *3.5.9 Save to Drawings Library*

3.7.1. 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 *3.5.8.1 Export drawings to a 3D PDF file - PDF module*

This page is intentionally left blank.

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).

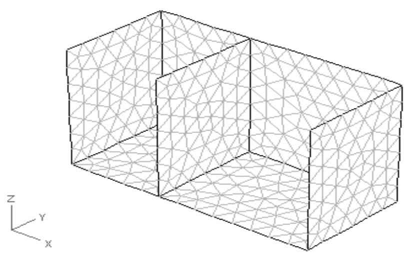
4.1. Geometry

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.

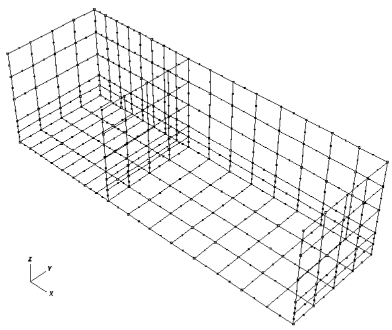
☞ ***Direct drawing of objects does not require drawing geometry in advance.***
See... 4.9.3 Direct drawing of objects

In the case of surface structures (plates, membranes, or shells) the mesh consists of quadrilaterals that represent the median plane of the elements.

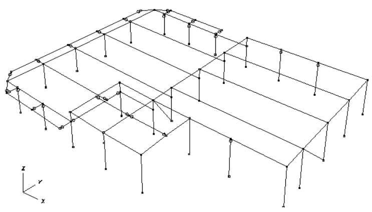
Automatic meshing on domains



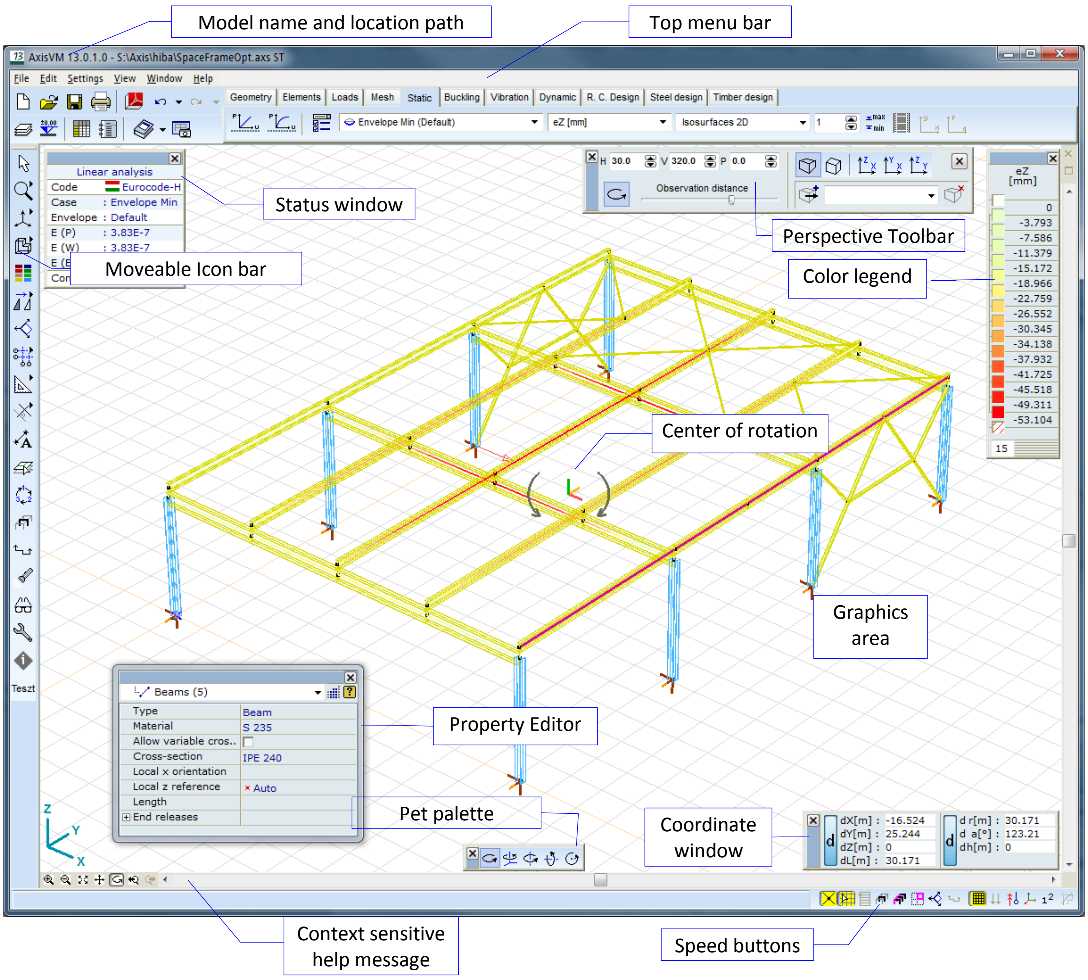
Automatic meshing on macro quads and triangles



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. You can access split commands from the Window menu.

Split horizontally Splits the active graphics window horizontally into two equal parts. The top window will become the active window.

See... *3.5.4 Split Horizontally*

Split vertically Splits the active graphics window vertically into two equal parts. The left window will become the active window.

See... *3.5.5 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.***

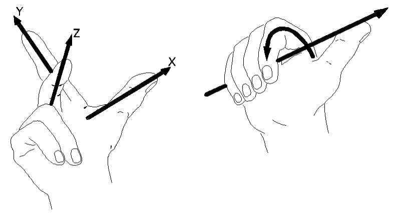
4.3. Coordinate systems

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... *4.9.20 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

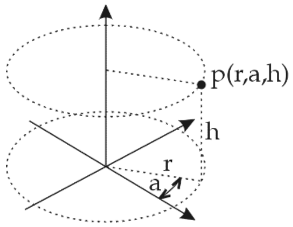
h: the value measured from the view plane to a point on the cylinder's main axis (that is perpendicular to the view plane) oriented outward from the screen

r: radius that is the distance on the view plane from the projection of the point to the cylinder's main axis

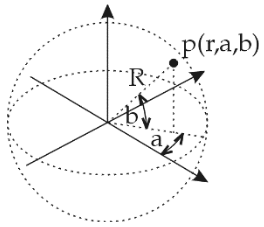
a: the angle between the line that joins the point with the origin and the horizontal

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).

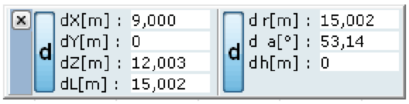


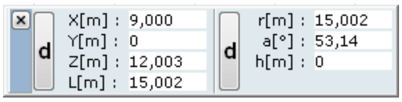
Cylindrical coordinate system



Spherical coordinate system

4.4. Coordinate window

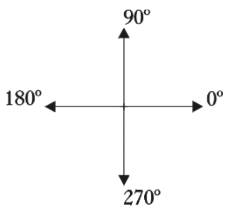




Displays the current absolute and relative values of the cursor position in the global coordinate system (Cartesian and cylindrical or spherical).
You can switch between absolute and relative coordinate displays, by clicking on the letters *d* in the Coordinate Window (delta switch). The display of the *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... *2.16.19.1 Grid and cursor*

4.6. Cursor step

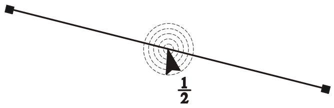
See in detail... *2.16.19.1 Grid and cursor*

4.7. Editing tools

Editing tools help the work by several features. **See...** *2.16.19.2 Editing*

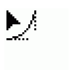
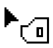
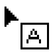
4.7.1. Cursor identification

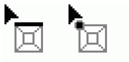
Sets the size of the cursor identification area (in pixels).

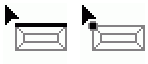


When you position the cursor over the graphics area, AxisVM finds the entity of the model that is closest to the center of the cursor from among the entities that are located in or intersect the identification area. The size of the identification area can be set at *Settings / Options / Editing / Cursor 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	
Text box, label	
Reinforcement domain, COBIAX solid area	

Edge, corner of a pad footing 

Edge, corner of a strip footing 

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 detect the lines on architecture background layers.

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 endpoint 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.***

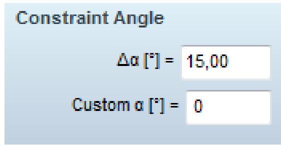
4.7.3. Measuring distance

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.

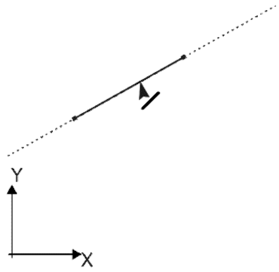
Custom α Holding the **[Shift]** key pressed, the cursor is moved a line that connects its current position with the origin, and that has an α or α + *n*90°* angle, where the value of n depends on the current cursor position.
$\Delta\alpha$ and α can be set in *Settings/Options/Editing/Constraint Angle.*

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.

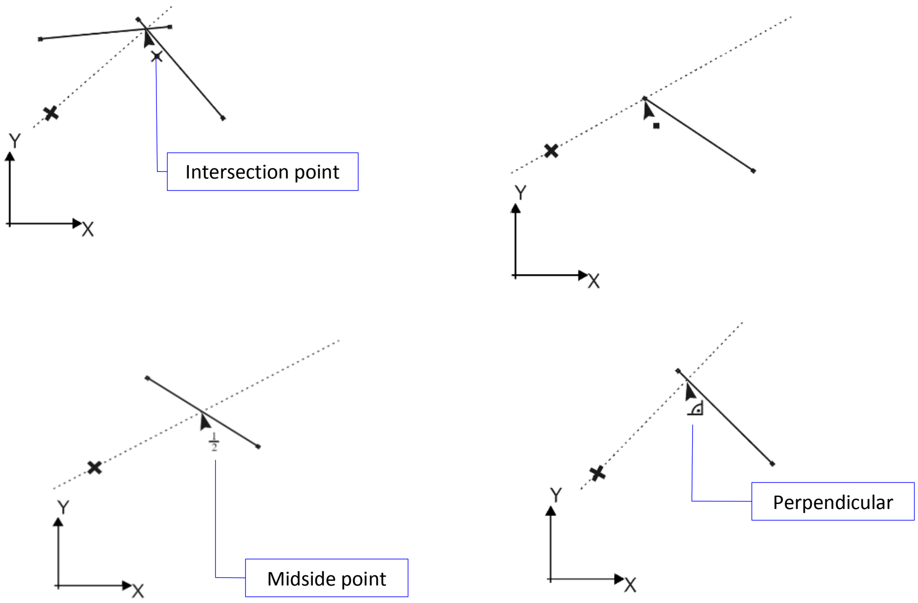
☞ ***You cannot use* Δα *and* Custom α *constraints in perspective view.***

If the cursor is over a line, holding the key **[Shift]** depressed, will constrain the cursor movement to the line and its extension .

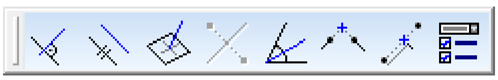
If the cursor identifies a point, holding the key **[Shift]** depressed, makes the cursor move along the line defined by the point and the relative origin..



When the cursor identifies a **domain or surface element** pressing **[Shift]** makes the cursor move in the plane of the element**.**



Geometry tools



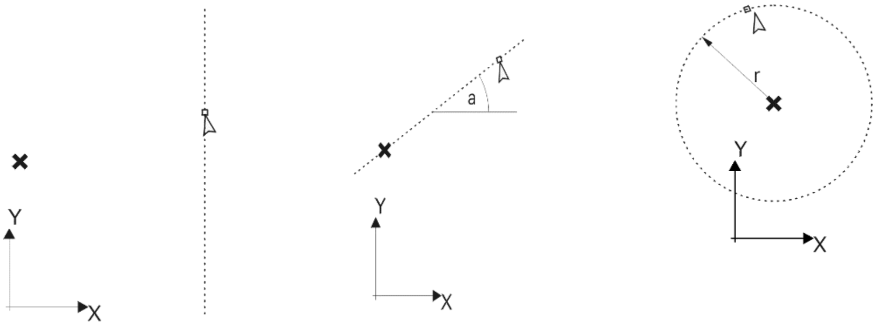
The icons of *Geometry tools* allow you to lock the direction of drawing a line.
See... *2.16.10 Geometry tools*

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.

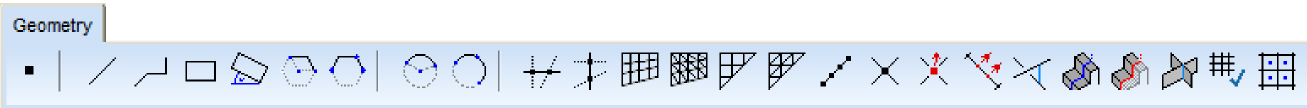
Frozen X coordinate *Frozen angle* *Frozen radius*



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...** *2.16.19.2 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

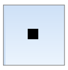


These tool buttons create new geometry or change the existing one..

☞ ***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.

4.8.1. Node (Point)



Lets you place new nodes or modify existing ones.
To place a node:

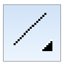
1. Move the graphics cursor to the desired location and press the **[Space]** key or the left mouse button (in perspective view you can place nodes only to special locations).
2. Enter the node coordinates numerically in the Coordinate Window, and then press **[Space]** or **[Enter]** (it works in all 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.***

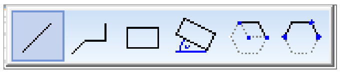
When working on parts with *Settings / Options / Editing / Auto / Part Management* ***turned on all geometric entities created will be automatically added to the active parts.***

4.8.2. Line



The Line tool is to construct lines or other simple shapes. The line type can be chosen by clicking on the arrow at the bottom-right corner of the currently used Line tool icon, 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 endpoints (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... *2.16.4 Workplanes*.

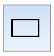
Polyline

Constructs a series of connected straight lines (a polyline). You must specify the vertices.

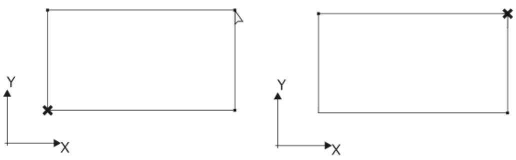
Exit current polyline by any of the following

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



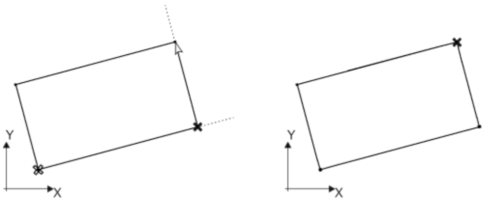
Constructs a rectangle (its corner points (nodes) and edge lines). You must specify two opposite corner points.



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 endpoints), and then the other side.

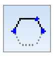


After you specify the first corner you can cancel the command pressing the **[Esc]** key. In perspective view, you can draw skewed rectangles using only the existing points.

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

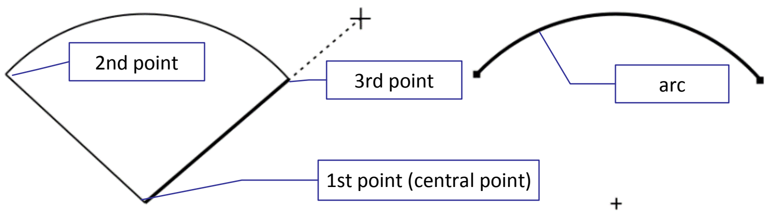


Number of sides has to be defined in a dialog. Polygon has to be defined by entering three points of the arc.

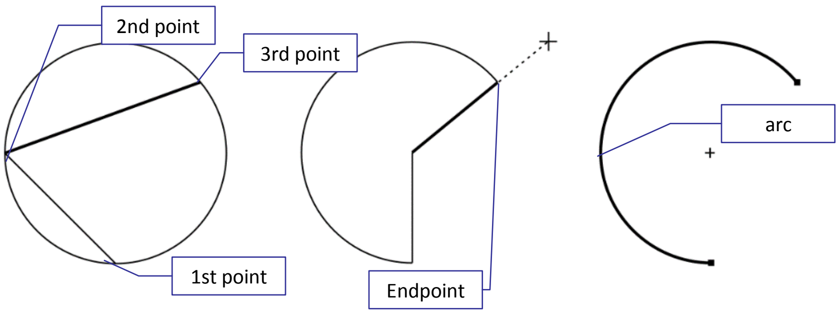
4.8.3. Arc

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.

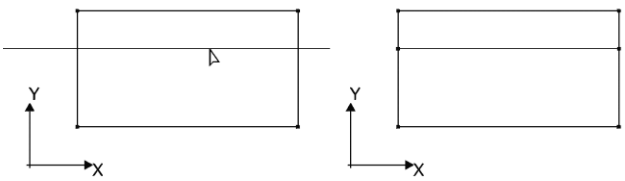


Defining an arc by three points. The command can be applied in perspective setting as well.



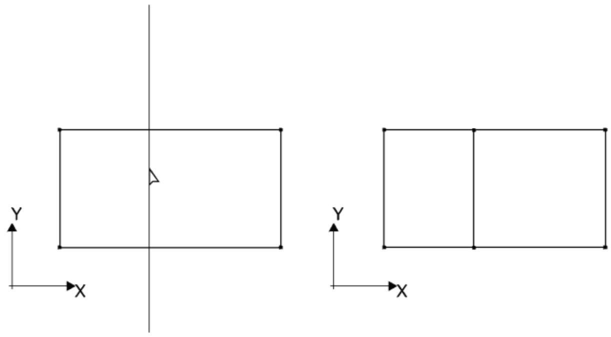
4.8.4. Horizontal division

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

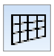
This function creates a vertical divider line passing through the cursor position. This line is in a plane parallel to the X-Y, X-Z or Y-Z plane depending on the actual view (or parallel to the workplane if a workplane is used). Creates new nodes at the intersections. If finite elements are intersected new elements inherit properties and loads of the original element.



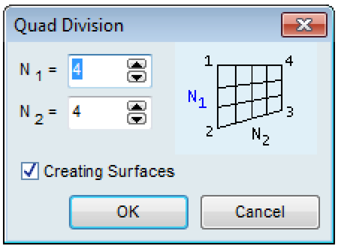
4.8.6. Quad/triangle division

Constructs a mesh of quads/triangles over a quad or triangle. Use this command 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.

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.

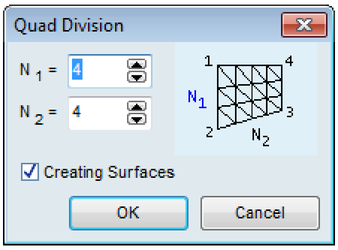


The quad and the mesh are displayed with solid grey lines.

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.

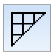
Quad-to-triangles

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.

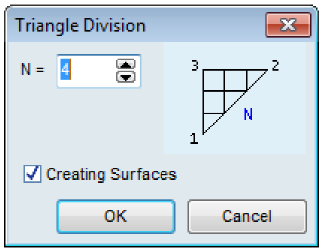


If a quad shape is entered that is not allowed (e.g. concave), the quad is displayed with red dotted lines.

Triangle-to-quads



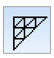
Constructs a mesh between the corners of a triangle (not necessarily with any side lines). The mesh will also contain triangles along the side that corresponds to the first two corners entered.
You must graphically select the corners successively (three points), and specify the number of segments N between corners.



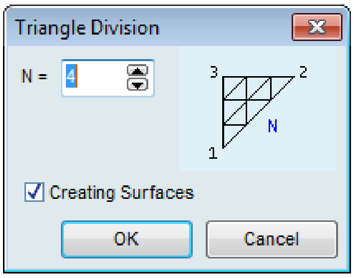
The triangle and the mesh are displayed with solid grey lines.

If the mesh leads to quad subdivisions that are distorted (have an angle smaller than 30º or greater than 150º), or to triangle subdivisions that are too distorted (has an angle smaller than 15º or greater than 165º), 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

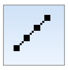


The command is similar to the triangle-to-quads command, except that each generated quad is divided into two triangles by its diagonals which are parallel to the side first entered.

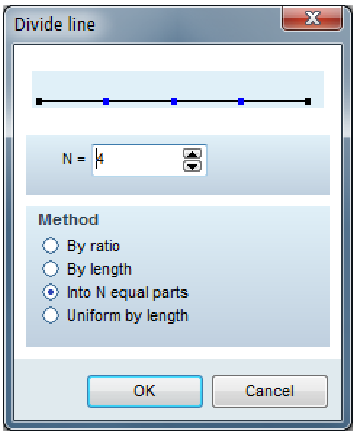


Same as for triangle-to-quads.

4.8.7. Line division



Lets you create new point (nodes) on the selected lines.
The following input options are available:



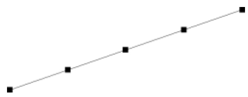
By Ratio: Lets you divide the selected lines into two segments. You must specify the parameter a of the location of the inserted node relative to the first node (i). The parameter a must be between 0 and 1. a = 0.5 represents a division of the selected lines into two equal segments.

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

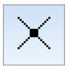


after division

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.***

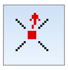
4.8.8. Intersect



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.***

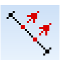
4.8.9. Remove node



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.

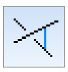
☞ ***Intersection nodes can be removed only if the number of connecting lines are even and lines can be joined.***

4.8.10. Remove intermediate nodes



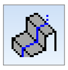
Removes unnecessary intermediate nodes on lines. Nodes with two connecting lines are removed provided they can be joined.

4.8.11. Normal transversal



Creates a connection between two lines along their normal tranversal.

4.8.12. Intersect plane with the model

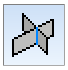


After defining the intersecting plane intersection lines and nodes will be added to the model. Domains, beams and ribs will be divided.

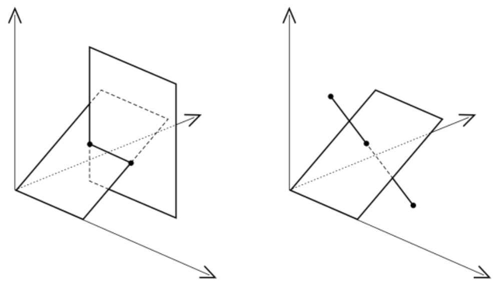
4.8.13. Intersect plane with the model and remove half space

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.

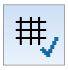
4.8.14. Domain intersection



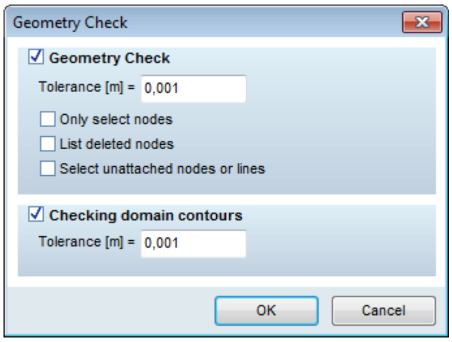
Creates intersection lines of domains and line elements. After clicking the tool button select domains to create their intersection or select a domain and a line to create the intersection.



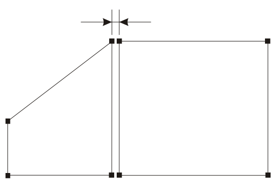
4.8.15. Geometry check



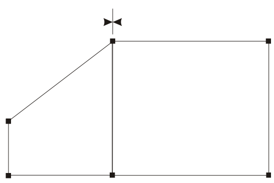
This function selects (if *Only select nodes* is checked) or eliminates extra nodes and lines within a given tolerance and fixes domain contours forcing contour segments into the same plane and adjusting arcs if radius is not the same at the startpoint and the endpoint. You can specify the maximum tolerance (distance) for merging points. The default value is $\Delta L = 0.001$ [m].
Points that are closer together than this distance are considered to be coinciding.



If *Only select nodes* is checked, nodes closer than *Tolerance* will be selected but the model remains unchanged. If it is not checked, nodes closer than *Tolerance* will be deleted and a new node will be created with averaged coordinates. Lines connected to the nodes will be replaced with a single line to the new node. 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. If *Select unattached nodes or lines* is checked a warning will be displayed if there are independent lines or nodes not connected to the rest of the structure.



before checking

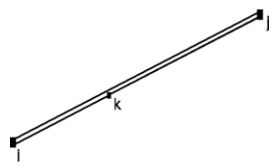


after checking

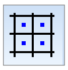
Select unattached nodes or lines:
If this check-box is enabled, AxisVM will send a warning message if unattached (independent) parts are encountered.

☞ ***The following case is not identified by the Check command.***

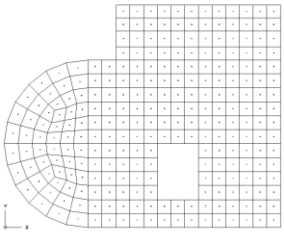
To avoid having hiding lines check** Settings / Options / Editing / Auto / Intersect **or click** Intersect **on the** Geometry **toolbar.

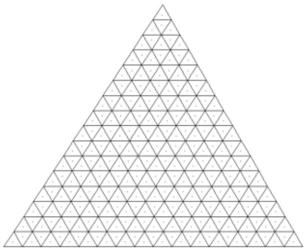


4.8.16. Surface



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.





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.*

4.8.17. 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

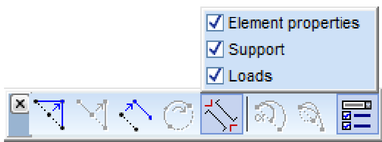
 Lengthening or shortening connecting arcs

 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 startpoint and midpoint of the original arc.

 Enabled only in detaching mode. It pops up a list of properties to be copied.

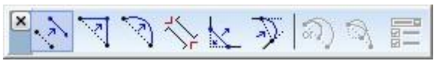


Entering node coordinates: Clicking a node the table of nodes appears where coordinates can be changed. After selecting one or more nodes their coordinates can be edited in the property editor as well.

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



 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 cutoff parallel to its original position

 Replacing a straight line with an arc based on two endpoint tangents.

 See *Dragging nodes*

Modifying arcs

 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

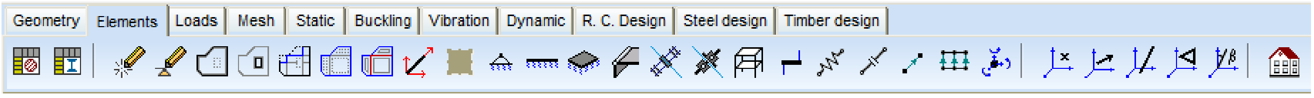
See... *2.16.6 Geometric tranformations on objects*

4.8.18. Delete

[Del] **See in detail...** *3.2.8 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:

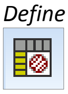
Finite element	*Material*	*Cross-section*	*Reference*	*Stiffness*	*Surface*
Truss	•	•	**o**		
Beam	•	•	•	**o**	
Rib	•	•	**o**		
Membrane	•		•		•
Plate	•		•		•
Finite element	*Material*	*Cross-section*	*Reference*	*Stiffness*	*Surface*
Shell	•		•		•
Support			•	•	
Rigid					
Spring			**o**	•	
Gap				•	
Link				•	
Edge hinge				•	

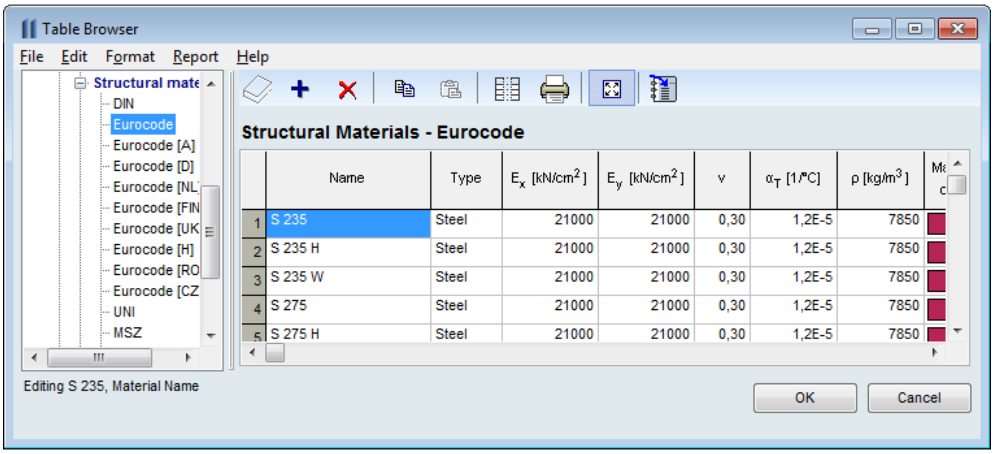
o: optional

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. Material

Define Materials





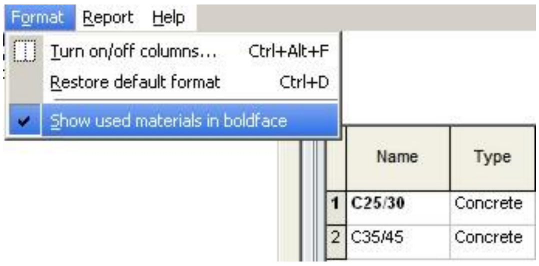
Lets you define and save material property sets or load them from a material library.
If you delete a material the definition of the elements with the respective material will be deleted.

Browse Material Library

[Shift+Ctrl+M]

The material library contains material properties of civil engineering materials based on Eurocode, DIN, NEN, SIA and other specifications.

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:

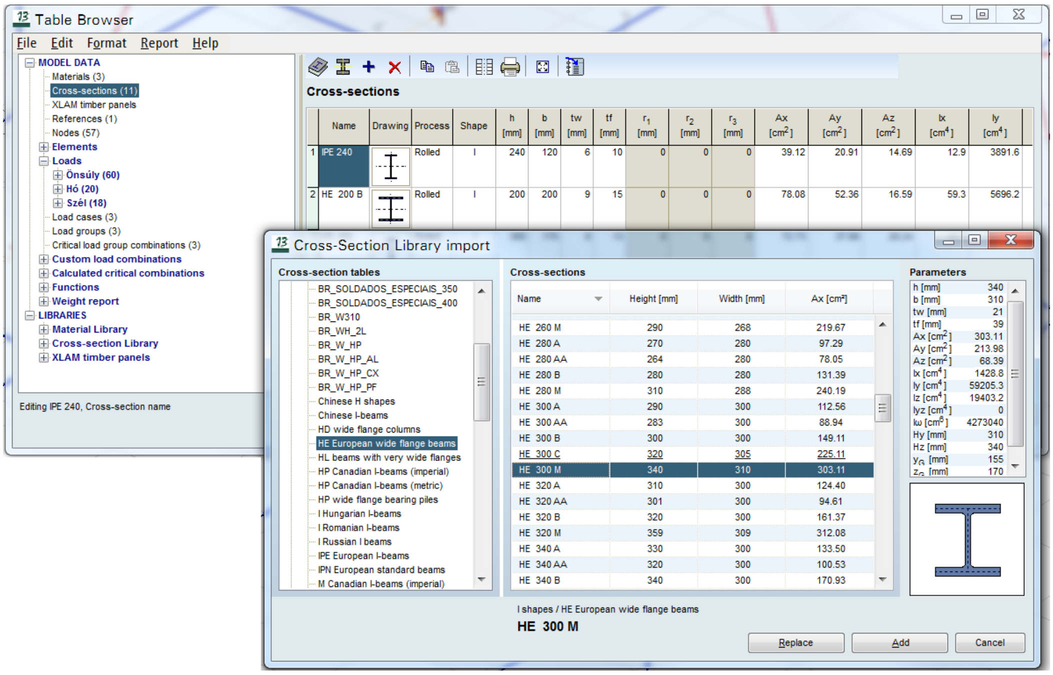
Finite Element	*E*	*ν*	*α*	*ρ*
Truss	•		•	•
Beam	•		•	•
Rib	•		•	•
Membrane	•	•	•	•
Plate	•	•	•	•
Shell	•	•	•	•
Support				
Rigid				
Diaphragm				
Spring				
Gap				
Link				

Displaying and changing material properties is described in *3.1.13 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.

4.9.2. Cross-section

Define Cross-sections



Lets you define and save cross-sectional property sets or load them from a cross-section library. The beam, truss, and rib elements require a cross-section. The properties are related to the element's local coordinate system.
For cross-section properties **see**... *3.1.14 Cross-Section Library*

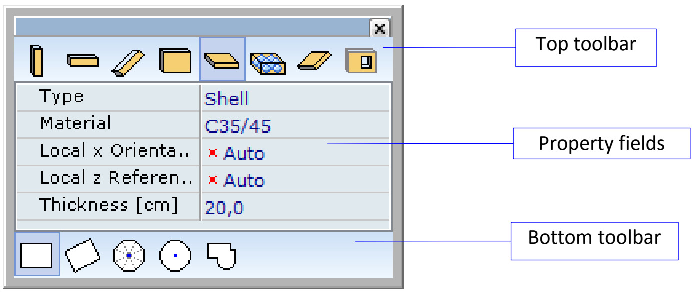
The *Browse cross-section libraries* command displays a category tree of available cross-section library tables. The list in the middle shows the cross-sections of the selected table. Clicking on column headers sorts the list into ascending/descending order by the respective parameter: *Name, Height, Width, Ax* (cross-section area).
Cross-section parameters and the drawing appears on the right.
See also *3.1.14 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.

4.9.3. Direct drawing of objects



After clicking the icon a direct drawing toolbar and property editor appears. With the help of this window coloumns, 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)

COBIAX slab Slab domain (parallel to the global X-Y plane)

Surface domain (spatial)

Hole

Object dragpoints

Column upwards / downwards

Wall upwards / downwards

Object geometry

Column

Single segment beam or wall

Beam or wall polyline

Arced beam with centerpoint, start and endpoint

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

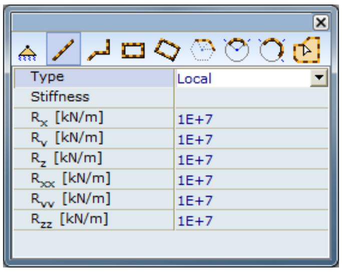
Polygon shaped slab / opening

Round shaped slab / opening

Complex slab / opening

4.9.4. 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 centerpoint and two points

Line support, arc by centerpoint and two points

Line support, arc by three points

Line support on domain edges

4.9.5. 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.

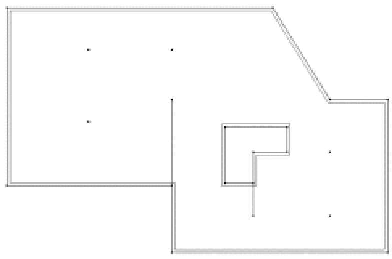
A domain has the following parameters:

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)

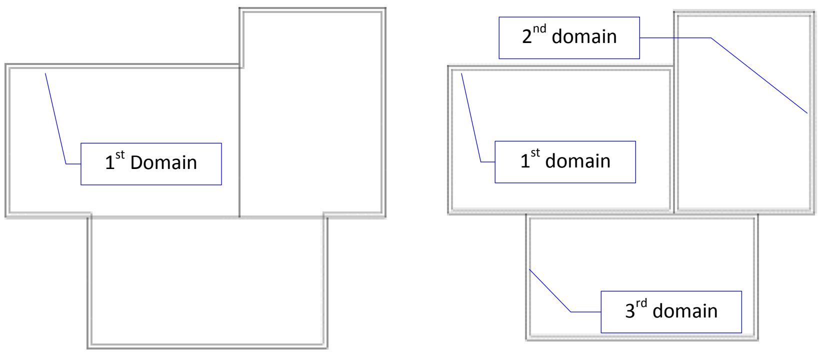
A domain is displayed by a contour line inside of the domain's polygon, with a color corresponding to the domain's element type (blue for membrane, red for plate, and green for shell).



Domains can be defined for floors, walls, and any other complex structural surface element.

The domain can be meshed automatically. **See...** *4.11.1.2 Meshing of domain*

More than one domain can be used to model a structural element.



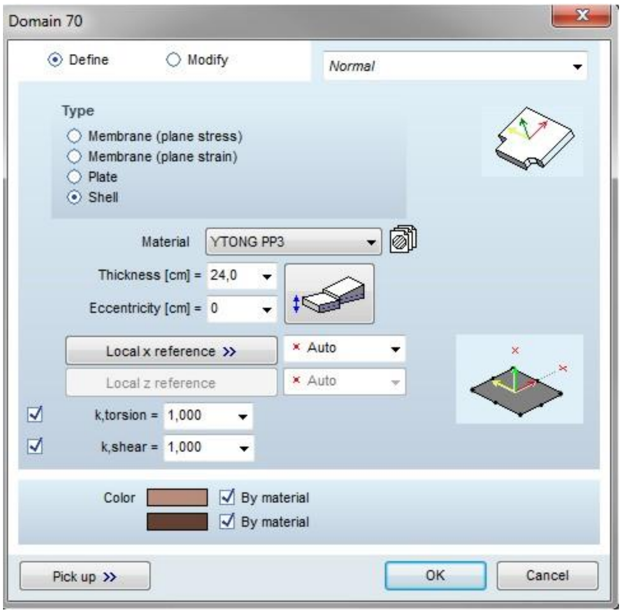
A domain can contain other (sub-) domains.

Different domain types are available

- *Normal domain* (for simple slabs, walls, etc.)
- *Cobiax domain* (for voided slabs, requires **CBX** module)
- *Airdeck domain* (for voided slabs, requires **ADK** module)
- *Ribbed domain* (for parametric ribbed slabs)
- *XLAM domain* (for cross-laminated timber panels)

4.9.5.1. Defining a normal domain

Type Domains have different types (membrane, plate, shell) according to the finite elements used for modelling. For the meaning of these types see… *4.9.9 Surface 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 k,shear If the material is masonry, it is possible to set reduced shear strength.

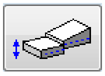
k,shear factor must be between 0.1 and 1, i.e. the shear strength of the masonry wall can be 10%-100% compared to the elastic, isotropic material.

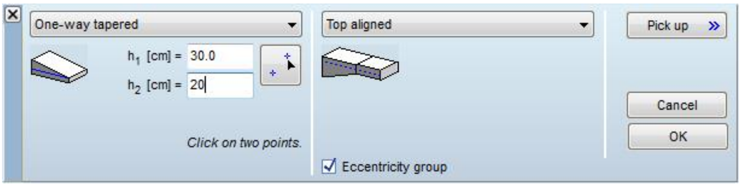
If the material is concrete, it is possible to set reduced shear strength of RC wall.

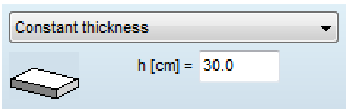
k,shear factor must be between 0.1 and 1, i.e. the shear strength of the RC wall can be 10%-100% compared to the elastic, isotropic material.

If the material is concrete, it is possible to set reduced torsional strength of RC plate.

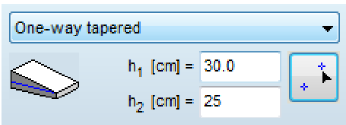
k,torsion factor must be between 0.1 and 1, i.e. the torsional strength of the RC plate can be 10%-100% compared to the elastic, isotropic material.



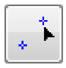


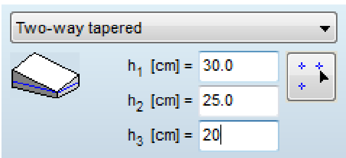


In case of *Constant thickness* the domain thickness *h* must be entered.

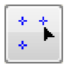


For *One-way tapered* domains enter h_1, h_2 and click on the button then click on two thickness reference points. Domain thickness will change linearly between the reference points, being h_1 at the first point and h_2 at the second one.





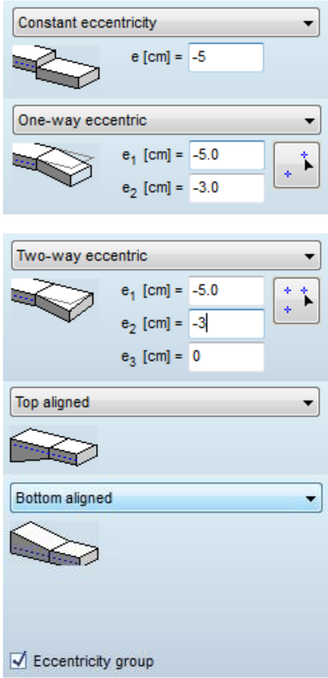
For *Two-way tapered* domains enter h_1, h_2, h_3 and click on the button then click on three thickness reference points. Domain thickness will change linearlz between reference points, being h_1 at the first point h_2, at the second one, h_3 at the third one. The three reference points cannot be on 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.***

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. 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 the plane defined by the domain.
Eccentricities and eccentricity groups can serve as a base for color coding (**see…** *2.18.3 Color coding*), and eccentricity groups can de displayed as parts (**see…** *2.16.14 Parts*).

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.

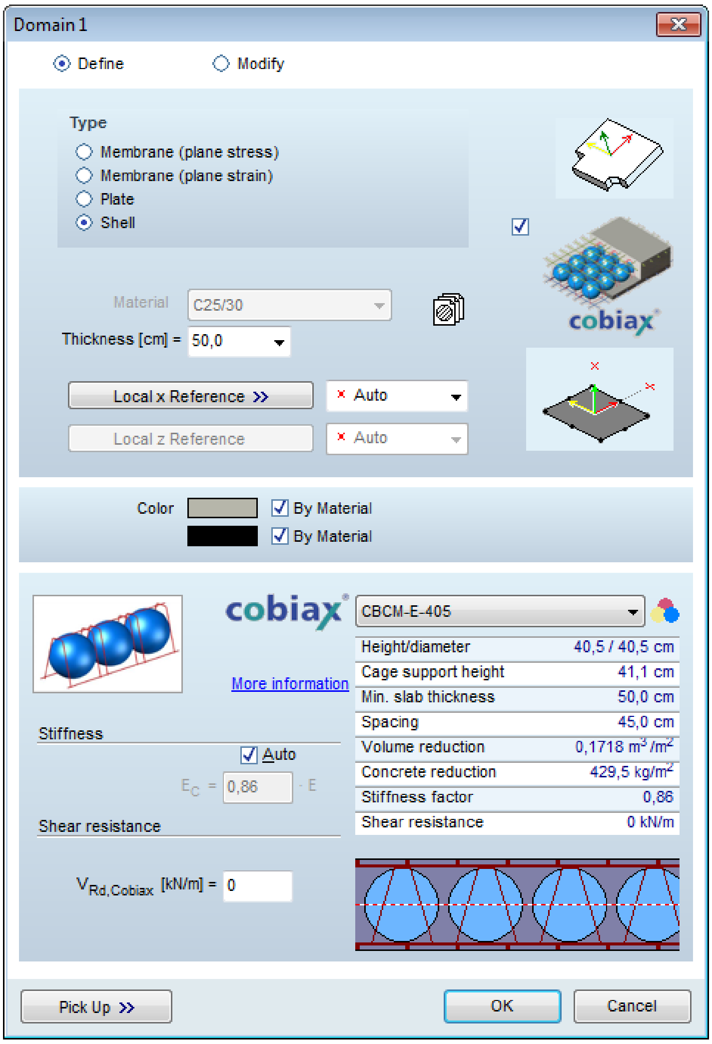
Delete a domain Press the **[Del]** button, select the domains (click on the contour line of the domain) you want to delete and click OK in the dialog.

4.9.5.2. COBIAX-domain – CBX module

If the package includes the COBIAX module (**CBX**), void formers can be placed into slabs reducing self weight and the total amount of concrete, making larger spans available. COBIAX slabs can be designed according to Eurocode, DIN 1045-1 and SIA (Swiss) design code.

COBIAX domain parameters

Clicking on the checkbox beside the graphics showing a COBIAX slab we can turn the void formers on or off. This checkbox is enabled only if the material is concrete and the thickness of the slab is at least 200 mm.
Models avaliable for the given thickness are listed in the dropdown combo box. Element parameters and the schematic diagram of the slab is displayed under the combo.
Void formers reduce the stiffness and shear resistance of the slab. If we choose *Automatic*, factors will be set to their default values. These can be overridden after clearing the checkbox.
Domain self weight will be automatically reduced and analysis will be performed with reduced stiffness and shear resistance.
Definition of shear resistance depends on the current design code.



Eurocode, DIN 1045-1

These design codes require specification of the $V_{Rd,Cobiax}$ shear resistance.
To estimate its value build the model with solid slabs and read the ($V_{Rd,c}$) shear resistance of the slab. Sheer resistance of COBIAX slabs is about half of the solid ones.

SIA 262

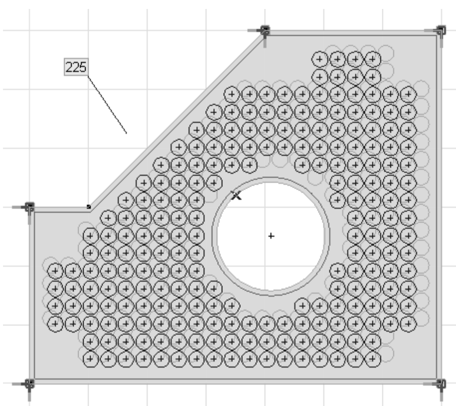
Swiss design code allows two options. It is possible to enter the actual shear resistance or only the shear factor.

☞ ***If more than one COBIAX domains were selected, their COBIAX parameters can only be redefined. Modifying COBIAX parameters of multiple domains is not allowed.***

Void formers appear as circles drawn in the slab plane in wireframe mode and balls placed into a partially transparent plate in rendered view.
Colours assigned to COBIAX-slabs and void formers can be customized by clicking on the button right to to the element type combo.

Move void formers

Void formers are positioned according to a raster depending on element type. Certain design rules are applied near holes, edges, and supports. Shifting the origin of the raster void former positions will change accordingly.
Right-clicking the domain outline choose Move Cobiax elements from the popup menu. Enter the base point of the translation vector then its end point. Number of the void formers in the resulting raster is displayed while moving the mouse.



Cobiax parameters in the output

Table Browser shows COBIAX slabs of the model and their parameters in one table under Elements. Two additional tables appear in the Weight Report section. A table titled COBIAX-elements lists elements by type with the number of void formers, the total area covered and the total weight reduction. COBIAX Weight Report displays and sums the weight reduction of individual slabs.

For details of COBIAX slab design **see...** *6.5.13 Design of voided slabs – CBX/ADK module*

4.9.5.3. AIRDECK-domain – ADK module

If the package includes the AIRDECK module (**ADK**), void formers can be placed into slabs reducing self weight and the total amount of concrete, making larger spans available. AIRDECK slabs can be designed according to Eurocode.

AIRDECK-domain parameters

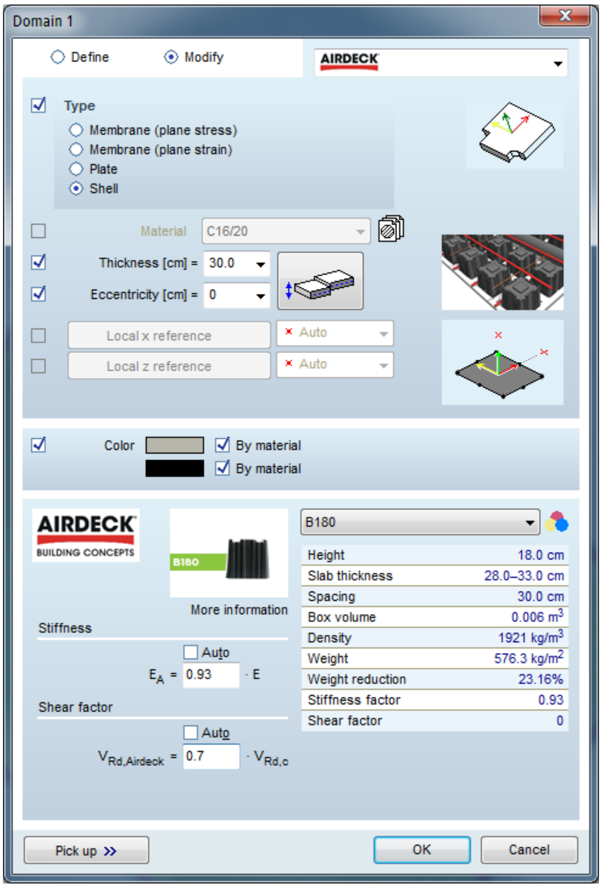
Clicking on the checkbox beside the graphics showing an AIRDECK slab we can turn the void formers on or off. This check-box is enabled only if the material is concrete and the thickness of the slab is at least 200 mm.

Models avaliable for the given thickness are listed in the dropdown combo box. Element parameters and the schematic diagram of the slab is displayed under the combo.

Void formers reduce the stiffness and shear resistance of the slab. If we choose *Automatic*, factors will be set to their default values. These can be overridden after clearing the checkbox.

Domain self weight will be automatically reduced and analysis will be performed with reduced stiffness and shear resistance.

Definition of shear resistance depends on the current design code.



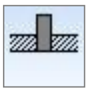
For details of AIRDECK slab design **see...** *6.5.13 Design of voided slabs*

4.9.5.4. Parametric ribbed plates

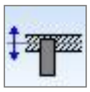
Beyond basic domain parameters the following parameters can be specified:

Rib type

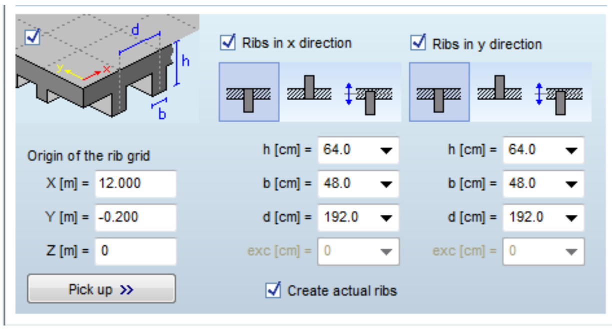
Bottom rib



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.

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***

Calculation AxisVM calculates equivalent orthotropic stiffness values for the material stiffness matrix modeling the ribbed domain with a two dimensional normal domain. This method converts the geometric orthotropy into material orthotropy. This is a homogenization process so its effectivity depends on the ratio of the characteristic size of the representative element and the domain dimensions.

☞ ***Distance between ribs must be much smaller than domain dimensions.***

Material stiffness of a general shell element can be described with the following system of equations

$$\begin{Bmatrix} \{N\} \\ \{M\} \end{Bmatrix} = \begin{bmatrix} \mathcal{A} & \boldsymbol{\mathcal{B}} \\ \mathcal{B}^{\mathrm{T}} & \boldsymbol{\mathcal{D}} \end{bmatrix} \begin{Bmatrix} \{\epsilon_0\} \\ \{\kappa\} \end{Bmatrix}; \ \begin{Bmatrix} Q_y \\ Q_x \end{Bmatrix} = K_s \begin{bmatrix} A_{44} & A_{45} \\ A_{45} & A_{55} \end{bmatrix} \begin{Bmatrix} \gamma_y \\ \gamma_x \end{Bmatrix}$$

where

$$\mathcal{A} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{12} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix}; \ \mathcal{B} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{12} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix}; \ \mathcal{D} = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{12} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix}$$

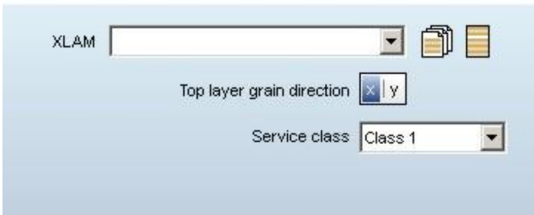
matrices can be derived from the 6x6 stiffness matrix of the Hooke model for orthotrop materials. $\mathcal{B}$ represents the material relation between normal forces and bending. As the statical framework of a ribbed plate is in the midplane of the domain, this matrix will have nonzero elements however some of the effects of these components to the displacements are ignored assuming that the normal forces have no eccentricity. Other stiffness values are calculated from the equations of equilibrium applied to the representative element and from the compatibility of shear stress at the connecting surfaces of the plate and the rib grid.
Another consequence of the homogenization is that stiffness peaks at the rib intersections will be smoothed so the domain with an equivalent stiffness will be more sensitive to local loads.

☞ ***Parametric ribbed plates provide reliable results for distributed loads. Large concentrated loads can cause considerable inaccircacy.***

Create actual ribs This 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.

4.9.5.5. XLAM domain

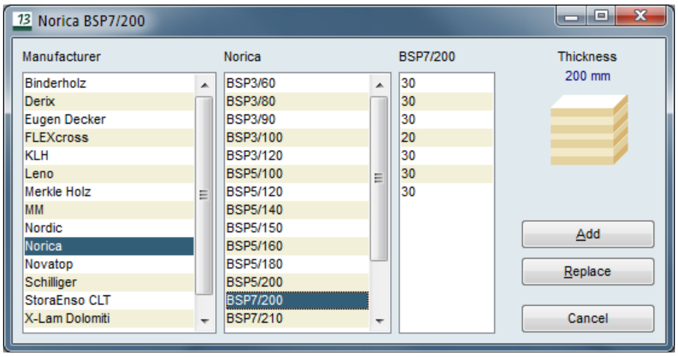
This domain type requires the XLM module handling XLAM (cross-laminated timber) panels. AxisVM offers a library of many common products but custom layer structure can also be entered.



Analysis provides displacements, forces and stresses in XLAM domains.

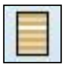
Definition

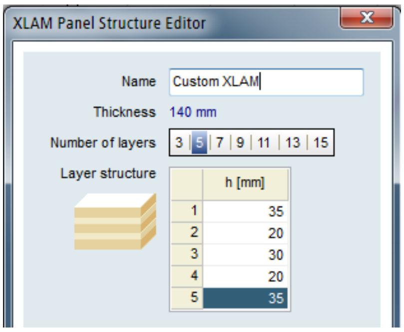
Browse XLAM libraries



XLAM layer structure can be loaded from libraries

Panel Structure Editor





The *Name* field contains the name of the panel structure. *Thickness* is the calculated total thickness of the panel. *Number of layers* is always odd, layer structure must be symmetrical. So editing a line of the *Layer structure* immediately changes the value of its symmetric counterpart.

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 Timber cross-section optimization – TD9 module*

Top layer grain direction

Grain direction of the topmost layer must be specified it can be the local *x* or *y* direction.

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

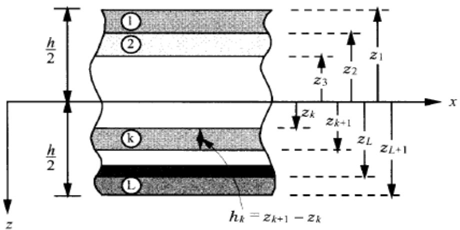
$$\mathcal{A} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{12} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix}; \quad \mathcal{B} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{12} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix}; \quad \mathcal{D} = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{12} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix}$$

matrices can be derived from the 6x6 stiffness matrix of the Hooke model for orthotrop 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 only symmetric laminated structures, so the layer thickness values and the grain direction pattern must be symmetrical to the midplane of the panel, and grain directions must be parallel. The above matrices are calculated as

$$A_{ij} = \sum_{k=1}^{L} (\bar{Q}_{ij})_{(k)} (z_{k+1} - z_k),$$

$$B_{ij} = \frac{1}{2} \sum_{k=1}^{L} (\bar{Q}_{ij})_{(k)} ({z_{k+1}}^2 - {z_k}^2) = 0,$$

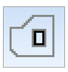
$$D_{ij} = \frac{1}{3} \sum^{L} (\bar{Q}_{ij})_{(k)} ({z_{k+1}}^3 - {z_k}^3).$$



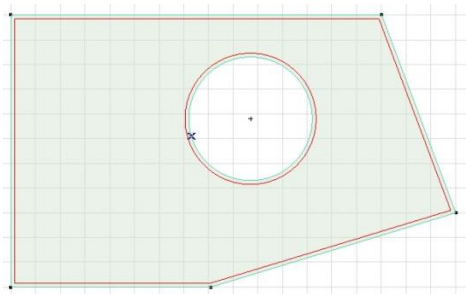
Lamination pattern. (Reddy, J. *Mechanics of Laminated Composite Plates and Shells.* CRC Press, 2004.)

where $\bar{Q}_{ij}$ is the Hooke-modell value transformed into the panel coordinate system.

4.9.6. Hole



Holes can be defined in domains. Holes have to be inside the domain and in the domain's plane.



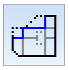
Select the (closed) polygons that are the edges of the holes you want to define. More than one outline can be selected. If an outline is not in the plane of the domain no hole will be created. You can move holes from one domain to another, or change their shape. If the hole outline intersects the domain outline the hole is.

Holes are displayed by a contour line with the color of the domain in which they are located.deleted.

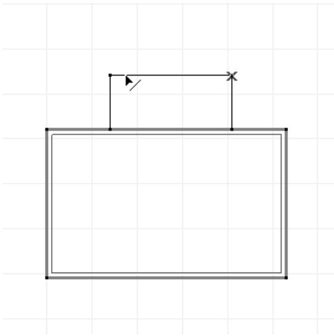
4.9.7. Domain operations

Domain contours can be changed, cut and a union of domains can be calculated.

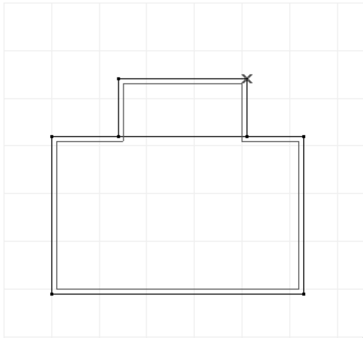
Change domain contour



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

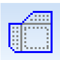


After

☞ ***Domain properties (material, thickness, local system) will be retained but the existing mesh will be removed.***

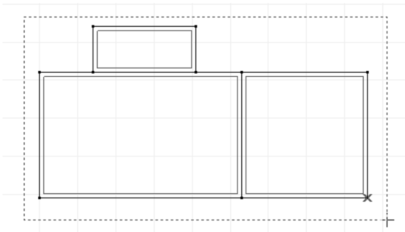
If loaded areas are removed from the domain, loads will automatically be removed.

Union of domains

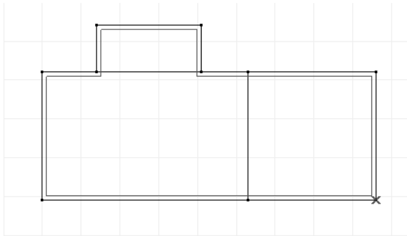


Union can be created from adjacent domains.

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.

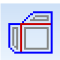


Before



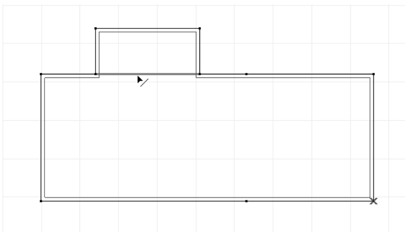
After

Cut domains

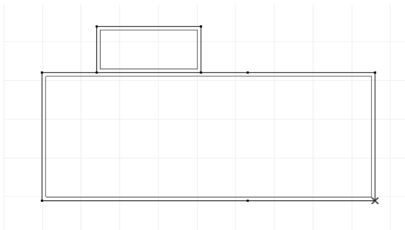


To cut domains along en existing line:

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



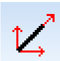
After

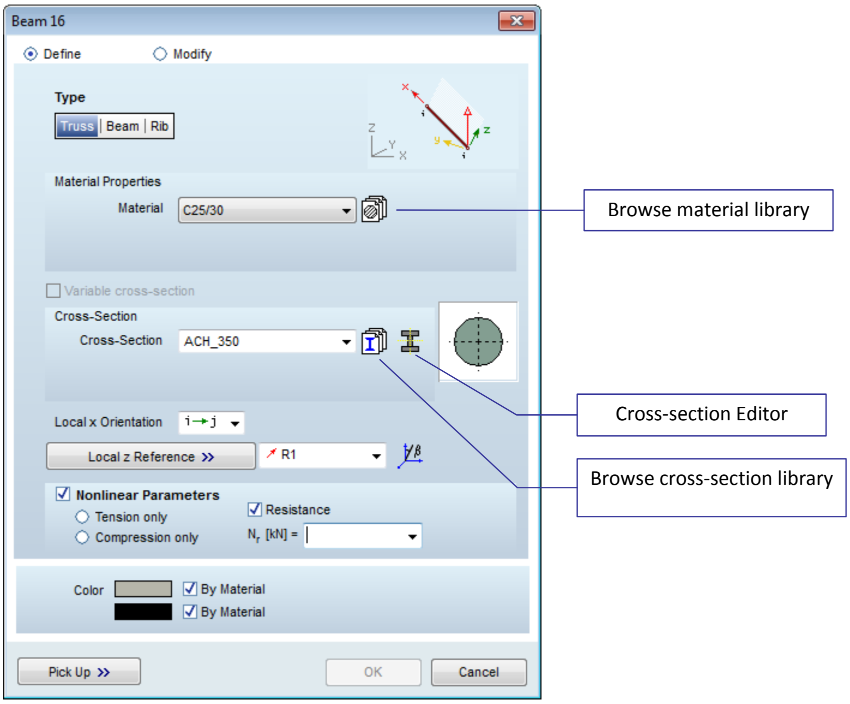
4.9.8. Line elements

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...** *3.2.13 Assemble structural members*, *3.2.14 Break apart structural members*

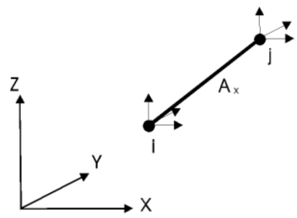
Color Elements can have their own fill and outline color used in rendered display mode. Default colors are taken from the material colors. If a color coding is applied the color of a line element is determined by the color coding both in wireframe and rendered modes.
See… *2.16.5 Color coding.*

Truss





Truss elements can be used to model truss structures. Trusses are two node, straight elements with constant cross-section properties 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).



Axial internal forces N_x are calculated for each truss. The variation of the axial force 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.

Allows browsing of the cross-section library to assign a cross-section to the element. The cross-section selected will be added to the cross-section table of the model.

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 and a timber material is selected, the service class can be set here. **For details see**…*6.7 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 vica 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 endpoint 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... *4.9.20 References*

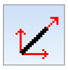
Reference angle Rotation of cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel to the global Z direction, the angle is relative to the global X axis. In 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.

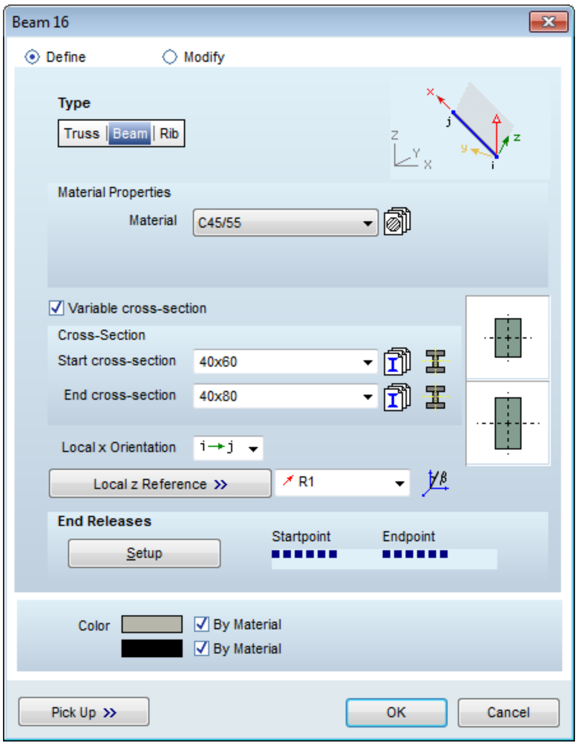
The nonlinear parameters are taken into account only in a nonlinear analysis.

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.

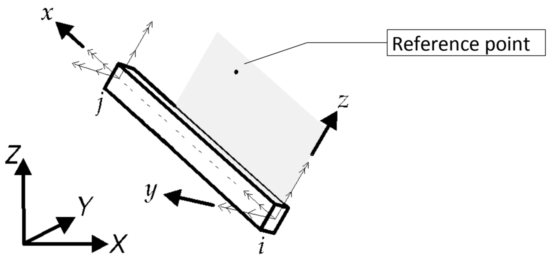
Beam



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.
The variation of the internal forces along the beam are: constant axial force, constant torsion, constant shear forces and linear moments.
The displacements and internal forces are calculated at intervals of at least 1/10 of the element length.



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

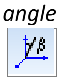
Defining material, cross-section and local direction X are similar to truss elements.

Automatic reference

The reference vector will be generated by the program according to the section *4.9.20 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

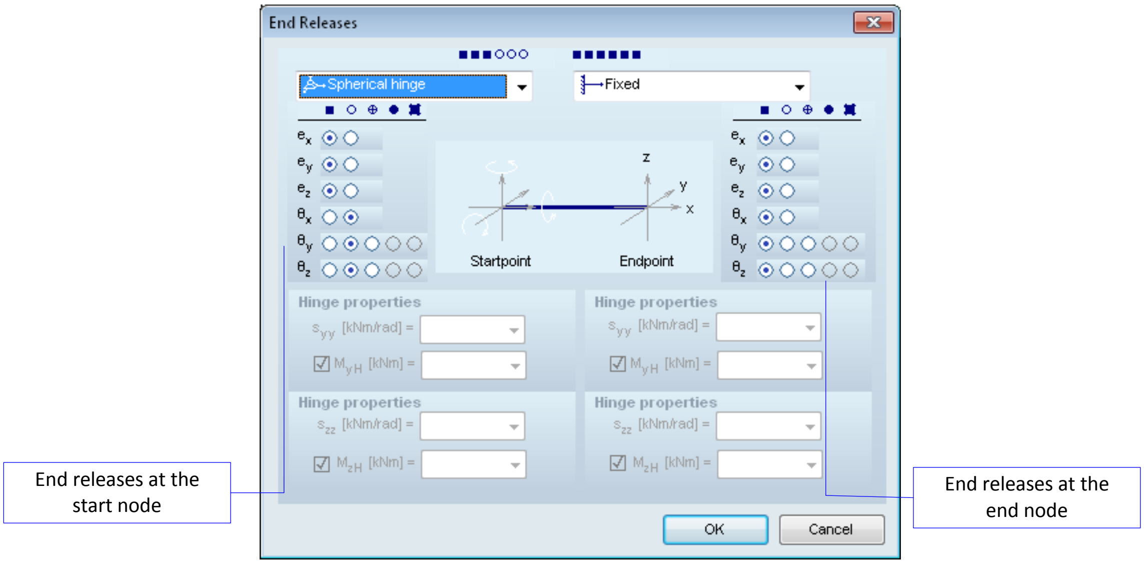


Rotation of cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel to the global Z direction, the angle is relative to the global X axis. In any other case the angle is relative to the global Z axis.

The beam elements are displayed on the screen as blue lines.

End releases

You can specify releases that remove the connection between the selected elements' degrees of freedom (in the local coordinate system) and the nodes. The end-releases are set by a six code set for each end. Each code corresponds to one internal force component. 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.



 Graphical symbol of a rigid connection code (the corresponding local displacement component of the beam end is transferred to the node)

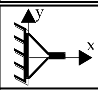
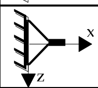
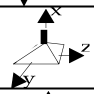
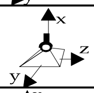
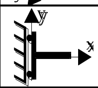
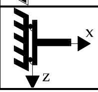
 Graphical symbol of a hinged connection code (the corresponding local displacement component of the beam end is not transferred to the node)

 Graphical symbol of a semi-rigid connection code (the corresponding local displacement component of the beam end is partially transferred to the node)

 Graphical symbol of an elastic-perfectly plastic connection: the maximum value of the moment at the endpoints is calculated from the material and cross-section properties.

 Graphical symbol of customizable pushover hinge: the corresponding moment-rotation relationship is defined by the user..

The table below demonstrates the use of end releases for some common cases:

End Release	**Symbol**
Hinge in ***x-y*** plane. Can’t transmit M_z moment.	
Hinge in ***x-z*** plane. Can’t transmit M_y moment.	
Hinge in ***x-y*** and ***x-z*** plane. Can’t transmit M_z and M_y moments.	
Hinge in ***x-y*** and ***x-z*** plane and free rotation about local ***x*** axis (spherical hinge). Can’t transmit M_x, M_y, and M_z moments.	
Free translation along local ***y*** axis. Can’t transmit V_y shear force.	
Free translation along local ***z*** axis. Can’t transmit V_z shear force.	

☞ ***Care must be taken not to release an element or group of elements such that rigid body translations or rotations are introduced.***

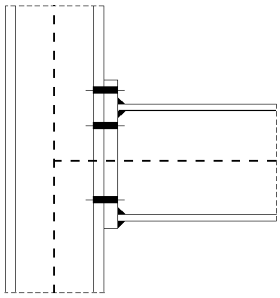
For example, if you specify spherical hinges at both ends (code: 000111), a rigid body rotation about element axis is introduced. In this case at one of the ends you may not release the element degree of freedom corresponding to the rotation about local *x* axis (e.g. ***i*** end numerical code: 000011; ***j*** end numerical code: 000111).

Example: Start node ■■■■○○ End node ■■■○○○

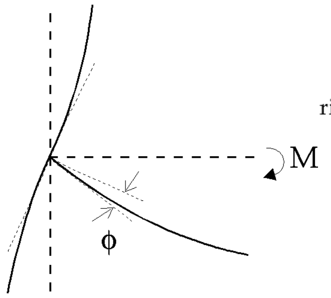
Semi-rigid connection

To define semi-rigid hinges set the radio button to semi-rigid and enter the torsional stiffness of the linear elastic spring modeling the connection about the local axis *y* or *z*. The value should be the initial stiffness of the real connection M-ϕ characteristics.
The moment - relative rotation diagram of a connection is modeled by a linear or nonlinear elastic rotational spring. The nonlinear characteristic can be used only in a nonlinear static analysis. In a linear static, vibration, or buckling analysis, the initial stiffness of the connection is taken into account.

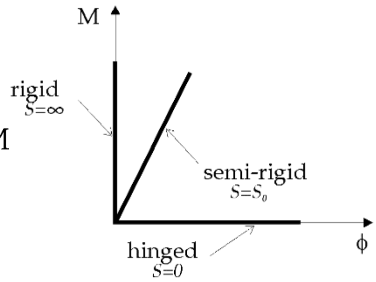
Connection:



Model:



Moment - Relative Rotation Diagram



☞ ***For example, in the case of steel frame structures, Eurocode 3 Annex J gives the details of application.***

Moment Resistance

To fixed or semi-rigid connections a moment resistance can be assigned, that is the maximum moment that can develop in the connection.

☞ ***The moment resistance parameter is used only in case of a non-linear analysis.***

Steel plastic hinge

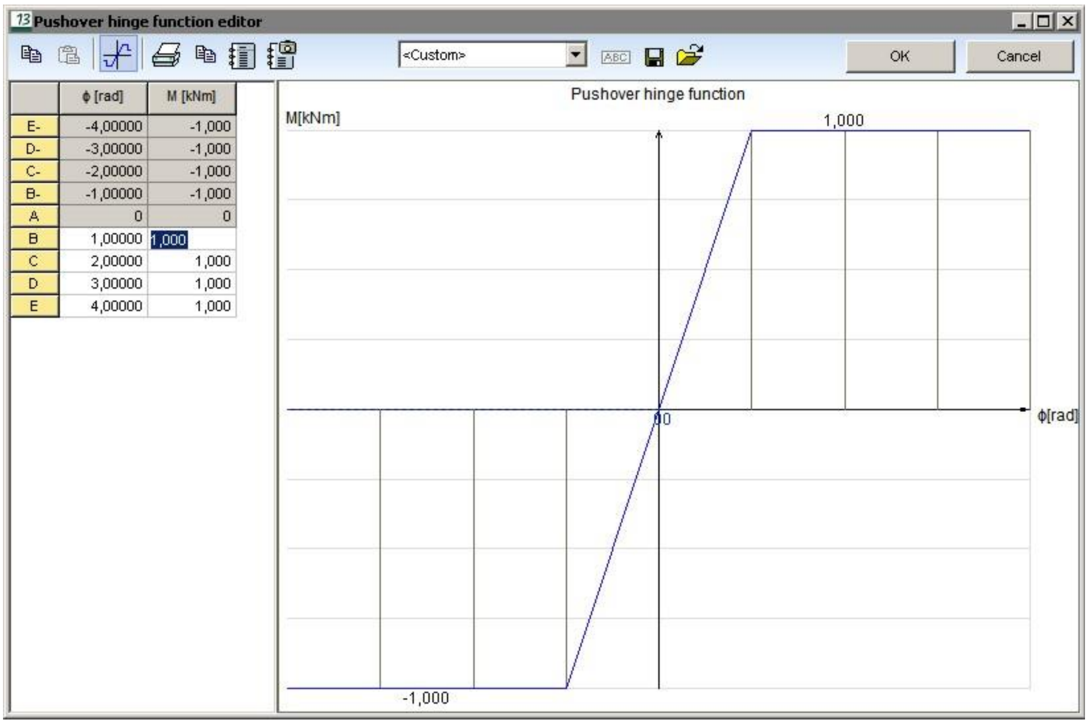
To define steel plastic hinges set the radio button to steel plastic.. Moment resistance will be displayed but cannot be edited. If elements with different materials or cross-sections are selected no value will appear in the edit field but hinges will be defined with the appropriate moment resistance.
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.***

Pushover hinge

To define pushover hinges set the radio button to pushover hinge. A custom moment-resistance relationship can be defined by clicking on the *Function editor* button under the appropriate pushover hinge characteristic title.

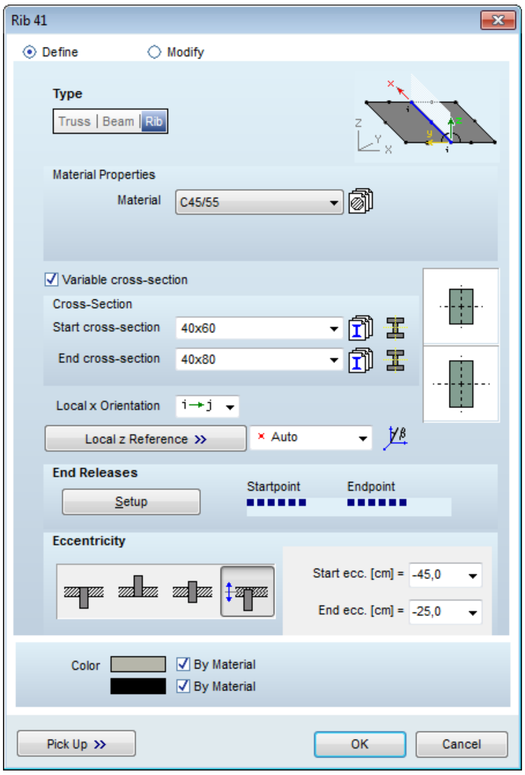
A total of five points can be defined for both directions of the moment-rotation diagram. This allows for modeling of complex connection behaviour including the possibility of hardening, softening and strength degradation. Behaviour after the last point is extrapolated based on points D and E. The diagram 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.

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.

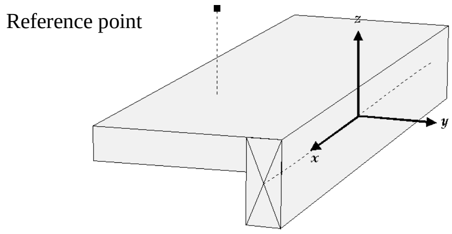
Material The material of the rib can be different from the surface material (if it is connected to a surface).

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 *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.

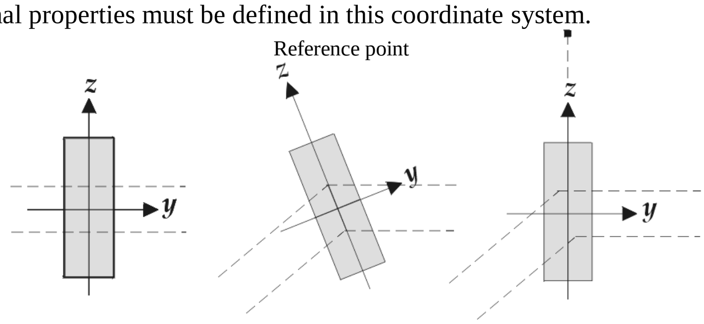


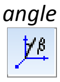
Rib connected to a surface element:

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 beam 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 available when defining the rib.

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.

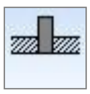
End releases End releases can be defined for ribs the same way as for beams. By default both ends are fixed.

Eccentricity You can specify eccentricity for a rib only if it is on the edge of one or two surfaces. If more than two surfaces are connected to the edge select one or two of them to define eccentricity for the rib.

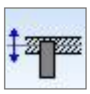
The eccentricity (ecc) of a rib is given by the distance of the center of gravity of its cross-section to the plane of the model of the surface (neutral plane). It is positive if the center of gravity is on the positive direction of its local z axis.

There are four options to set the rib eccentricity. Bottom rib, top rib, rib in the midplane or custom eccentricity.


Bottom rib


Top rib

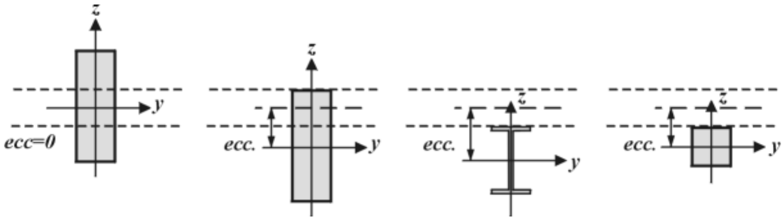

Rib in midplane


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 are 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:***

$$I_y^* = I_y + A \cdot exc^2$$

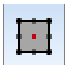
For shells, due to the eccentric connection of the rib to the shell, axial forces will appear in the rib and shell.

Ribs appear as blue lines.

Modifying Selecting elements of the same type and clicking the tool button Modifying will be actived. Properties of elements can be changed if the checkbox before the value is checked. If a certain property is does not have a common value its edit field will be empty. If a value is entered it will be assigned to all selected elements.

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.

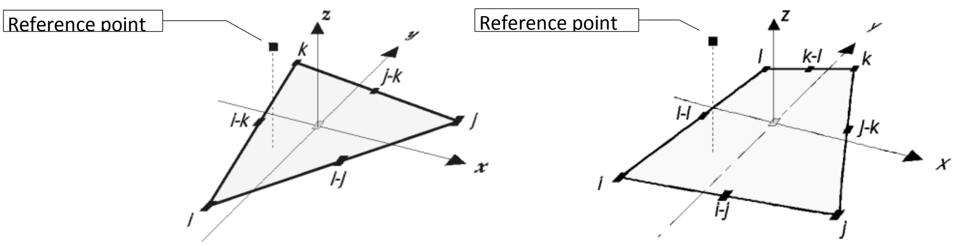
4.9.9. Surface elements



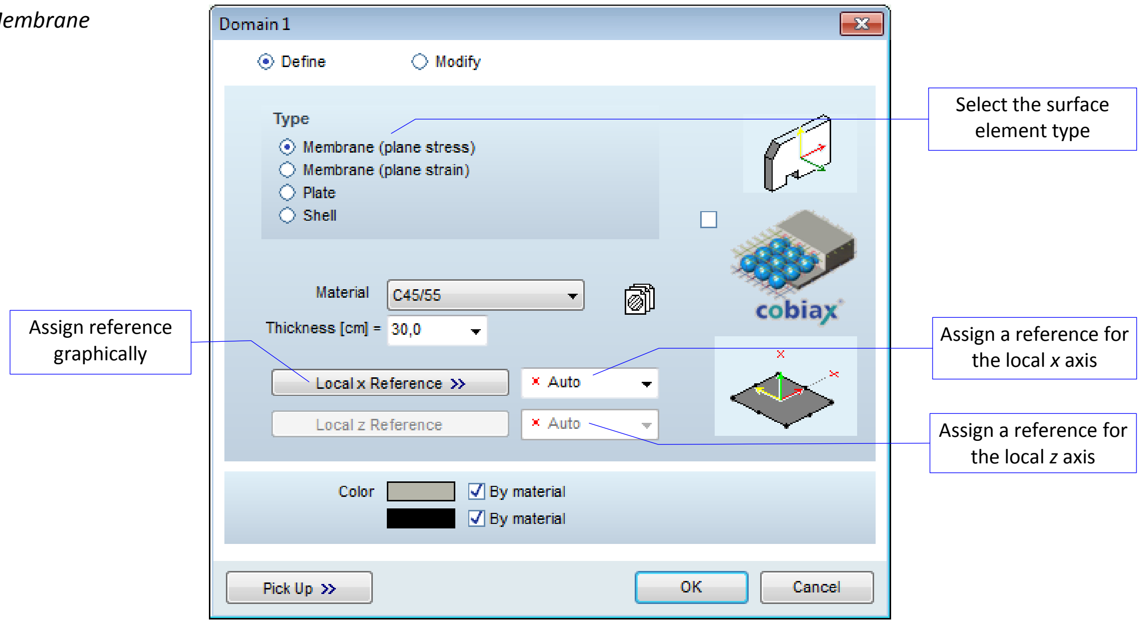
Surface elements can be used to model membranes (membrane element), thin and thick plates (plate element) and shells (shell element) assuming that the displacements are small.
As surface elements you can use a six node triangular or eight/nine node quadrilateral finite elements, formulated in an isoparametric approach. The surface elements are flat and have constant thickness within the elements.

☞ ***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.



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 *4.9.20 References* or can be set automatically.

The center of the membrane elements is displayed on the screen in blue.

Plate

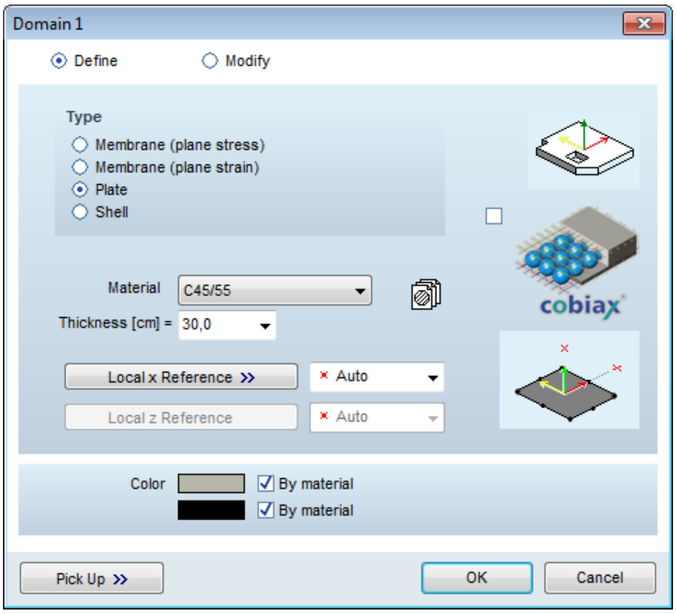


Plate elements may be used to model flat structures whose behavior is dominated by flexural effects.

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.

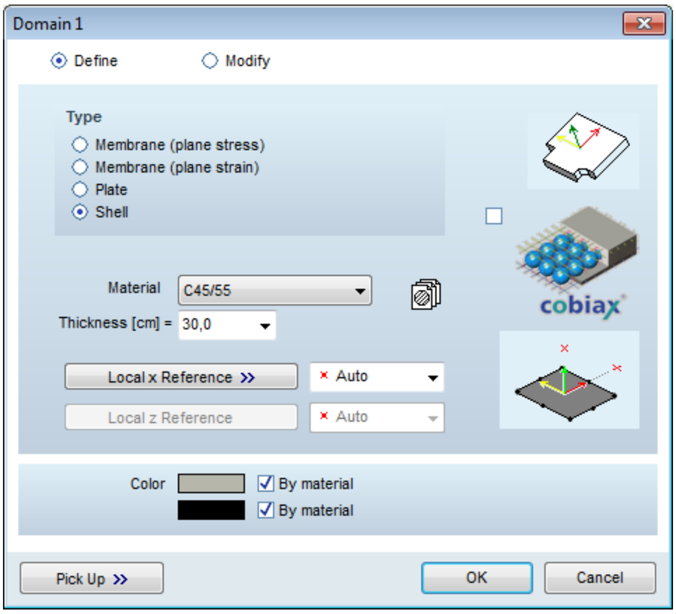
The following parameters should be specified:

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** *4.9.20 References* or can be set automatically.

The center of the plate elements is displayed on the screen in red.

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** *4.9.20 References* or can be set automatically.

The center of the shell elements is displayed on the screen in green.

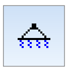
Modifying

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.

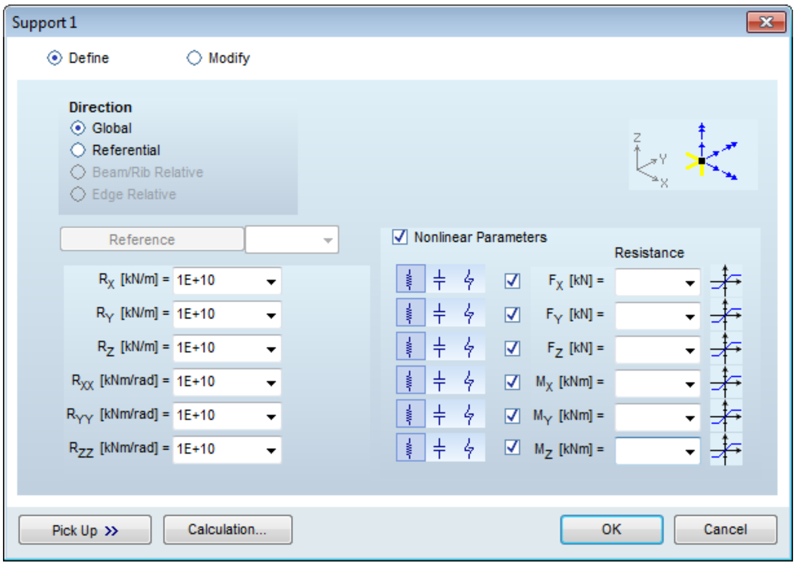
Pick up>>

See... *Pick up* at *4.9.8 Line elements*

4.9.10. 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) stiffness values about the element axes. 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 default stiffness values are 1E+10 [kN/m], [kNm/rad].

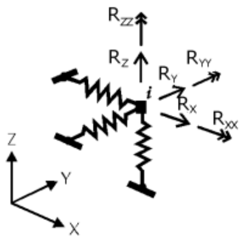
The support elements are displayed on the screen in yellow (translational spring) or orange (rotational spring).

The support can be defined in the following systems:

- Global *- Beam/rib relative*

- Reference *- Edge relative*

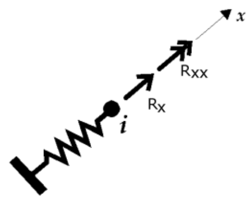
Global Defines nodal support elements parallel to global coordinate axes. You must select the nodes that are identically supported, and specify the corresponding translational (R_X, R_Y, R_Z) and rotational (R_{XX}, R_{YY}, R_{ZZ}) stiffnesses.

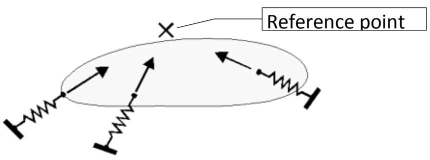


You can define only one global support for a node. You cannot define nodal support for a midside node of a surface element.

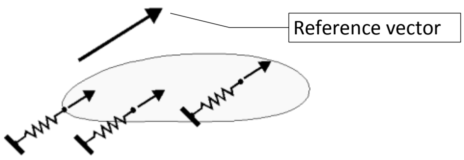
Reference Defines nodal support elements in the direction of a reference (point or vector). You must select the nodes that are identically supported, and specify the corresponding stiffness (translational R_x, and rotational R_{xx}).

The direction of the reference vector is defined by the element node and its reference point or reference vector in the following way:





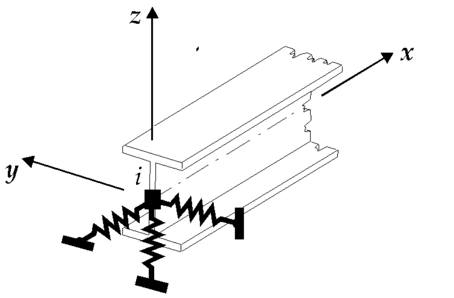
Support elements oriented toward a reference point



Support elements parallel to a reference vector

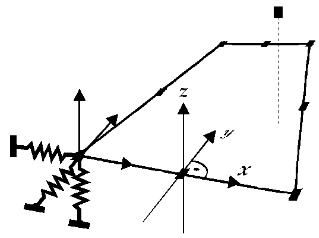
Beam/rib relative

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 R_x, R_y, R_z and rotational R_{xx}, R_{yy}, R_{zz} stifnesses.



Edge relative

Defines nodal support elements about local coordinate axes of surface element edges. You must select the surface elements and the nodes that are identically supported, and specify the corresponding translational R_x, R_y, R_z and rotational R_{xx}, R_{yy}, R_{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.

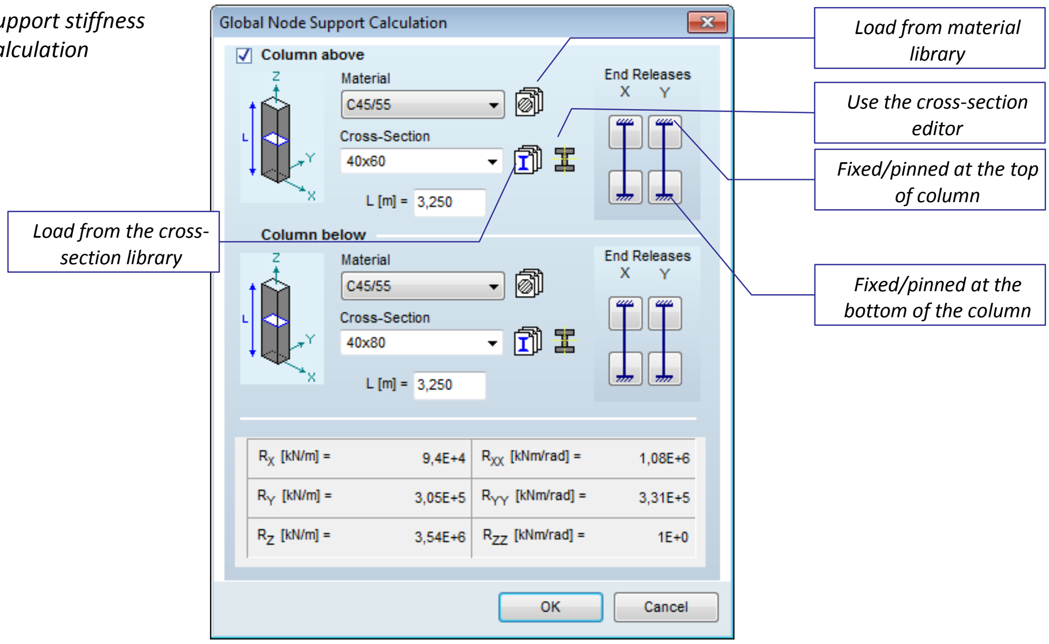
Nonlinear behavior

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 be also be entered.

☞ ***The non linear 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.***

Nodal supports appear as brown (R_X, R_Y, R_Z) and orange (R_{XX}, R_{YY}, R_{ZZ}) pegs in 3 orthogonal direction.

Support stiffness calculation



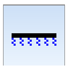
Use the *Calculate...* button to calculate the support stiffness (including the rotational stiffness) due to a column type support. The support stiffnesses are determined based on the end releases, material, and geometry of the column.

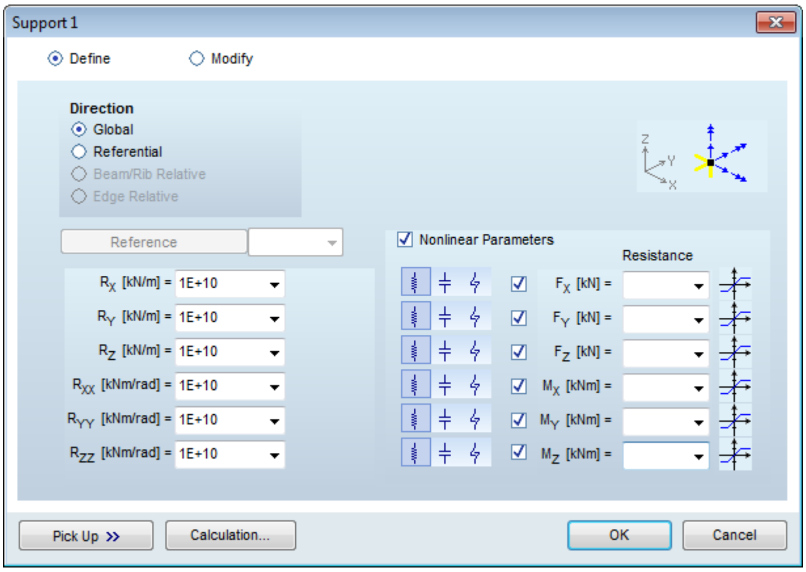
Calculating nodal support stiffness a column below and a column above the node can be specified separately. These column parameters can also be used in punching analysis (especially in the case of intermediate slabs). The columns and walls modeling the supports also appear in rendered view and the cursor can identify them.

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 *4.9.8 Line elements*

4.9.11. Line support





Line support elements may be used to model the line support conditions of a structure. Line support elements (Winkler type) are elastically supporting beams, ribs, or surface edges, while the internal forces are the support reactions.

You can specify the translational and/or rotational (torsional) stiffness values about the element axes. 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*

☞ ***The default stiffness values are 1E+7 [kN/m/m], or [kNm/rad/m].***

Global Defines line support elements parallel to global coordinate axes. You must specify the corresponding translational (R_X, R_Y, R_Z) and rotational (R_{XX}, R_{YY}, R_{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 R_x, R_y, R_z and rotational R_{xx}, R_{yy}, R_{zz} stifnesses.

☞ ***The beams/ribs with line supports must be divided into at least four elements.***

In addition, the following condition must be satisfied:

$$L \le l_k = \frac{1}{2}\min\left(\sqrt[4]{\frac{4E_x I_z}{k_y}}, \sqrt[4]{\frac{4E_x I_y}{k_z}}\right)$$

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.***

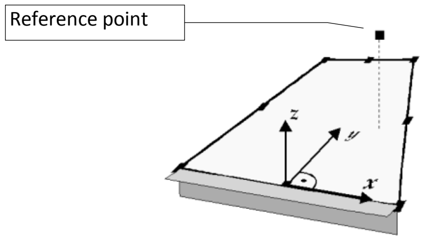
If you specify line supports the internal forces are linearly interpolated between the ends of the element, therefore the division of the elements is required.

Edge relative

Defines edge support elements relative to local coordinate axes of the edges. You must specify the corresponding stiffness (translational R_x, R_y, R_z and rotational R_{xx}, R_{yy}, R_{zz}).
If one surface is connected to the edge the local coordinate axes of the edge are:

- $\boldsymbol{x}$ = the axis of the edge
- $\boldsymbol{y}$ = the axis is oriented toward inside of the surface element in its plane
- $\boldsymbol{z}$ = 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 deter-mined based on the selected surfaces.



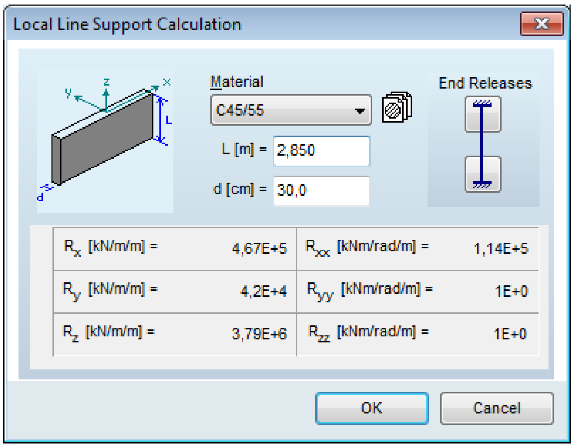
Nonlinear behavior

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 non linear 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.***

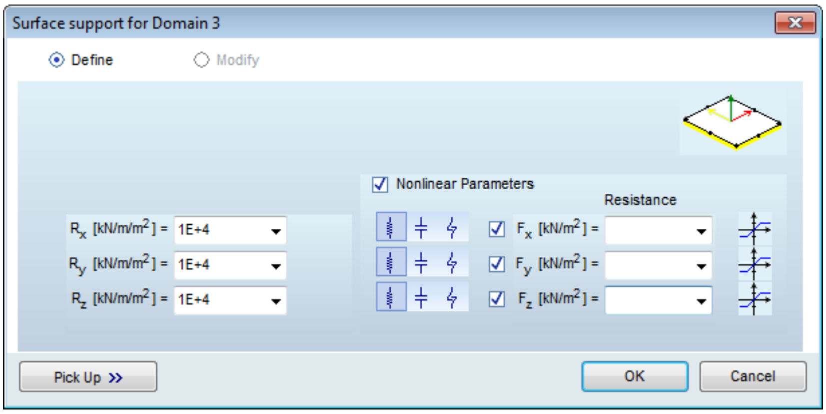
Line supports appear as brown (R_X, R_Y, R_Z) and orange (R_{XX}, R_{YY}, R_{ZZ}) lines.

Support stiffness calculation



Use the *Calculate*... button to calculate the global or edge-relative line support stiffness (including the rotational stiffness) due to a wall type support. The support stiffnesses are determined based on the end releases, material, and geometry of the wall.

4.9.12. Surface support



Surface support

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 R_x, R_y, R_z (Winkler's modulus) about the surface element local x, y, and z axes.

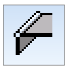
☞ ***The default stiffness values are 1E+4 [kN/m/m], or [kNm/rad/m].***

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).

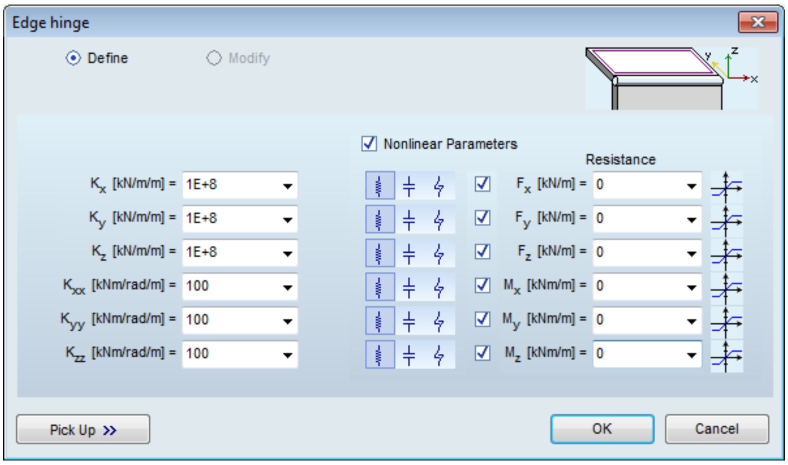
☞ ***The non linear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account.***

Surface supports appear as an orange square-hatched fill.

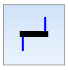
4.9.13. Edge hinge



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.



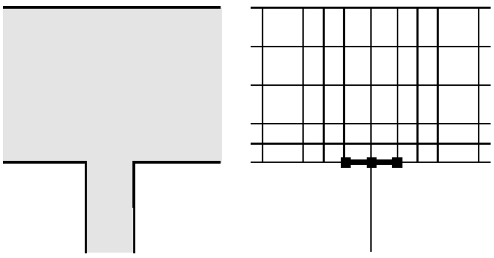
4.9.14. Rigid elements



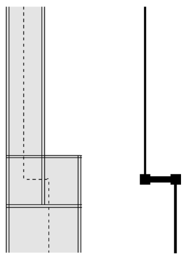
Rigid elements may be used to model parts with a rigid behavior relative to other parts of the structure.
The elements can be defined by selecting the lines that connect its nodes. The selected lines that have common nodes define the same rigid element. There is no limit to the number of nodes of any element.

☞ ***The degrees of freedom of the nodes of a rigid element cannot be constrained (fixed).***

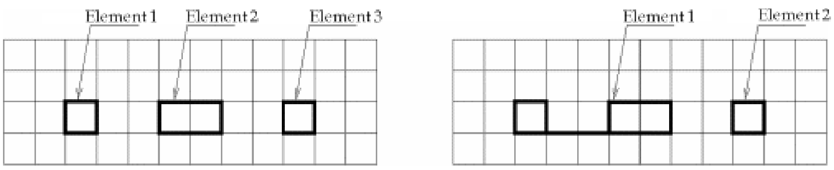
Modeling membrane–beam element connection:



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 rigid elements are displayed on the screen with thick black lines.

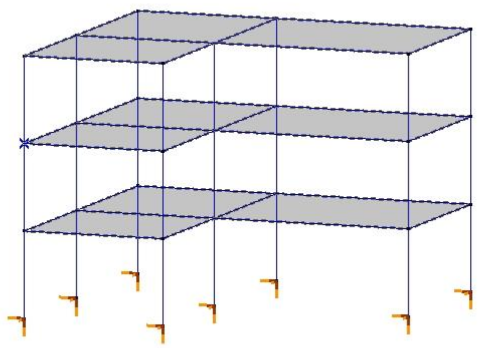
4.9.15. Diaphragm

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.

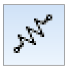
Definition Select lines to define diaphragms. Each set of connecting lines will form a diaphragm.

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.

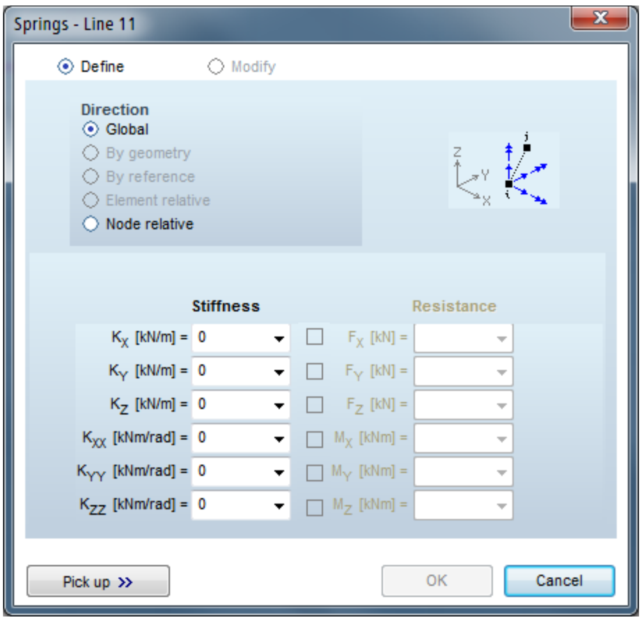


4.9.16. Spring

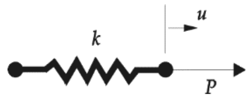


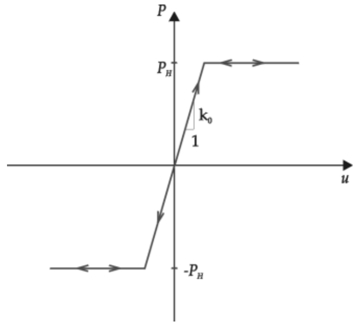
The spring element connects two nodes of the model. The element has its own coordinate system. You can specify the translational and/or rotational (torsional) stiffness values about the element axes. The element can have nonlinear elastic stiffness properties.
Springs can be defined in the following systems:

Global
By geometry
By reference
Element relative
Node relative



Define You must select the nodes that are connected, and specify the corresponding stiffness (translational K_X, K_Y, K_Z and rotational K_{XX}, K_{YY}, K_{ZZ}). If a nonlinear elastic spring is to be defined, you can specify resistance values, for each internal force component.





☞ ***Resistances will be taken into account only in a nonlinear static analysis, otherwise they will be ignored.***

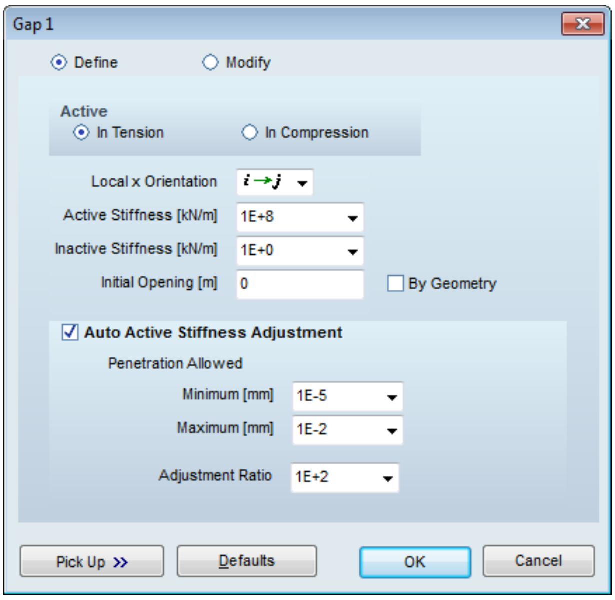
The nonlinear parameters are taken into account only in a nonlinear analysis. In any other case in the analysis (Linear static, Vibration I/II, Buckling) the initial stiffnesses are taken into account (that stay constant during the analysis).

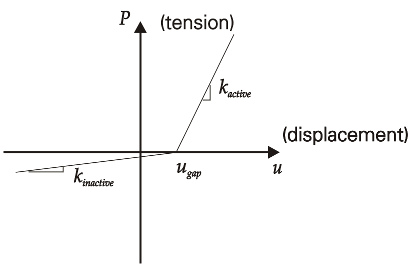
4.9.17. Gap

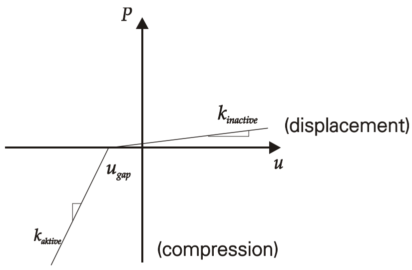
The gap element is used to model point-to-point contact. **The element has two states:**

- one ***active***, when it has a large stiffness value (simulates that a contact is achieved)
- one ***inactive***, when it has a small stiffness value (simulates that no contact is achieved).

This contact model is approximate.
The gap element can be active in tension or compression. Typical force-displacement diagrams of gaps active in tension and compression are shown below correspondingly.







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 the element is used to model regular contact problems, you may allow the element to auto adjust its stiffness, in order to smooth the large stiffness variations (at status changes) that can cause even divergence of the iterative solution process.

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.

Auto active stiffness adjustment:
If no adjustment is selected, the values below are not taken into account.

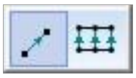
Minimum allowed penetration: You can set a minimum value for the penetration of the contact condition that is allowed. By default is 1E-05.

Maximum allowed penetration: You can set a maximum value for the penetration of the contact condition that is allowed. By default is 1E-05.

Maximum adjustment ratio: If the penetration is below the minimum, the active stiffness is softened by a maximum ratio entered here. If the penetration is between the two limits, no action is taken. If the penetration exceeds the allowed maximum, the active stiffness is hardened by a maximum ratio entered here. The default value is 100. In this case, the value of the adjustment ratio is the taken as: 1/100, 1/10, 1, 10, or 100.

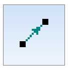
☞ ***If the gap element is used in an analysis different from a nonlinear static analysis, the element will be taken into account as a spring with a stiffness corresponding to its initial opening. If the initial opening is zero, the active stiffness will be taken into account.***

4.9.18. Link



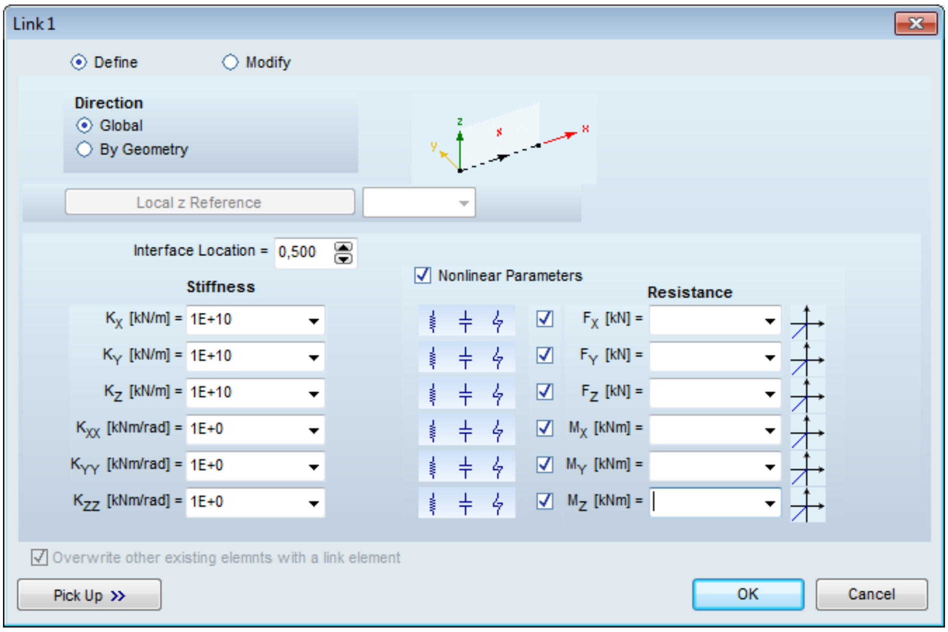
Link elements

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.



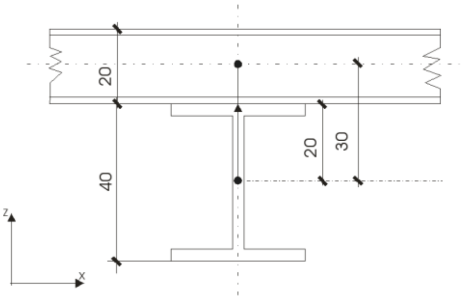
Node-to-Node (N-N) Link

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.

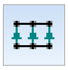
Example**:** A main girder-purlin connection (**see**… SteelFrame.axs in the examples folder)
Let assume that the vertical axis is Z being parallel to the local z axis. The main girder is an IPE-400 in X-Z plane, the purlin is an I-200. You would like to transfer forces from the purlin to the main girder 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.

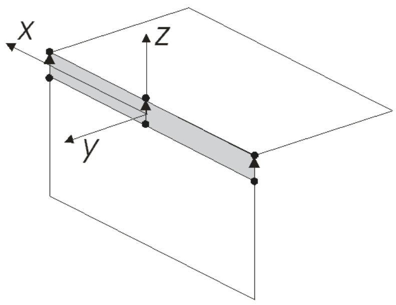
Nonlinear parameters

Nonlinear parameters can be assigned to each nonzero stiffness component. To change the character-istics click one the three buttons (bidirectional, compression only, tension only) and set the resistance checkbox and specify a value if necessary.



Line-to-Line Link

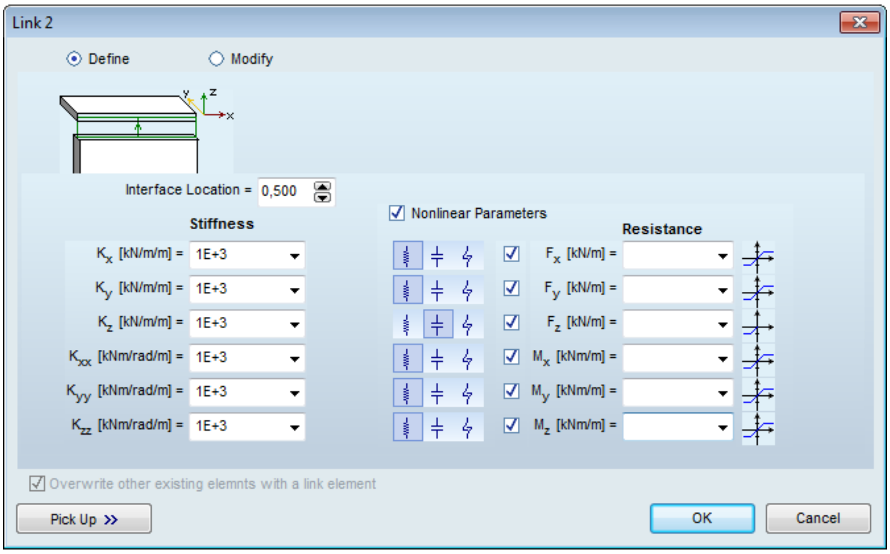
Connects two lines with three nodes each that can be rib elements and/or edges of surface elements. A line-to-line link has 6 nodes. The stiffness compon-ents are defined in the local coordinate system of the link that is in the plane of the link element with the x local axis parallel to the master line, and the local z axis oriented toward the other line in the plane of the link and is orthogonal to the local x axis.



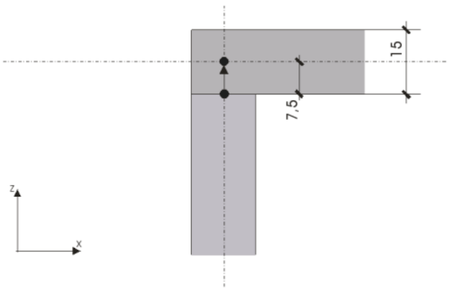
The position of the interface can vary from 0 to 1 relative to the master line (selected by the user).

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.

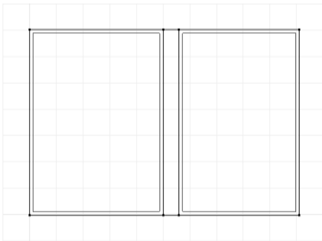


Let's assume that the vertical axis is Z, the wall is in Y-Z plane, the floor is parallel to the X-Y plane and walls are represented by shell elements. Floor thickness is 15 cm.
You would like to transfer forces from the floor to the wall but not the moments.
Elements are represented by their middle 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.

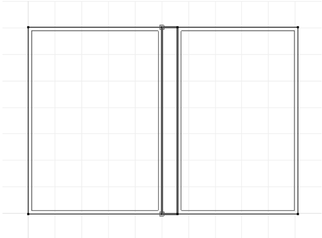
Nonlinear parameters

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:



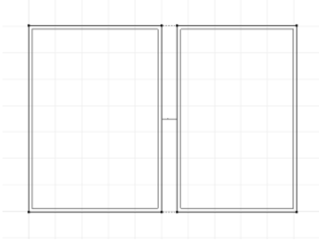
1. Define the domains **(See...** *4.9.5 Domain***)** and connect the cor-responding opposite nodes of the domains with lines (the number of nodes on the edges of the domains should be equal).



2. Select the quadrilateral between the domains. Click **OK** on the Selection Toolbar.

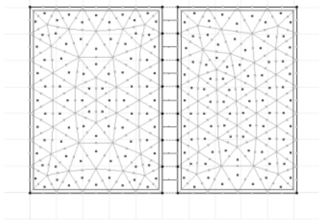
3. Select the master line of the link element. Click **OK** on the Selection Toolbar.

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... *4.11.1.2 Meshing of domain*

6. Link elements are divided according to the domain mesh.



4.9.19. Nodal DOF (degrees of freedom)

Lets you constrain the six nodal degrees of freedom that are: translations (e_X, e_Y, e_Z and rotations (θ_X, θ_Y and θ_Z).

In the default setting no nodes have constrained degrees of freedom.
In the calculations, 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:

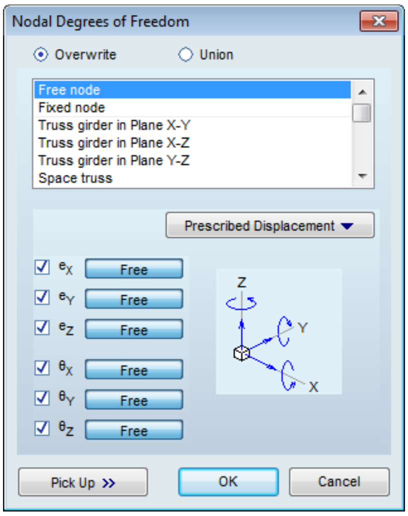
Plane truss girder / Space truss / Plane frame/ Grillage / Membrane / Plate

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
Performs a union set operation with the set of the new degrees of freedom codes and the set of existing degrees of freedom codes of the selected nodes. This option is useful in the definition of symmetry conditions.



Example of union

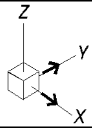
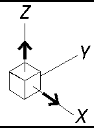
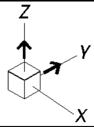
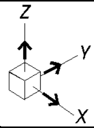
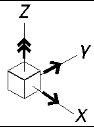
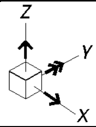
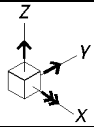
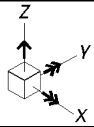
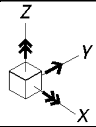
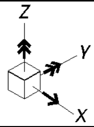
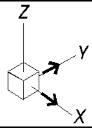
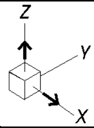
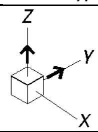
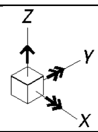
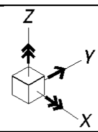
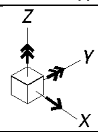
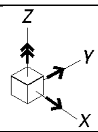
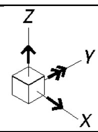
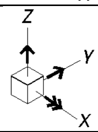
	ex	eY	eZ	ΘX	ΘY	ΘZ
initial value	*free*	*constr.*	*free*	*constr.*	*free*	*constr.*
new value	*free*	*free*	*free*	*constr.*	*constr.*	*constr.*
result	*free*	*constr.*	*free*	*constr.*	*constr.*	*constr.*

The six nodal degrees of freedom $(e_X, e_Y, e_Z, \theta_X, \theta_Y, \theta_Z)$ 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].

☞ ***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.***

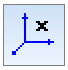
The nodes with DOF different from [f f f f f f] are displayed on the screen in cyan.

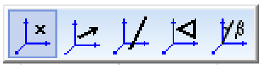
Notations: ↑ free translation, ↟ free rotation about the specified axis.

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			
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.

4.9.20. 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:

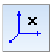
Automatic references

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 centerpoint.

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.

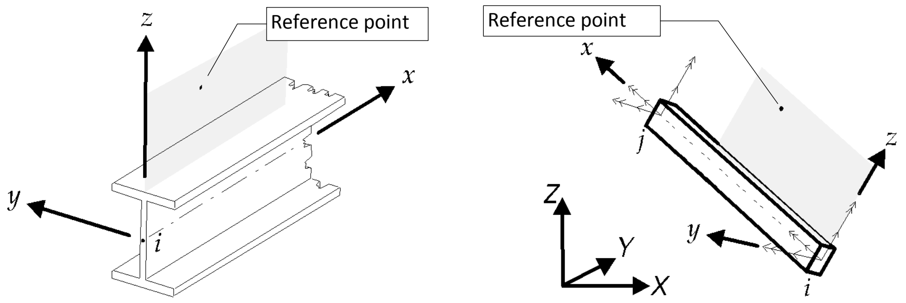
Reference point



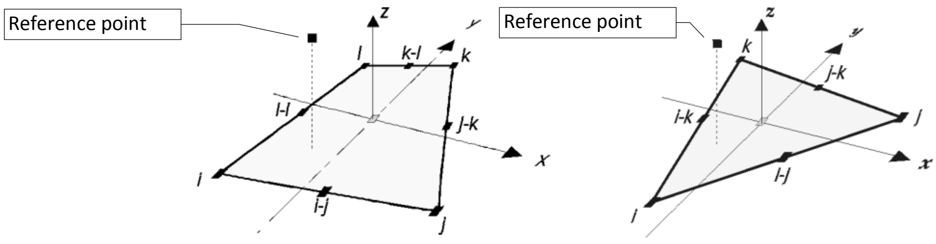
Reference point is used to define the orientation (local coordinate system) of beam, rib, support, and spring elements or to define the positive local x and z axes of surface elements.
The reference points are defined (by its coordinates) in the global coordinate system.

The reference points are displayed on the screen as small red + symbols.

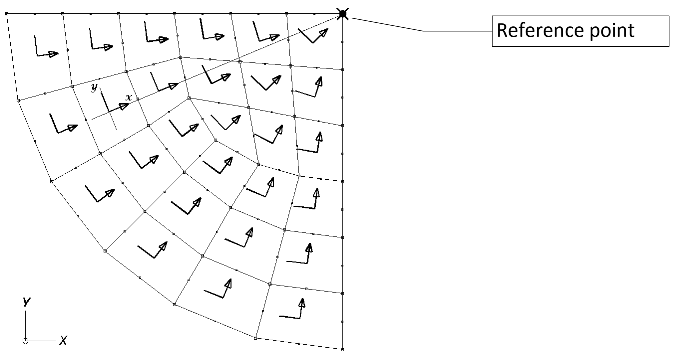
Beams, ribs, and springs:
The reference point and the element's local ***x*** axis defines the local ***x-z*** plane. The positive local ***y*** and ***z*** axis direction is determined by the right-hand rule.



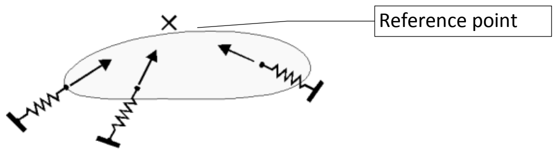
Surface elements:
The positive local *z* axis is oriented toward the half-space in which the reference point is located, and is perpendicular to the element's plane. Once the local *x*-axis is defined local *y*-axis is determined according to the right hand-rule.



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.

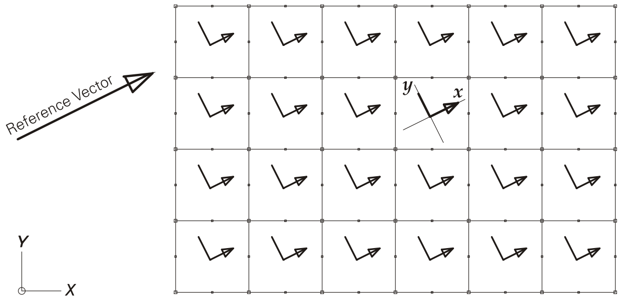


Reference vector

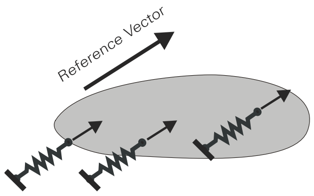
Lets you define the local ***x*** axis for surface, support, and spring elements. Also defines the orientation of local z coordinate axis of beam, rib and spring elements.

The reference vectors are displayed on the screen as red arrows.

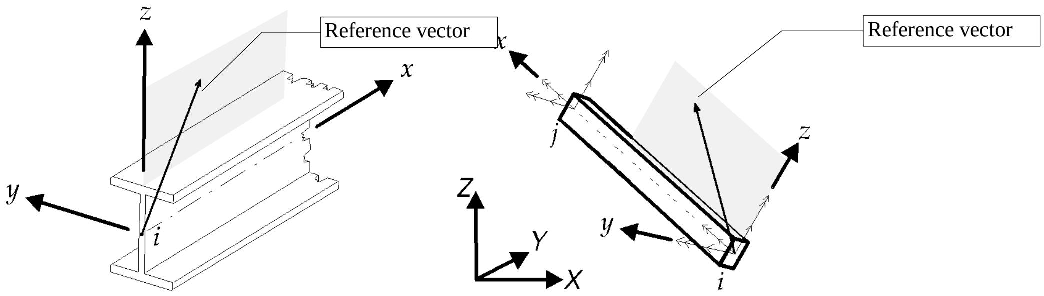
Surfaces:
The local ***x*** axis will be parallel to the reference vector. In the case of a surface element the reference vector must be parallel to the plane of the element.
The orientation of local *z*-axis can also be defined by a reference vector.



Supports:
In the case of a support element you can use a reference vector to define local x axis.

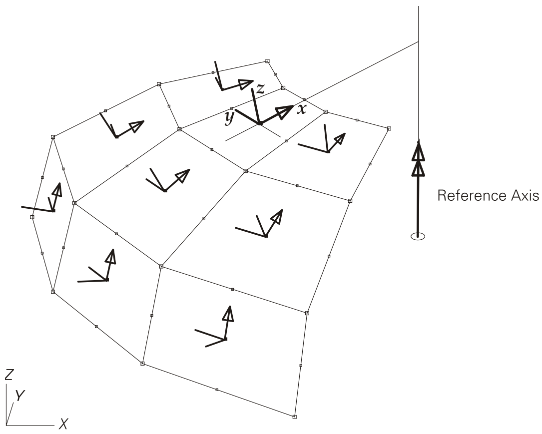


Beams, ribs, and springs:
The reference vector and the element's local ***x*** axis defines the local ***x-z*** plane.
The positive local ***y*** and ***z*** axis direction is determined by the right-hand rule.



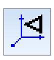
Reference axis

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.

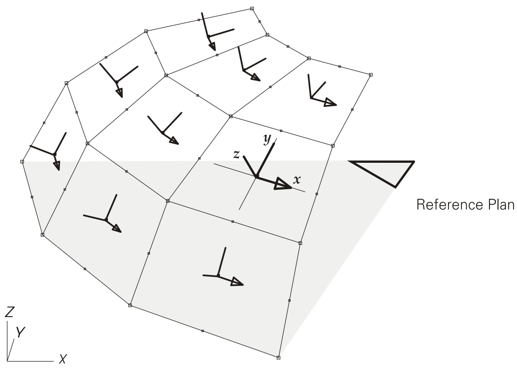


The reference axises are displayed on the screen as red arrows.

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

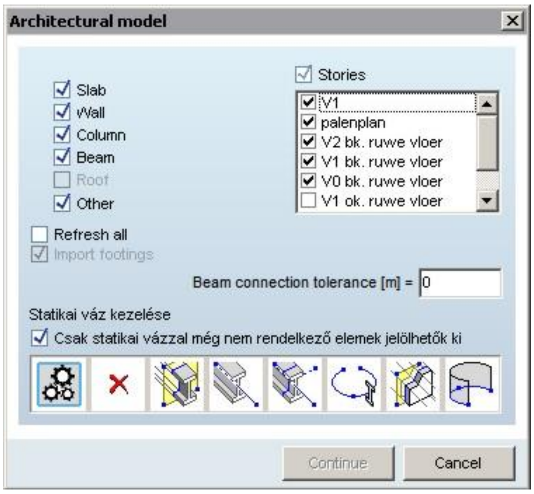
Rotation of truss / beam / rib cross-sections is made easy by the reference angle. The automatic local coordinate system (and the cross-section) can be rotated around the element axis by a custom angle. If the element is parallel to the global Z direction, the angle is relative to the global X axis. 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.***

4.9.21. Creating model framework from an architectural model

This icon starts the conversion operation of the architectural model if previously an IFC file (*.IFC) was loaded by *File / Import* (**See ...** *3.1.6 Import*) as a background layer. **Creating model framework requires IFC module.**



Display Select architectural project stories 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.

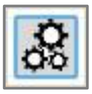
Refresh all Makes the architectural model visible in all windows.

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.

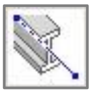
When importing ifc files, it is not always possible to detect the statical frames of objects. Those objects whose statical frame detection failed, are drawn to the screen with a dashed line. User now has many possibilites to create, modify or delete statical model though:



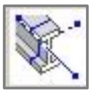
Automated detection of statical frame: in as much as the automated detection method has been successful while importing, the program can use those precalculated data to create statical frame. **More than one item can be selected for this operation.**

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.**

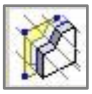


Handling beam-like objects by supplying their middle lines: user must support two points that will form a line. This line then is imprinted to the selected bodies, and the resulting segments will serve as the middle lines of the objects. Taking the middle points of these segments and their direction 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.**

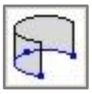


Handling beam-like objects by supplying their middle lines and position of the cross-section: almost the same like the previous operation, except for that here, the position of the cross-section must be given by the user, too. Then the center line of the selected body is modified in a way, that the startpoint is left unchanged, and the endpoint is the mirage of the startpoint 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.**

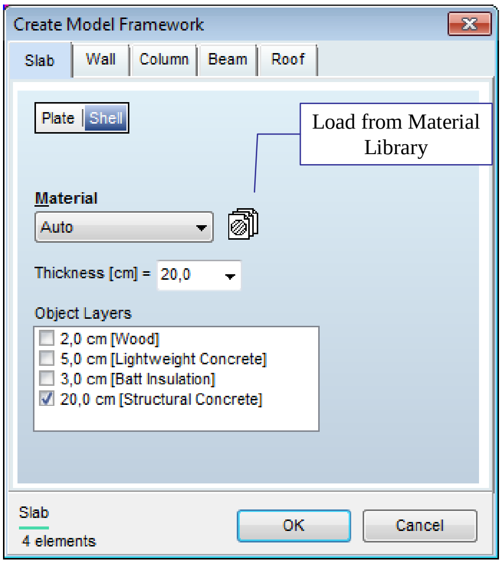
In all editing operations, user is given a real-time visual feedback about the current statical frame, thus it is possible to edit the correct statical model of the object. Sometimes some preediting can be necessarry, though. An optional filter is given to filter out ifc objects that already have statical 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.

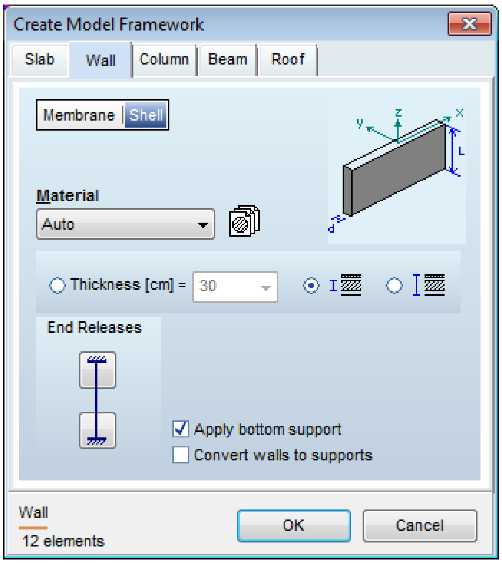
If the *Material* field is set to *Automatic*, and the IFC file includes material data and assigments the model will import them.
You can assign properties to the selected architectural objects as follows:

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



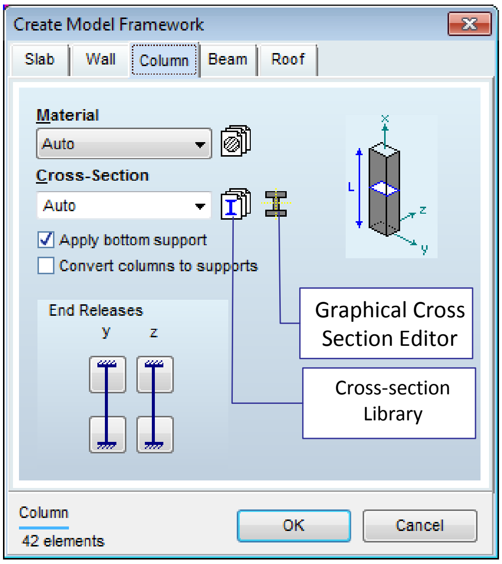
Walls can be defined as membranes or shells. Assign a material and a thickness. For layered walls you can choose to apply the thickness of the load bearing layer, the total thickness or a custom value.

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

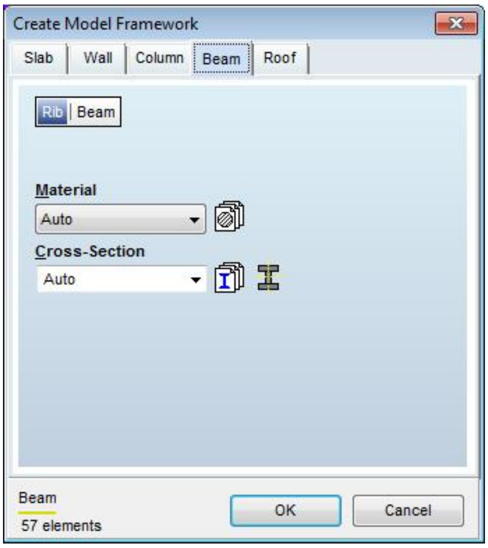


Column objects are always converted to beam elements. Assign a material and a cross-section. If *Auto* is selected the cross-section is created based on the geometrical description of the architectural object.

You can assign a support to the bottom of the column.

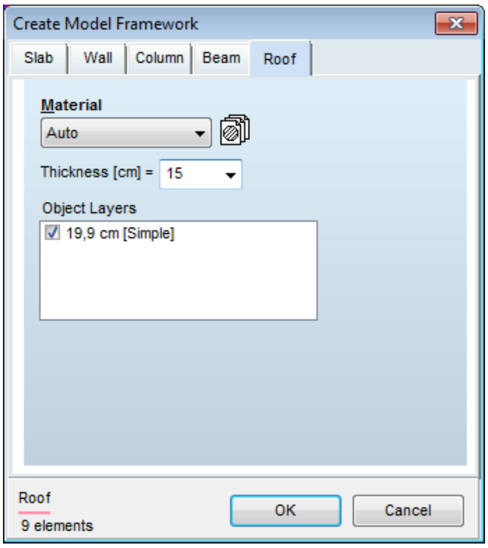
Convert columns to supports: The selected column objects can be converted to supports. Support stiffness is established based on the end releases. Supports will be placed at the top of the column.

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.

4.9.22. 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.

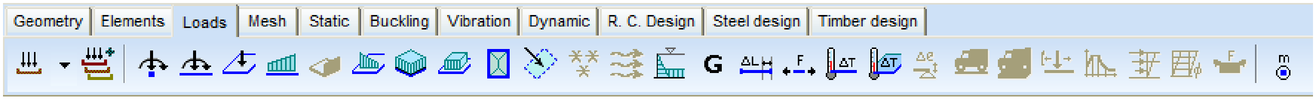
☞ ***In fact, the modification is similar to the element definition, but does not assign properties to undefined geometrical elements and allows access to a specific property without altering others. You can switch to the element definition radio button to define all properties of all the selected elements, 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... *3.5.1 Property Editor*

4.9.23. Delete

[Del] **See...** *3.2.8 Delete*

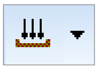
4.10. Loads



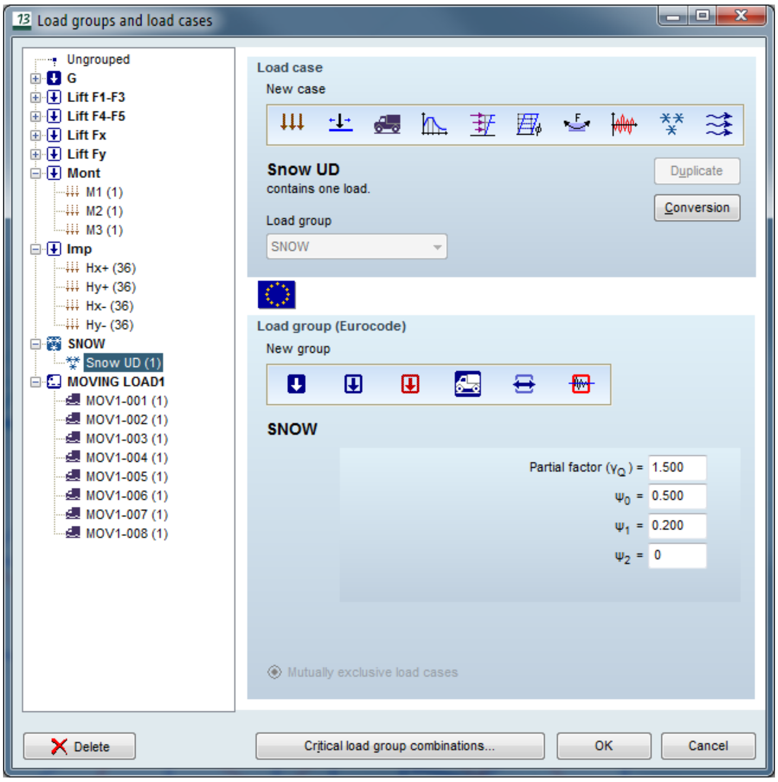
Lets you apply various static loads for static, dynamic and buckling analysis, and define concentrated masses for vibration analysis.

4.10.1. Load cases, load groups

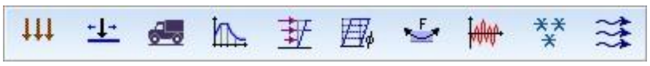
Load Case

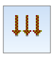


Lets you set the current, create new, and modify or delete existing load cases. Any load you create will be stored in the current load case. In the professional version the number of load cases is not limited. In the standard version a maximum of 99 cases can be created. Load groups can also be created from the different load cases.



New Case You must assign a different name to each case. The following are the possible types of load cases that you can choose from when you want to create a new load case:



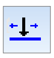


1. Static

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 $i+1, i+2, \ldots, k$.

See details... *4.10.27 Moving loads*

When selecting moving load case the only icon available on the Toolbar will be Moving Load.

4. Seismic

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.

See details... *4.10.23 Seismic loads – SE1 module*

When selecting seismic load case the only icon available on the Toolbar will be Seismic parameters.

5. Pushover

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… *4.10.24 Pushover loads – SE2 module*

When selecting pushover load case the only icon available on the Toolbar will be Pushover parameters.

6. Global imperfection

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 … *4.10.25 Global imperfection*

Imperfection load cases does not contribute to critical combinations of linear analysis results.

7. Tensioning

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…** *4.10.26 Tensioning - PS1 module*

When selecting tensioning load case the only icon available on the Toolbar will be Tensioning.

8. Dynamic load case

Dynamic load cases can be used only if **DYN** module is available. After defining a dynamic load case and selecting it the *Loads* tab will allow definition of dynamic loads and nodal acceleration.
See details…*4.10.28 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

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 counterclockwise).
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 Dialouge 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 the load panel and snow load definition.***

10. Wind load cases

AxisVM can calculate and apply wind loads on the structure. 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.* 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 Dialouge 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. These are the the load panel and wind load definition.***

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)*

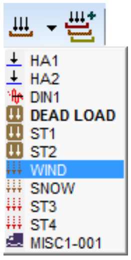
Duplicate Lets you make a copy of the selected load case under another name. You must specify the new name, and a factor that will multiply the loads while copying. The factor can be a negative number as well.

☞ ***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.

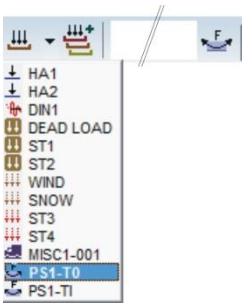
Delete Lets you delete the selected load case.

You can change the current load current case by selecting from the drop down list near the load case icon. Selection can be moved using the up and down arrow keys. This is the best way to overview moving load cases.



☞ ***The name of the selected load case will appear in the Info window and the loads you define will get to this load case.***

In case of choosing Tensioning load case only the *Tensioning* icon will be active on the toolbar.
Click on it then select the proper beam or rib elements, so the *Tensioning* dialog will appear.
See... *4.10.26 Tensioning - PS1 module*



Click right mouse button over the list, select *Order of load cases* to get to a dialog setting the load case order. This dialog is also available in the Table Browser *(Format / Order of load cases).*

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*

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...

Can be simultaneous with exceptional goups

If checked load case(s) from the group can act together with a load case from an exceptional group in critical combinations.

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 .

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)

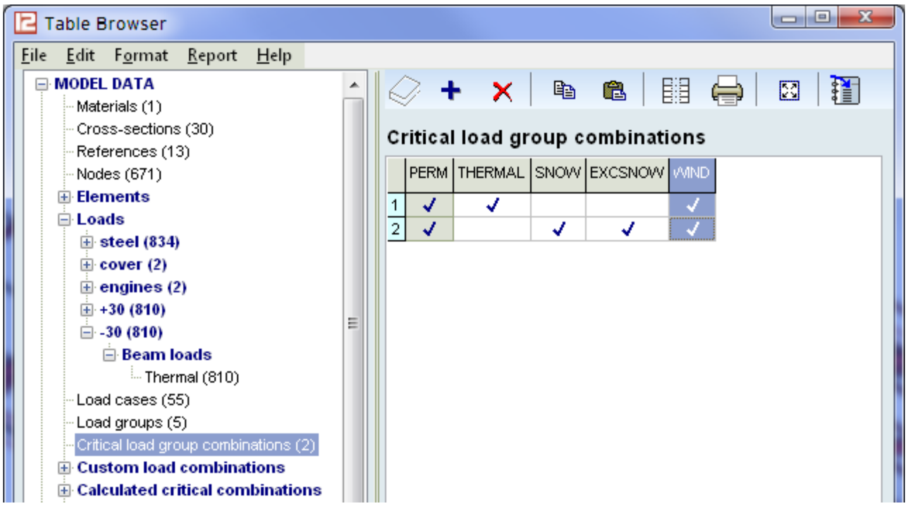
Tensioning load group is handled as a permanent load group. It can contain only tensioning load cases. Both load cases for the same tensioning (name-T0 and name-TI) cannot be included in any load combination.

Seismic load group (Eurocode, SIA 26x, DIN 1045-1, STAS 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 $\alpha = 0$.

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 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.*

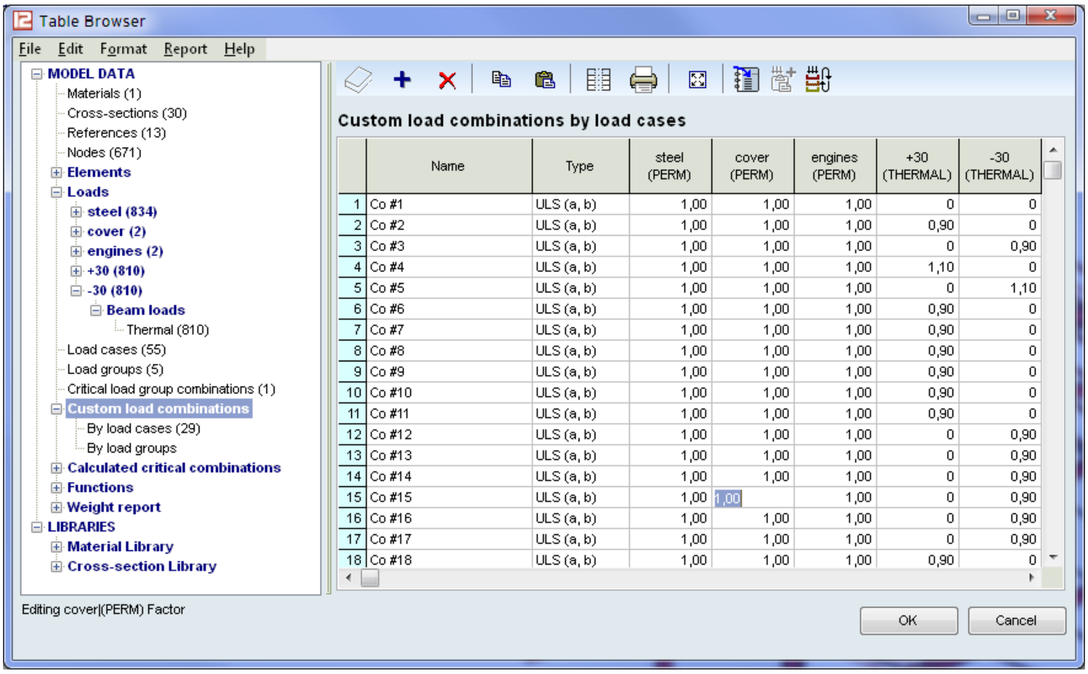


Each line describes a possible combination of load groups. Load cases in checked load groups will be combined in critical combinations.
By default the table contains only one line where all load groups are checked. New lines can be added by clicking the + button and check box states in existing lines can be changed but permanent load groups cannot be turned off.
The program finds the extreme values according to the possible combinations.

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
Snow	beam, rib, plate, shell
Wind	beam, rib, 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.

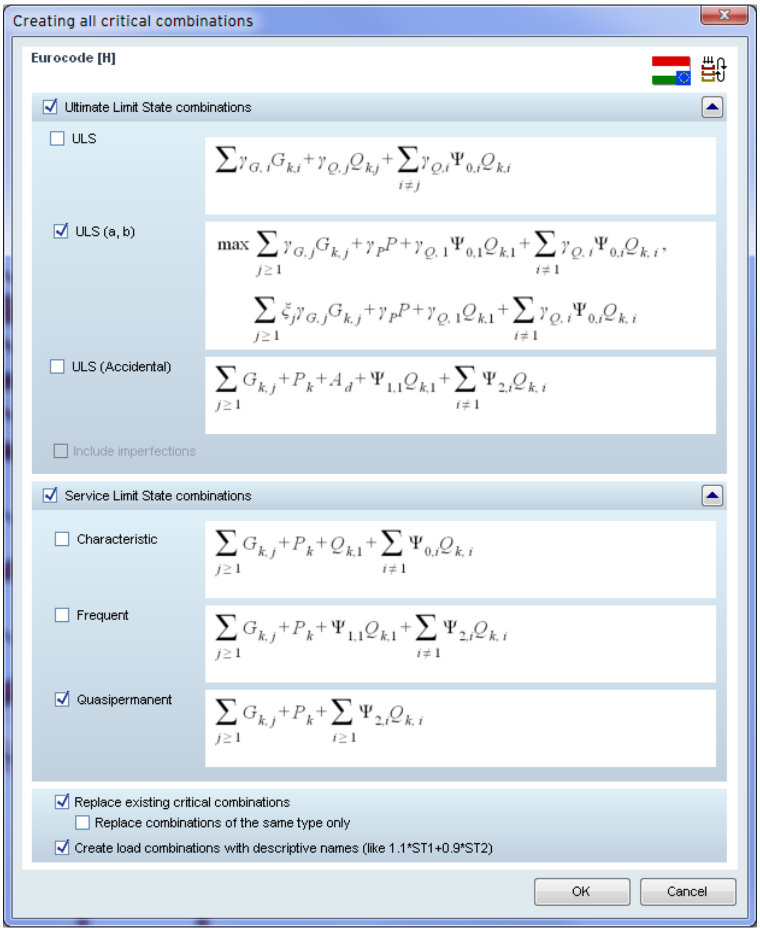
☞ ***It is not necessary to create load combinations to determine the critical combination.***

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 imperfctions* is available only if the model contains an imperfection load group.

If *Replace critical combinations* is checked all previously generated critical combinations will be deleted and replaced with the new combinations. If *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, STAS, DIN, 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 1990:2005 (6.10.a) and (6.10.b). These formulas result in smaller forces and displacements making the design more economical.

$$\max\left\{\begin{array}{l}\sum_{j\geq 1}\gamma_{G,j}G_{k,j}+\gamma_P P+\gamma_{Q,1}\Psi_{0,1}Q_{k,1}+\sum_{i>1}\gamma_{Q,i}\Psi_{0,i}Q_{k,i}\\ \sum_{j\geq 1}\xi_j\gamma_{G,j}G_{k,j}+\gamma_P P+\gamma_{Q,1}Q_{k,1}+\sum_{i>1}\gamma_{Q,i}\Psi_{0,i}Q_{k,i}\end{array}\right\}\begin{array}{l}6.10.a\\ 6.10.b\end{array}$$

In certain countries (e.g. Austria) 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), DIN, SIA26x, Italian code

$$\sum_{j\geq 1}\gamma_{G,j}G_{k,j}+\gamma_P P+\gamma_{Q,1}Q_{k,1}+\sum_{i>1}\gamma_{Q,i}\Psi_{0,i}Q_{k,i}$$

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

$$\sum_{j\geq 1}G_{k,j}+P+E+\sum_{i\geq 1}\Psi_{2,i}Q_{k,i}$$

DIN

$$\sum_{j\geq 1}G_{k,j}+A_{Ed}+\sum_{i\geq 1}\Psi_{2,i}Q_{k,i}$$

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

$$\sum_{j\geq 1}G_{k,j}+P_k+\sum_{i\geq 1}\Psi_{2,i}Q_{k,i}$$

STAS, 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.

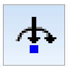
Italian code

Combination of seismic loads with other load types: $\sum G_K + \gamma_I \cdot E + \sum \Psi_{ji} Q_{Ki}$

Where:

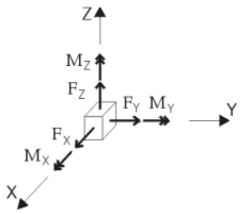
γ_I	importance factor
E	seismic load
G_K	characteristic value of permanent loads
Q_{Ki}	characteristic value of variable loads
Ψ_{ji}	Ψ_{2i}(ULS) combination factor for quasipermanent Q_i
Ψ_{0i}	(DLS) combination factor for rare Q_i

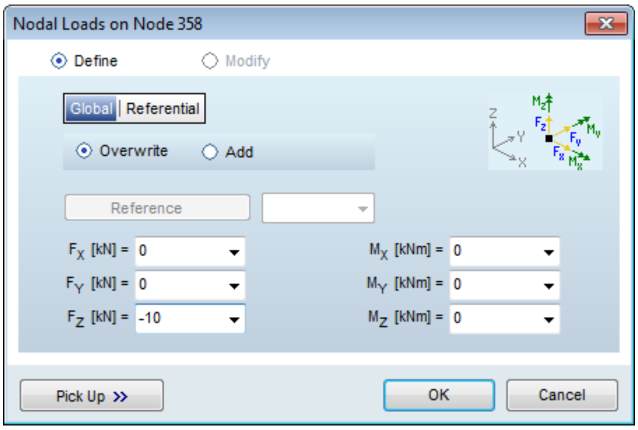
4.10.3. Nodal loads



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

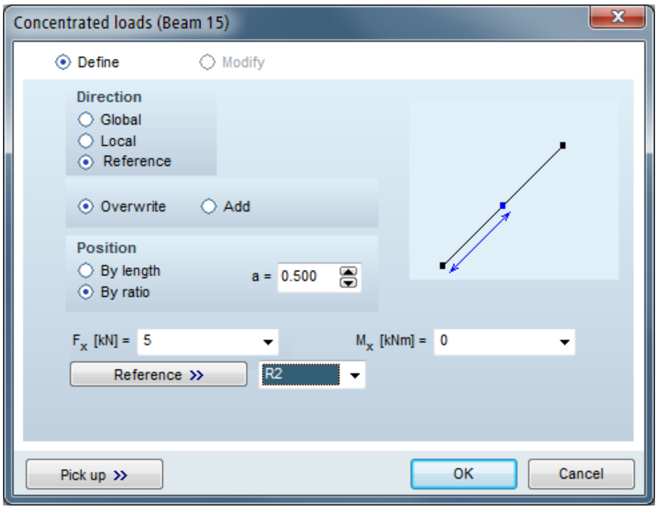
1. Select the load.
2. Click the *Nodal Load* icon on the toolbar.
3. Change the values

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



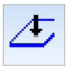
Lets you apply concentrated forces / moments to the selected beam or rib elements. You must specify the values of the load components *(Fx, Fy, Fz, Mx, My, Mz)* in the local or global coordinate system. For a referential load (where the direction is given by the reference) enter an *Fx* and an *Mx* component.
The reference can be selected from the list or picked up from the view (if display of references is activated, **see** *2.16.18 Display options*) clicking on the *Reference>>* button.
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.

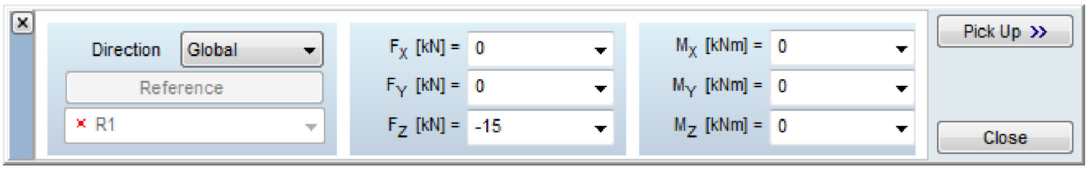
4.10.5. Point load on domain or load panel

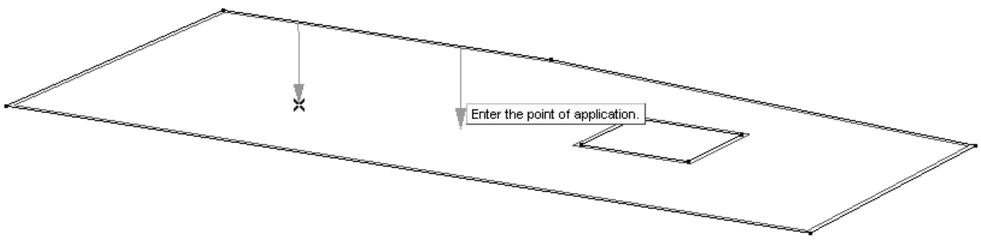


Applies a point (concentrated) load at the location of the cursor if it is over a domain or a load panel. You can also enter the location of the load by its coordinates. Place loads by clicking the left mouse button or pressing any of the command buttons.
See... *4.7.2 Entering coordinates numerically*

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.

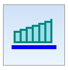
Modify value

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.***

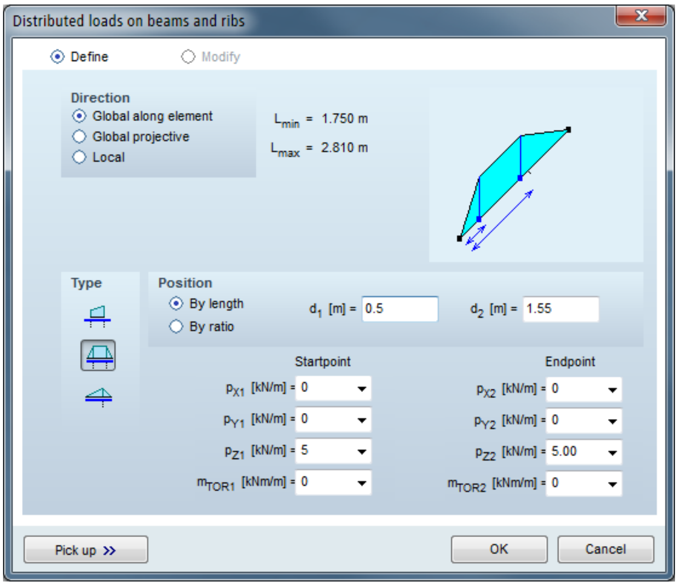
4.10.6. Distributed line load on beam/rib



Lets you apply constant or linearly distributed forces and torque to the selected beam / rib elements. You can apply multiple distributed loads to a beam/rib in the same load case.

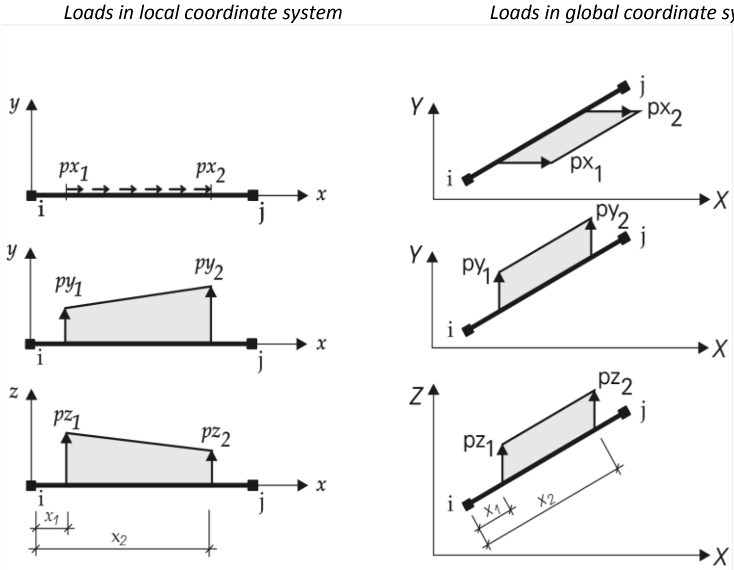
Line loads can be selected, moved, copied, modified independently of the beam or rib. Modify load values like in case of nodal loads.

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:

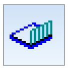
Type	 *segment*  *trapezoid*  *triangle*
Position	*By length:* between d_1 and d_2 [m], *By ratio:* between a_1 and a_2
Starting location	d_1[m] or a_1 in the local coordinate system
Starting value	p_{x1}, p_{y1}, p_{z1} [kN/m] forces, m_{TOR1} [kNm/m] torque
End location	d_2[m] or a_2 in the local coordinate 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

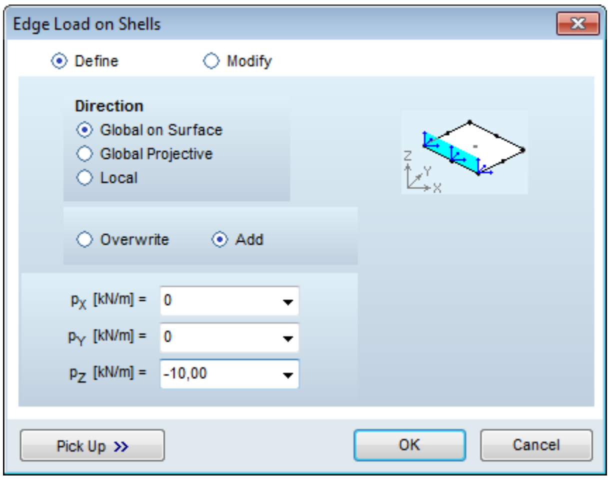
If the load is projective, the value of the load that is applied to the beam/rib is p*sin α, where α is the angle of the load direction and the beam/rib axis.

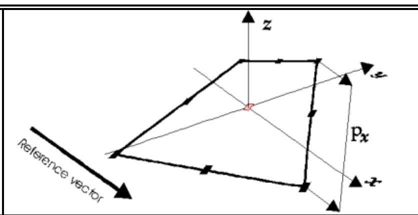
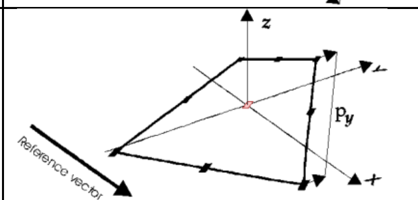
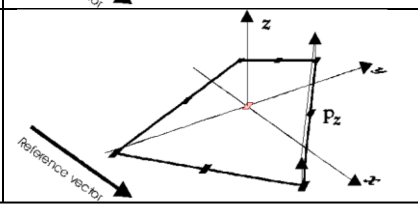
☞ ***For rib elements you can apply line loads distributing along the entire length of the rib only.***

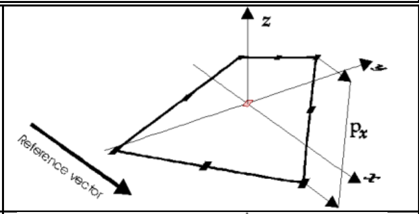
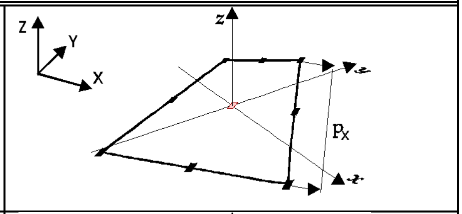
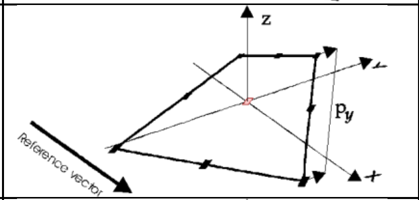
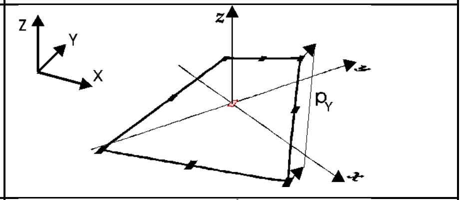
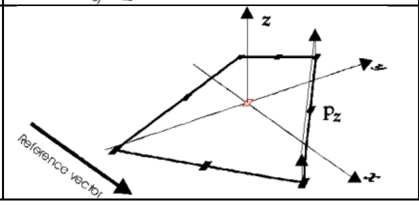
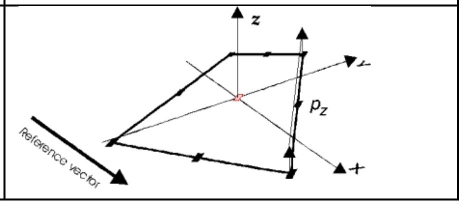
4.10.7. Edge load



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		-	-

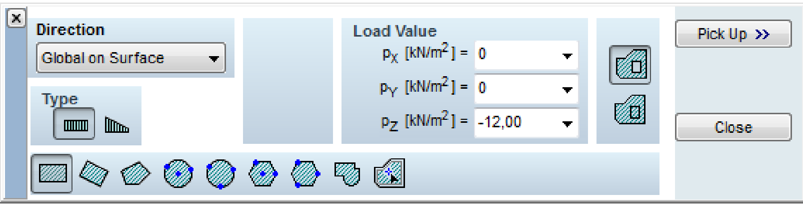
Element		Load in local directions (in local coordinate system)		Load in global directions (in global coordinate system)
Shell	x		X	
	y		Y	
	z		Z	

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.

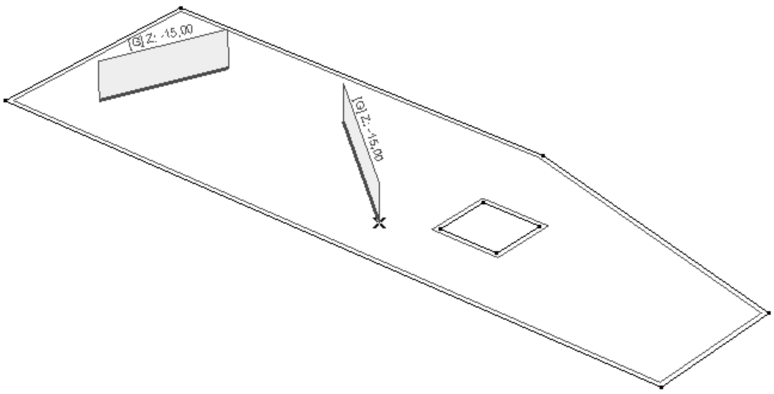
4.10.8. Domain / Load panel line load

Applies a uniform or linear distributed line load over a domain or a load panel.

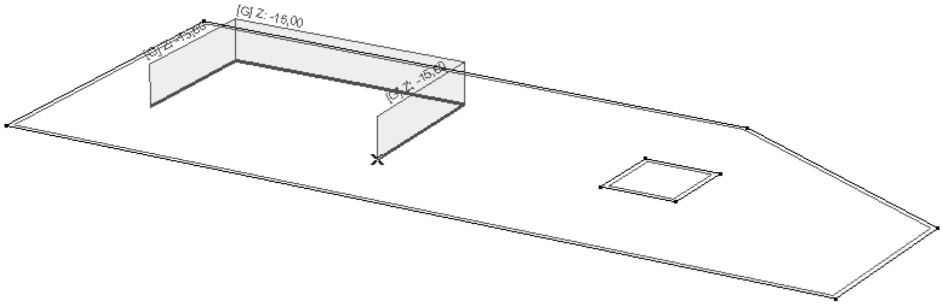
The direction of the load can be global projective, global along element, edge relative or surface relative. The m_x is always the torsional moment (around the application line of the load). Set load components and placement method then draw the load (or click the lines) to place it.



Line load between two points



Line load along a polyline



Line load along a rectangle

Line load along a rotated rectangle

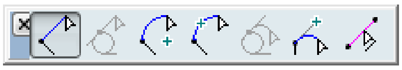
Line load along an arc defined by its centerpoint and two points

Line load along an arc defined by three points

Line load along an arc polygon defined by its centerpoint 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 centerpoint, drawing an 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

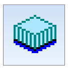
1. Select the load with the cursor (a load symbol appears beside the cursor).
2. Click the left mouse button.
3. Enter new load values in the dialogue window.
4. Click on the Modify button to apply the changes and close the window.

The load value can also be changed in the Table Browser.

Delete Select the loads you want to delete and press **Delete**.

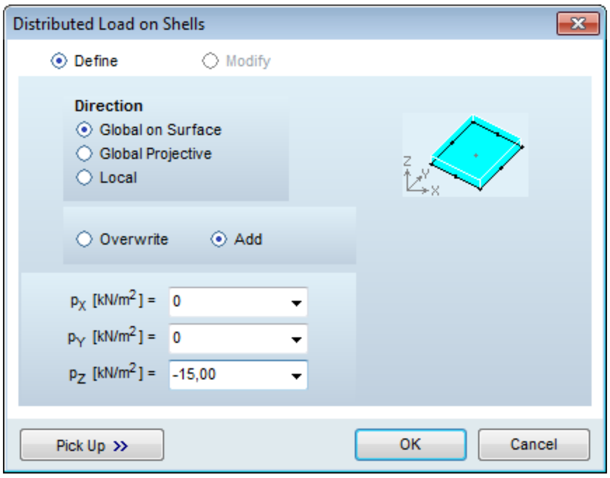
☞ ***Modifying domain mesh leaves line loads (applied on the domains) unchanged.***

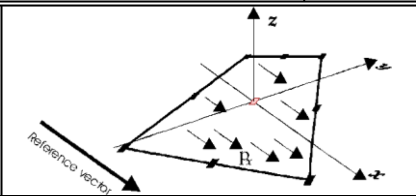
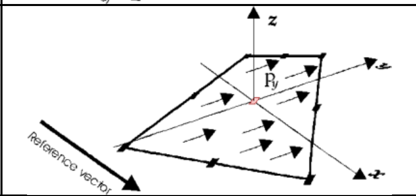
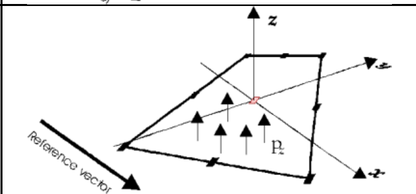
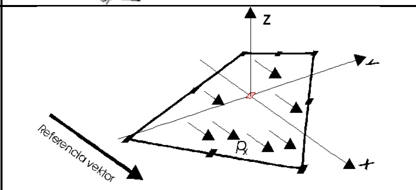
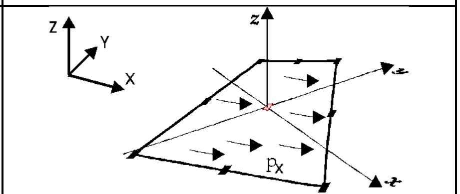
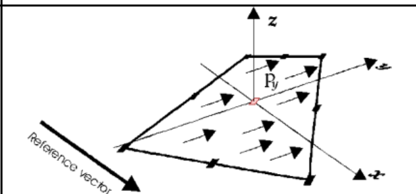
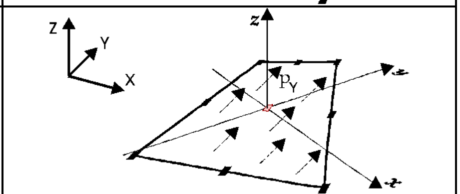
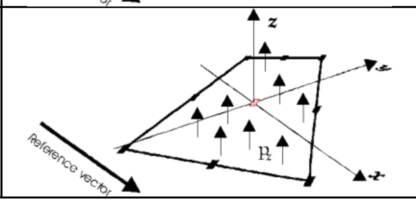
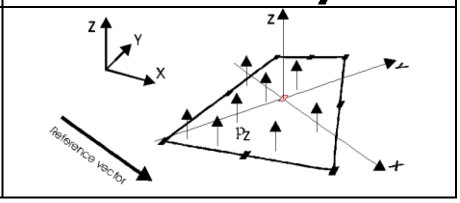
4.10.9. Surface load



Surface load can be applied on surface finite elements, domains and load panels.
The intensity of a distributed load on a surface element is constant.

Modifying domain mesh leaves the loads (applied on the domains) unchanged.

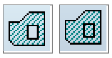


Element	*Load in local directions (in local coordinate system)*		*Load in global directions (in global coordinate system)*	
Membrane	x		-	-
	y		-	-
Plate	z		-	-
Shell	x		X	
	y		Y	
	z		Z	

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.

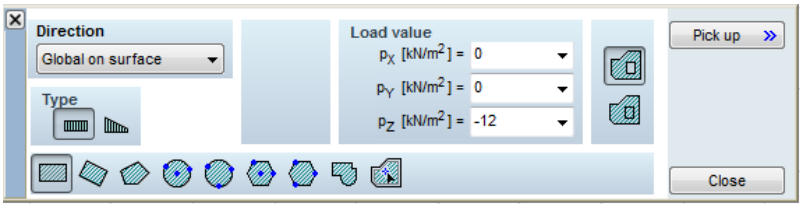


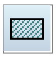
Loads disappear/ allowed on holes

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.

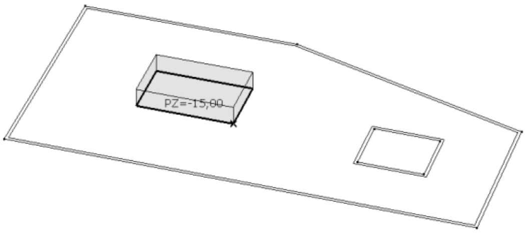
Constant load

Steps of load definition in case of constant load:



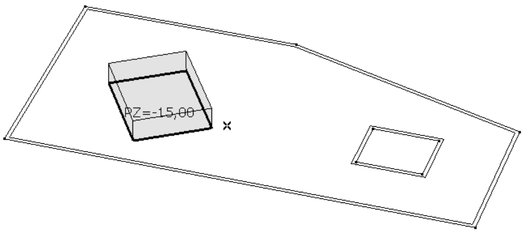


Rectangular area load



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)

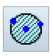
Skewed rectangle area load



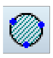
1. Enter load components (p_X, p_Y, p_Z)
2. Enter three corners of the rectangle by clicking or by coordinates.
(This function is available only on the X-Y, Y-Z and X-Z planes)

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

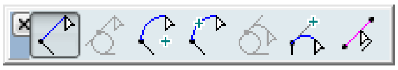


Disk or sector load defined by three points

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 centerpoint, drawing an 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.

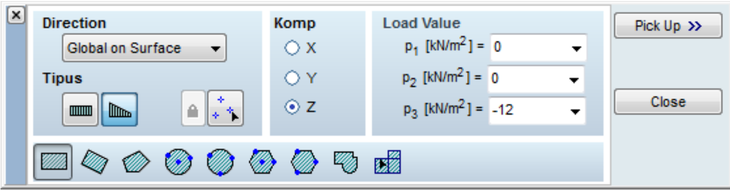
Distributed domain load

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

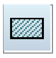
Steps of load definition in case of linear load:



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.

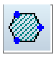
Polygon load

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 centerpont and two points

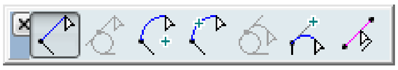
Sector or disc shaped load defined by three points

Arc polygon shaped load defined by centerpont 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 centerpoint, drawing an 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.

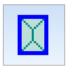
Area load intensity and shape can also be changed in the Table Browser by changing the appropriate values in the load table.

Delete

Select the loads to delete and press **[Del]**

☞ ***Mesh-independent loads are not affected by removing meshes or re-creating meshes on domains.***

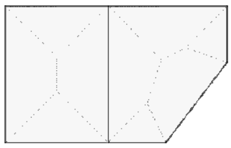
4.10.11. Surface load distributed over line elements



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.

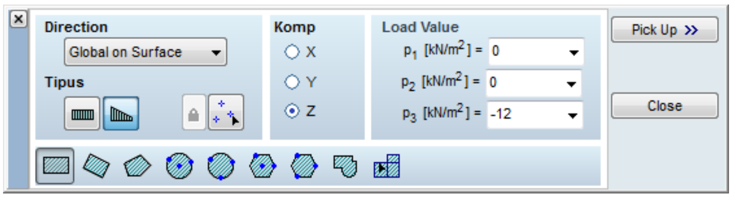
1. Click the icon and select the load distribution range in the dialog.

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.

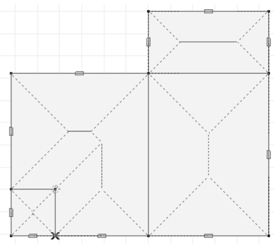
2. Define load polygon the same way as for a constant or linearly changing domain area load.



Load direction can be global on surface, global projective or local. Local directions are defined like automatic references for domains **See...** *4.9.20 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 individal beam loads.

4.10.12. Load panels

In order to define snow and wind loads load panels must be defined over the structure. Load panels are load bearing surfaces used to apply snow and wind loads. The only function of a load panel is to distribute the loads over the domains, beam and rib elements under the panel. It is possible to select load bearing rib / beam elements or domains controlling the load transfer.
To define load panels you have to draw shapes over elements or select domains or select outlines of several planar regions. Choose a function by selecting a tool from the toolbar.

Shape tools

 *Rectangular*

 *Slanted rectangle*

 *Sector or full circle by centerpoint and two points*

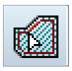
 *Sector or full circle by three points*

 *Sector or regular polygon by centerpoint and two points*

 *Sector or regular polygon by three points*

 *Complex polygon*

Special selection tools

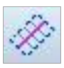
 Clicking on a domain creates a load panel over the domain

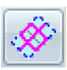
 Creating load panels for the selected domains.

 This function creates load panels based on selected lines. Load panel outlines are determined from the outlines of coplanar subsets of selected lines.

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.

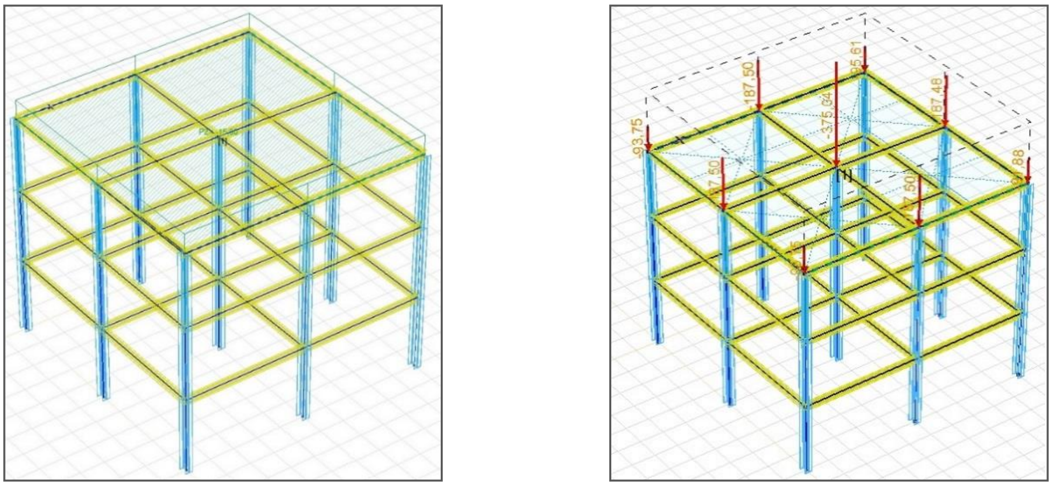
 Load is distributed to **selected domains and selected ribs and beams.**

Select this option to distribute the load on **domains, ribs and beams in active parts** under the load panel.

Loads from the selected load panels can be converted to individual loads. **See…** *3.2.18 Convert loads of the selected load panels to individual loads.*

☞ ***Selected beams / ribs / domains can be out of the plane of the load panel.***

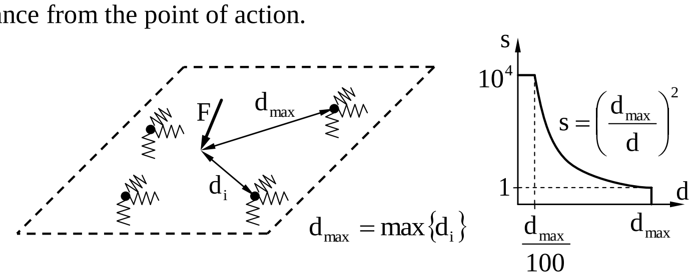
If multiple domains are selected and their outlines projected to the plane of the load panel overlap, loads will be transferred only to the domain closest to the load panel.



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.*

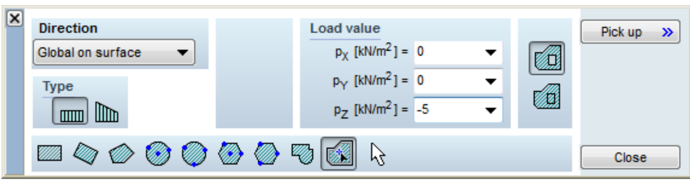
Defining loads

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 *Distrubited 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 standards in the program for which snow load generation is available are listed below. The national design standards that served as the basis of calculation for the given standard in the program are also listed.

Eurocode general	*EN 1991-1-3:2003 (EC 1-3)* *Eurocode 1 Action on Structures* *Part 1-3 General Actions – Snow loads*
EC German	*EN 1991-1-3:2003* *DIN EN 1991-1-3/NA December 2010*
NTC Italian	*EN 1991-1-3:2003* *UNI EN 1991-1-3/NA July 2007*
EC Dutch	*EN 1991-1-3:2003* *NEN EN 1991-1-3/NB November 2007*
EC Hungarian	*EN 1991-1-3:2003* *MSZ EN 1991-1-3:2016/NA September 2016*
EC Romanian	*CR 1-1-3/2012* *conform with SR EN 1991-1-3*
EC Czech	*EN 1991-1-3:2003* *CSN EN 1991-1-3/NA July 2012*
EC Slovakian	*EN 1991-1-3:2003* *STN EN 1991-1-3/NA December 2004*
EC Polish	*EN 1991-1-3:2003* *PN EN 1991-1-3/NA September 2005*
EC Danish	*EN 1991-1-3:2003* *DS/EN 1991-1-3 DK NA October 2015*
EC Austrian	*EN 1991-1-3:2003* *ÖNORM B 1991-1-3 September 2013*
Swiss	*SIA261:2003 Einwirkungen auf Tragwerke* *SIA261/1:2003 Ergänzende Festlegungen*

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 not considered.
- Snow load on snowguards is not considered.

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 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 algorithm provides reliable results only for altitudes below 1500 m. For higher altitudes the snow load shall be calculated as per 5.1 in EC 1-3 NA.
- 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.
- The total area of the roof is assumed to be less than 2000 m^2. For larger roof areas see 9.2.1.2 in EC 1-3 NA and apply an appropriate increase in s_k for the analysis.
- 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.

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.

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 C_e 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 μ_1 in Table 5.2 in EC 1-3. Each panel has its own μ_1 value that is calculated using the slope of the panel.
- Snow load shape coefficients for the drifted load case are based on μ_2 in Table 5.2. μ_2 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.

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 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 ont he 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 on the ground is calculated automatically based on Eq. (C.1) in EC 1-3 NA using the zone and altitude specified by the user.

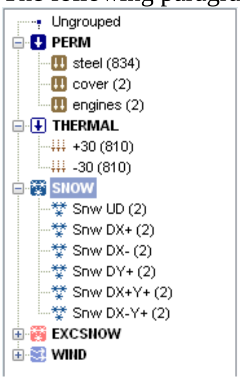
- Snow load shape coefficients for multi-span roofs are calculated as per 9.2.3 in EC 1-3 NA.
- Snow load shape coefficients for vaulted roofs are calculated as per 9.2.4.3 in EC 1-3 NA.
- The modifications for calculating the effect of taller construction works in 9.2.5.1 in EC 1-3 NA are taken into account.

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 C_e 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.
- μ_2 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 and 6.2 in EC 1-3 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.
- 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.*

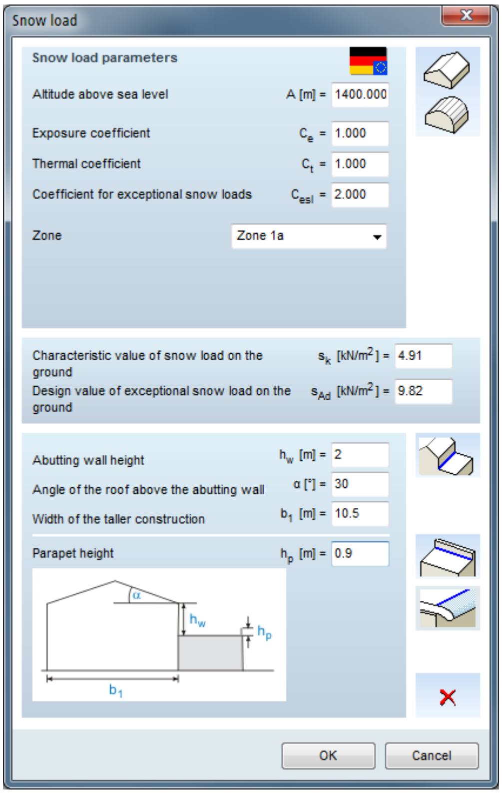
To set the snow load parameters select one of the snow load cases as the current load case. It makes the snow load icon enabled in the *Loads* tab.

If no load panels have been created before, draw the load panels according to *4.10.12 Load panels*.

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 characteristic snow load on the ground depends on the climatic region and the altitude of the site. Higher altitude generally corresponds to higher load intensity. The program calculates the snow load intensity from the parameters.

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

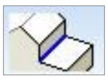
Coefficient for exceptional snow loads C_{esl}
In countries where the design code requires checking exceptional snow loads the exceptional load intensity is calculated by multiplying the normal intensity by C_{esl}. Custom values can also be entered. In this case the program asks for confirmation then uses the custom value.

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.

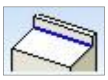
Importance factor
An importance factor can be entered depending on the classification of the building if it is required by the design code. Nonstandard values can also be used after confirmation.

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.

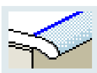
AxisVM calculates snow load shape coefficients for roofs abutting and close to taller construction works or having a parapet which acts as an obstruction. It can take into account the effect of overhanging snow on roof edges.
Parameters are stored with the edges so different roof edges can have different parameters.



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.

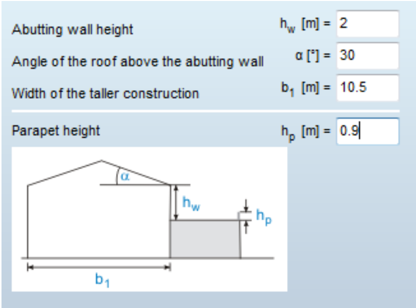


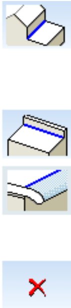
Select parapet edges
To define parapets enter the h_p parameter then click on the second icon to select the lines.



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.



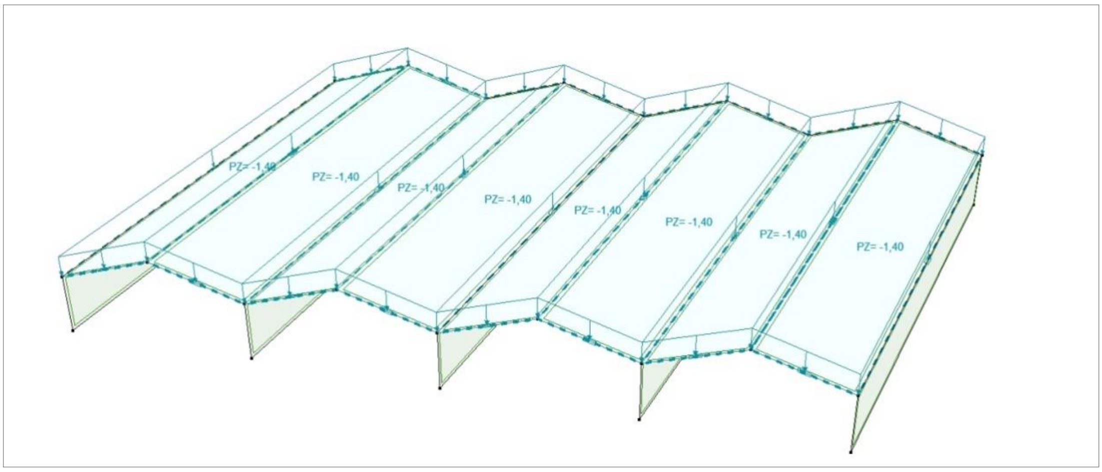


Abutting wall height [m]
h_w height of the abutting wall relative to the roof level

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

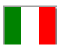
Parapet height
h_p is the height of the parapet or other obstruction relative to the roof level .



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 standards in the program for which wind load generation is available are listed below. The national design standards that served as the basis of calculation for the given standard in the program are also listed.

	Eurocode general	*EN 1991-1-4:2005 (EC 1-4)* *Eurocode 1 Action on Structures* *Part 1-4 General Actions – Wind Actions*
	EC German	*EC 1991-1-4:2005* *DIN EN 1991-1-4/NA December 2010*
	NTC Italian	*EC 1991-1-4:2005* *UNI EN 1991-1-4/NA July 2007*
	EC Dutch	*EC 1991-1-4:2005* *NEN EN 1991-1-4/NB November 2007*
	EC Hungarian	*EC 1991-1-4:2005* *MSZ EN 1991-1-4/NA January 2016*
	EC Romanian	*CR 1-1-4/2012* *conform with SR EN 1991-1-4*
	EC Czech	*EC 1991-1-4:2005* *CSN EN 1991-1-4/NA July 2013*
	EC Slovakian	*EC 1991-1-4:2005* *CSN EN 1991-1-4/NA July 2013*

EC Belgian — *EC 1991-1-4:2005*
NBN EN 1991-1-4/ANB December 2010

EC Polish — *EC 1991-1-4:2005*
PN EN 1991-1-4/NA October 2008

EC Danish — *EC 1991-1-4:2005*
DS/EN 1991-1-4 DK NA July 2015

EC Austrian — *EN 1991-1-4:2005*
ÖNORM B 1991-1-4 May 2013

Swiss — *SIA261:2003 Einwirkungen auf Tragwerke*
SIA261/1:2003 Ergänzende Festlegungen

Assumptions
Application limtis

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 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
- roofs of the following types are covered: flat, monopitch, duopitch, hipped, and vaulted
- building height < 200 m
- building span < 200m
- 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 influence of neighboring structures and obstacles is assumed negligible
- the building is assumed not to have a dominant face
- the structural factor c_sc_d 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

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 c_o, but the calculation of c_o 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 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 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 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. Should the user decide not to specify μ, two different cases are considered with c_{pi} = +0.2 and c_{pi} = -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.

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.

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_l 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 as per Table NA.3 in EC 1-4 NA.

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 cdir and cseason 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 1 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.

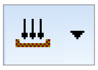
EC Austrian

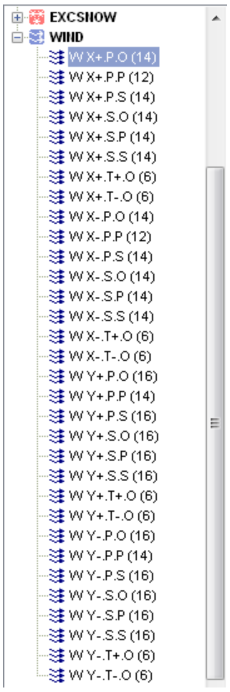
- Pressure coefficients for walls are calculated as per Table 3a, 3b and 4 in EC 1-4 NA.
- Zones F and G are not used for roofs when their cumulative area is less than 20% of the total roof area as per 9.2 in EC 1-4 NA.

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.
- The c_{red} reduction factor is assumed 1.0 in all cases.
- The c_d dynamic factor is assumed 1.0 in all cases. The corresponding limitations are listed in 6.3.5 in SIA 261.

Usage The following paragraphs explain the usage of the automatic wind load generator module.



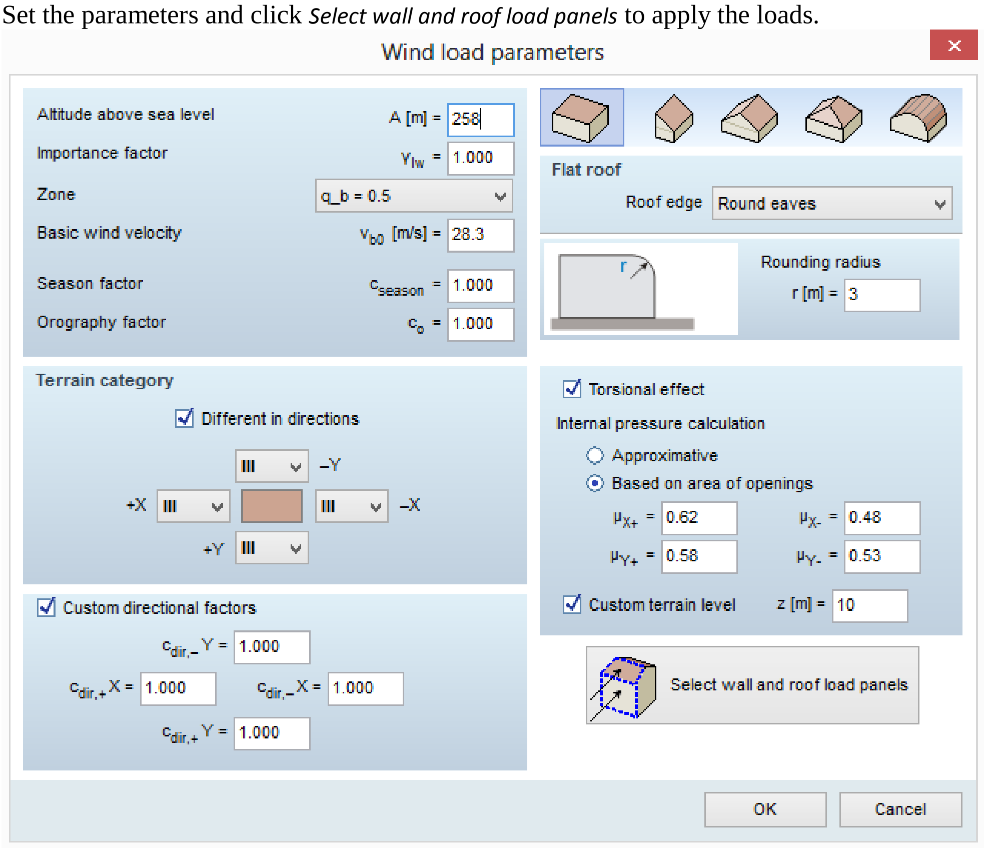


To apply wind loads according to Eurocode, first click on the Load cases / load groups button and define a wind load case by clicking on the wind load case button. A wind load group will be created automatically. As a first step a temporary wind load case is created in the wind load group and its name 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

Set the parameters and click *Select wall and roof load panels* to apply the loads.



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.

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.

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

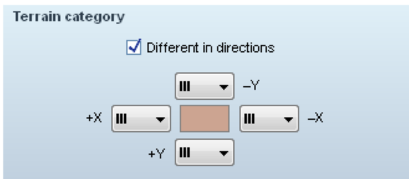
I Lakes or flat and horizontal area with negligible vegetation and without obstacles

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.

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*  *Monopitch*  *Duopitch*  *Hipped*  *Barrel*

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
Two options are available to determine internal pressure. The first one is the approximate method. It applies the critical pressure and suction loads recommended by Eurocode in separate load cases. The second option requires entering the μ factor based on the distribution of openings in different directions and calculates the internal pressure accordingly. Where μ=0 the program uses the approximate method in that direction.

Custom 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 allowsthe user to perform advanced analyses such as consideration of wind load for only the roof of a building.

Select wall and roof load panels
After clicking on this button select load panels representing walls and roofs of the structure. Load panels are automatically identified as walls or roofs by their geometry.

After clicking the OK button, the wind loads are generated automatically for the selected load panels.

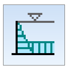
Wind load data

After the wind loads have been generated all the data related to their calculation is available in the Table Browser under *Loads/ Wind load parameters* and *Wind load case parameters.*

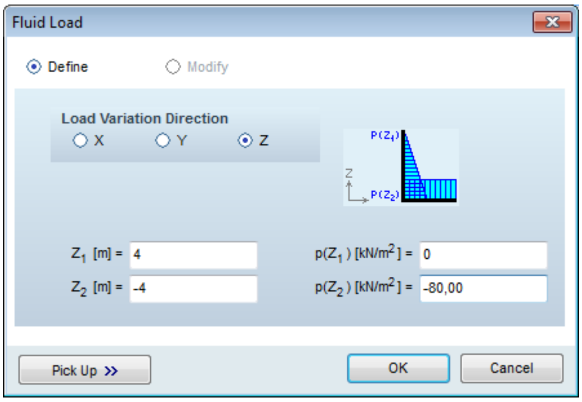
Wind load parameters summarizes the values that are not load case specific. Default wind load settings lead to direction independent wind loading, thus identical parameters in all four directions. Should the user specify direction dependent terrain categories and/or custom direction factors, these values became 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 type 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.

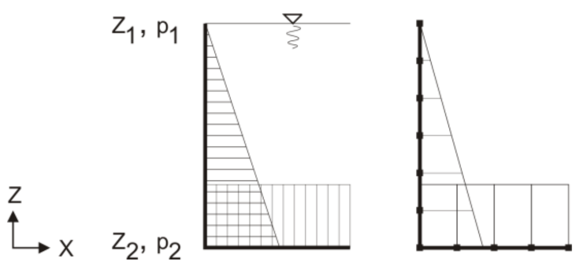
4.10.15. Fluid load



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.
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.



To change a fluid load only on certain elements use partial selection, i.e. draw a selection frame around the elements.

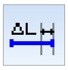


4.10.16. Self-weight

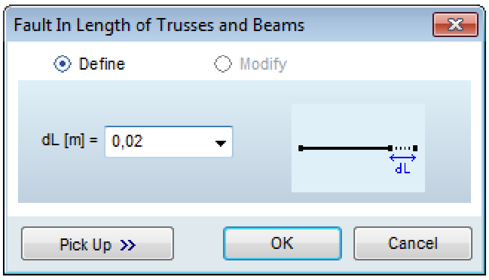
Lets you take the self-weight of the elements (that have materials assigned) and domains into account in the analysis. Self-weight is computed based on the cross-sectional, the mass density of the material, the gravitational acceleration g, and the length or area of the element. The load is applied as 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.

4.10.17. Fault in length (fabrication error)

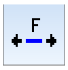


This load type is used when a structural beam element is shorter or longer than required due to a fault in manufacturing.
Lets you apply the load, which is required to force the shorter/longer beams to fit the distance of the corresponding nodes, to the selected elements. You must specify the value of the manufacturing fault, *dL [m]*. A positive *dL* means that the beam is longer by *dL*.

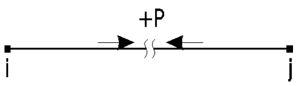


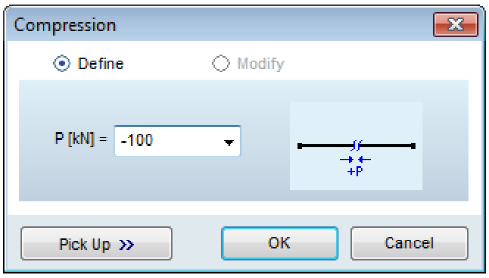
The load has the same effect as the $dT^{=} = dL/(\alpha \cdot L)$ thermal load.

4.10.18. Tension/compression

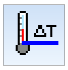


Lets you define an initial axial internal force in truss/beam elements. The load has the same effect as a $dT^{=} = -P/(\alpha \cdot E \cdot A)$ thermal load.

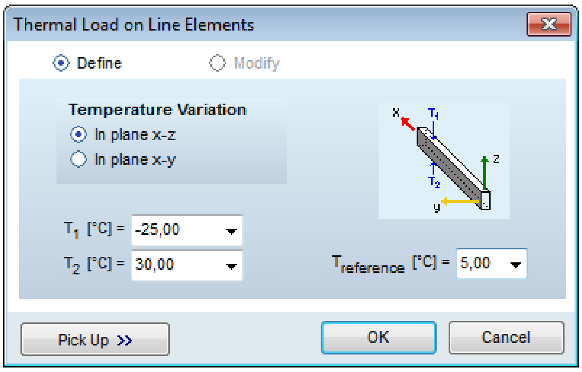




4.10.19. Thermal load on line elements



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.

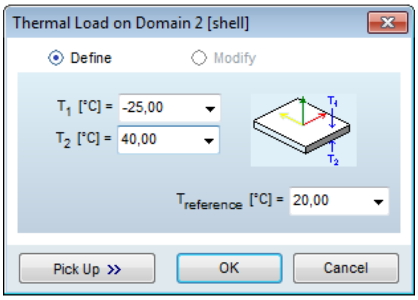
4.10.20. Thermal load on surface elements

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)

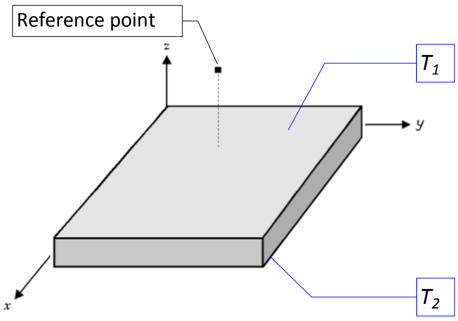
T_1 the temperature of the top cord (in the positive local z direction)

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.***

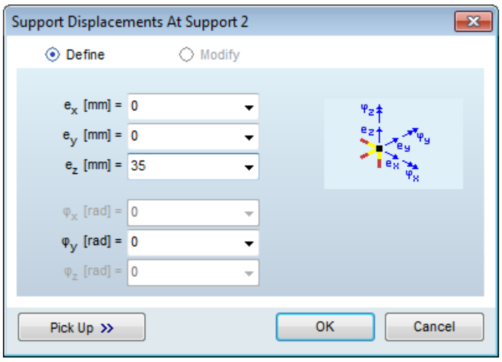
4.10.21. Forced support displacement

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]*).

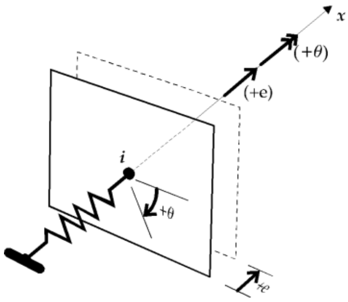
AxisVM approximates the problem, by applying a force $P_{support}$ in the direction of the support element so as to produce the forced displacement e.

$P_{support} = K_{support} \cdot e$,

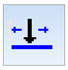
where $K_{support}$ is the corresponding support stiffness.



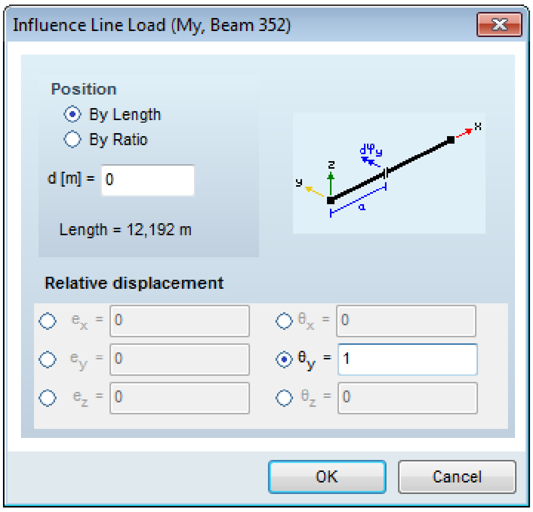
If the stiffness of the support element is large enough, the secondary deflections due to other loads will be negligible. Therefore, you may apply forced displacements only to the supports stiff enough relative to the stiffness of the structure (at least 10^3 times larger) in the corresponding direction. 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.



4.10.22. Influence line



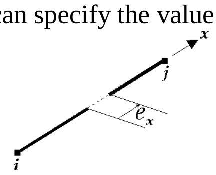
Lets you apply a relative displacement load to obtain the influence line of an internal force component, on the selected truss/beam elements. You must specify the value of the relative displacement *e* as +1 or –1.



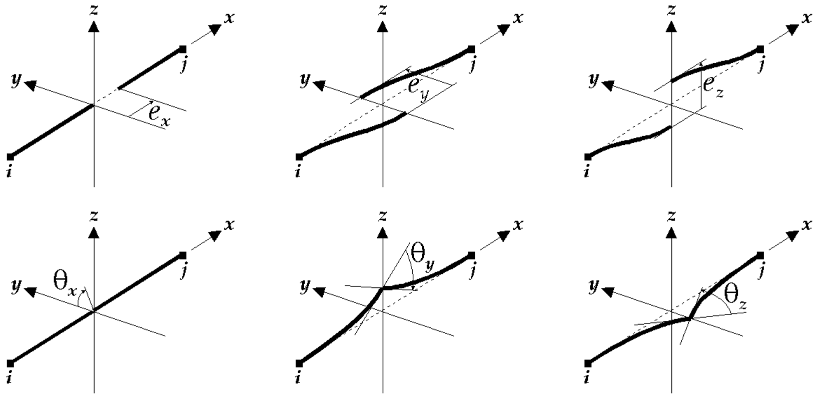
☞ ***You can define influence line load, only in an influence line type load case.***

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.



4.10.23. 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. Exceptions to this approach are the Swiss and Hungarian National Standards, which are not based on the Eurocodes. The standards in the program for which MRSA calculation is available are listed below.

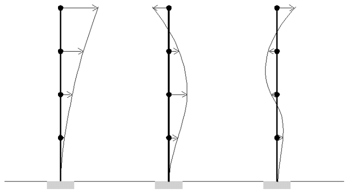
Eurocode general	*EN 1998-1:2004 (EC 8-1)* *Eurocode 8 Design of structures for earthquake resistance* *Part 1: General rules, seismic actions and rules for buildings*
EC German	*EN 1998-1:2004* *DIN EN 1998-1/NA:2011-01*
EC Dutch	*NPR 9998:2015*
EC Hungarian	*EN 1998-1:2004*
EC Romanian	*P100-1-2013*
EC Czech	*EN 1998-1:2004*
EC Slovakian	*EN 1998-1:2004*
EC Polish	*EN 1998-1:2004*
EC Danish	*EN 1998-1:2004*
EC Austrian	*EN 1998-1:2004*
EC British	*EN 1998-1:2004*
EC Finnish	*EN 1998-1:2004*
EC Belgian	*EN 1998-1:2004*
EC Swedish	*EN 1998-1:2004*
NTC Italian	*DM 14/01/2008*
Swiss	*SIA 261:2003*
Hungarian National Standard (MSZ)	
Romanian STAS	
German DIN 1045-1	*DIN 4149:2005-04*

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. The following parts provide details on the calculation process. First, the general Eurocode procedure is presented followed by a summary of country-specific modifications.

4.10.23.1. Seismic load calculation according to Eurocode 8

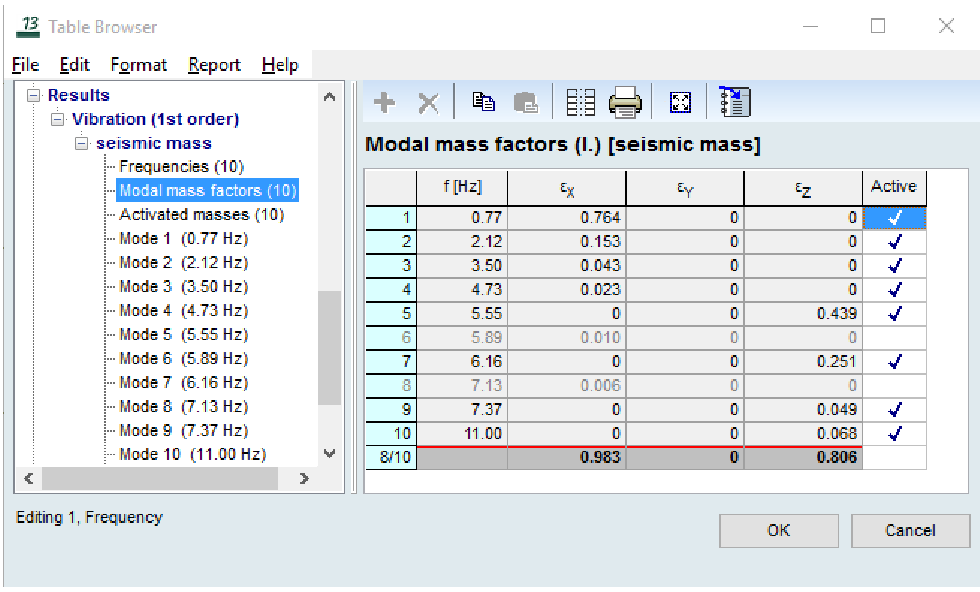
1. Calculate vibration modes



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.

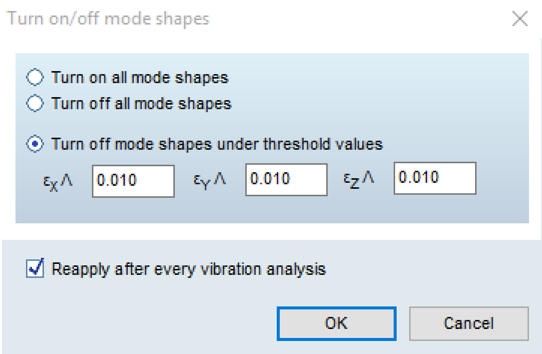


	f [Hz]	ε_X	ε_Y	ε_Z	Active
1	0.77	0.764	0	0	✓
2	2.12	0.153	0	0	✓
3	3.50	0.043	0	0	✓
4	4.73	0.023	0	0	✓
5	5.55	0	0	0.439	✓
6	5.89	0.010	0	0	
7	6.16	0	0	0.251	✓
8	7.13	0.006	0	0	
9	7.37	0	0	0.049	✓
10	11.00	0	0	0.068	✓
8/10		0.983	0	0.806	

The user can indicate which mode shapes shall be used for seismic load generation by putting a tick in the *Active* column of the table.

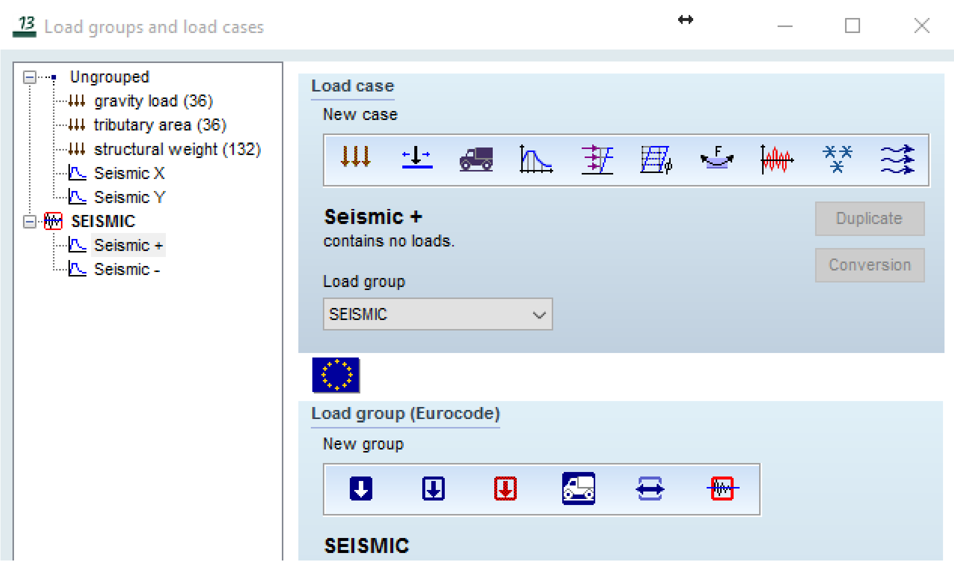
Right-clicking any cell in the *Active* column triggers a popup menu, where the *Turn on/off mode shapes* dialog can be selected. This dialog window lets the user define threshold values for automatic mode shape filtering in each direction. The checkbox at the bottom of the window allows the user the 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.



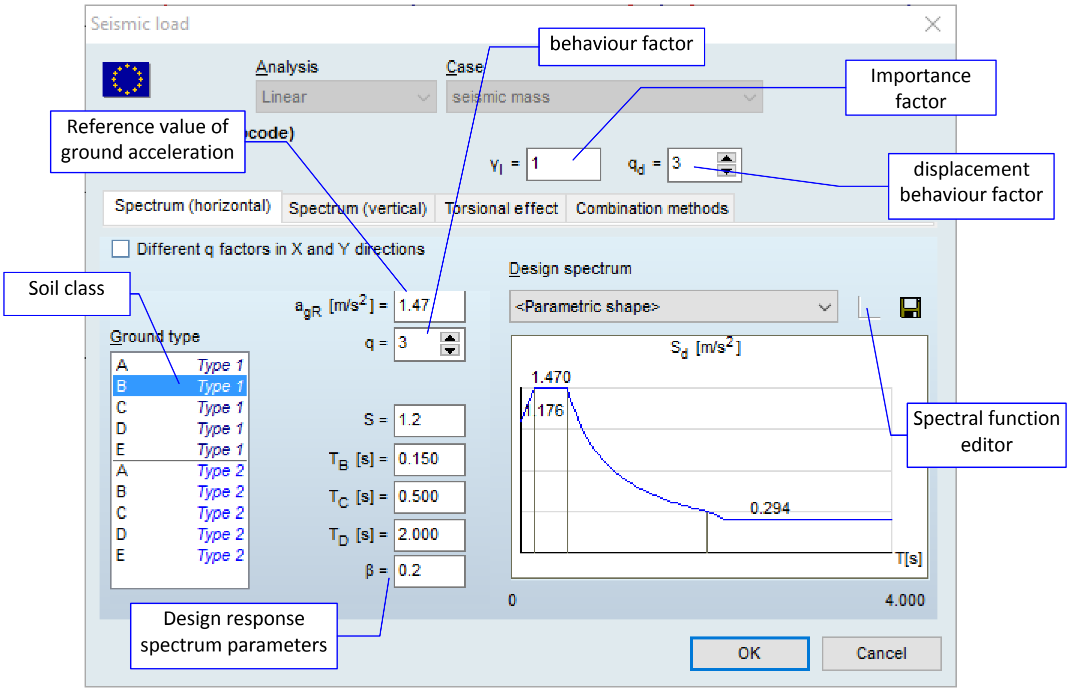
2. Create a new seismic load case

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.



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*.

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.

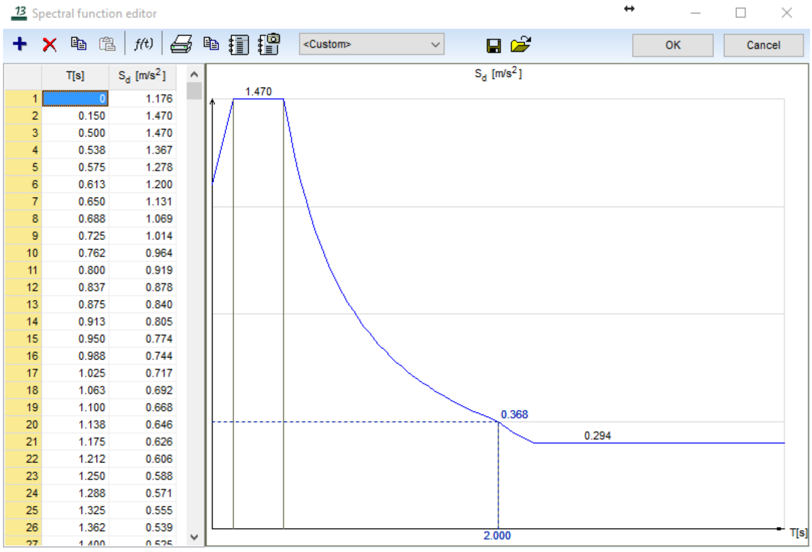
S, T_B, T_C, T_D – Soil-specific parameters that define the spectral shape. If the type of soil is selected from the list of soil classes, these parameters are automatically loaded according to Tables 3.2 and 3.3 in EC 8-1. Every value can be overridden by the user.

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

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.



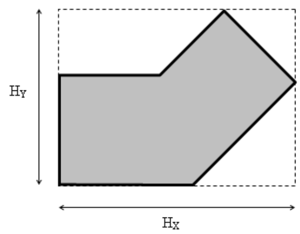
The second tab of the *Seismic Load* dialog window corresponds to the parameters of the **vertical response spectrum**. Note that the effect of seismic excitation in the vertical (Z) direction is only taken into account if a vertical response spectrum is defined. Vertical loads are not generated by default. Parameters and the layout of this tab are identical to the first one.

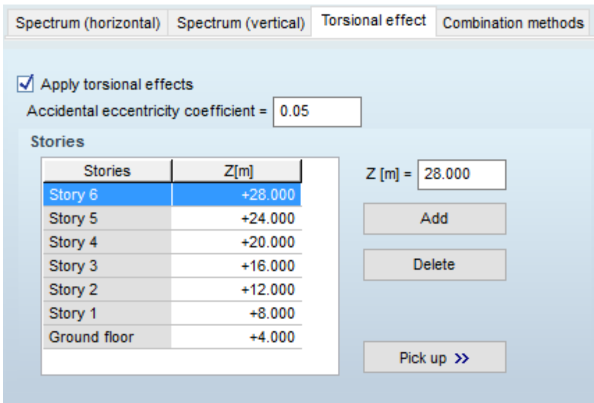
The third tab lets the user decide if **torsional effects** due to accidental eccentricity shall be taken into account. If the user wishes to consider such effects, then the program calculates additional torsional moments around the vertical axis for every story and every mode shape. The magnitude of torsional moments depends on the horizontal load and eccentricity of each story.
Horizontal loads are retrieved from load cases corresponding to each vibration mode and horizontal direction.
The size of eccentricity relative to the total size of each story 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:

$$T_{X,i,j} = F_{X,i,j} \cdot (\pm e H_{Y,i})$$
$$T_{Y,i,j} = F_{Y,i,j} \cdot (\pm e H_{X,i})$$

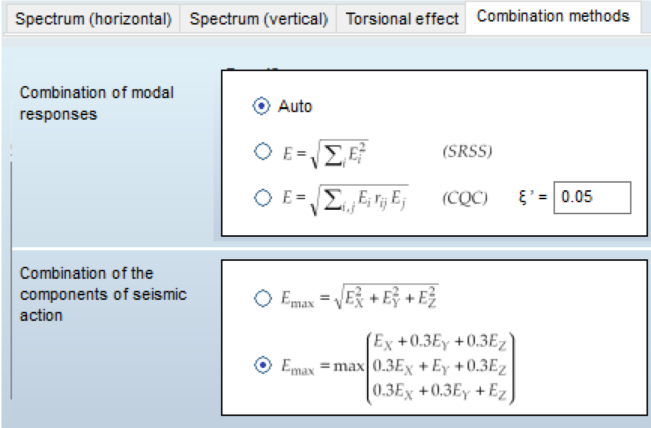
where F is the horizontal force in mode j at the i^{th} story; H is the size of the i^{th} story in the given X or Y direction. Torsional moments are considered with either + or – signs, but always with the same sign at all stories.





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: $T_j / T_i < 0.9$.
Results in the two horizontal and the vertical direction can be combined using either of the two commonly used combination methods displayed in the dialog window.



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

MRSA 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.

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 stories* table in the *Table Browser*. Independent results are available in the table for each horizontal direction.

The following quantities are calculated by the program:

θmax – *plastic stability index, also known as the interstory 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

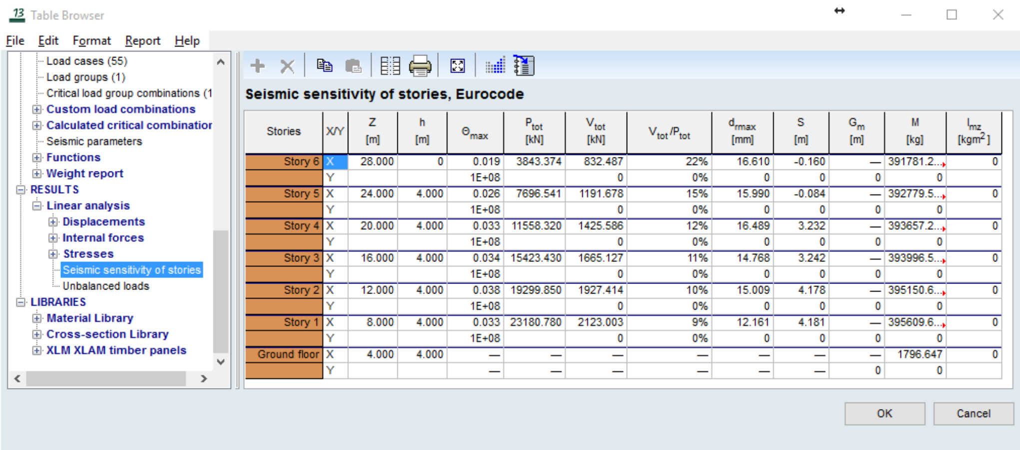
d_{max} – design interstory displacement calculated as the relative displacement of corresponding storey centroids

S – location of the shear center

G_m – location of the centroid

M – storey mass

I_{mz} – moment of inertia at the centroid about the Z axis



Seismic sensitivity of stories, Eurocode

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} [kgm²]
Story 6	X	28.000	0	0.019	3843.374	832.487	22%	16.610	-0.160	—	391781.2...	0
	Y			1E+08		0	0%	0	0	0	0	
Story 5	X	24.000	4.000	0.026	7696.541	1191.678	15%	15.990	-0.084	—	392779.5...	0
	Y			1E+08		0	0%	0	0	0	0	
Story 4	X	20.000	4.000	0.033	11558.320	1425.586	12%	16.489	3.232	—	393657.2...	0
	Y			1E+08		0	0%	0	0	0	0	
Story 3	X	16.000	4.000	0.034	15423.430	1665.127	11%	14.768	3.242	—	393996.5...	0
	Y			1E+08		0	0%	0	0	0	0	
Story 2	X	12.000	4.000	0.038	19299.850	1927.414	10%	15.009	4.178	—	395150.6...	0
	Y			1E+08		0	0%	0	0	0	0	
Story 1	X	8.000	4.000	0.033	23180.780	2123.003	9%	12.161	4.181	—	395609.6...	0
	Y			1E+08		0	0%	0	0	0	0	
Ground floor	X	4.000	4.000	—	—	—	—	—	—	—	1796.647	0
	Y			—	—	—	—	—	—	0	0	

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 – RC1 module

6.5.9.1 Check of reinforced columns according to Eurocode 2

6.5.10.3 Checking actual beam reinforcement

6.5.11.1 Punching analysis according to Eurocode2

6.5.12 Footing design – RC4 module

6.6.1 Steel beam design according to Eurocode 3 – SD1 module

6.6.2 Bolted joint design of steel

The f_{se} factor scales only the internal forces from seismic load.

4.10.23.2. 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.

4.10.23.3. Seismic load calculation according to German EC8-1 NA

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 DIN EN 1998-1/NA (EC8-1 NA)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 EC8-1 NA NDP 3.2.2.5(4)P.
- Soil-specific parameters are set according to Tables NA.4 and NA.5 in EC8-1 NA.

4.10.23.4. Seismic load calculation according to Italian DM2008

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 DM2008 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 DM2008 3.2.3.2.
- Soil-specific parameters are set according to Tables 3.2.V-VII in DM 2008 3.2.3.2.

4.10.23.5. 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).

4.10.23.6. Seismic load calculation according to Dutch NPR 9998:2015

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 9998:2015 standard.

3. Set seismic load parameters

Seismic effects and the definition of corresponding response spectra in NPR 9998 are heavily affected by soil conditions and the maximum peak ground acceleration value. Three distinct soil types are identified by the standard and this approach is also implemented in AxisVM:

- **normal soil**: *normal soil conditions* as per NPR 9998:2015 3.2.2.1. Horizontal response spectra are calculated using equations 3.21-3.23 in NPR 9998:2015.
- **special soil**: *special ground conditions* as per NPR 9998:2015 3.2.2.1. Horizontal response spectra are calculated by scaling the spectra defined by equations 3.21-3.23 in NPR 9998:2015 with a 1.50 amplification factor.
- **stiff soil**: *normal soil conditions* as per NPR 9998:2015 3.2.2.1 and $v_{s,30}$ > 250 m/s and no individual soil layers with v_s < 200 m/s. Horizontal response spectra are calculated using equations 3.21-3.23 in NPR 9998 if $a_{g,ref} \geq 0.2$ g. Otherwise, equations 3.24-3.26 are used.

The k_{ag} parameter shall be specified by the user based on the Consequence Class (CC) corresponding to the structure in Tables 2.1 and 2.2 in NPR 9998:2015. Because the NPR 9998:2015 standard uses the k_{ag} factor to consider the importance of the building, the γ_I importance factor is not used in this case. The vertical response spectrum is calculated using equations 3.27-3.29 in NPR 9998:2015.

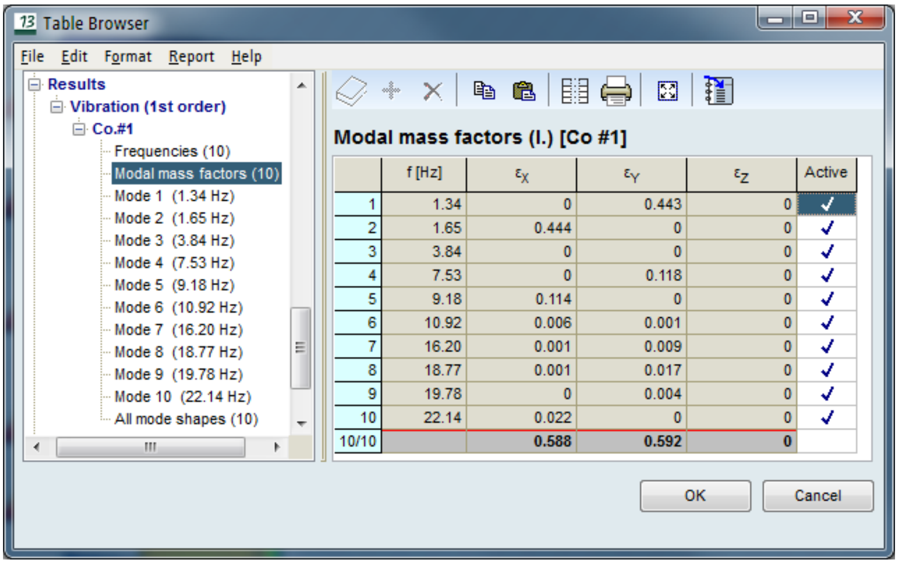
4.10.24. Pushover loads – SE2 module

Pushover loads are generated according to the regulations of Eurocode 8 (EN 1998-1:2004) 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.

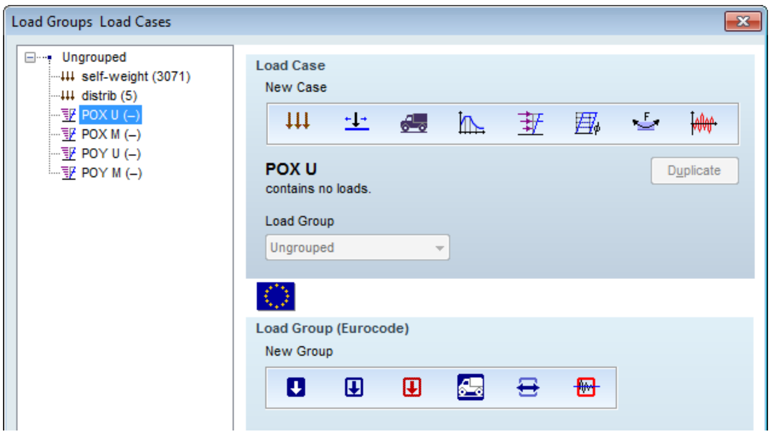
1. Calculate vibration mode shapes and frequencies.
When running the vibration analysis be sure to use the convert loads to masses option with the appropriate load case if there are loads defined that need to be considered static. Check the table of *Modal mass factors* in the Table Browser. Vibration results will appear only if the *Vibration* tab is selected.



	f [Hz]	ε_X	ε_Y	ε_Z	Active
1	1.34	0	0.443	0	✓
2	1.65	0.444	0	0	✓
3	3.84	0	0	0	✓
4	7.53	0	0.118	0	✓
5	9.18	0.114	0	0	✓
6	10.92	0.006	0.001	0	✓
7	16.20	0.001	0.009	0	✓
8	18.77	0.001	0.017	0	✓
9	19.78	0	0.004	0	✓
10	22.14	0.022	0	0	✓
10/10		0.588	0.592	0	

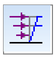
Although there is no requirement in Eurocode 8 for the minimum value of seismic equivalence coefficient, it is strongly advised to perform standard pushover analysis only on structures having clearly dominant mode shapes in each horizontal direction. The coefficients for each mode shape are listed in the *Modal mass factors* table (see above). 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.

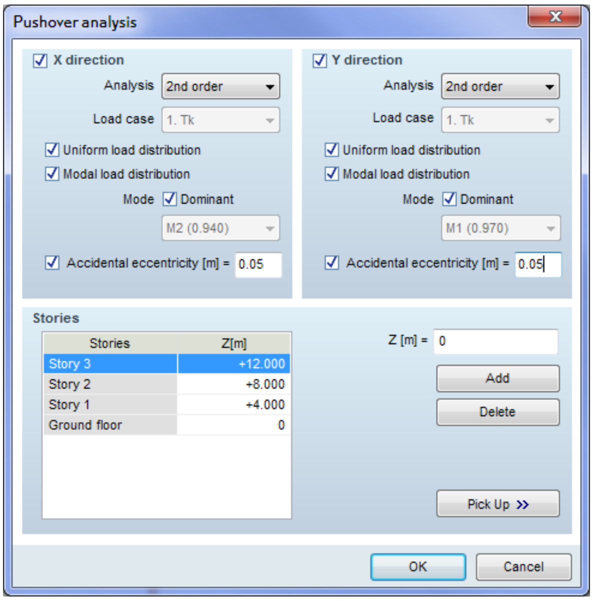
2. Create a new pushover load case.
Pushover load cases can be created, renamed and deleted in the *Load cases & load groups* dialog window. The initial configuration of four load cases is created by clicking on the *Pushover load* button.



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 story levels used for interstory drift calculation are specified at the bottom part of the window. (Previously defined story 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.

Pushover loads are generated only after closing the dialog window. Unnecessary load cases are also removed at this time.

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 story. The sum of the signed pushover loads will be still 1 kN but the resultant force will be eccentric causing torsion.

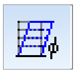
4. Run a Nonlinear Static Analysis

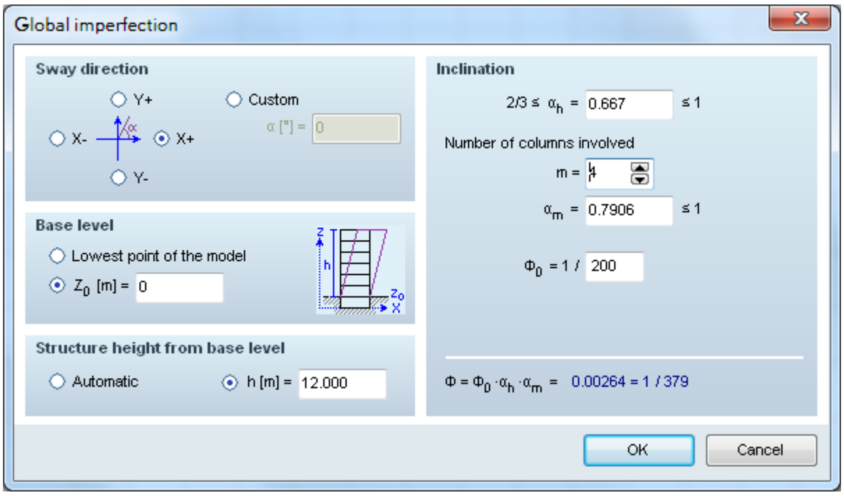
After defining loads for pushover load cases the pushover analysis shall be run using the Nonlinear Static Analysis button under the Static tab of the main window. Setting the solution control to Pushover lets the user define a parametric and a constant load case. The parametric load case is typically 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 *6.1.4 Pushover capacity curves* chapter.

4.10.25. 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 from the following formula: $\Phi = \Phi_0 \alpha_h \alpha_m$, where

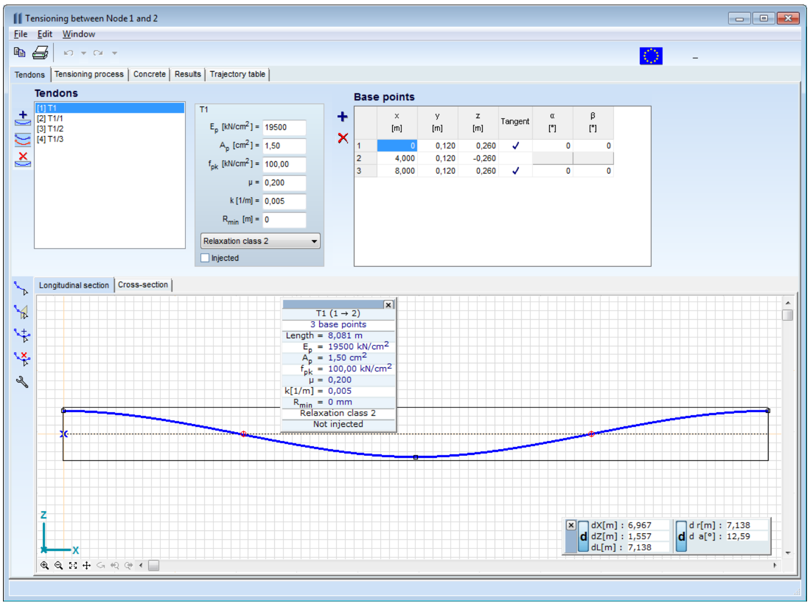
α_h is a reduction factor: $\alpha_h = \frac{2}{\sqrt{(h[m])}}$ with $\frac{2}{3} \le \alpha_h \le 1$

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

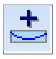
4.10.26. Tensioning - PS1 module

Tendons can be assigned to a continuous selection of beam or rib elements. After defining tendon properties and the tensioning process AxisVM determines the immediate losses of prestress and the equivalent loads for the end of tensioning (load case *name-T0*). After completing a static analysis it 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.

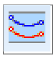
Tendons The first tab is to define tendon parameters and geometry.



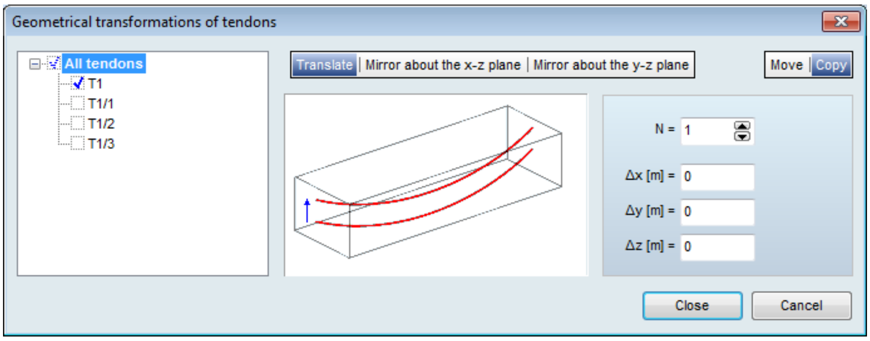
Icons on the vertical toolbar beside the tendon list are



Add new tendon. Geometry for the new tendon can be defined using the toolbar beside the diagram.



Geometrical tansformations 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.

Parameters of the selected tendon appear beside the tendon list. Parameter values can be edited.

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 $0.005 < k < 0.01$.
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.

To draw tendon geometry 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 make the base point the last one. 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* plane containing the first base point.

Steps of drawing a tendon in 2D

1. Select the postion of the cross-section where you want to define the tendon basepoint.
   Settle the tendon onto the proper position in the cross-section view.
   You can position the tendon onto the top or at the bottom of the cross-section considering the concrete cover.

   Position the tendon onto an optional point

   Position the tendon onto the neutral axis

   Position the tendon onto the top of the cross-section

   Position the tendon onto the bottom of the cross-section

2. Following the first location you can position the other points of the tendon onto the longitudinal section.

Draw tendon in 3D. The tendon position within the cross-section has to be specified at every basepoint. You can close a tendon geometry with using Mouse Right Button/Complete.
Steps of drawing a tendon in 3D:

1. Select the postion of the cross-section where you want to define the tendon basepoint.
2. Settle the tendon onto the proper position in the cross-section view.

Following the first location repeat Step 1 and Step 2 to define all basepoint.

Add new base point. Click the cable to add a new base point. In case of several tendons this function only works with the active tendon.

Delete base point. Clicking an existing base point deletes it. After deleting the second base point the tendon geometry is deleted. In case of several tendons this function only works with the active tendon.

Table of base points — Base point properties can be edited in the table.

The following functions are available on the toolbar:

Add a new base point

Delete selected lines

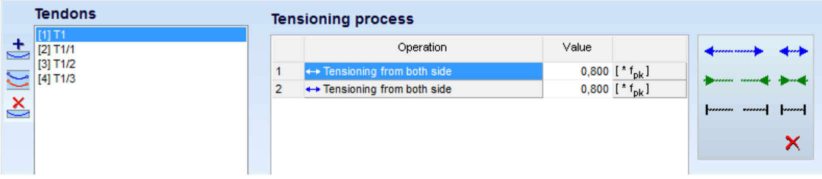
Copy tendon data to the Clipboard

Paste tendon data from the Clipboard

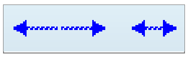
Options. Grid and cursor settings of the longitudinal and the cross-section diagram can be set.
See… *2.16.19.1 Grid and cursor*

Tensioning process

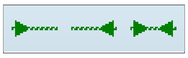
The second tab is to define the tensioning process for tendons by determining the order of certain operations.



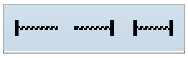
Possible operations and parameters:



Tensioning from left / right / both side — Force as a fraction of the characteristic value of tendon steel tensile strength (f_{pk}).



Release from left / right / both side

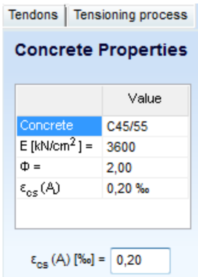


Anchor on left / right / both side — Wedge draw-in of the anchorage device

Deletes the last operation from the list.

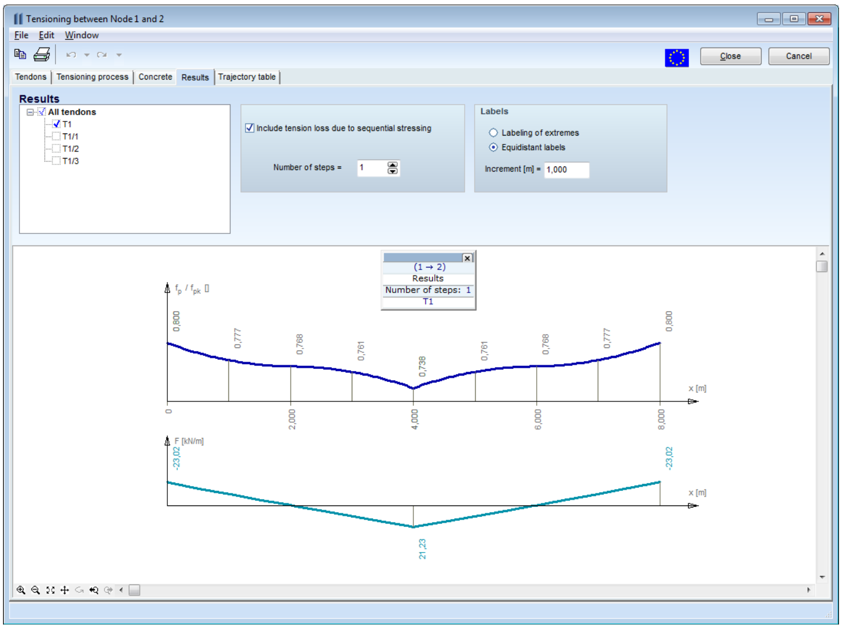
Concrete

The third tab is to check the material properties of the concrete. $e_{cs}(\infty)$ is the long term value of the concrete shrinkage strain. Its value can be entered here.



Results

If valid parameters, geometry and tensioning process is assigned to every tendon, result diagrams are displayed on the fourth tab. If one tendon is selected in the tree two diagrams are shown. The first one is the actual tension along the tendon (f_p/f_{pk}), and the equivalent load for the tendon *(F)*. If more than one tendon is selected the diagram shows the resultant equivalent load for the selected tendons only.



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

$$\Delta P_{c+s+r} = A_p \Delta\sigma_{c+s+r} = A_p \frac{\epsilon_{cs} E_p + 0.8\Delta\sigma_{pr} + \frac{E_p}{E_{cm}}\varphi\sigma_{c,QP}}{1 + \frac{E_p}{E_{cm}}\frac{A_p}{A_c}\left(1 + \frac{A_c}{I_c} z_{cp}^2\right)[1 + 0.8\varphi]},$$

where

$\Delta\sigma_{c+s+r}$ 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 centre 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

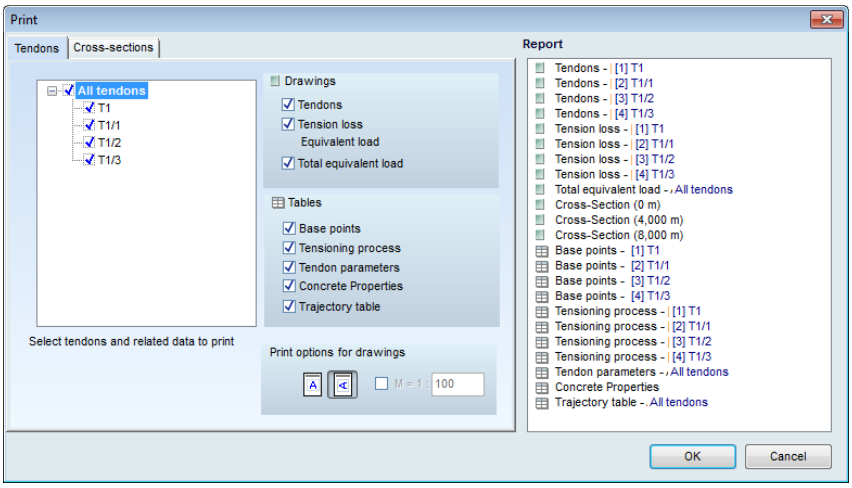
The main toolbar has two buttons.

Copy diagram
Ctrl+C

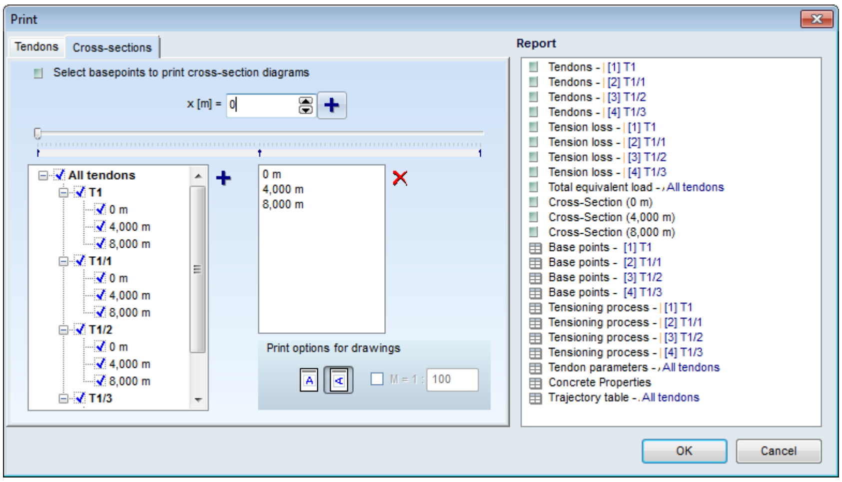
Copies the drawing on the active tab to the Clipboard as a Windows metafile. This way the diagram can be pasted to other applications (e.g. Word).

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).*

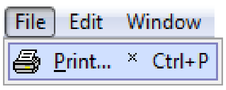


Cross-sections can be selected to print cross-section diagrams.



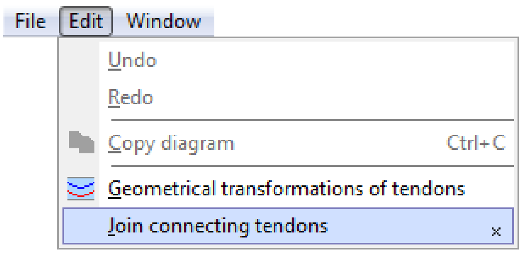
Menu *You can reach the following functions via the menu:*

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*

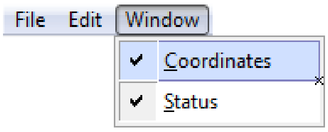
Geometrical transformations of tendons

See… *Tendons / Geometrical transformations of tendons*

Join connecting tendons

If more than one beam or rib element has been selected and these elements contain connecting tendons this function joins the connecting tendons. The joining works in case of single element, too.

Window



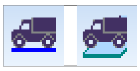
Coordinates

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

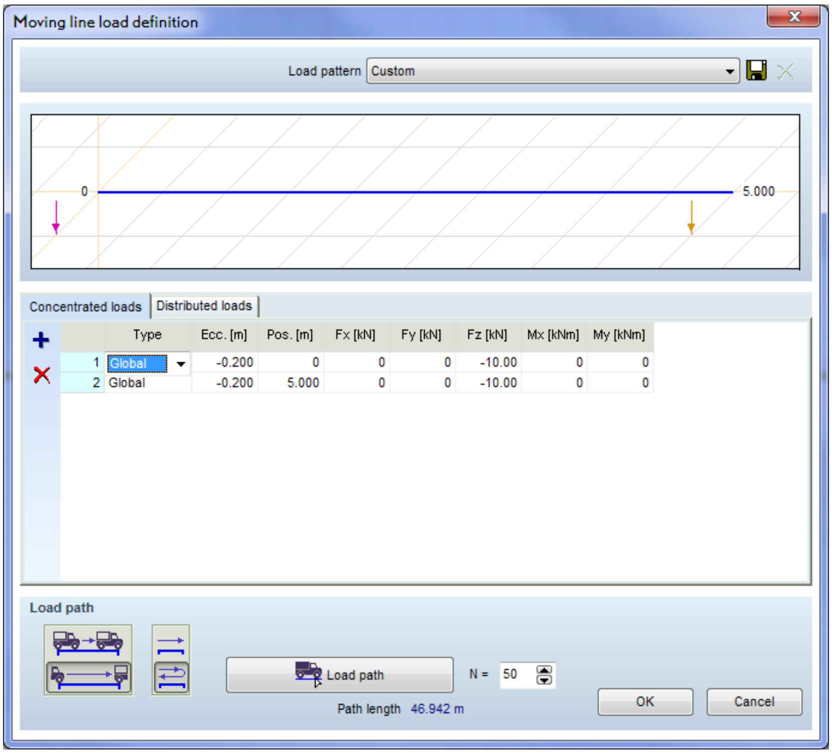
On diagrams an information window appears displaying diagram-specific information. Display of this window can be turned on and off.

4.10.27. 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* . 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 cannot be deleted or moved into another load group individually. Increasing the number of load steps will create additional load cases. Decreasing this number will make 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 gray.
Open Table Browser to see the load and the load path in tabular format. These tables can be also used for reporting purposes.

4.10.27.1. Moving loads on line elements

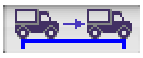


Moving load on line elements is a load pattern moving on a user-defined load path in *N* steps. The load pattern can contain any combination of concentrated and distributed loads.

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 axis. If it is on the left side when moving along the path its eccentricity is negative. If it is on the right side, its eccentricity is positive.
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 startpoint and endpoint has to be selected. These points must be nodes along the path.
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.

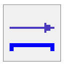
The local *z* direction of the load pattern will always be the local *z* direction of the line elements it is placed on.
Lengthening, shortening or breaking a line element of the path will lead to an automatic recalculation of the load phases.


Crane runway mode

In the first phase the load with the lowest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the highest coordinate in the pattern will be placed over the endpoint.


Bridge mode

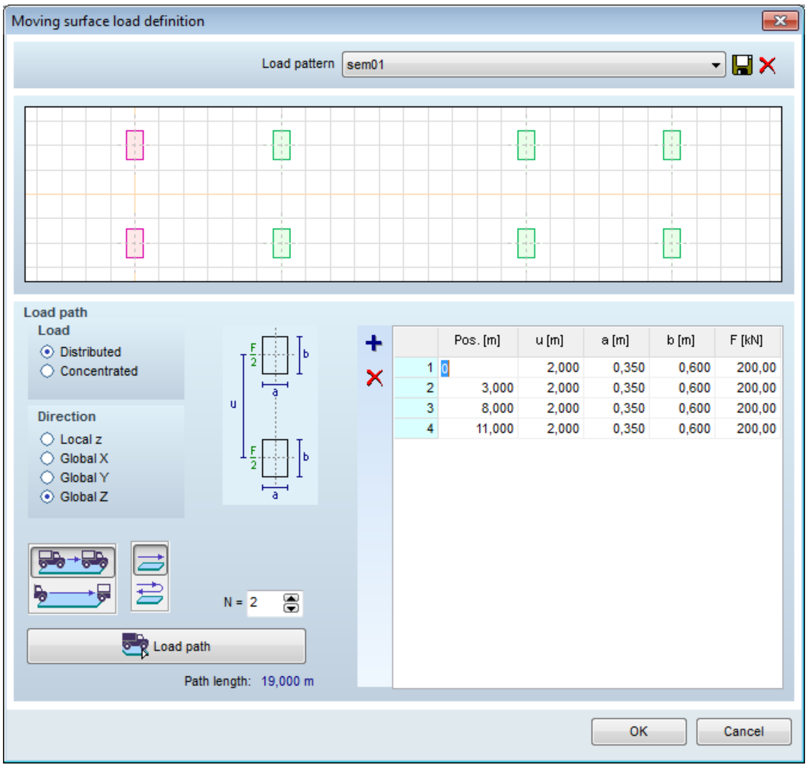
In the first phase the load with the highest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the lowest coordinate in the pattern will be placed over the endpoint.



One way: Load moves from startpoint to endpoint in *N* steps.

Round trip: Load moves from startpoint to endpoint and back in *2N* steps.

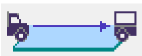
4.10.27.2. Moving loads on domains



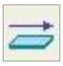
This load type is convenient when vehicle loads has to be defined. The load pattern consists of concentrated or rectangular surface loads pairs representing the wheels on the axles.
u is the vehicle gauge, *a* and *b* refers to the rectangle dimensions. Axle load *F* will be distri-buted evenly on the 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.
The load path does not have to stay in the same plane and can cross holes or empty areas between domains.
Path startpoint and endpoint is the first and last point of the polyline.
Each phase will contain only the loads actually falling on a domain. The local *z* direction of the load pattern will be the local *z* direction of the domain 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 startpoint. In the last phase the load with the highest coordinate in the pattern will be placed over the endpoint.
Changing domain geometry will lead to an automatic recalculation of the load phases.


Crane runway mode

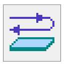
In the first phase the load with the lowest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the highest coordinate in the pattern will be placed over the endpoint.


Bridge mode

In the first phase the load with the highest coordinate in the pattern will be placed over the startpoint. In the last phase the load with the lowest coordinate in the pattern will be placed over the endpoint.



One way: Load moves from startpoint to endpoint in *N* steps.



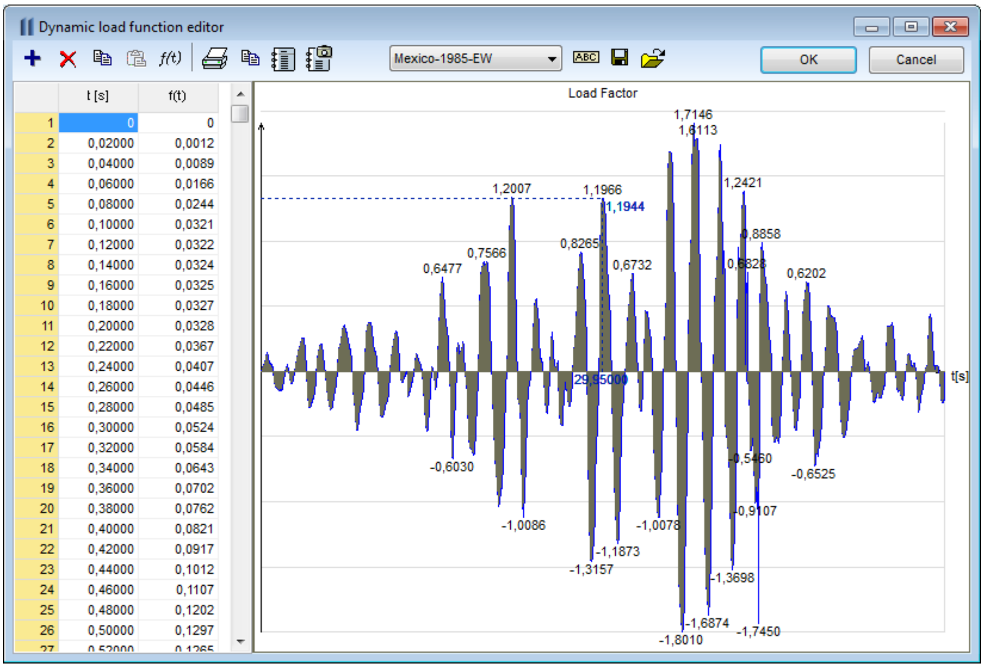
Round trip: Load moves from startpoint to endpoint and back in *2N* steps.

4.10.28. 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 a ***dfn*** folder created under the main folder of the program.

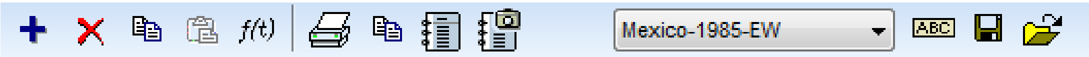


Table editing functions

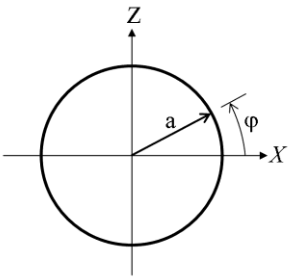
Add a new row to the table.

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. The follwing 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.

Diagram and report functions

Print the diagram and the 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*

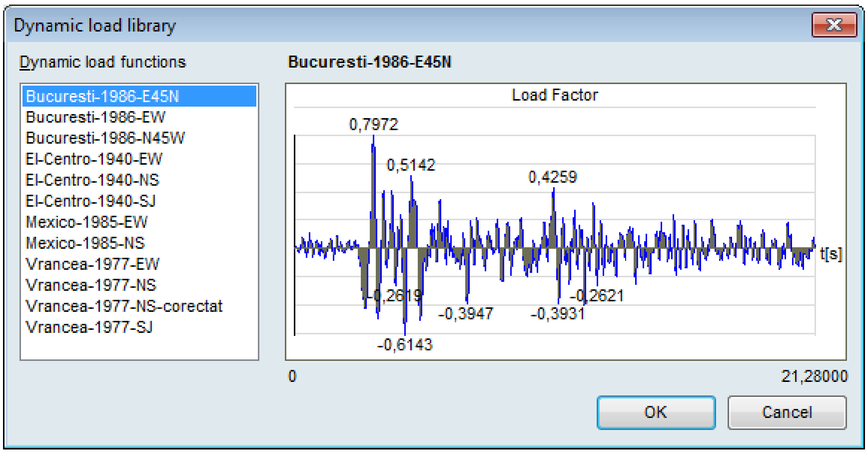
Mexico 1985 EW

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.

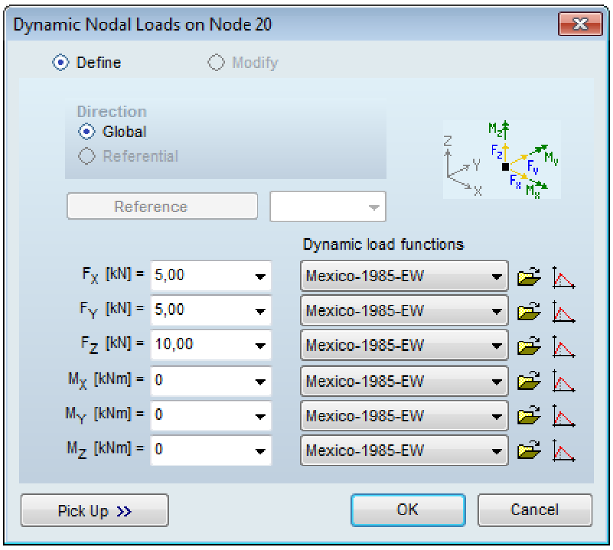
Load a function from the library.



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.

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.

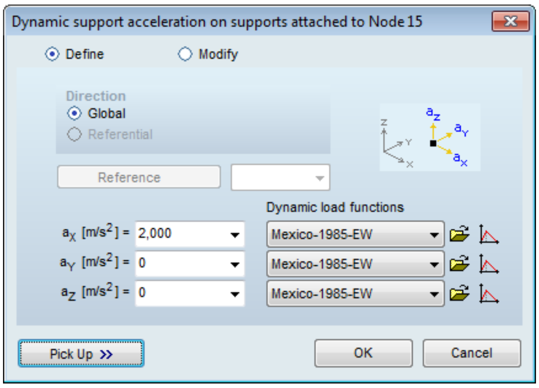
Dynamic loads are displayed as dashed yellow arrows.

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

$$a_i(t) = a_i \cdot f(t)$$

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.

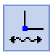
If multiple nodal supports are attached to a node, acceleration acts on all supports.

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.

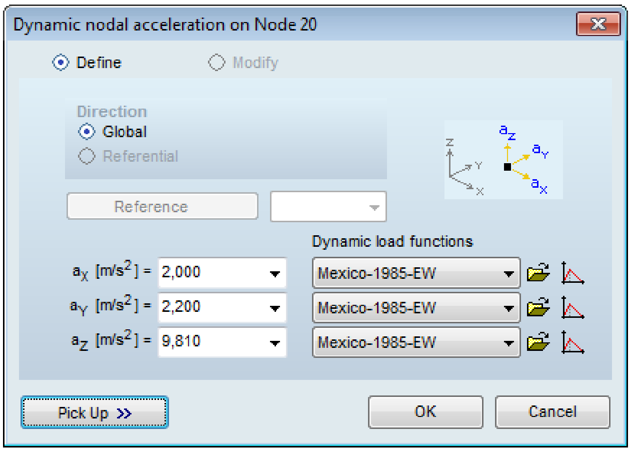
Dynamic nodal acceleration



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

$$a_i(t) = a_i \cdot f(t)$$

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.

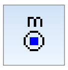
To specify ground acceleration for seismic analysis nodal support accelerations must be defined.

Modify, delete

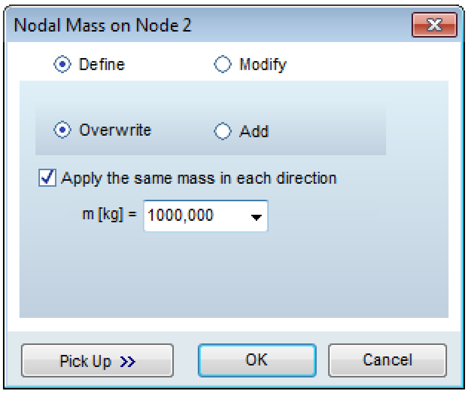
Dynamic nodal acceleration can be modified or deleted the same way as a static load.

Dynamic nodal acceleration is displayed as a circle and a yellow arrow.

4.10.29. 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.

4.10.30. 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

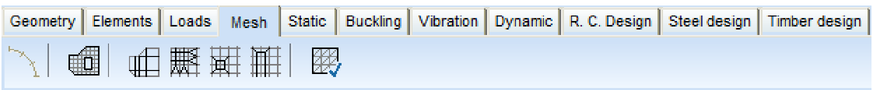
If the *Loads* tab is active click a finite element to modify its loads. If the element has more than one load only one of them will come up. If you have placed different concentrated and distributive loads on a beam and click the beam the load nearest to the click position will come up. If more finite 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.

4.10.31. Delete

[Del] **See...** *3.2.8 Delete*

4.11. Mesh

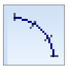


Clicking the mesh tab mesh toolbar becomes available with mesh generation for line elements and domains, mesh refinement functions and a finite element shape checking.

4.11.1. 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.

4.11.1.1. 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:

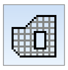
Maximum deviation from arc: Chord height cannot exceed the value specified.

Maximum element size: Length of the mesh lines cannot exceed the value specified.

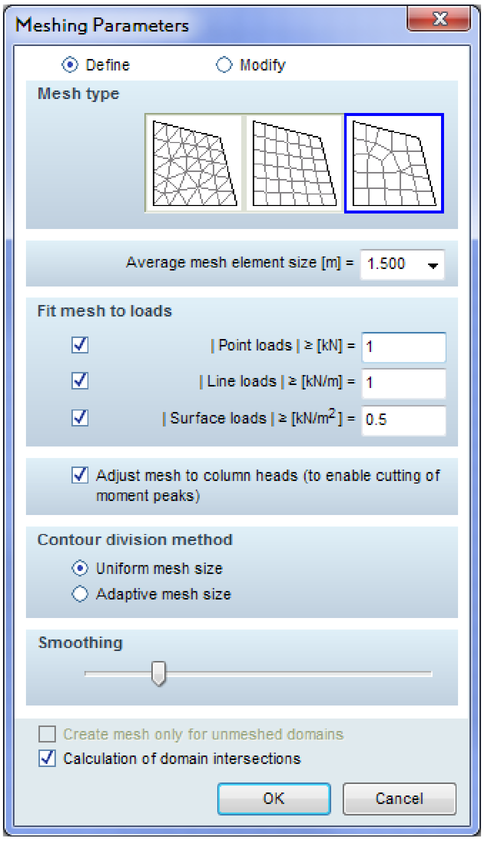
Division into N segments: Line elements are divided into N parts.

By angle: Central angle of arced mesh segments cannot exceed the value specified.

4.11.1.2. 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 of moment peaks.



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 of moment peaks. 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 moment peaks over columns* option of the *Display Parameters* dialog. **See...** *6.1.10 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.

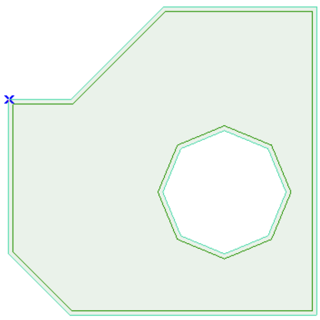
Smoothing Track bar controls the smoothing of the mesh. Smoothing slows down mesh generation a bit. Moving the handle to the left end sets minimum smoothing and fast processing while the right end sets maximum smoothing with slower processing. The result of smoothing depends on domain geometry and other mesh parameters, so setting higher smoothing does not necessarily result in a better mesh quality.

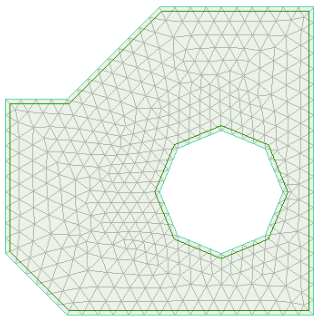
If *Create mesh only for unmeshed domains* is checked no mesh will be created for domains already meshed.

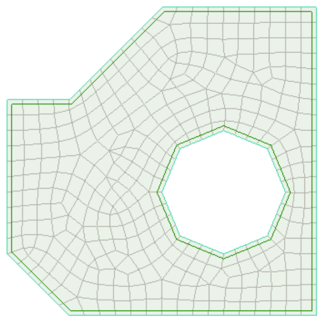
If *Calculation of domain intersections* is turned on domain intersections are automatically calculated before meshing.

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.

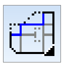

Before meshing

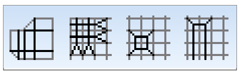

Triangle mesh


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.***

4.11.2. Mesh refinement



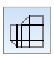


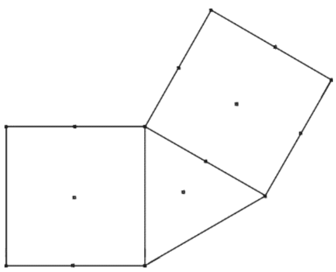
Lets you refine the finite element mesh of surfaces. The elements in the refined mesh have the same properties (material, cross-section / thickness, references, etc.) as those in the coarse mesh.

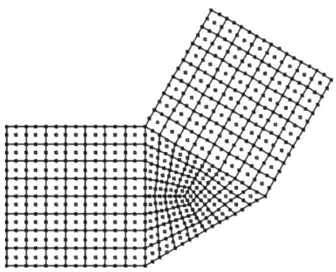
☞ ***You have to manually set the nodal degrees of freedom of the newly generated mesh that were not set automatically during the process of mesh generation.***

The following options are available:

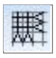
Uniform Lets you refine the entire selected mesh. You must specify the maximum side length of a surface element in the refined mesh.



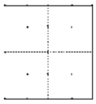

Before mesh refinement


After mesh refinement

Bisection

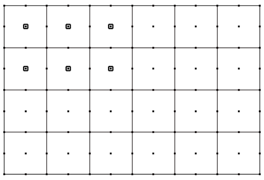


Lets you refine the selected mesh by bisecting the elements as shown in the figure.

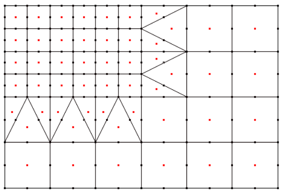


Quadrilateral element

Triangular element

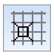


before

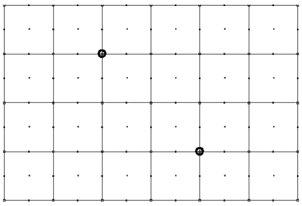


after

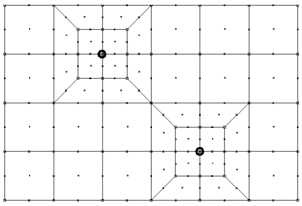
Node relative



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.

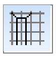


before

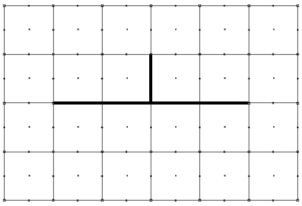


after

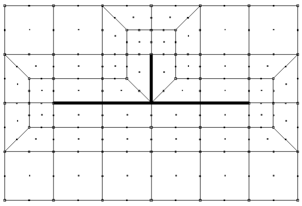
Edge relative



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.

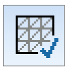


before



after

4.11.3. Checking finite elements



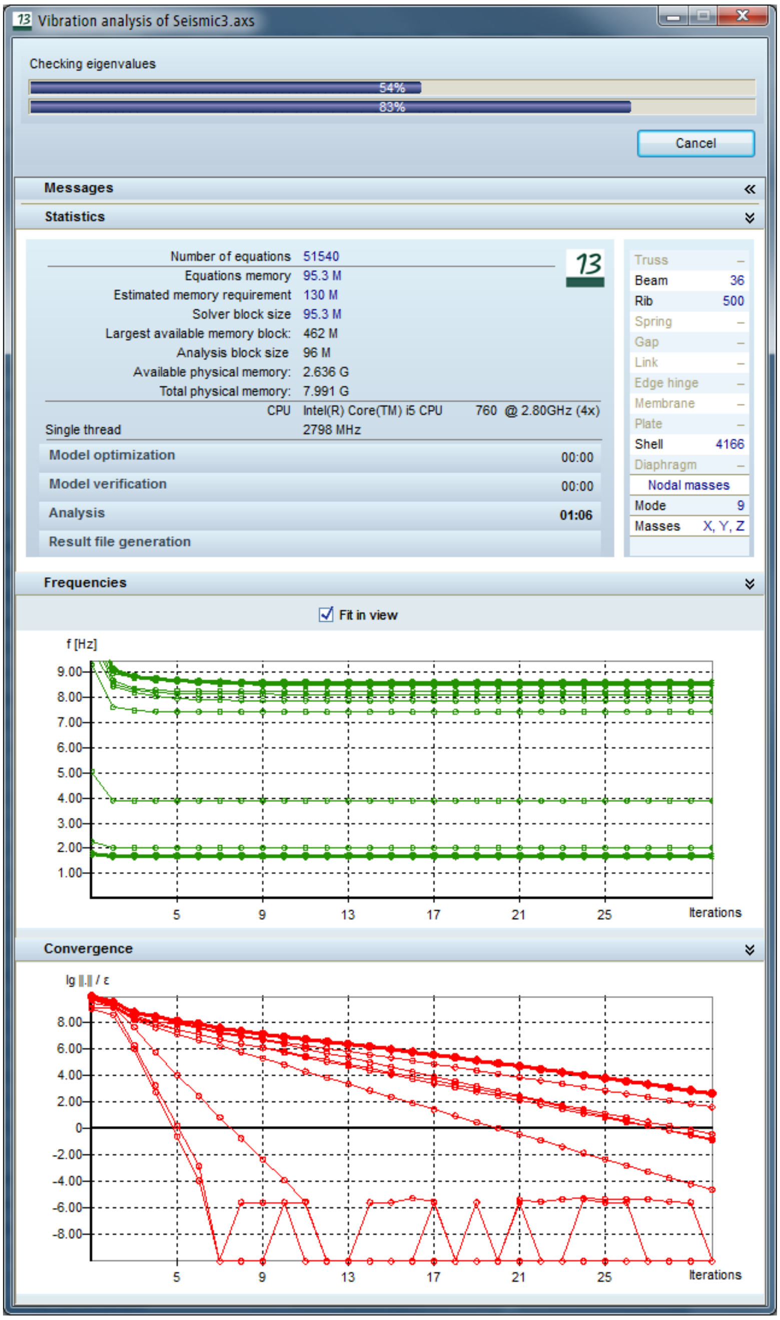
Program checks the minimum angle of surface finite elements (α).

A triangular finite element is distorted if $\alpha \leq 15$.
A qudrilateral finite element is distorted if $\alpha \leq 30$.

This page is intentionally left blank.

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 three 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 requierements, 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* displays 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...** *2.16.20 Model info*Model info

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

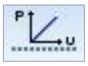
During the processing of the results the program sorts the results according to the original order of the nodes and prepares them to graphical display.

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.

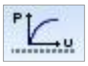
☞ ***See the description of the gap, and spring elements in Chapter 4, on how to use these elements in a linear analysis.***

The relative errors at the end of the iteration process appear in the info window.

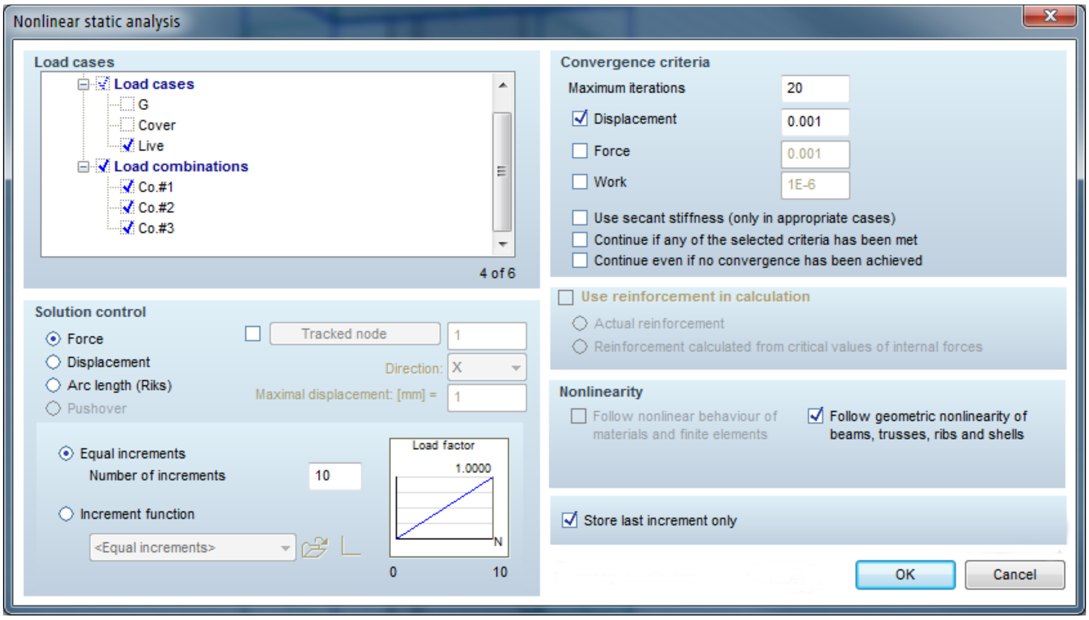
E(U): relative error of the displacement convergence
E(P): relative error of the force convergence
E(W): relative error of the work convergence

Values indicating instability appear in red.

Nonlinear static



Performs a nonlinear-elastic 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 gap, link or non-linear support, truss or spring elements, or taking into account the geometric nonlinearity of truss, beam, rib and shell elements.
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.

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.

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...** *4.10.23 Seismic loads – SE1 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.

Load factor

Load factor can be used to multiply loads of the selected load case or combination for the nonlinear analysis.

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.

☞ ***Increment function must be monotonous (loads cannot decrease).***

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

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.

Use reinforcement in calculation

When analyzing reinforced concrete plates it is possible to take into account the calculated or actual reinforcement.
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 deflection of RC plates).

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 concretre beam and column elements).

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 or compression-only supports) 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

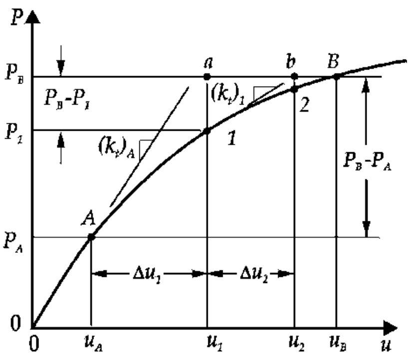
The equilibrium is established with respect to the deformed line elements. Depending on the magnitude of displacement second or third order analysis is performed. Geometric nonlinearity can be taken into account only for truss, beam, rib and shell elements. If there are no elements with nonlinear characteristic 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.

☞ ***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.

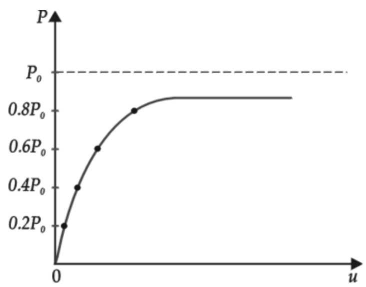
☞ ***You should disable this check-box if you want to trace the load-displacement or other (nonlinear) response of the structure.***



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.

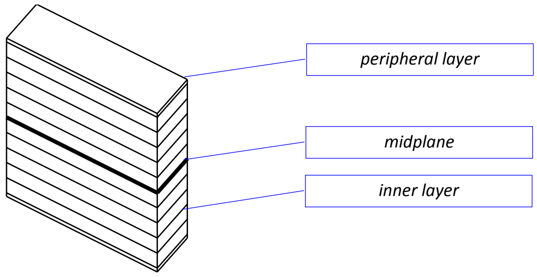
Displacement control

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.



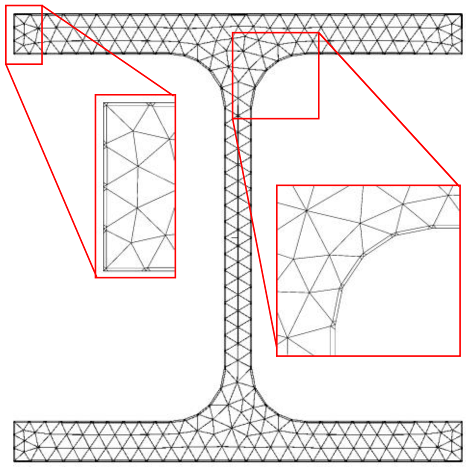
Finite elements with nonlinear material

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.

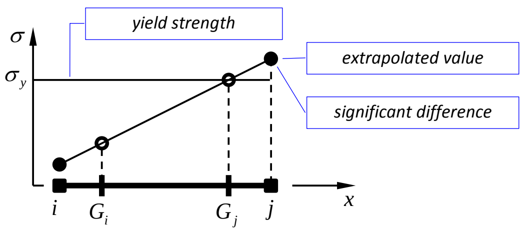


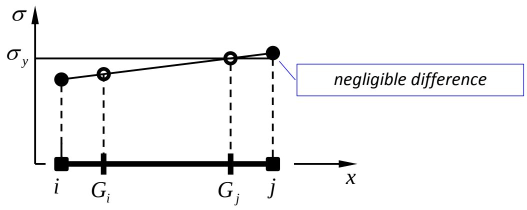
Stress components are the same as in the linear material model. For surface elements results are obtained in top, center and bottom plane, for beam and rib cross-sections in the stress points and along the outline.

To obtain accurate results in the extreme fiber the thickness of the layers are 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.



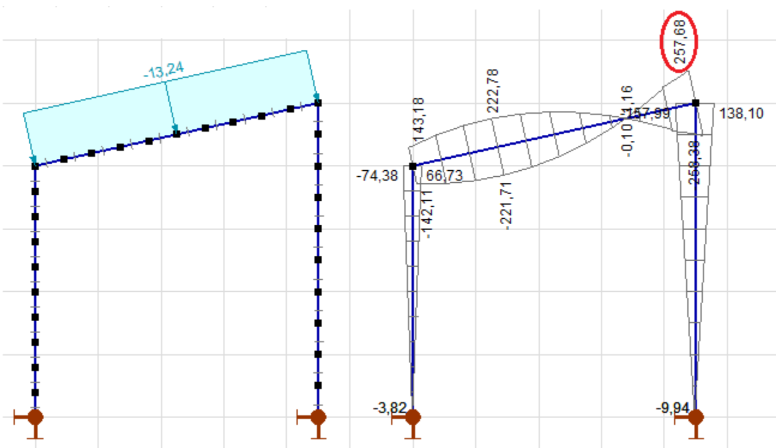
The nodal values of strain, stress and internal force components are extrapolated on the basis of the exact values calculated at the Gaussian integration points. This procedure causes a typical over-estimation of the results exceeding the yield strength at the boundary of plastic zones. The extrapolated values are more accurate if the variation of the result component along the element is small compared to its average value.



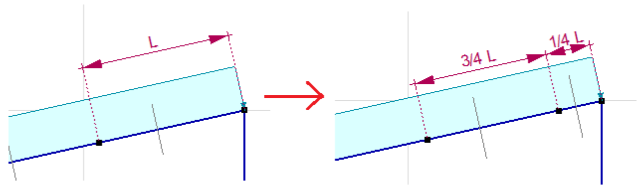


To get more accurate results the refinement of the mesh in the plastic zones is necessary. Obtaining the new results further refinement may also be necessary. This kind of iteration process is common in case of models with nonlinear material. Increasing the mesh density the overestimation of strains, stresses and internal forces can significantly be reduced but it cannot be eliminated.

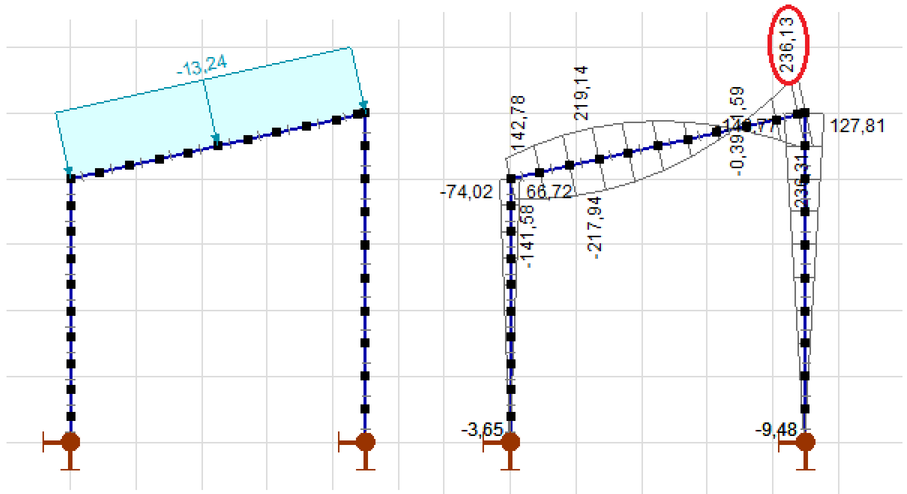
In case of beam structures the insertion of a small element to the plastic hinge occurred can be enough in the first step. The suggested length for the small element is ¼ of the length of the original element.



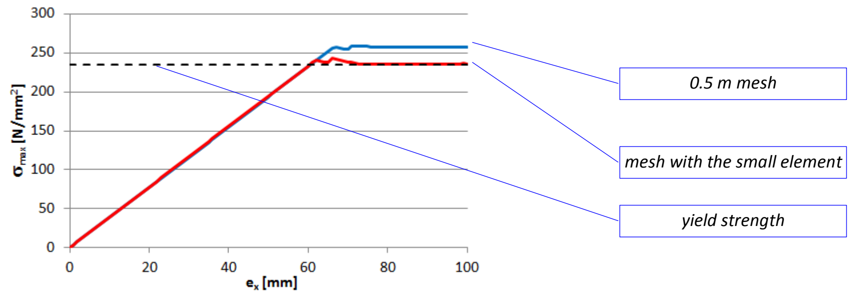
Model and stress distribution with a 0.5 m mesh. The marked stress value is significantly higher than the yield strength: 257,86 MPa > 235 MPa.



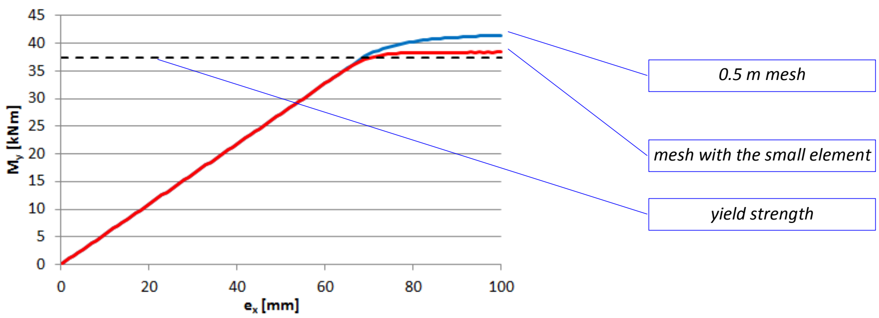
Inserting a small element to the location of the plastic hinge



Model and stress distribution after inserting a small element.
The marked stress value is only slightly higher than the yield strength: 236,13 MPa = 1,0048 · 235 MPa.

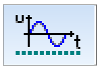


Stress results with respect to the control displacement



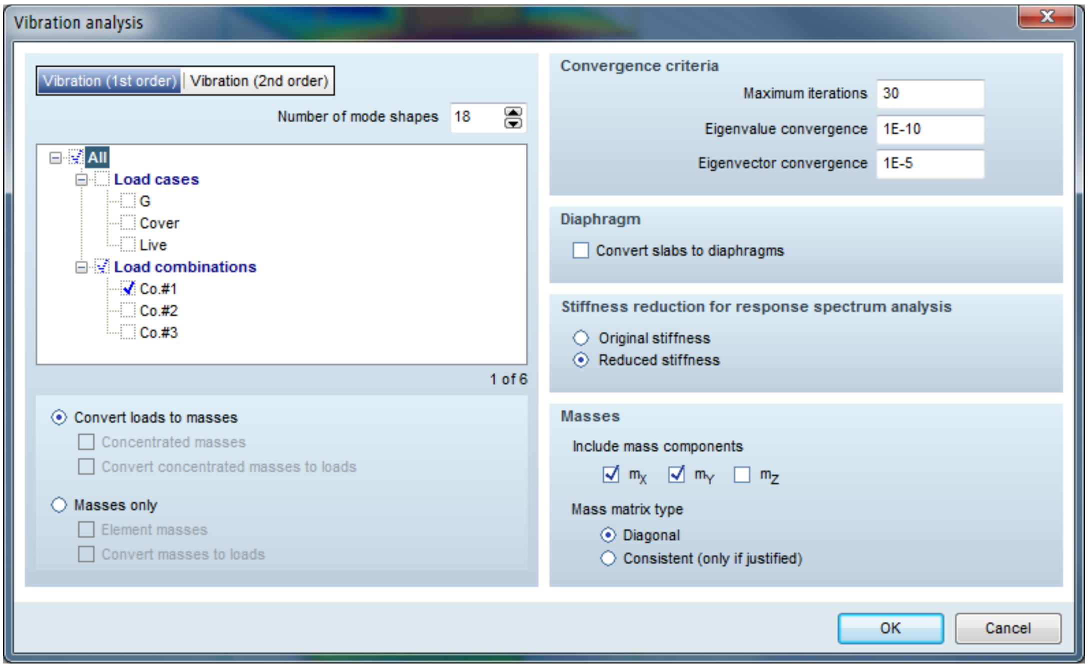
Bending moment with respect to the control displacement

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.***



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. A maximum number of 99 can be requested. The default value is 6. 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 gravitational loads into masses, and take these concentrated masses into account.

Masses only
You can analyze models without loads, but with masses, and take element masses into account.

Include mass components
Only checked mass components will be used in the analysis. It is useful when calculating modal shapes only 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

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.

Stiffness reduction for response spectrum analysis

Further information in chapter *3.3.10 Stiffness reduction.*

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.

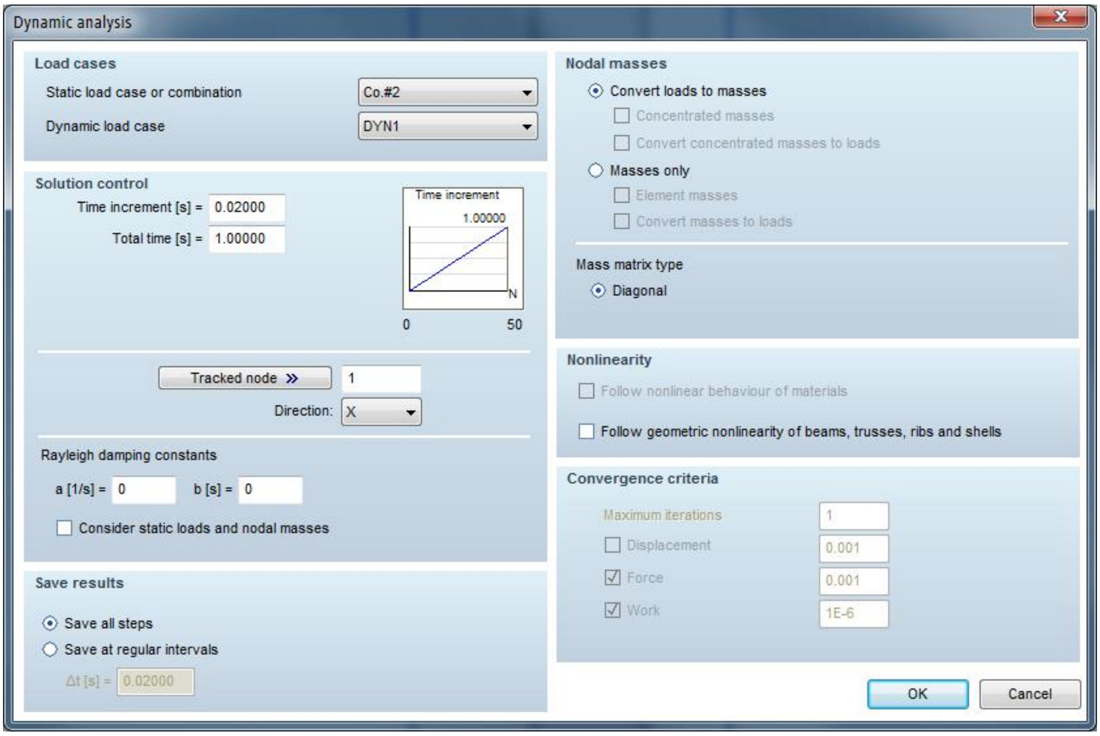
☞ ***The program uses a diagonal mass matrix by default. Due to the lumped mass modeling technique to achieve the required accuracy the elements must be divided into more elements (by refining the mesh). Usually at least four finite elements must correspond to each half wave.***

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, $\{U\}^T[M]\{U\} = 1$

5.3. 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.



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 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 contants according to the following formulas:

$$\mathbf{M}\underline{\ddot{u}} + \mathbf{C}\underline{\dot{u}} + \mathbf{K}\underline{u} = P(t)$$
$$\mathbf{C} = a\mathbf{M} + b\mathbf{K}$$

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.

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 *Perform with equilibrium iterations* 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.

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:

$$\mathbf{M}\underline{\ddot{u}}_{t+\Delta t} + \mathbf{C}\underline{\dot{u}}_{t+\Delta t} + \mathbf{K}\underline{u}_{t+\Delta t} = P(t)$$

where **C** is the damping matrix, **M** is the mass matrix, **K** is the stiffness matrix.

$$\underline{U}_{t+\Delta t} = \underline{U}_t + \Delta t \cdot \underline{\dot{U}}_t + \frac{\Delta t^2}{2}\left[(1-2\beta)\underline{\ddot{U}}_t + 2\beta\underline{\ddot{U}}_{t+\Delta t}\right]$$

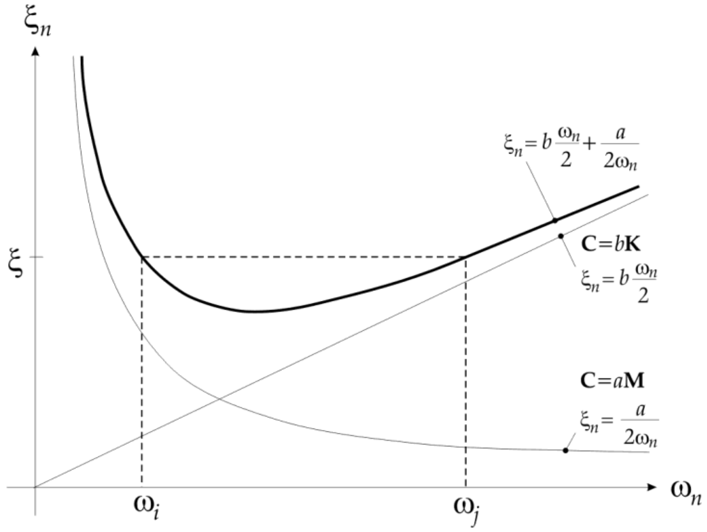
$$\underline{\dot{U}}_{t+\Delta t} = \underline{\dot{U}}_t + \Delta t\left[(1-\gamma)\underline{\ddot{U}}_t + \gamma\underline{\ddot{U}}_{t+\Delta t}\right]$$

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:



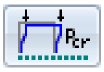
$$a = \xi\frac{2\omega_i\omega_j}{\omega_i + \omega_j}$$

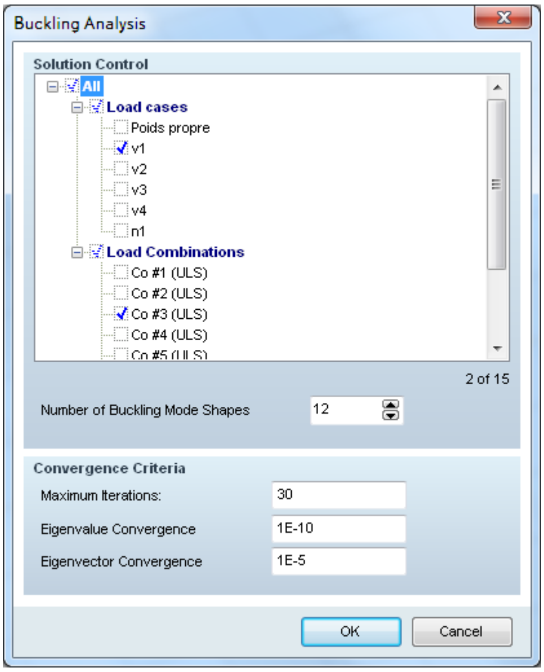
$$b = \xi\frac{2}{\omega_i + \omega_j}$$

where ω_i and ω_j are angular frequencies relating to f_i and f_j:

$$\omega_i = 2\pi f_i$$
$$\omega_j = 2\pi f_j$$

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, 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.***

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

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.***

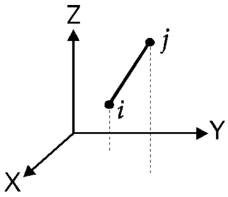
Trusses

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.

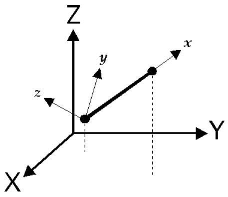
☞ ***If the model contains trusses the critical load parameter of global structural buckling will be computed only. Buckling of individual trusses is not analysed.***

5.5. Finite elements

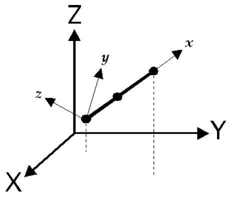
All finite elements may be used in a linear static, nonlinear static, vibration, buckling and dynamic analysis. Note that ***not*** all elements have geometric stiffness.



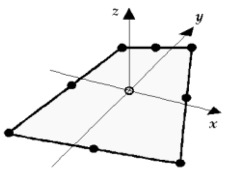
Truss



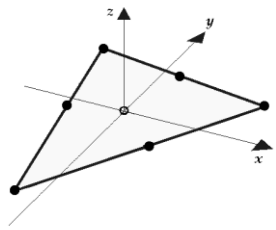
Beam



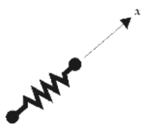
Rib



Quadrilateral membrane, plate, or shell

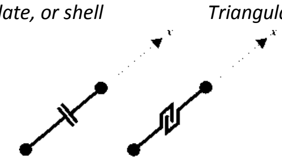


Triangular membrane, plate, or shell



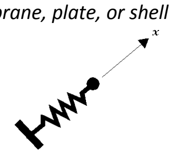
Spring

(only one component is shown)



Gap

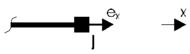
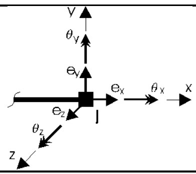
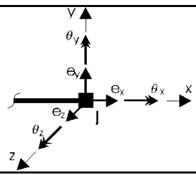
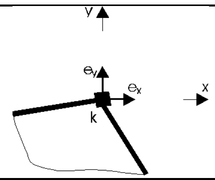
(active in compression and tension respectively)

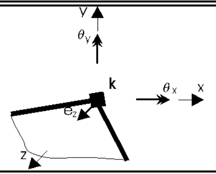
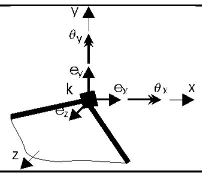
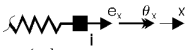
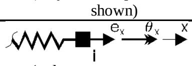
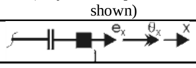
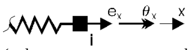


Support

(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						

Finite element	ex u	ey v	ez w	Θx	Θy	Θz	
Plate			*	*	*		
	Hughes type, 9-node, Heterosis isoparametric plate element						
Shell	*	*	*	*	*		
	Flat shell superimposed membrane and plate element						
Support	*	*	*	*	*	*	 (only two components are shown)
Spring	*	*	*	*	*	*	 (only two components are shown)
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}
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		

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

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...** *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 ...

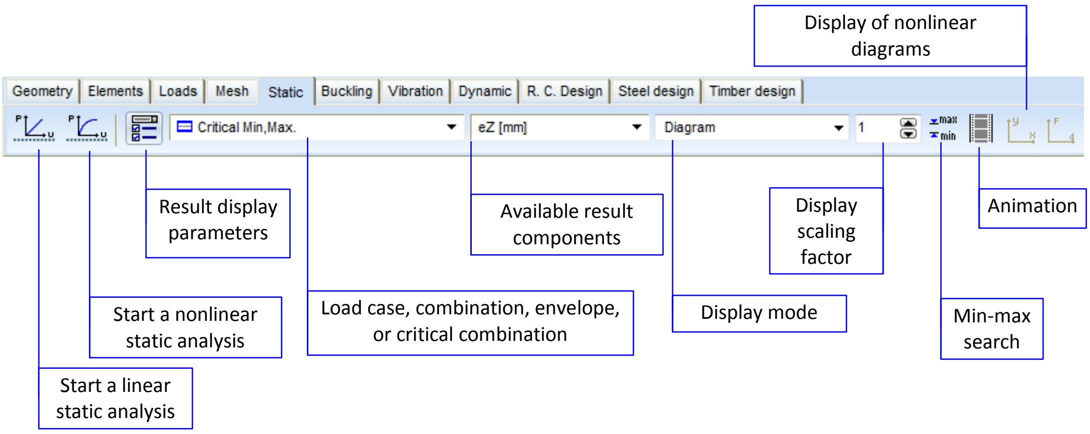
There is a singularity in the system stiffness matrix corresponding to that degree of freedom. You should check the support and degrees of freedom (DOF) settings of your model.

6. The Postprocessor

Static	Lets you display the results of a static analysis. (*6.1*)
Vibration	Lets you display the results of a vibration analysis. (*6.2*)
Buckling	Lets you display the results of a buckling analysis. (*6.2*)
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 Beam Design	Lets you display the results of a timber design analysis. (*6.7*)
XLAM design	Lets you display the results of an XLAM-domain design analysis. (*6.8*)

6.1. Static

The Static menu item allows you to display the tools for displaying and interpreting the static analysis results.

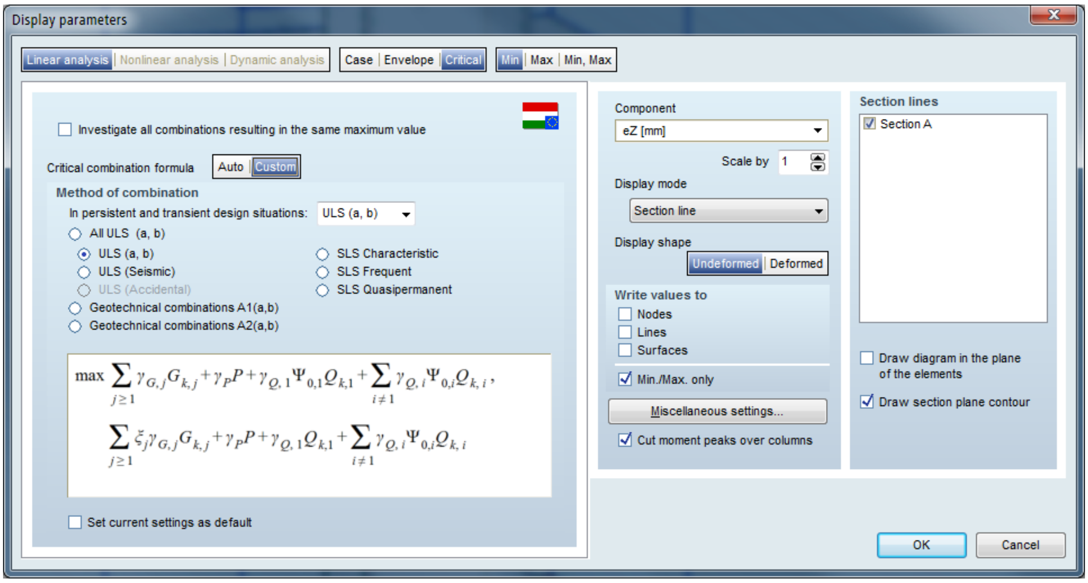


Start a linear static analysis **See...** *5.1 Static*

Start a nonlinear static analysis **See...** *5.1 Static*

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.

Display Parameters dialog shows the following options.



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.

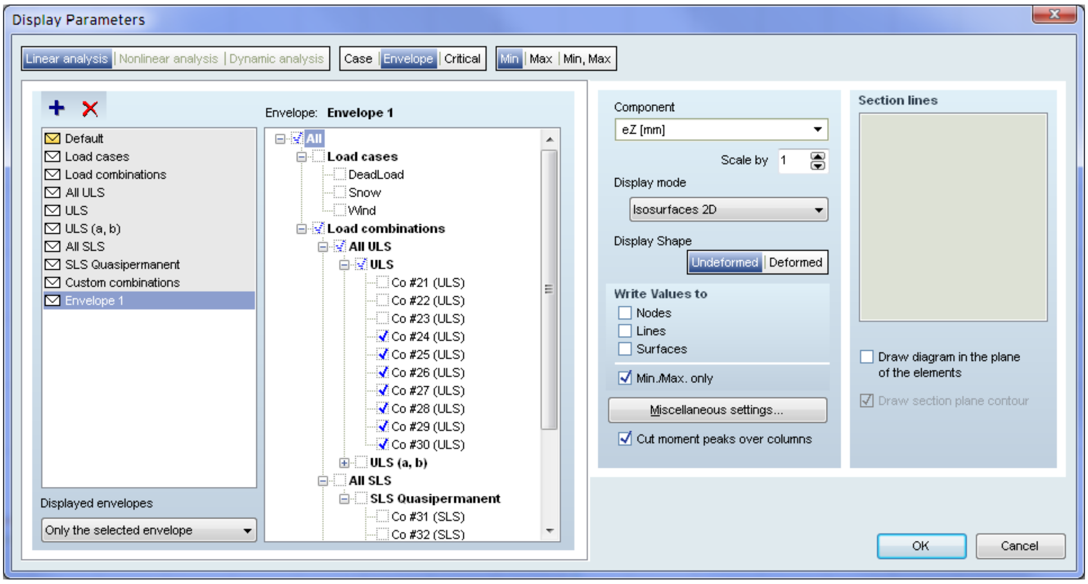
Envelope

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



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

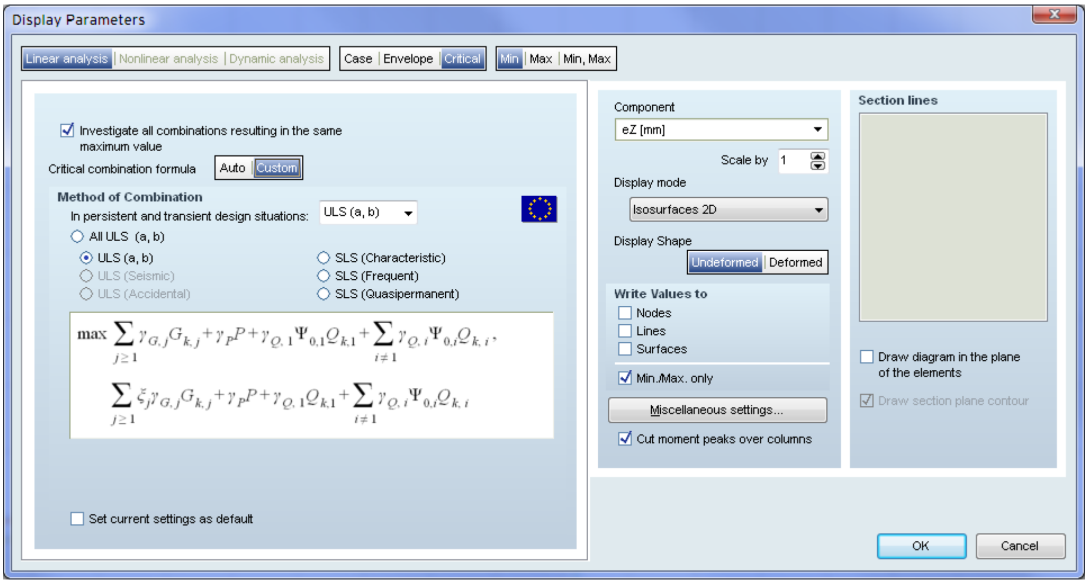
Delete a custom envelope (only custom envelopes can be deleted)

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.

Critical



Investigate all combinations resulting in the same maximum value

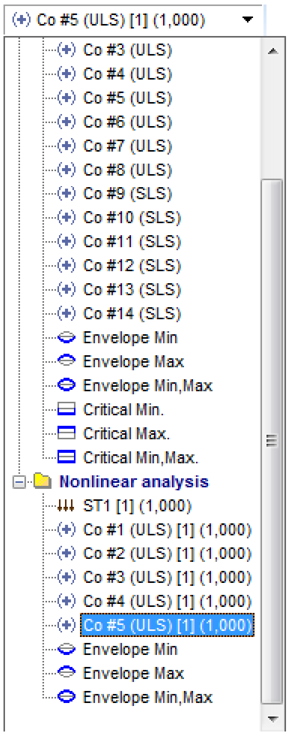
By default this option is off. AxisVM takes into account combinations resulting in an extreme for any result component. In certain design methods however a combination which produces no extremes can be more unfavorable.
In this case turn this option on. In design calculations AxisVM will build all possible 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, DIN 1045-1, SIA 262 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).



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.

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.

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 continuos 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.

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...** *2.18.4 Color legend*

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... *2.16.15 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.

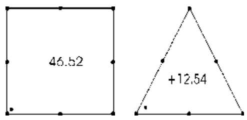
Write Values to ... *Nodes*

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.

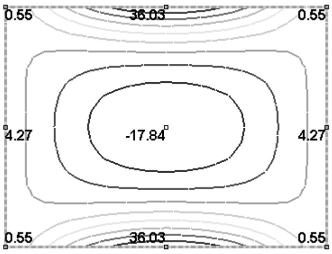
All surfaces



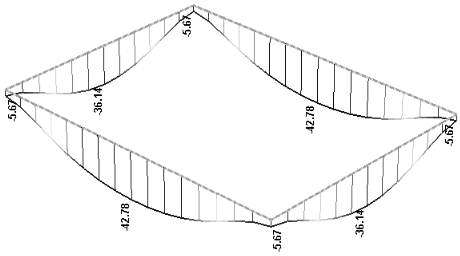
Writes the values of the current result component to the surface elements. The maximum absolute value of the nine values computed at the nodes of each surface is displayed, and the respective node is marked by a small black circle.

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

After clicking the *Miscellaneous Settings...* button the following options are available:

Result Smoothing Parameters

None

The values of the internal forces of the surface elements computed at the nodes are not averaged.

Selective

The values of the internal force components of the surface elements computed at the nodes are averaged in a selective way, depending on the local coordinate systems, the support conditions and the loads of the elements that are attached to a node.

All

The values of all internal force components of the surface elements computed at the nodes are averaged.

Intensity Reference Value

Lets you display the variation of the current internal force component within the surface elements in a filled color contour plot form. The numerical values are displayed if a *Show value labels on* option is enabled.

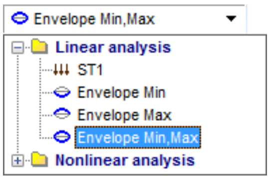
See... *6.1.10 Surface element internal forces*

Cut moment peaks over columns

If the mesh on a domain was created with the option *Adjust mesh to column heads* moment peaks can be averaged and cut over column heads by activating this option.

See... *6.1.10 Surface element internal forces*

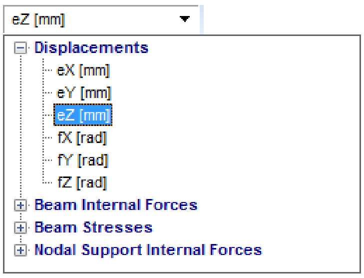
Case selector to display



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

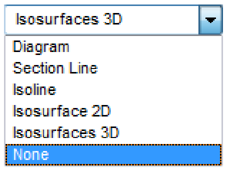
Available result component



You can select a result component from the drop-down list for display:

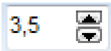
Displacement (eX, eY, eZ fX, fY, fZ,eR, fR)
Beam/rib internal force (Nx, Vy, Vz, Tx, My, Mz)
Beam/rib stress (Smin, Smax,Tymean, Tzmean)
Surface internal force (nx, ny, mx, my, mxy, vxz, vyz, vSz, n1, n2, an, m1, m2, am, nxD, nyD, mxD, myD)
Surface stress (Sxx, Syy, Sxy, Sxz, Syz, Svm, S1, S2)
Nodal support force (Rx, Ry, Rz, Rxx, Ryy, Rzz)
Line support force (Rx, Ry, Rz, Rxx, Ryy, Rzz)
Surface support force (Rx, Ry, Rz)
Spring internal force (Rx, Ry, Rz, Rxx, Ryy, Rzz)
Gap internal force (Nx)

Display mode *You can select a display mode from the drop-down list:*



☞ ***If Min,Max envelope or critical load combination is selected, the Isoline and Isosurface 2D cannot be selected.***

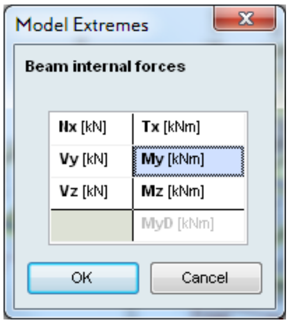
Display scaling factor



Lets you scale the display of the diagrams.

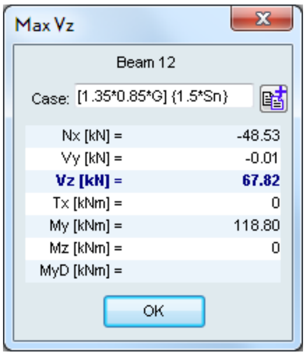
6.1.1. Minimum and maximum values

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.

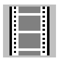


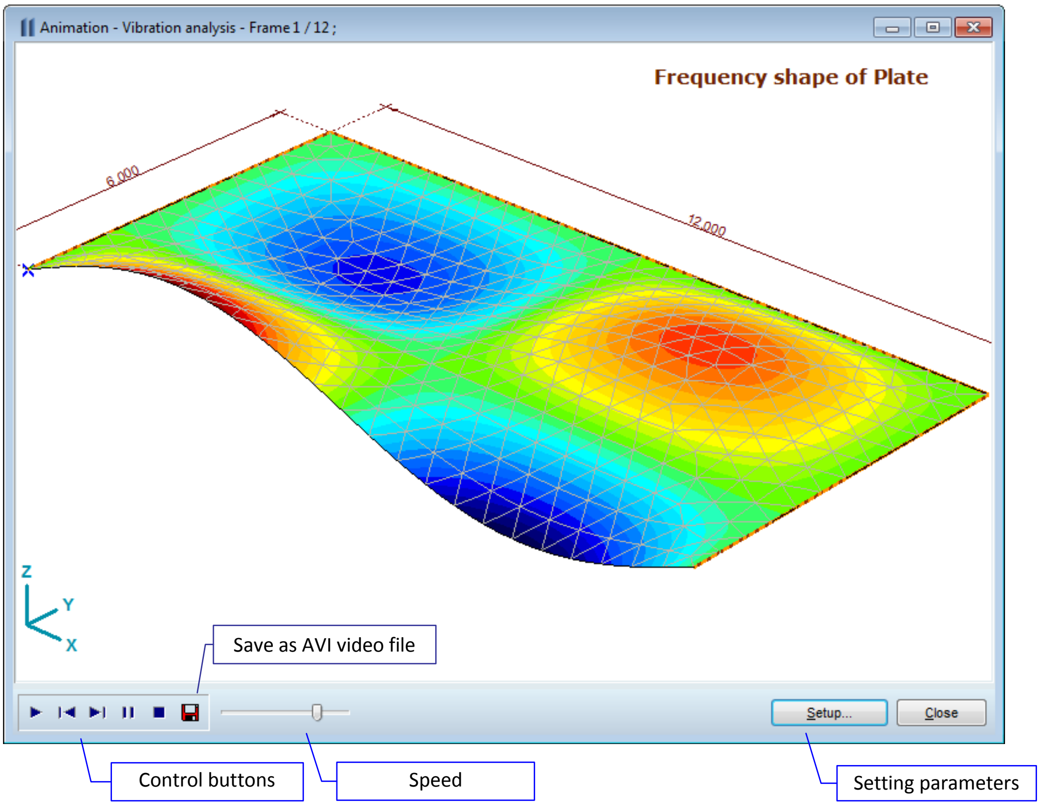
☞ ***If parts are displayed extreme values are determined from the displayed parts only.***

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*.



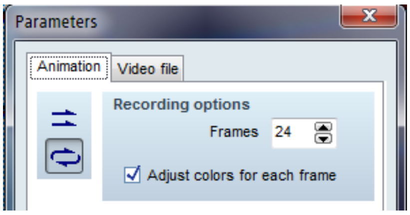
6.1.2. Animation

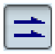




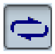
Lets you display the displacements, internal forces, and mode shapes in animated form (frame by frame). The animation consists of a sequence of frames that are generated by linear interpolation between initial values (frame 0) and the actual values of the current result component (frame *n*), according to the number of frames (*n*).

Animation





Unidirectional play
Plays the frames starting from frame zero and ending with frame *n*.



Bi-directional play
Plays the frames starting from frame zero and ending with frame *n* and then the reverse.

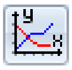
Frames
Lets you set the number of animation frames. You must specify a value between 3 and 99. More frames produce smoother but slower animation.

Adjust color for each frame
If frames are made from an iso-line/surface display and this option is selected the color range is recalculated for each frame.

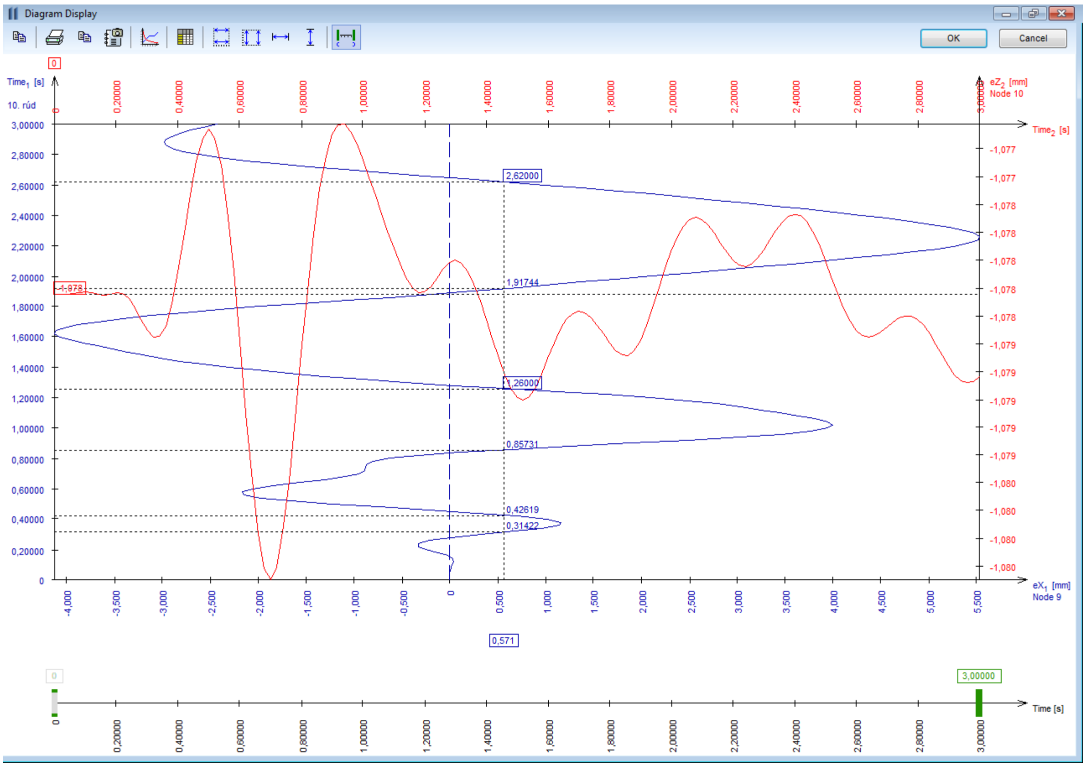
Video File You can create a video file, *name.avi*.
Click Save button to save the parameters of the video file.

You can set the duration of displaying a frame. Lower duration will result in a bigger number of frames. A number of 30 frames/second is usual, therefore you should not normally enter less than 30 ms for the duration of a frame.

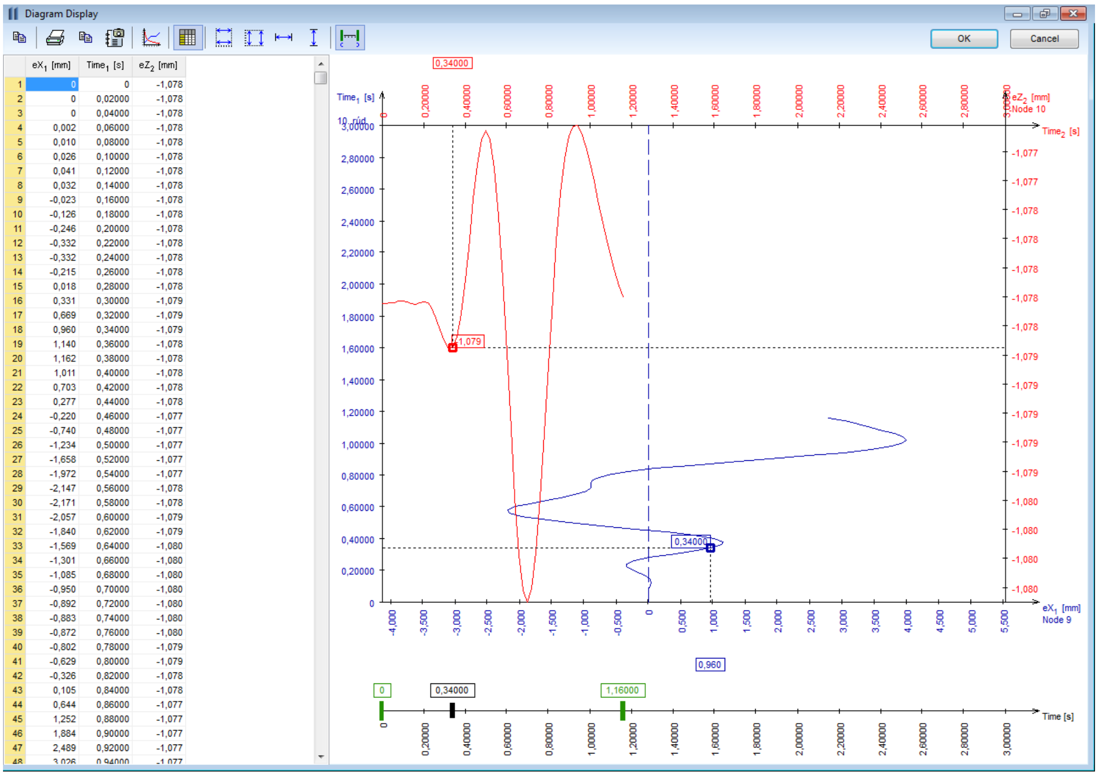
6.1.3. Diagram display



This dialog displays 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 dynamic analysis the bottom trackbar displays time instead of increment numbers.



Toolbar

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.

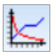


Diagram Display Parameters
Components to be displayed can be selected from combo boxes.
If a result component is selected clicking the *Node* button allows selecting the node where the result is read.
The *x1-y1* diagram is in blue, with ticks and labels on the left and bottom axes.
The *x2-y2* diagram 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.

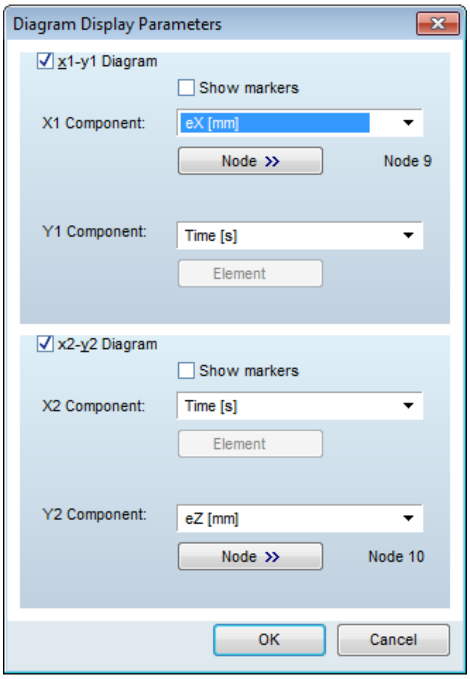
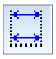
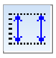


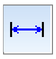
Table
Turn on/off the table displaying numerical values.



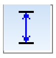
Same range on the two X-axes
If the same X-component is chosen for the two horizontal axes their ranges can be set to the same.



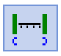
Same range on the two Y-axes
If the same Y-component is chosen for the two vertical axes their ranges can be set to the same.



Fit in view in X-direction
Sets the horizontal range between minimum and maximum of X values.

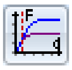


Fit in view in Y-direction
Sets the vertical range between minimum and maximum of Y values.

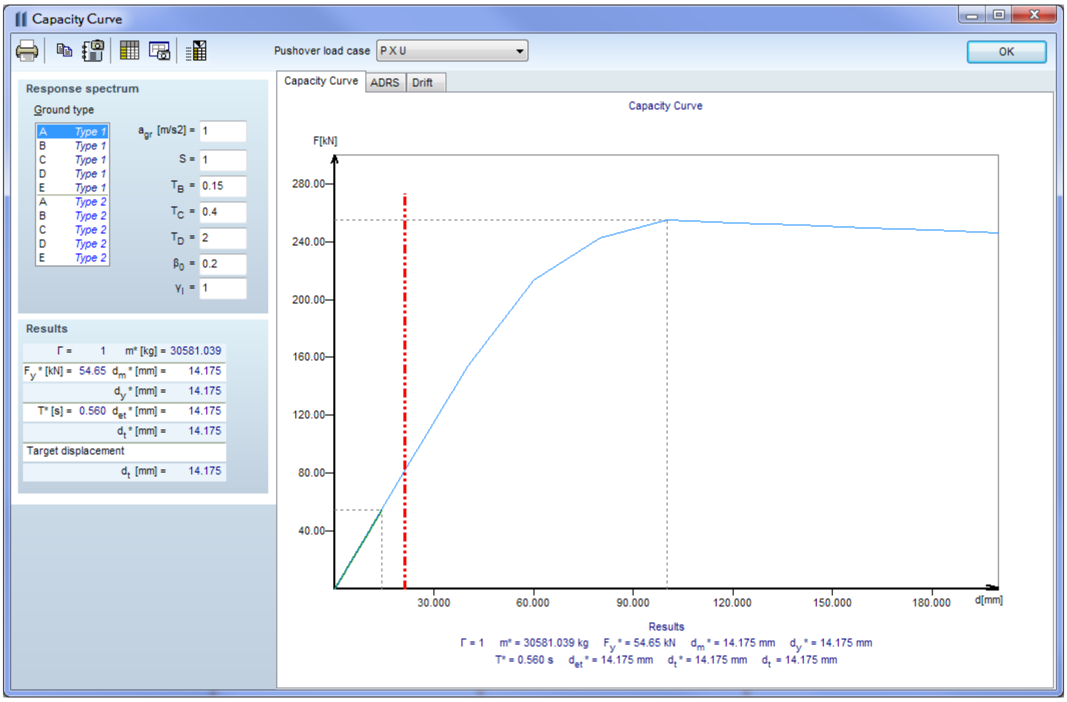


Interval controls
Turns on/off the green interval control rectangles of the bottom trackbar. Dragging them changes the displayed range of increments or time.

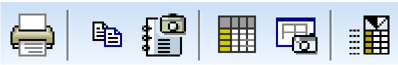
6.1.4. Pushover capacity curves



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 analyzed. 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...** *4.10.23 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 for the shear force can only be determined by running another analysis limited by a larger displacement and checking if the curve reached a maximum after which the base shear started to decrease. If so, then the maximum value is at the maximum of the curve. If no maximum has been reached, the displacement has to be increased even further if necessary.



Toolbar




Print drawing
Prints the current diagram


Copy to clipboard
Copies the current diagram to the Clipboard.

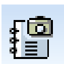
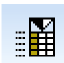

Add drawing to Gallery
Saves the current diagram to the Gallery to make it available for reports.


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 stories and other parameters.

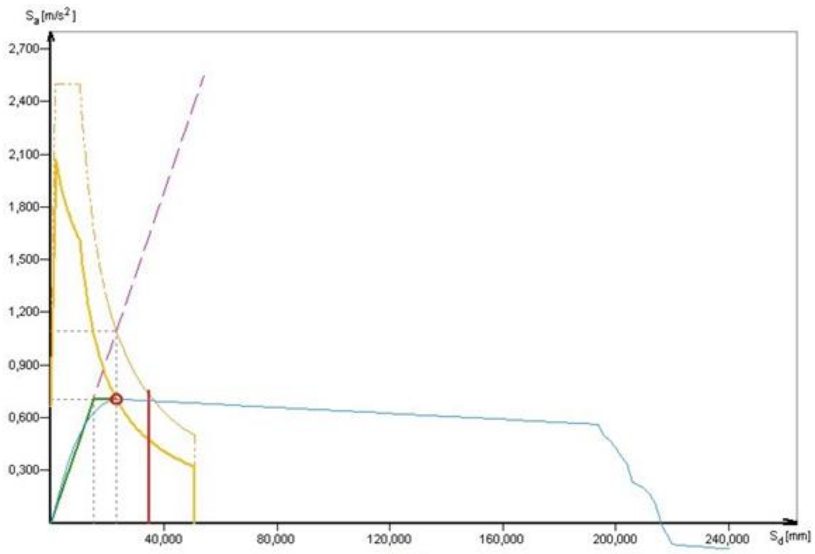
6.1.4.1. Capacity curves according to Eurocode 8

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 fulfills the deformation capability requirements, otherwise (the lines style is continuous) it fails these requirements.

6.1.4.2. 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.



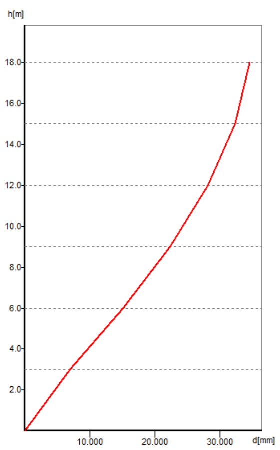
Results The variables marked by an asterisk (*) represent the SDOF system's behavior, while the others correspond to the MDOF system.

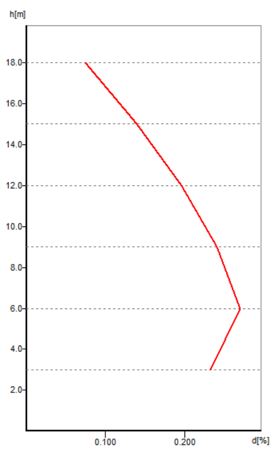
Γ	transformation factor for computing SDOF characteristics
m^*	mass of equivalent SDOF system
F_y^*	base shear force at d_m^* displacement of the equivalent SDOF system and yield force of the elasto-perfectly plastic force-displacement relationship
d_m^*	ultimate displacement of the idealized bilinear 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

6.1.4.3. Drift

On the *Drift* tab the diagrams of absolute and relative story displacement (interstory drift) are displayed. The diagram of absolute drift shows the horizontal displacement of the centre of gravity of stories relative to the soil. The relative diagram shows the interstory drift expressed as a percentage of the story 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 stories.





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

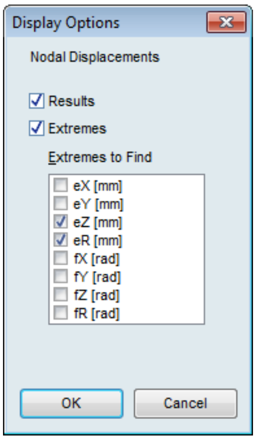
6.1.5. Result tables

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.*

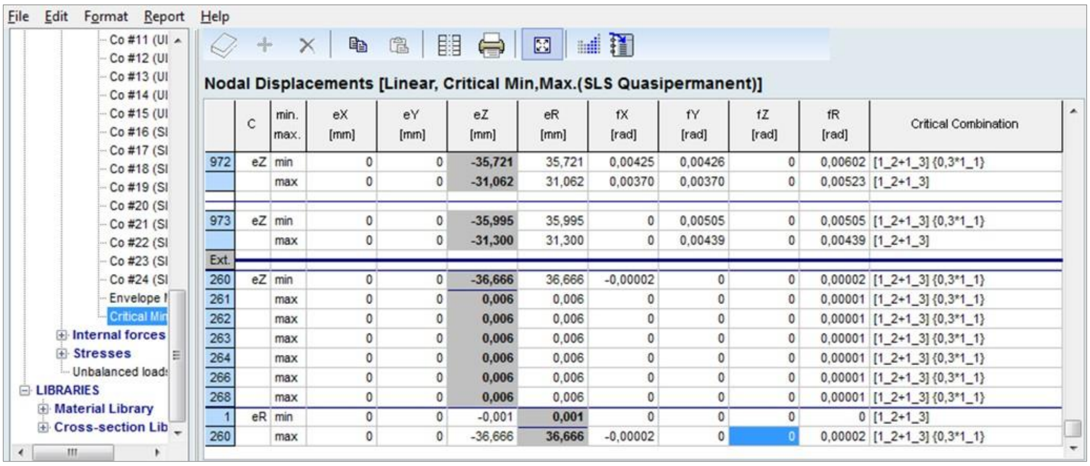
Results Unchecking this option removes the detailed results leaving the extremes as the only content of the table.

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:

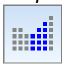
[...]   represents the results of a permanent load case.
{ ... }   represents the results of an incidental load case.
(...)   represents the results of an exceptional load case.



Nodal Displacements [Linear, Critical Min,Max.(SLS Quasipermanent)]

	C	min. max.	eX [mm]	eY [mm]	eZ [mm]	eR [mm]	fX [rad]	fY [rad]	fZ [rad]	fR [rad]	Critical Combination
972	eZ	min	0	0	**-35,721**	35,721	0,00425	0,00426	0	0,00602	[1_2+1_3] {0,3*1_1}
		max	0	0	**-31,062**	31,062	0,00370	0,00370	0	0,00523	[1_2+1_3]
973	eZ	min	0	0	**-35,995**	35,995	0	0,00505	0	0,00505	[1_2+1_3] {0,3*1_1}
		max	0	0	**-31,300**	31,300	0	0,00439	0	0,00439	[1_2+1_3]
Ext.											
260	eZ	min	0	0	**-36,666**	36,666	-0,00002	0	0	0,00002	[1_2+1_3] {0,3*1_1}
261		max	0	0	**0,006**	0,006	0	0	0	0,00001	[1_2+1_3] {0,3*1_1}
262		max	0	0	**0,006**	0,006	0	0	0	0,00001	[1_2+1_3] {0,3*1_1}
263		max	0	0	**0,006**	0,006	0	0	0	0,00001	[1_2+1_3] {0,3*1_1}
264		max	0	0	**0,006**	0,006	0	0	0	0,00001	[1_2+1_3] {0,3*1_1}
266		max	0	0	**0,006**	0,006	0	0	0	0,00001	[1_2+1_3] {0,3*1_1}
268		max	0	0	**0,006**	0,006	0	0	0	0,00001	[1_2+1_3] {0,3*1_1}
1	eR	min	0	0	-0,001	**0,001**	0	0	0	0	[1_2+1_3]
260		max	0	0	-36,666	**36,666**	-0,00002	0	0	0,00002	[1_2+1_3] {0,3*1_1}

Property Filtering



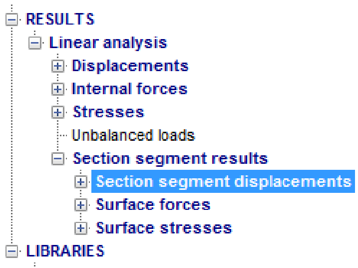
See in detail... *2.9 Table Browser*

Print
[Ctrl]+[P]

Clicking the Print tool button or choosing the *File / Print* menu item the print dialog appears.
See... *3.1.10 Print*.

6.1.5.1. Section segment result tables

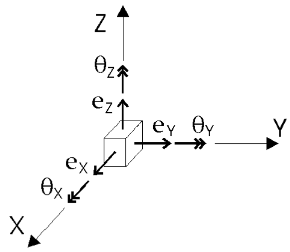
If section segments are defined in the model section segment result tables appear under the RESULTS node.
These tables list the values of result components along the active (displayed) section segment. Internal segment result points will be created where the segment plane intersected the finite element edges.



6.1.6. Displacements

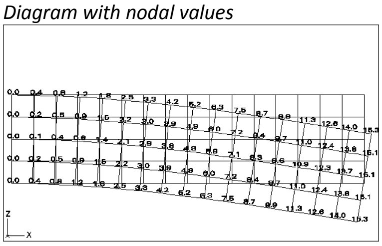
Nodes

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.

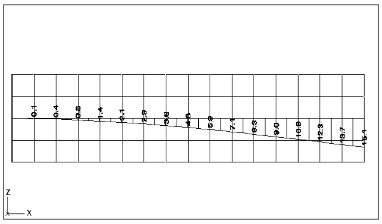


Displaying the displacements of a cantilever (membrane model):

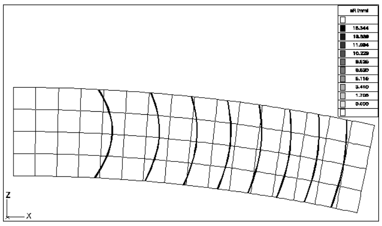
Diagram with nodal values

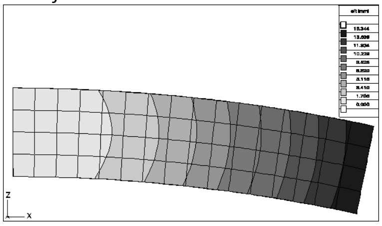


Section line with nodal values



Isolines *Isosurfaces 2D*

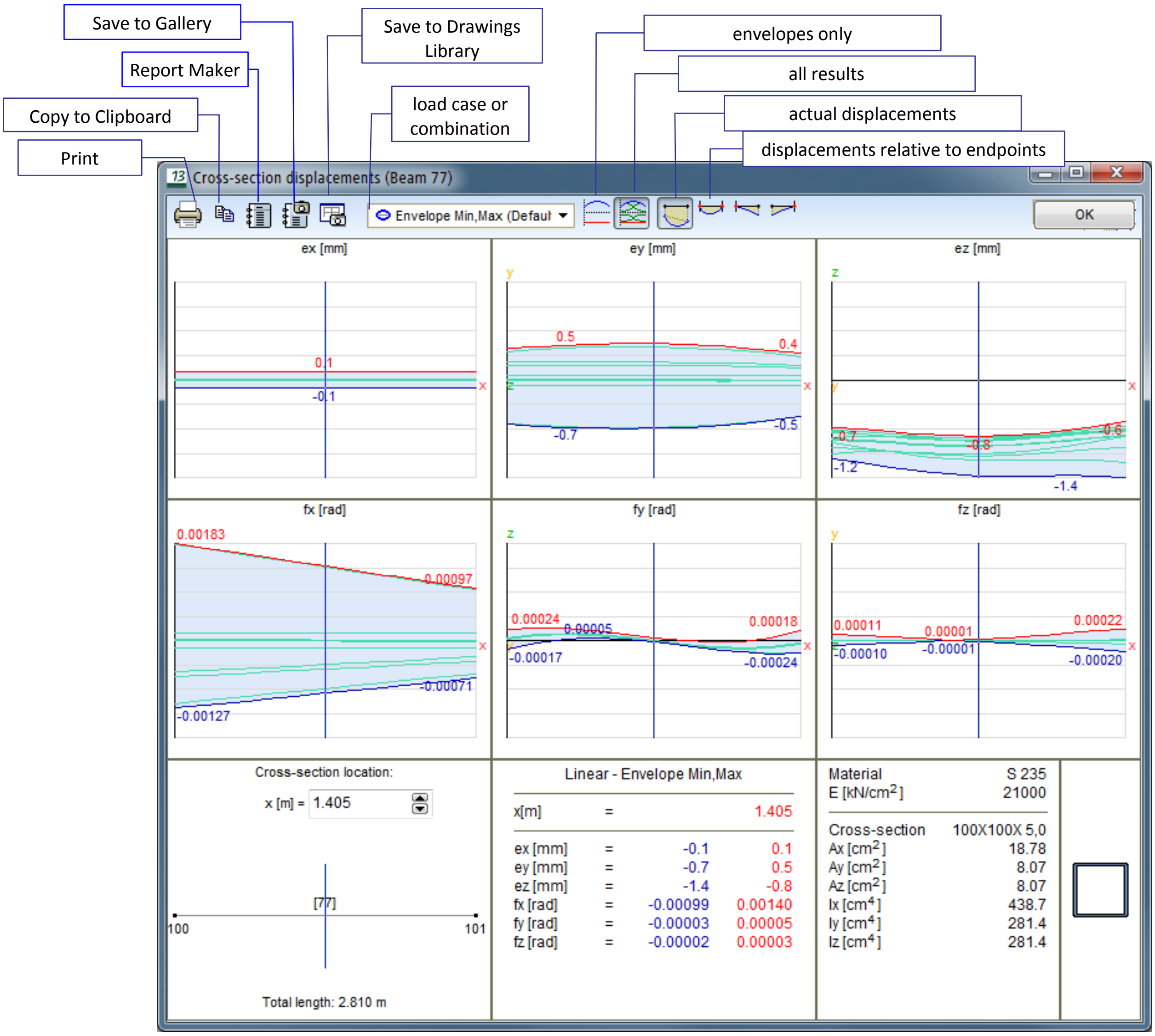


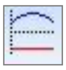


Beams For each beam element the intermediate displacements are obtained in the local and global coordinate systems. When displaying the displacements of the structure the beam displacements are related to the **global** coordinate system. If you pick the cursor on a beam element the six beam displacement components related to the element **local** coordinate system are displayed in a diagram form.

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... *2.16.19.3 Drawing*/ Contour line angle
b) The local x orientation is the same.
c) The elements have the same material



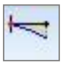


Display the envelope only

Display results of all load cases / combinations plus the envelope

Actual displacements

Displacements relative to endpoints



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

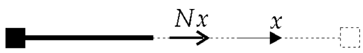
See... *6.1.5 Result tables*

6.1.7. Truss/beam internal forces

Truss

Axial internal forces (*Nx*) are calculated for each truss element.

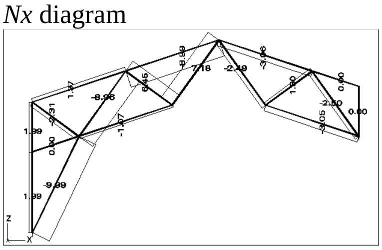
A positive axial force corresponds to tension, a negative axial force corresponds to compression.



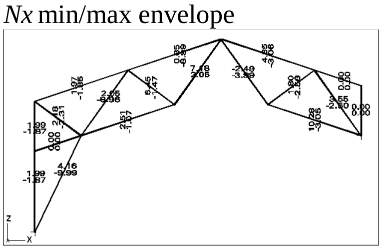
When displaying the *Envelope* and *Critical* results the minimum and maximum values can concomitantly be displayed.

Displaying the internal forces of a truss girder:

Nx diagram

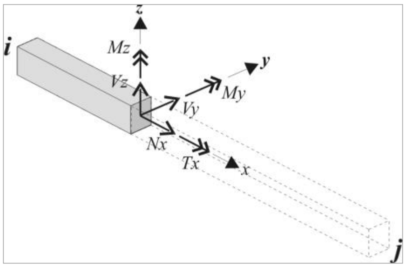


Nx min/max envelope



Beam

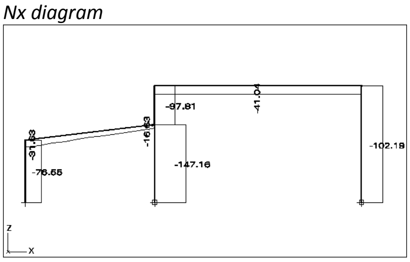
Three orthogonal internal forces, one axial and two shear forces (*Nx* , *Vy* , *Vz*) and three internal moments, one torsional and two flexural *(Tx, My, Mz)* are calculated at the intermediate cross-sections of each element.



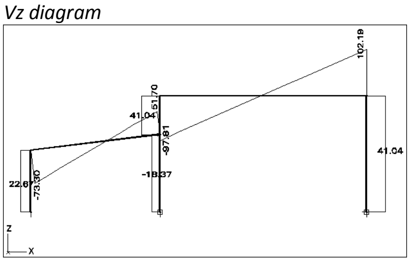
The internal forces are related to the element local coordinate system, and the positive sign conventions apply as in the figure above. The moment diagrams are drawn on the tension side of the beam elements.

Displaying the internal forces of a frame:

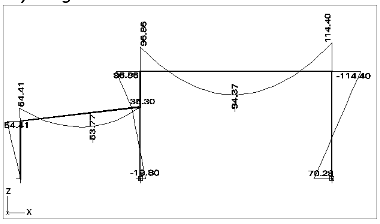
Nx diagram



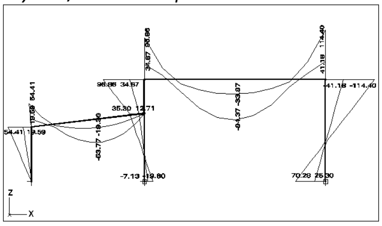
Vz diagram



My diagram



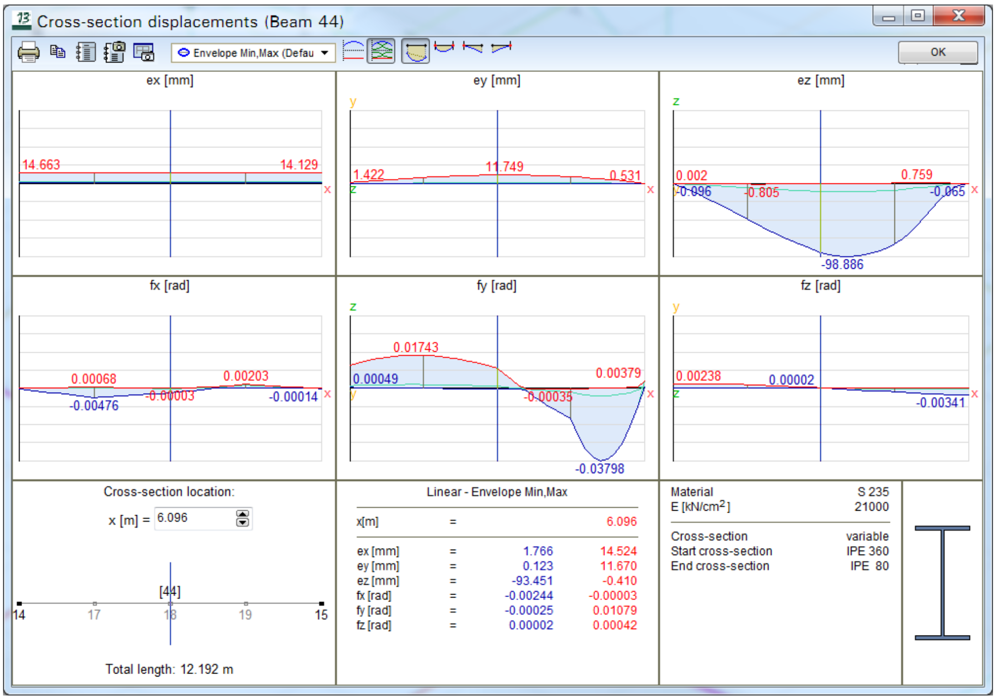
My min/max envelope



If you click a beam element all six beam internal force components are displayed in a diagram form.

You can display internal forces of more than one beam element if:

a) The local coordinate system of the elements are almost or entirely identical.
See... *2.16.19.3 Drawing*/ Contour line angle
b) The local x orientation is the same.
c) The elements have the same material.



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.

Result tables Three different types of table are available.

Beam internal forces:	displays internal forces along the beam
Beam end internal forces:	displays internal forces only in start and endpoint of the beams
Forces for connection design:	lists internal forces of connecting ribs / beams / trusses per node

See… 6.1.5 Result tables

Save diagrams to the Drawings Library

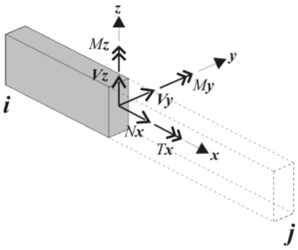
Associative diagrams can be saved to the Drawings Library. Drawings from this library can be inserted into reports. After changing and recalculating the model diagrams in the library and reports change accordingly.

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... *6.1.5 Result tables*

6.1.8. 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.

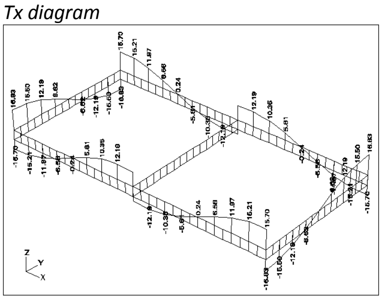


The internal forces are related to the element local coordinate system positioned in the center of gravity of the cross-section, and the positive sign conventions apply as in the figure below. The moment diagrams are drawn on the tension side of the beam elements.

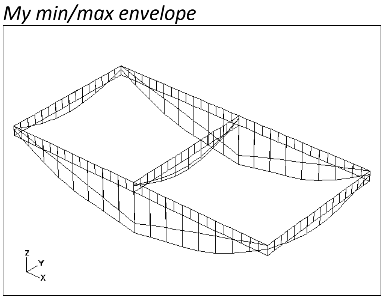
If the rib is connected eccentrically to a shell element, axial forces will appear in the rib and in the shell. In this case the design moment can be calculated as follows: $M_{yD} = M_y + e_z \cdot N_x$

Displaying the internal forces of a ribbed plate:

Tx diagram



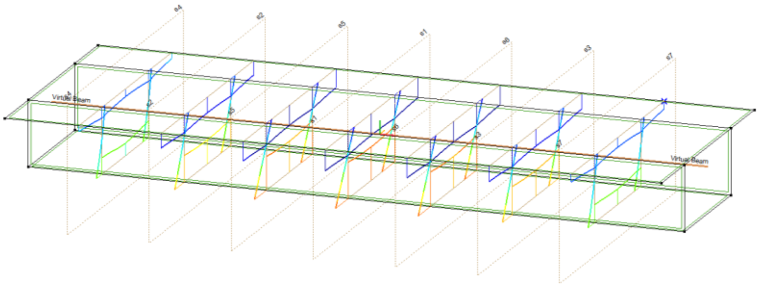
My min/max envelope

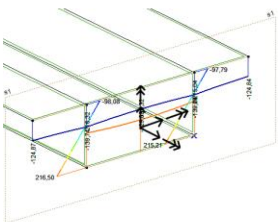


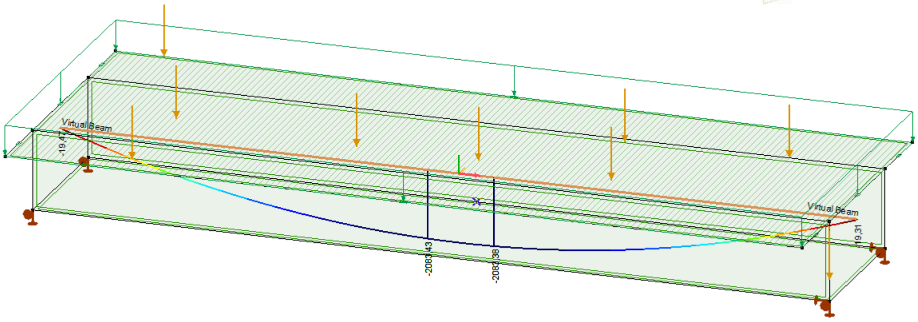
Result Tables **See...** *6.1.5 Result tables*

6.1.9. Virtual beam internal forces

Calculation After defining virtual beams (**see…** *2.16.16 Virtual beams*), the program calculates center of gravity in each section to determine the centre line, on which a finite number of section 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.

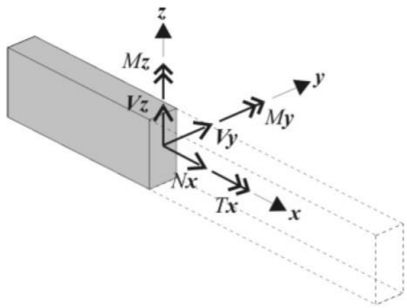






Components of internal forces

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.



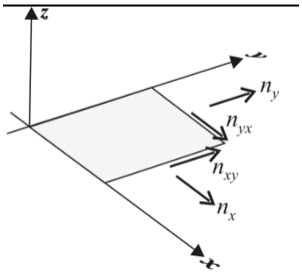
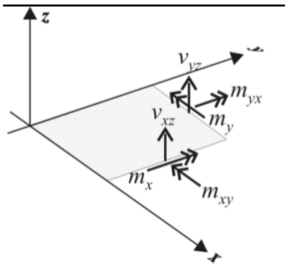
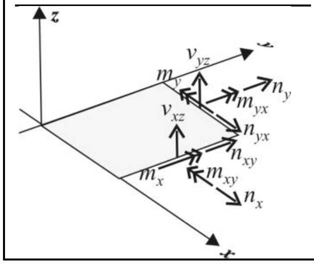
Result tables

The results are listed in all sections, with the global coordinates of the center of gravity.

6.1.10. Surface element internal forces

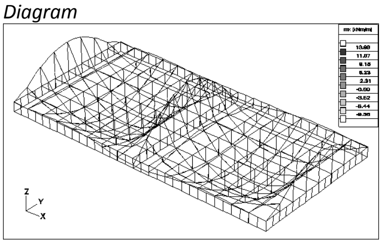
Internal forces

The internal forces and the positive sign conventions of each surface element type are summarized in the table below.

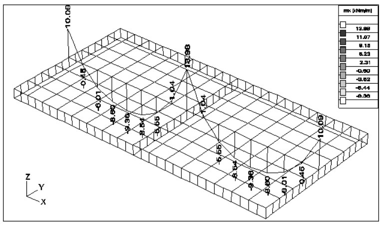
Surface elements			
Membrane 	n_x n_y n_{xy}	*Plate* 	m_x m_y m_{xy} v_{xz} v_{yz}
Shell 	n_x n_y n_{xy} m_x m_y m_{xy} v_{xz} v_{yz}		

Displaying the internal forces of a ribbed plate:

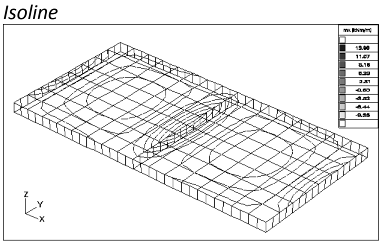
Diagram



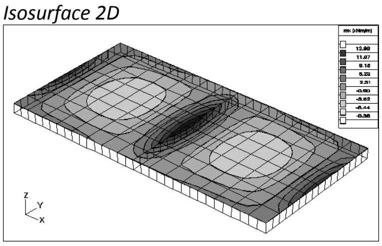
Section line



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.

Cut moment peaks over columns

If we model columns connecting to slabs as nodal supports, moment peaks 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 moment peaks can be averaged. If we checked the option *Adjust mesh to column heads* (*4.11.1.2 Meshing of domains*), the mesh already follows the column cross-section. After turning on *Cut moment peaks over columns* on the *Display parameters* dialog (*6.1 Static*), moment diagrams will be displayed in Isosurfaces 3D mode like the right diagram below.

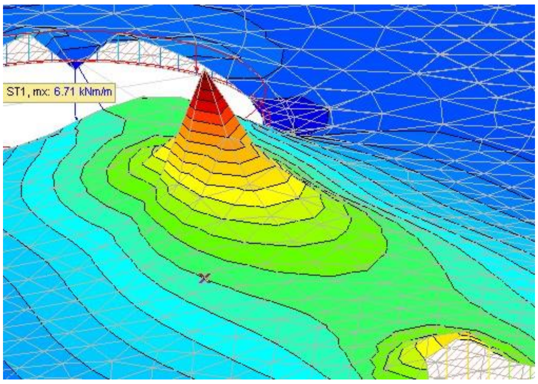


Diagram without cutting moment peaks

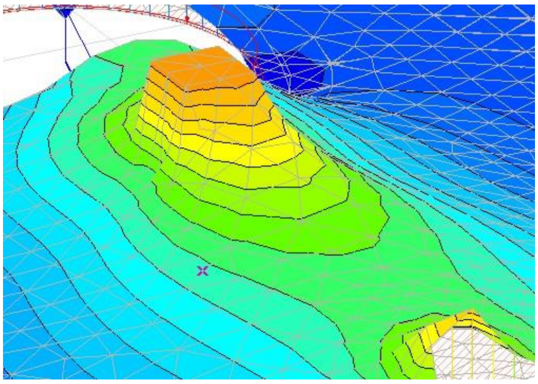


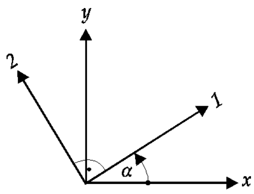
Diagram with moment peaks cut

Result Tables

See... *6.1.5 Result tables*

Principal forces

The *n1, n2, αn, m1, m2, αm* principal internal forces and the *qR* resultant shear forces are computed. 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)

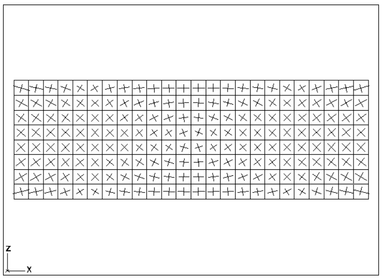
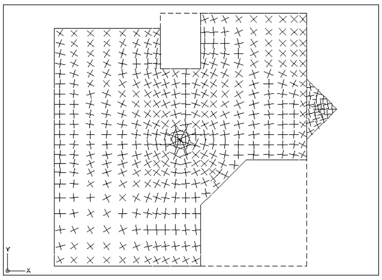


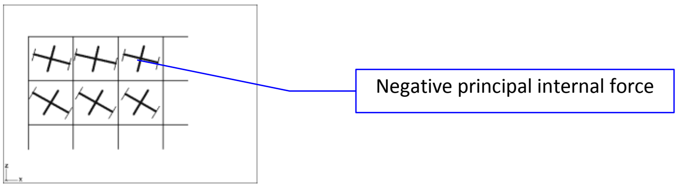
	Shell	
	Membrane	*Plate*
n_1	$n_1 = \frac{n_x + n_y}{2} + \sqrt{\left(\frac{n_x - n_y}{2}\right)^2 + n_{xy}^2}$	-
n_2	$n_1 = \frac{n_x + n_y}{2} - \sqrt{\left(\frac{n_x - n_y}{2}\right)^2 + n_{xy}^2}$	-
α_n	$\mathrm{tg}(2\alpha_n) = \frac{2n_{xy}}{n_x - n_y}$	-

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	-	$\mathrm{tg}(2\alpha_m) = \frac{2m_{xy}}{m_x - m_y}$
vSz	-	$vSz = \sqrt{v_{xz}^2 + v_{yz}^2}$

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 forms.
The principal directions *(αn, αm)* can be displayed only in diagram form.
The direction vector color and size 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...** *6.1.5 Result tables*

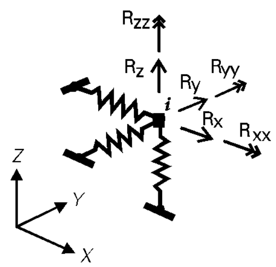
Reinforcement forces

For surface elements nxv, nyv, mxv, myv reinforcement (design) forces and moments are also calculated according to the following rules:

$$n_{xD} = n_x \pm |n_{xy}|, \qquad n_{yD} = n_y \pm |n_{xy}|$$
$$m_{xD} = m_x \pm |m_{xy}|, \qquad m_{yD} = m_y \pm |m_{xy}|$$

The reinforcement design forces can be displayed in diagram, section line and iso-line / surface colored form.

6.1.11. Support internal forces



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.

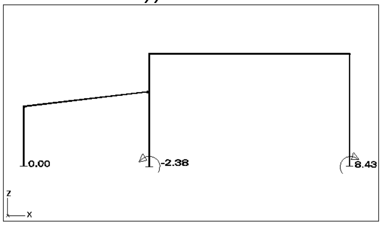
The resultant internal forces $R_{eR}, R_{\Theta R}$ are computed as follows:

$$R_{eR} = \sqrt{R_x^2 + R_y^2 + R_z^2}, \qquad R_{\Theta R} = \sqrt{R_{xx}^2 + R_{yy}^2 + R_{zz}^2}$$

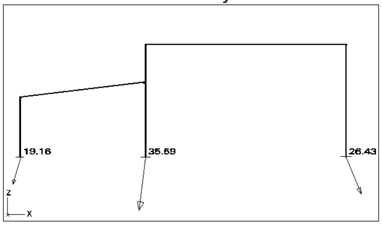
Rxyz and Rxxyyzz result components refer to a special display mode where the individual force or moment components are displayed simultaneously as three arrows pointing in the respective local direction.

Displaying the internal forces of supports in a frame and a shell structure:

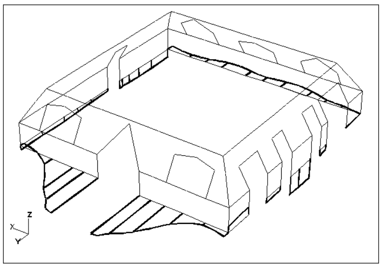
Ryy moments



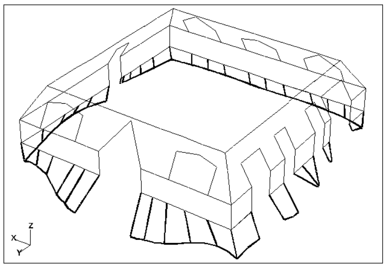
ReR resultant forces



Ry edge forces



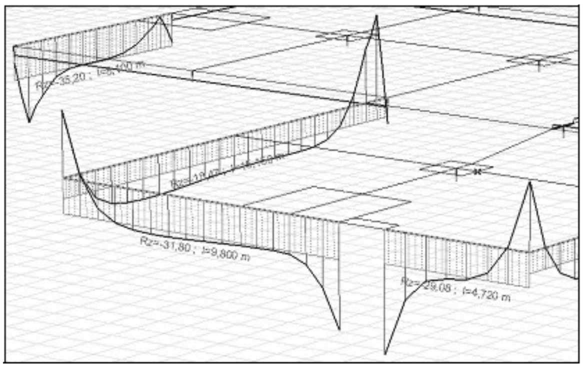
ReR edge resultant forces



Result Tables **See...** 6.1.5 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 continuos 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.



6.1.12. 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 stifness component is set to zero the related result component is zero and not displayed neither in the component combo nor in result tables.

6.1.13. 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*

6.1.14. 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
exz	Shear strain in local *xz* plane
eyz	Shear strain in local *yz* plane
eSz	Resultant shear strain normal to plane of the element

The following strain components are available for the surface elements:

Strain component	Membrane	Plate	Shell
exx	exx		exx
eyy	eyy		eyy
exy	exy		exy
kxx		kxx	kxx
kyy		kyy	kyy
kxy		kxy	kxy
exz		exz	exz
eyz		eyz	eyz
eSz		eSz	eSz

6.1.15. 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.

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 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 $S_{o,i} = \sqrt{S_{xi}^2 + 3V_i^2}$

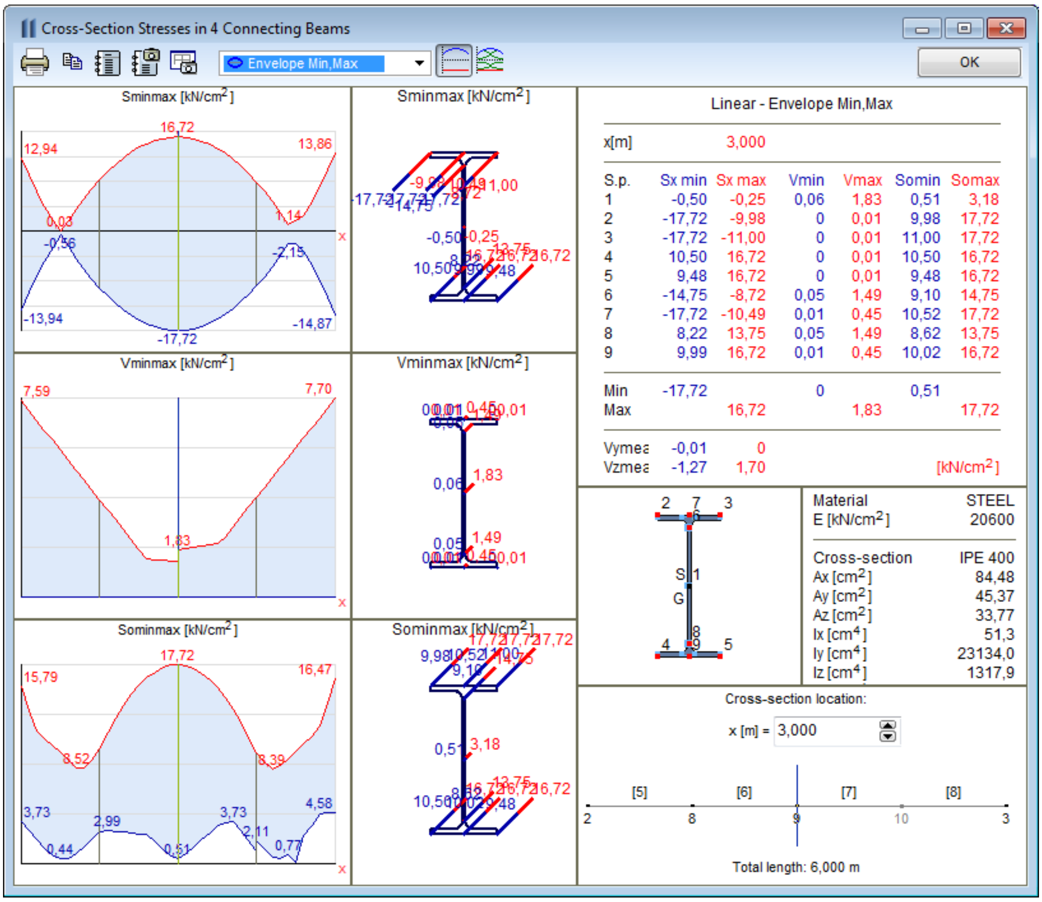
If a cross-section contains two or more separate parts V_i and $S_{o,i}$ is not calculated.

Mean shear stresses: $V_{y,mean} = \frac{V_y}{A_y}$, $V_{z,mean} = \frac{V_z}{A_z}$, if $A_y = A_z = 0$ then $A_y = A_y = A_x$

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 *6.1.7 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... *6.1.5 Result tables*

6.1.16. 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}		$s_{yz} = \frac{3v_{yz}}{2t}$	$s_{yz} = \frac{3v_{yz}}{2t}$

In the case of plane strain membrane elements $s_{zz} \neq 0$, and is determined as $s_{zz} = \nu \cdot (s_{xx} + s_{yy})$

In case of moments the x or y suffix refers to the direction of the section, therefore mx moment will make the plate rotate around the local y direction and my around the local x direction.

Von Mises stress The Von Mises stress is computed:

$$s_0 = \sqrt{0.5\left[(s_{xx} - s_{yy})^2 + (s_{yy} - s_{zz})^2 + (s_{zz} - s_{xx})^2\right] + 3(s_{xy}^2 + s_{yz}^2 + s_{zx}^2)}$$

Stress values can be displayed as a diagram, section diagram, as isolines or isosurfaces.

Result Tables **See...** *6.1.5 Result tables*

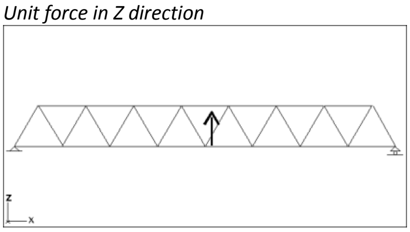
6.1.17. Influence lines

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.

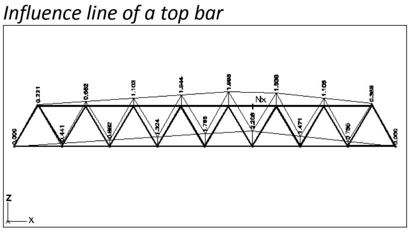
Truss Clicking a truss shows the elements' absolute maximum ordinate value.

Displaying the axial force influence line diagrams of a truss girder:

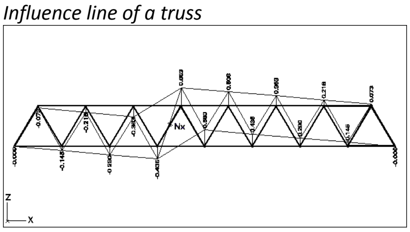
Unit force in Z direction



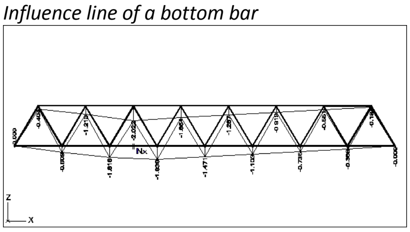
Influence line of a top bar



Influence line of a truss



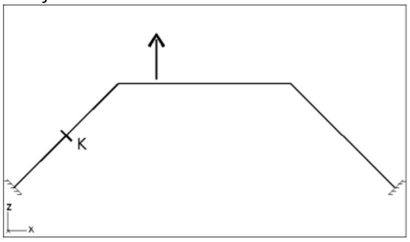
Influence line of a bottom bar



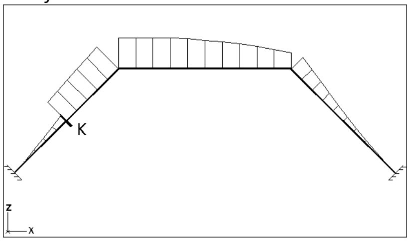
Beam Clicking a beam shows the elements' absolute maximum ordinate value and its location.

Displaying the internal force influence line diagrams of a frame:

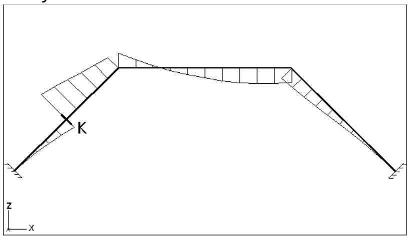
Unit force in Z direction



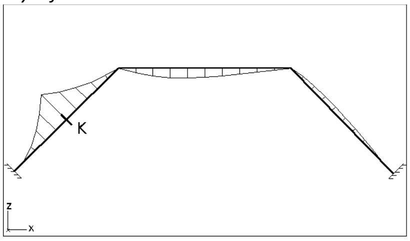
Nx influence line



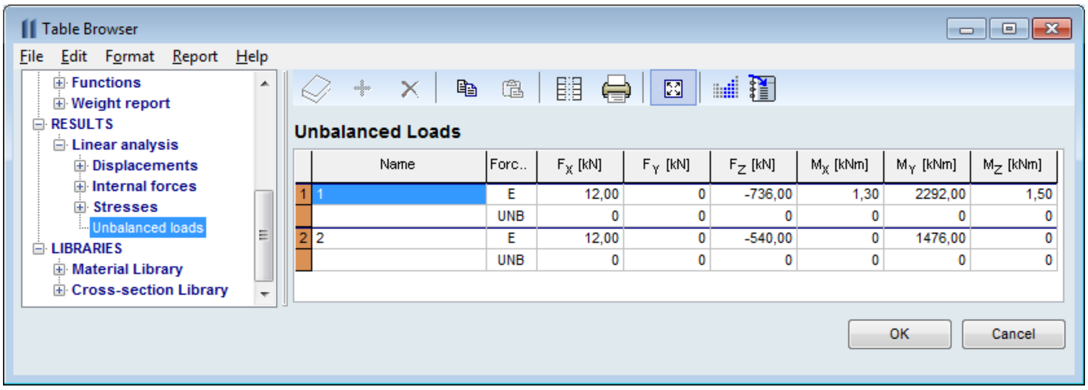
Vz influence line



My influence line



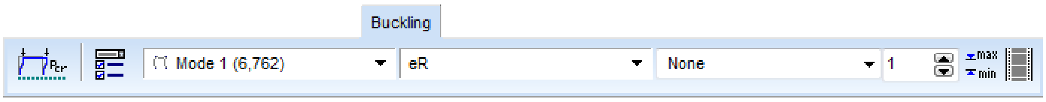
6.1.18. Unbalanced loads



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.***

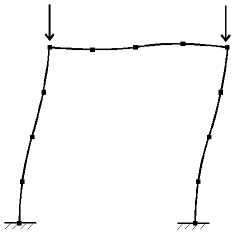
6.2. Buckling



Displays the results of a buckling analysis (buckling mode shapes and critical load parameters).

In the Info Window the following will appear:

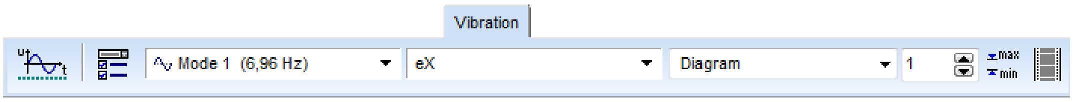
Buckling of a frame:



ncr	the critical load multiplier
Error	relative Error of the eigenvalue
Iteration	the number of iteration performed until convergence was achieved

☞ ***AxisVM stores the buckling analysis results corresponding to each case.***

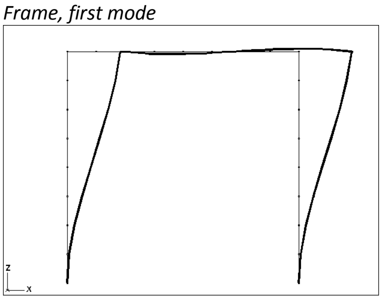
6.3. Vibration



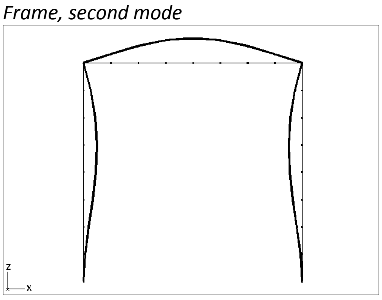
Displays the results of a vibration analysis (mode shapes and frequencies).
You must specify the mode shape number.
The mode shapes are normalized with respect to the mass.

Displaying mode shapes:

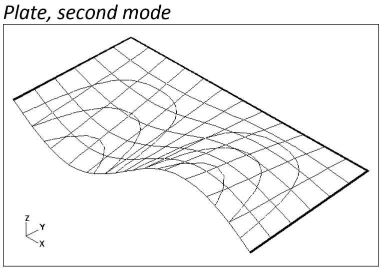
Frame, first mode



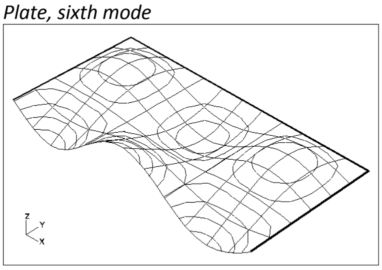
Frame, second mode



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 iteration performed until convergence was achieved

☞ ***AxisVM stores the vibration analysis results corresponding to each case.***

Result table **See...** *6.1.5 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:

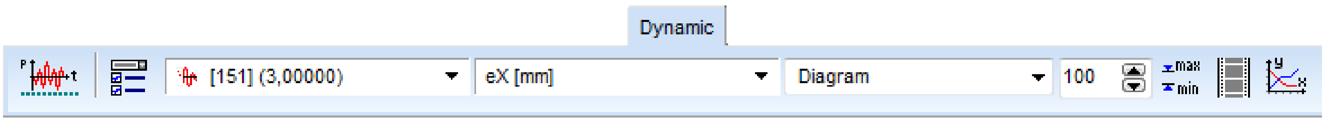
$$M_{mod} = \mu \int_{A} \delta^2(x, y) \cdot dA$$

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 directions 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:

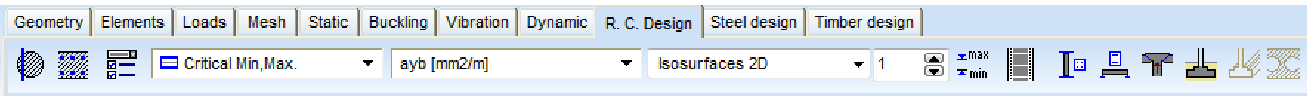
http://www.stb.rwth-aachen.de/projekte/2007/HIVOSS/docs/Guideline_Floors_EN02.pdf

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

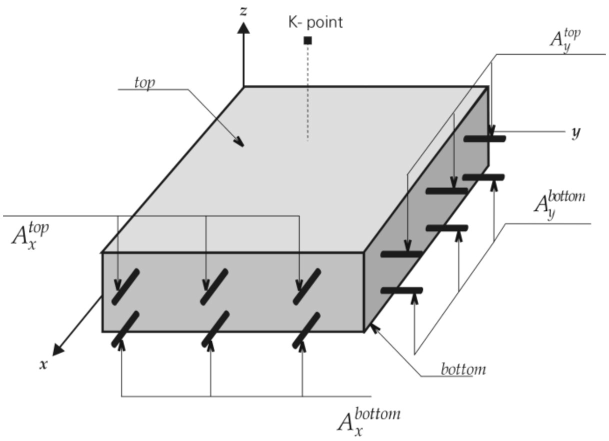


6.5.1. Surface reinforcement – RC1 module

Design Codes

Eurocode 2: *EN 1992-1-1:2004*
DIN: *DIN 1045-1:2001-07*
SIA: *SIA 262:2003*

Surface reinforcement can be calculated based on Eurocode 2. 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.

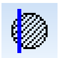


Result components

mxD, myD,	
nxD, nyD:	design forces
axb:	calculated reinforcement area at the bottom in x direction
ayb:	calculated reinforcement area at the bottom in y direction
axt:	calculated reinforcement area at the top in x direction
ayt:	calculated reinforcement area at the top in y direction
xb:	actual (applied) reinforcement at the bottom in x direction
yb:	actual (applied) reinforcement at the bottom in y direction
xt:	actual (applied) reinforcement at the top in x direction
yt:	actual (applied) reinforcement at the top in y direction
xb–axb:	reinforcement difference at the bottom in x direction
yb–ayb:	reinforcement difference at the bottom in y direction
xt–axt:	reinforcement difference at the top in x direction
yt–ayt:	reinforcement difference at the top in y direction
vRd,c:	shear resistance
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

Reinforcement parameters

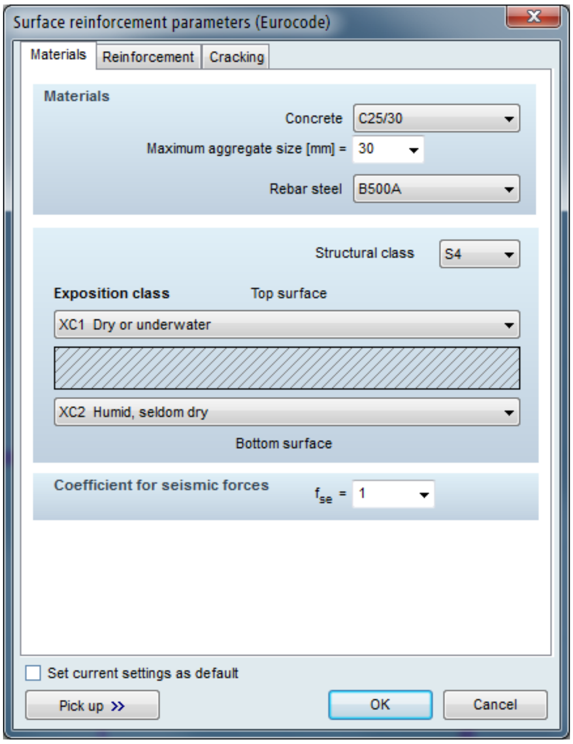
In the surface reinforcement design, the following parameters must be assigned to the finite elements:



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 *4.10.23 Seismic loads – SE1 module.*

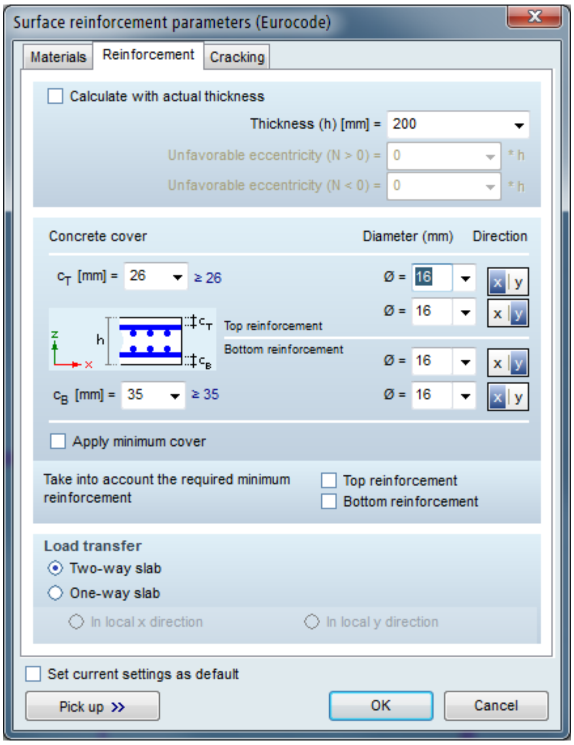


Reinforcement *Calculate with actual thickness :*
If this option is activated the program calculates reinforcement using the actual (constant or variable) thickness.
h is the total thickness used in the calculation, which may differ from the actual plate thickness.

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).

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.



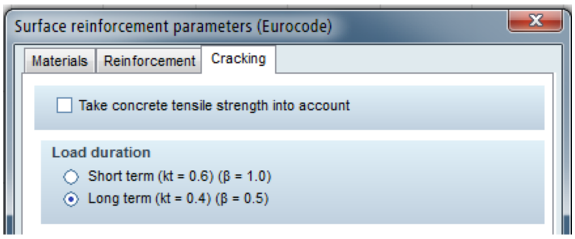
Apply minimum cover The program determines the minimum top and bottom concrete cover from the environment class according to the current design code.

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 value the required minimum is used.

Load transfer AxisVM takes into account the load transfer mode when calculating minimum and maximum rebar spacing according to the design code and the respective national annex.

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.

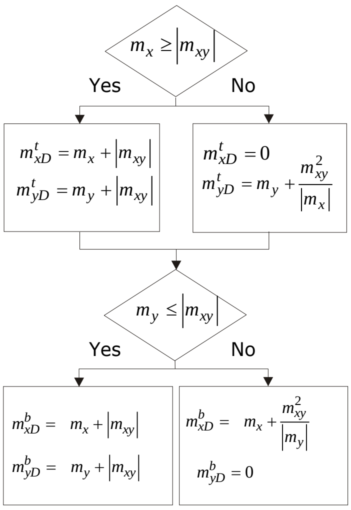
Cracking On this tab the load duration can be set.
This parameter is used in cracking analysis.



Take concrete tensile strength into account If this option is activated the program assumes that the stress in concrete remains below the concrete tensile strength and does not calculate crack width

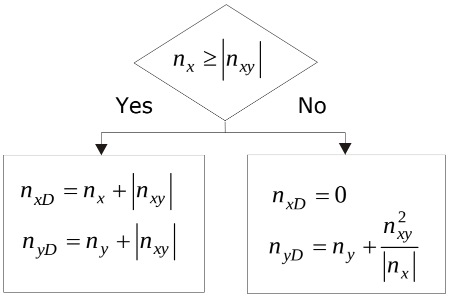
6.5.1.1. Calculation according to Eurocode 2

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: $\substack{\Delta m_2=0 \\ \Delta m_1=\min!}$ $m_x \geq m_y$



Results AxisVM calculates the tension and/or compression reinforcements (for doubly reinforced sections).

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: $\substack{\Delta n_2=0 \\ \Delta n_1=\min!}$ $n_y \geq n_x$



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: *axb, axt, ayb, ayt.*
These represent the calculated *t*op and *b*ottom reinforcement in *x* and *y* directions.
Total reinforcement in *x* direction: *Ax = axb+axt*
Total reinforcement in *y* direction: *Ay = ayb+ayt*

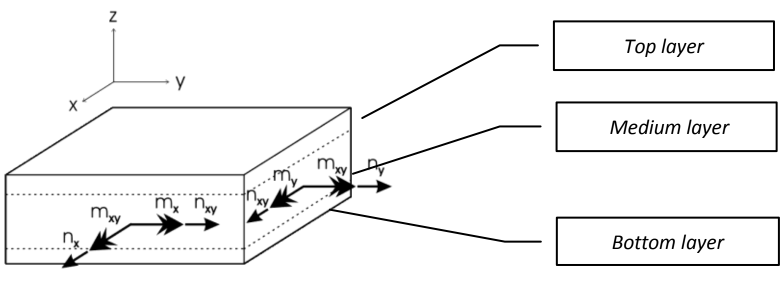
☞ ***The error message 'The section cannot be reinforced' appears if:***

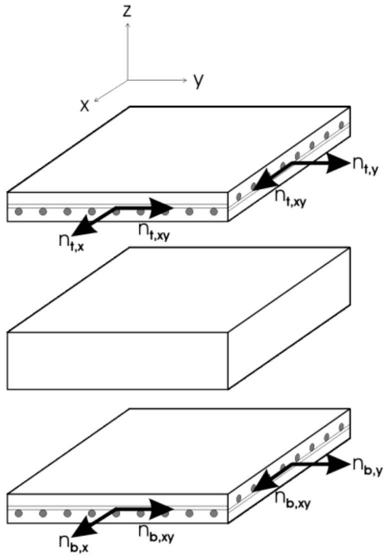
$A_x > 0.04A_c$, or $A_y > 0.04A_c$, where A_c is the concrete cross-section area.

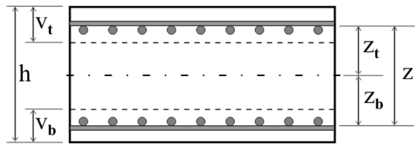
Tables The following symbols are used in tables:
(-) compression reinforcement bar
??? the section cannot be reinforced in the corresponding direction

6.5.1.2. Calculation according to DIN 1045-1 and SIA 262

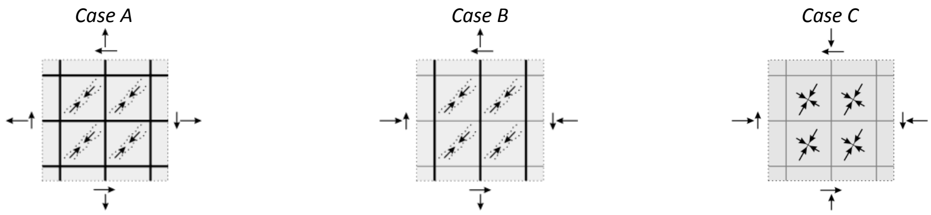
Plate, Membrane, Shell Reinforcement of membranes, plates and shells are calculated according to the three-layer method.
The internal forces ($n_x, n_y, n_{xy}, m_x, m_y, m_{xy}$) are calculated in the perpendicular directions of the reinforcement.
The surface is divided into three layers. Membrane forces 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.



Error message ***The error message 'The section cannot be reinforced' appears.***

If the compressed zone of the concrete fails due shear forces.

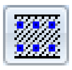
If the compression principal stress is higher than f_{cd}.

$A_x > 0.04A_c$, 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.

6.5.2. Actual reinforcement

Actual Reinforcement

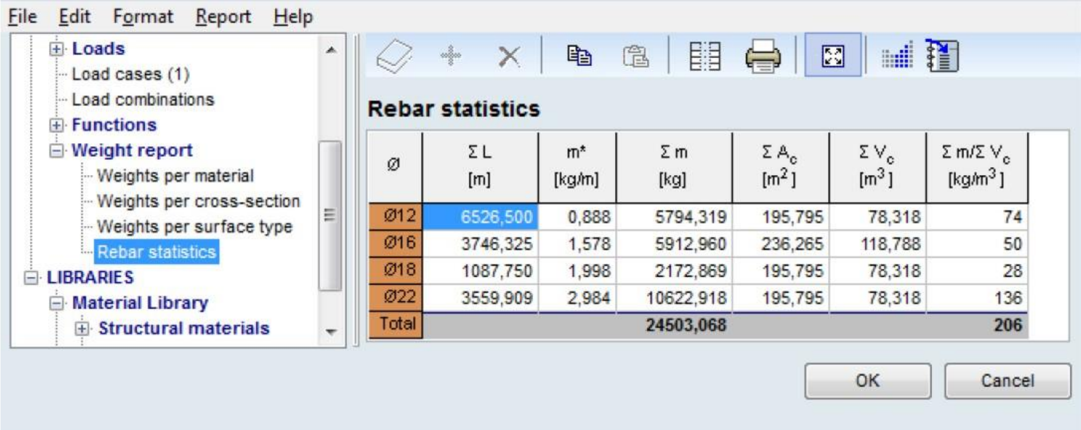


Lets you apply an actual reinforcement to the surface elements depending on the calculated reinforcements.
Using the actual reinforcement you can perform a non-linear plate deflection analysis.

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.

Rebar statistics The actual reinforcement within the model 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.

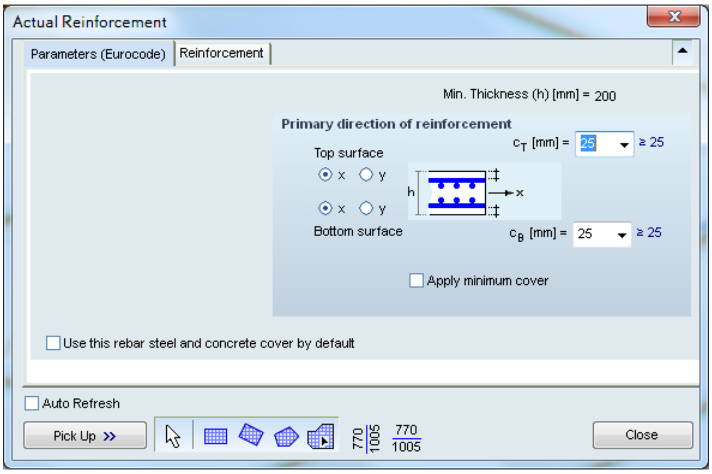


6.5.2.1. Reinforcement for surface elements and domains

Parameters

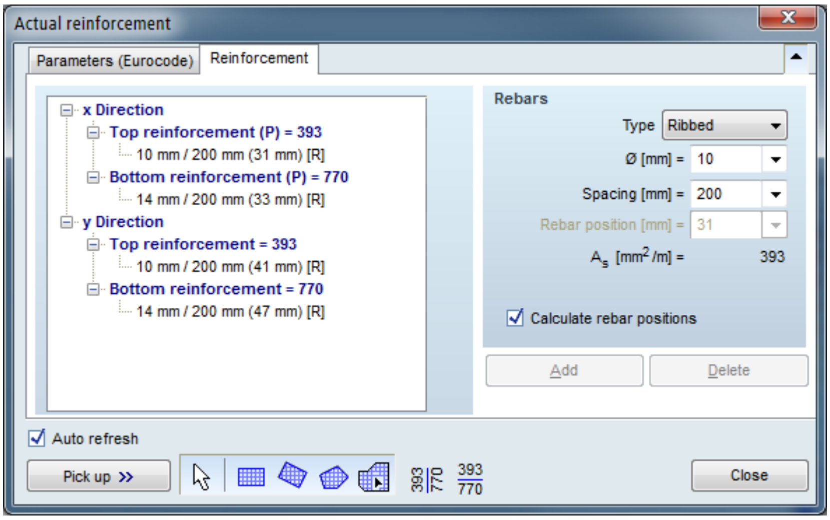
The first tab displays the parameters required by the design code for cracking calculation.

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*)



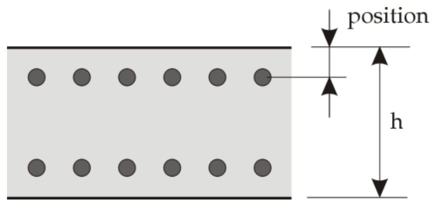
Min. Thickness Min. Thickness displays the minimum thickness entered as surface reinforcement parameter for the selected elements, and not the minimum thickness of the elements.

Reinforcement



The actual reinforcement of the selected surfaces is shown in the tree on the left. Selecting a reinforcement makes its parameters editable on the right. Changing the values updates the tree. *Calculate rebar positions* sets the rebar positions according to the actual concrete cover and primary directions.

☞ ***The position of the rebar is defined as the distance between the side of the concrete and the axis of the rebar.***

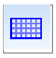


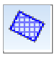
Add and Delete The applied reinforcement is shown in a tree view on the left. By selecting a reinforcement you can change its parameters in the right side. By selecting a location (e.g. x Direction / Top Reinforcement) you can set a new reinforcement on the right side and add it.

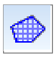
Use the *Delete* button (or **[Del]** key) to delete reinforcement or the *Add* button (or **[Ins]** key) to add reinforcement to a group. If you select a node of the tree view the *Delete* button (or **[Del]** key) will delete all the reinforcements under that node. The *Add* button (or **[Ins]** key) will add reinforcement to the corresponding group.

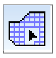
The reinforcement defined in the dialog can be applied to the selected elements or the bottom toolbar can be used to control the way the actual reinforcement is placed on the structure. Mesh-independent reinforcement can also be defined.


Displays the selection toolbar to select existing domains. The current reinforcement is applied when the selection is completed.


Option to draw rectangular reinforcement domains.


Option to draw skewed rectangular reinforcement domains.

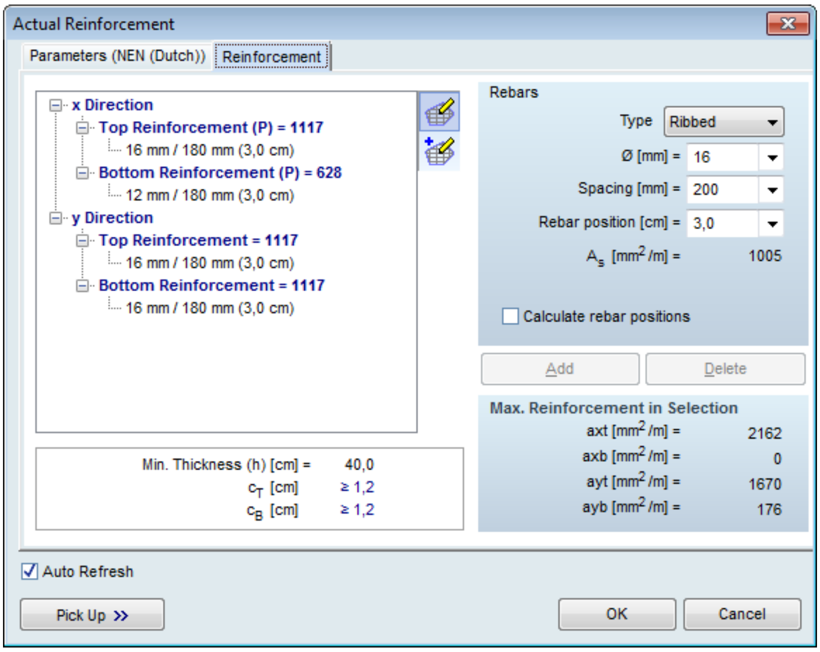

Option to draw polygonal reinforcement domains.


Option to apply reinforcement to domains just by clicking them.

Reinforcement is applied only where reinforcement domains fall on surface elements or domains.

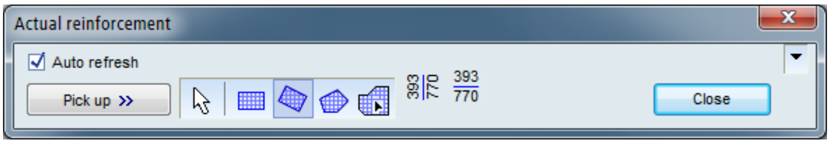
6.5.2.2. Mesh-independent reinforcement

To define mesh-independent reinforcement set the parameters and the reinforcement scheme first then draw rectangular or polygonal reinforcement domains.
If no surfaces or domains are selected clicking the button on the toolbar displays this dialog.



Reinforcement can be added or deleted the same way as above.

The dialog can be reduced to a toolbar. Clicking the triangle 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 the vertical line. The amounts of top and bottom *x* reinforcement are written along the horizontal line.

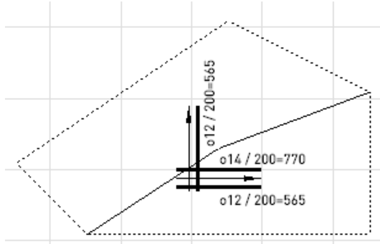


Toolbar icons remain the same.

Reinforcement is applied only where reinforcement domains fall on surface elements or domains.

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* and *y* directions appear at the center. Centerpoint is connected to two vertices of the domain polygon by continuous brown lines.



When modifying an existing reinforcement domain two methods are available:


Overwrite

New reinforcfement overwrites the existing one.


Add

New reinforcement is added to the existing one.

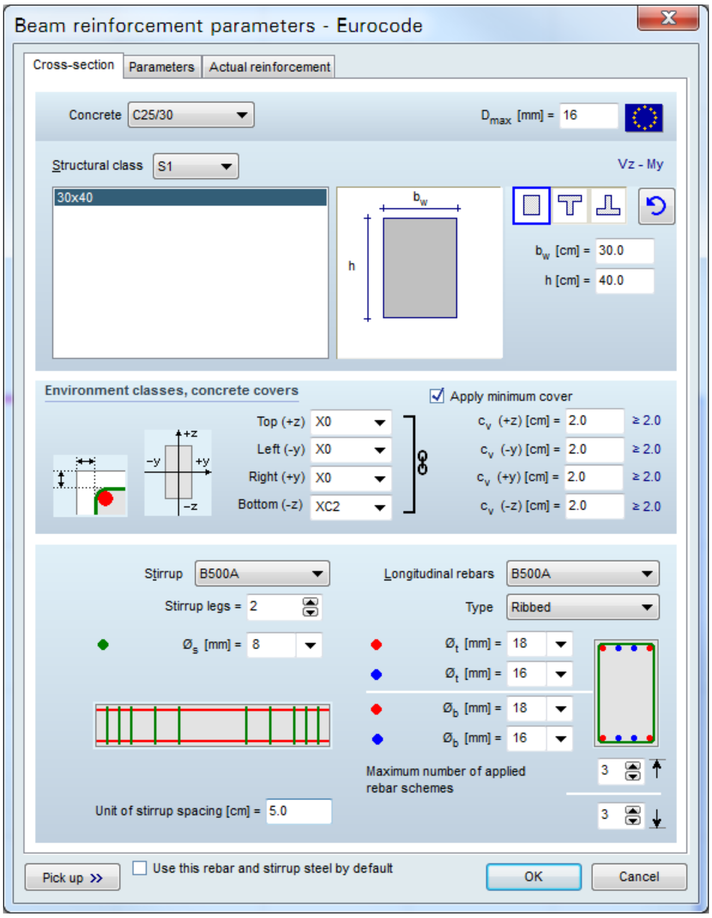
6.5.3. Beam reinforcement parameters (uniaxial bending)

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…** *6.5.10 Beam reinforcement design – RC2 module*) uses these parameters.

Cross-section

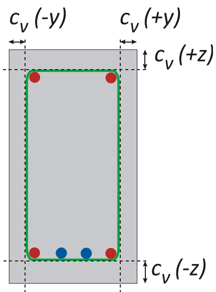


The list contains all beam design cross-sections found in the selection. The geometry parameters of the selected list item will be displayed and can be edited. Changes won't lead to recalculation of forces so cross-section dimensions should not be changed unless it is really necessary.
Concrete grade 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 covers

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.



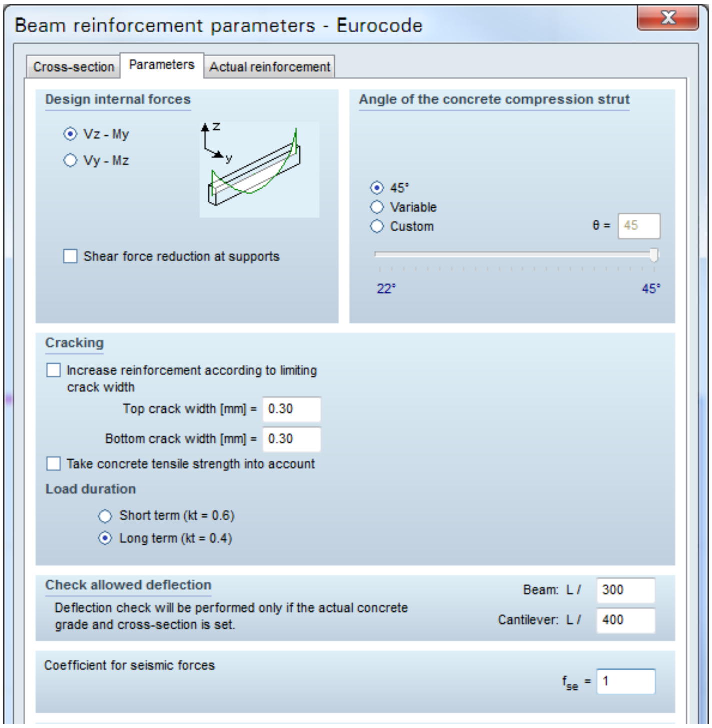
Reinforcement parameters

Stirrup parameters are on the left, parameters of longitudinal rebars are on the right. Stirrup is displayed in green, rebars at the corners are red, other longitudinal rebars are blue, their diameter at the top / bottom ($Ø_t$, $Ø_b$) can be set separately. 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.***

Stirrup spacing will always be a multiple of the value set as the *Unit of stirrup spacing*.
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.
Checking *Shear force reduction at supports* allows the application of shear force reduction methods according to the current design code.

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) $1 \leq \text{ctg}\,\Theta \leq 2.5$. 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 $A_s \leq 0.04A_c$) to reduce the crack width under the specified value. To perform cracking analysis the load duration must be specified. **See**... *6.5.3 Beam reinforcement parameters (uniaxial bending).* 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

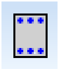
k_t is a factor depending on load duration.
For short term loads $k_t = 0.6$, for long term loads $k_t = 0.8$

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.

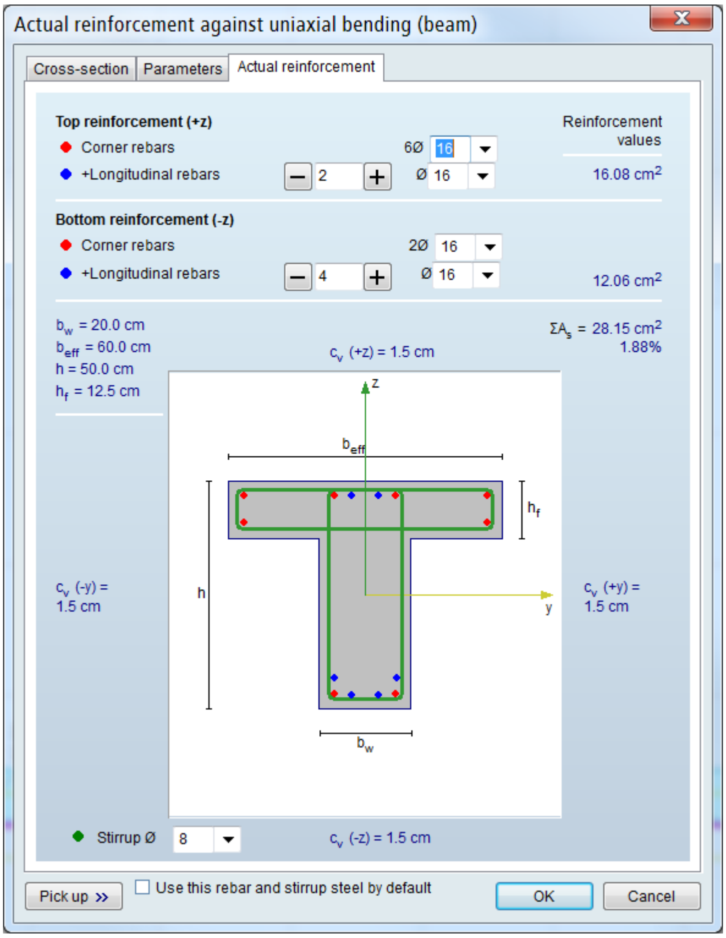
For coefficient of seimic forces see *4.10.23 Seismic loads – SE1 module*.

Actual reinforcement against uniaxial bending (beam)



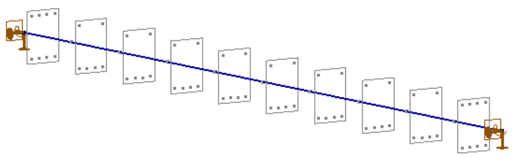
An actual reinforcement can be assigned to the selected finite elements. Nonlinear analysis can take into account this constant reinforcement when calculating deflection.

The necessary number of 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 rebars can be specified. Rebars are distributed automatically.

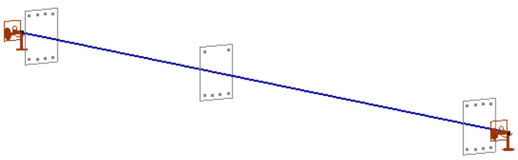


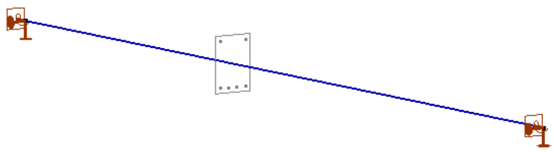
Pick up Clicking *Pick up* allows picking up beam reinforcement design parameters defined for another beam.

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…** *2.16.18 Display options)* actual reinforcement can also be seen in the model view.

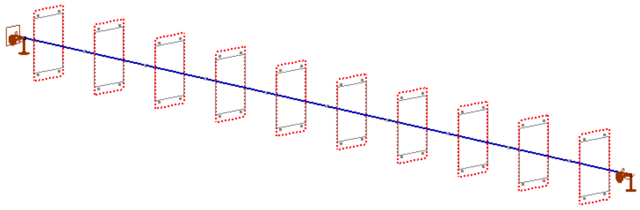


If a mesh is assigned to the line element (**see…** *4.11.1.1 Meshing of line elements*), each finite element can have different actual reinforcement (Turn on *Settings / Preferences / Editing / Enable selection of finite elements on lines*). If line mesh display is disabled, three cross-sections are 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.

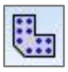




During the definition of beam reinforcement parameters it is possible to change the cross section. If the cross section assigned to the line element is not identical to the cross section specified in the beam reinforcement parameters dialog, the modified cross section is displayed red dashed line in the model view.



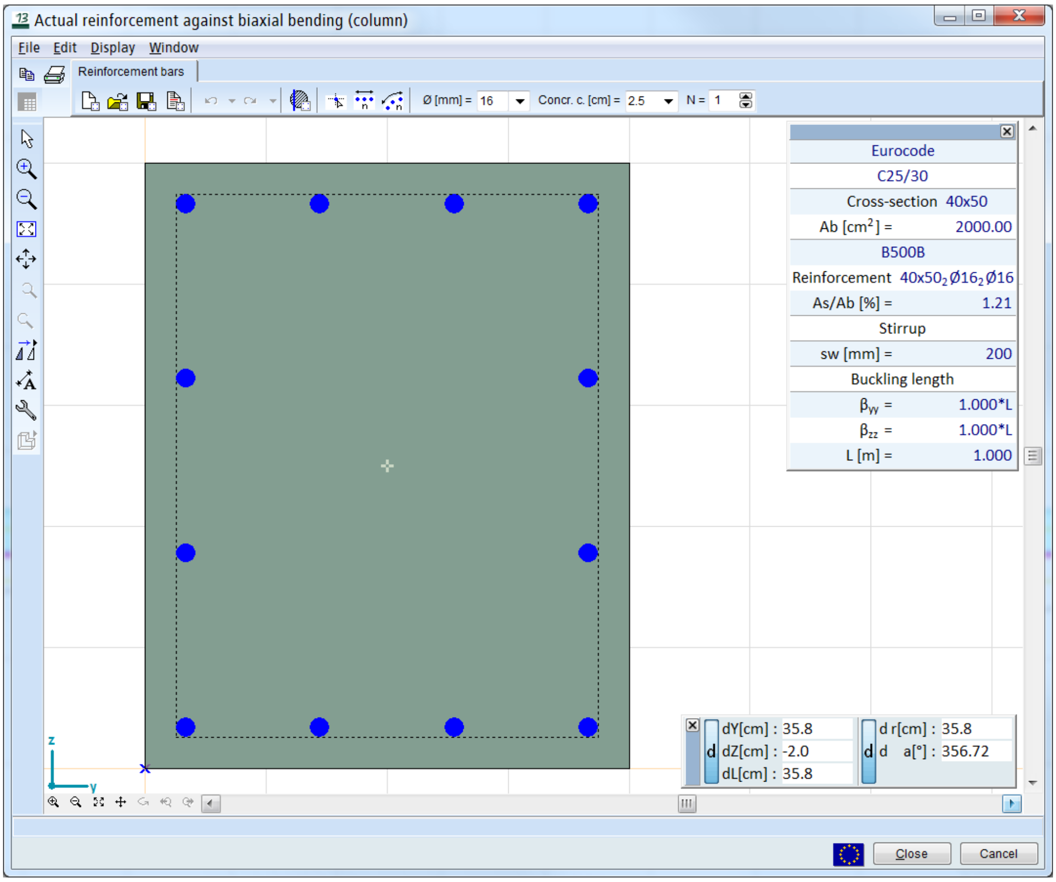
6.5.4. Actual reinforcement against biaxial bending (column)



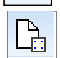
Parameters for reinforcement against biaxial bending can be specified in the dialog below.

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…** *6.5.9 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… 6.5.3 Beam reinforcement parameters (uniaxial bending)) may be more practical.***



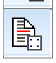

Saves the current drawing to the Drawings Library.


Defines a new reinforcement


Opens a new cross-section or reinforcement.

☞ ***Only cross-sections with graphics data can be opened.***

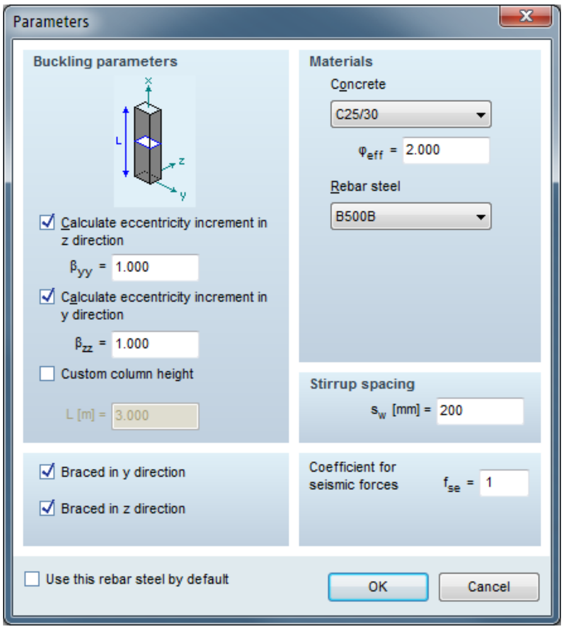

Saves the reinforcement under a name for further use.


List of existing column reinforcements. You can sort them and delete the marked rows.

Define reinforcement

The following icons are available on the Define reinforcement menu:

Parameters



Lets you specify the parameters for calculation of the load-moment strength interaction diagram.
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. 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.
Check *Braced in z (or y) direction* if the overall structure is braced in the respective direction.
You can see coefficient of seimic forces at *4.10.23 Seismic loads – SE1 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 taking into account the concrete cover.

By spacing

Inserts evenly *N+1* new rebars between two selected points.

On circular arch

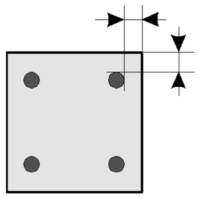
Inserts evenly *N+1* new rebars between a selected starting point and an end-point of a circular arch.

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

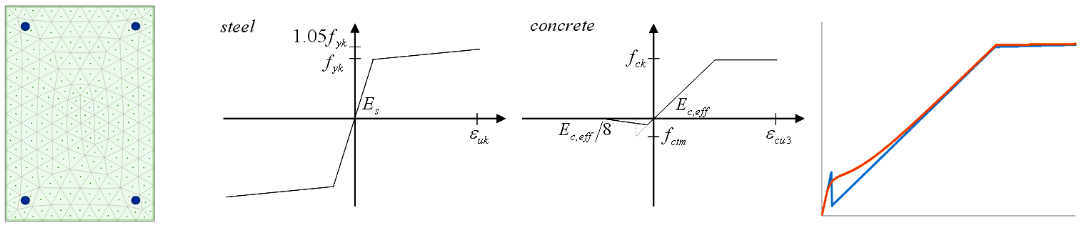
After actual reinforcement and its prameters have been assigned to the element and the display of cross-section shapes is turned on *(Display options / Symbols / Graphics symbols / Cross-section shape,* **see…** *2.16.18 Display options)* actual reinforcement can also be seen in the model view. See also *6.5.3 Beam reinforcement parameters (uniaxial bending)*

6.5.5. Nonlinear analysis of reinforced concretre beam and column elements

In case of nonlinear analysis actual reinforcements, concrete paramteres and steel/concrete nonlinear behaviour can be taken into account (**see…***5.1 Static analysis*) for the following standards:

Eurocode 2: EN 1992-1-1:2004
SIA: SIA 262:2003

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. In order to avoid convergence problems, concrete material model has been changed on tension side. 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.



☞ ***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. 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…*** *4.10.23 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.***

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”*
2. *„Increment X, Beam Y: Bending moment exceeds the bending resistance”*
3. *„Increment X, Beam Y: Longitudinal tension reinforcement is insufficient”*

6.5.6. Nonlinear deflection of RC plates

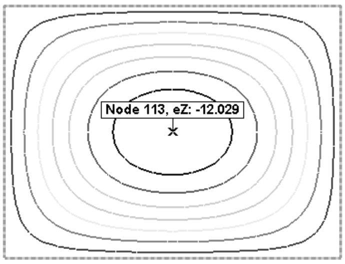
In case of the linear static analysis the plate deflection is calculated according to the elastic theory. In fact the behaviour of RC plates is non-linear due to two opposite effects. The actual reinforcement increases the bending strength but cracking decreases it.
The non-linear RC plate deflection analysis follows up these two effects with the actual reinforcement.
The program performs a non-linear analysis in an iterative way using the moment-curvature diagrams of RC cross-sections. The strength effect of the tensile concrete is also taken into account.
This non-linear analysis is available based on Eurocode, DIN 1045-1 (German), SIA-262 (Swiss), NEN (Dutch), MSz (Hungarian) and STAS (Romanian) design codes.

The main **steps of a plate deflection calculation** are

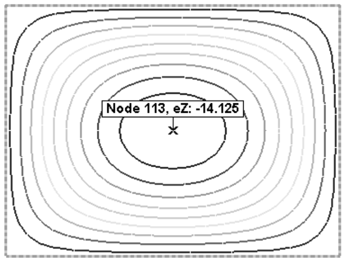
1.) performing a linear analysis of the plate
2.) calculating the required reinforcement
3.) applying the actual reinforcement
4.) performing a non-linear analysis of the plate

☞ ***When you start the non-linear analysis, check the*** *Use actual reinforcement in the calculation* ***checkbox.***

Plate deflection:



Linear (elastic) analysis

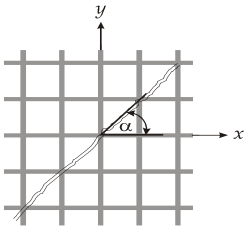


Non-linear analysis

6.5.7. Cracking

Design Codes

Eurocode 2: *EN 1992-1-1:2004*
DIN: *DIN 1045-1:2001-07*

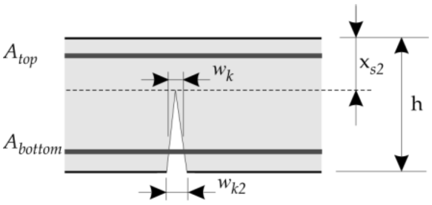


After the assignment of the actual reinforcement the program calculates the crack width and crack directions in the membrane, plate and shell elements. The direction of the reinforcement is relative to the surface element's local *x* and *y* axes.The program displays the crack openings in a color coded mode, can draw the crack map and the crack angles.

The set of the parameters can be seen in the previous section.

Results In the table of results the following information can be found:

Aax, Aay	actual reinforcement in ***x*** and ***y*** direction
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 calculation of crack width is based on the actual reinforcement assigned to the surfaces.

6.5.7.1. 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

$$\epsilon_{sm} - \epsilon_{cm} = \frac{\sigma_{s2} - \dfrac{k_t f_{ctm}}{\rho_{\rho,eff}}\left(1 + \dfrac{E_s}{E_{cm}}\rho_{\rho,eff}\right)}{E_s} \geq 0.6 \cdot \frac{\sigma_{s2}}{E_s}$$

$$s_{r,max} = 3.4 \cdot c + 0.425 \cdot k_1 k_2 \frac{\overline{\Phi}}{\rho_{\rho,eff}}$$

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_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.

6.5.7.2. Calculation according to DIN 1045-1

$w_k = s_{r,max}(\epsilon_{sm} - \epsilon_{cm})$, 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

$$\epsilon_{sm} - \epsilon_{cm} = \frac{\sigma_{s2} - 0.4 \cdot \frac{f_{ctm}}{\rho_{eff}}\left(1 + \frac{E_s}{E_{cm}}\rho_{eff}\right)}{E_s} \geq 0.6 \cdot \frac{\sigma_{s2}}{E_s}$$

$$s_{rmax} = \frac{\overline{d}}{3.6 \cdot \rho_{eff}} \leq \frac{\sigma_{s2}\overline{d}}{3.6 f_{ctm}}$$

where
$\overline{d}$ is the average rebar diameter
$\rho_{\rho eff} = A_s / A_{c,eff}$ is the effective reinforcement ratio

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 calculation for plates and shells

Design Codes

Eurocode 2: *EN 1992-1-1:2004*
DIN: *DIN 1045-1:2001-07*
SIA: *SIA 262:2003*

AxisVM calculates the shear resistance of the reinforced plate or shell without shear reinforcement, the normal shear force and the difference between them.

$v_{Sz} = \sqrt{v_{xz}^2 + v_{yz}^2}$ 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.

$d = (d_x + d_y)/2$ is the average effective height.

$\rho_l = \sqrt{\rho_x \rho_y}$ 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.

☞ ***The calculation of the shear resistance is based on the actual reinforecement assigned to the surfaces.***

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

$$C_{Rd,c} = \frac{0.18}{\gamma_c}, \qquad k = 1 + \sqrt{200/d} \leq 2.0, \qquad k_1 = 0.15$$

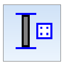
$$\sigma_{cp} = \frac{N_{Ed}}{A_c} \leq 0.2\, f_{cd}, \qquad v_{min} = 0.035 \cdot k^{3/2} \cdot f_{ck}^{1/2}$$

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 $\rho_l \leq 0.02$

The $V_{Rd,c}$ shear resistance and the difference between actual shear force and the shear resistance $v_{Sz} - V_{Rd,c}$ can also be displayed with isolines and isosurfaces.

6.5.9. Column reinforcement – RC2 module



The reinforced column check can be performed based on the following design codes:

Design Codes

Eurocode 2: *EN 1992-1-1:2004*
DIN: *DIN 1045-1:2001-07*
SIA: *SIA 262:2003*

Reinforcement bars

Actual reinforcement can be entered the same way as described in *6.5.4 Actual reinforcement against biaxial bending (column)*

Column Check

Calculates the interaction diagram 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_{xd}, M_{yd}, M_{zd} 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.

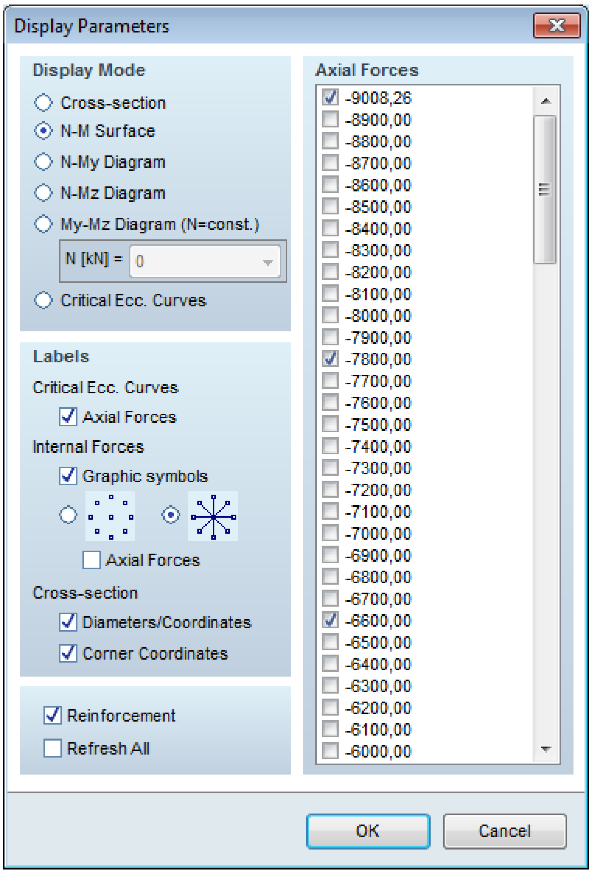
Display

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.

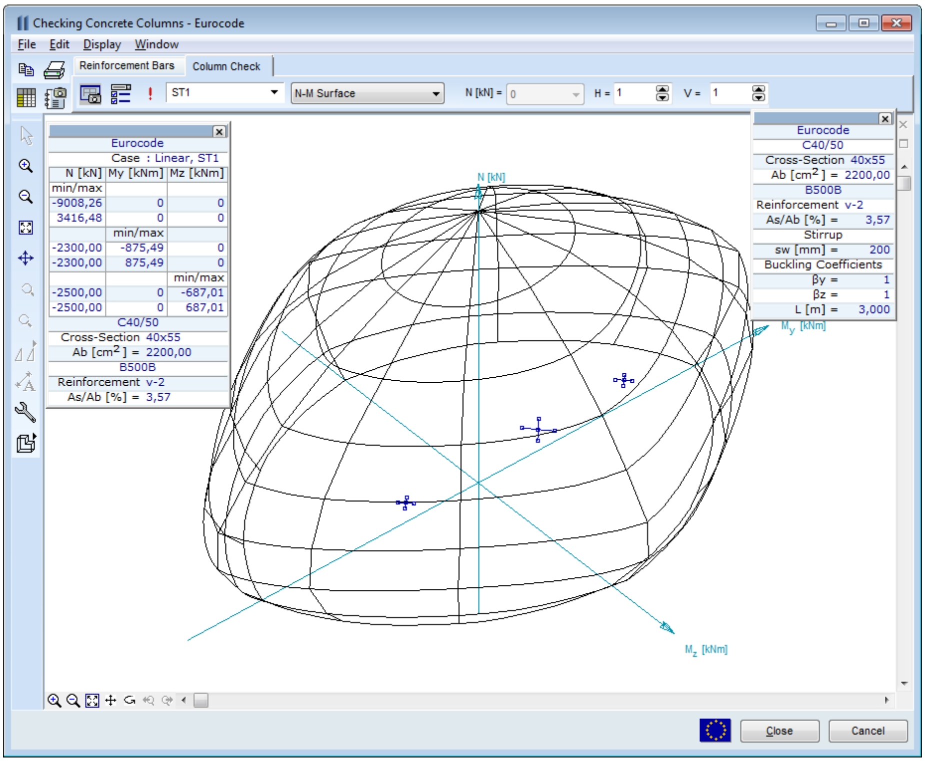
Select axial force values to use when drawing the 3D interaction diagram (N-M Surface) from the check list.

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.

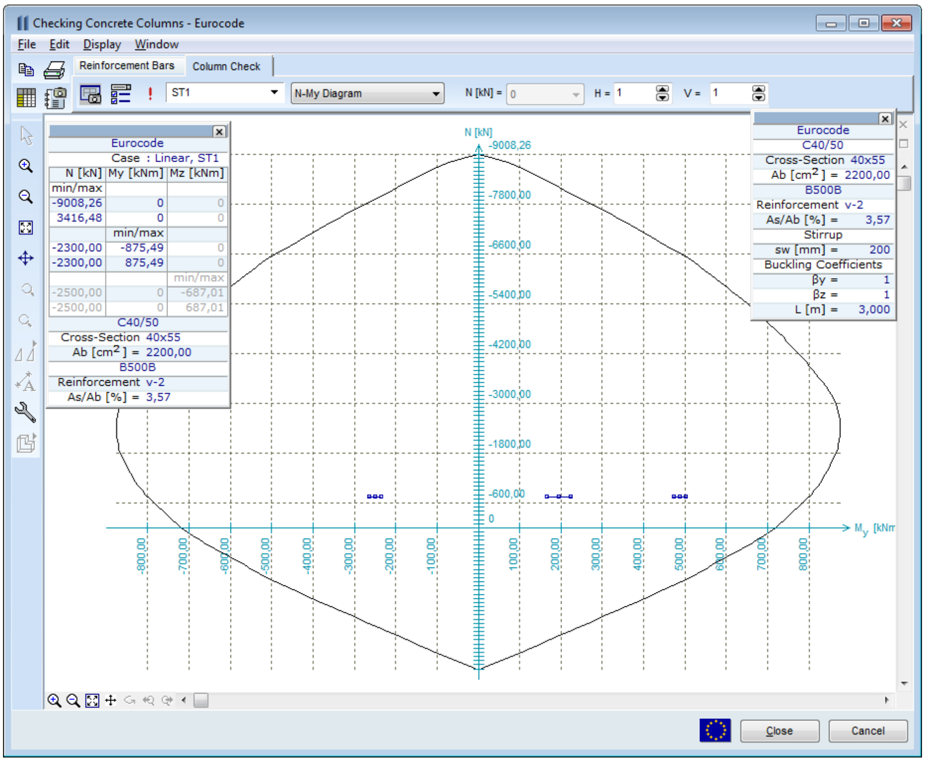


Blue color shows that the N_{xd}, M_{yd}, M_{zd} values are within the interaction diagram. Red color shows that N_{xd}, M_{yd}, M_{zd} values are ouside the interaction diagram. The normal forces for these points are always displayed.

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.

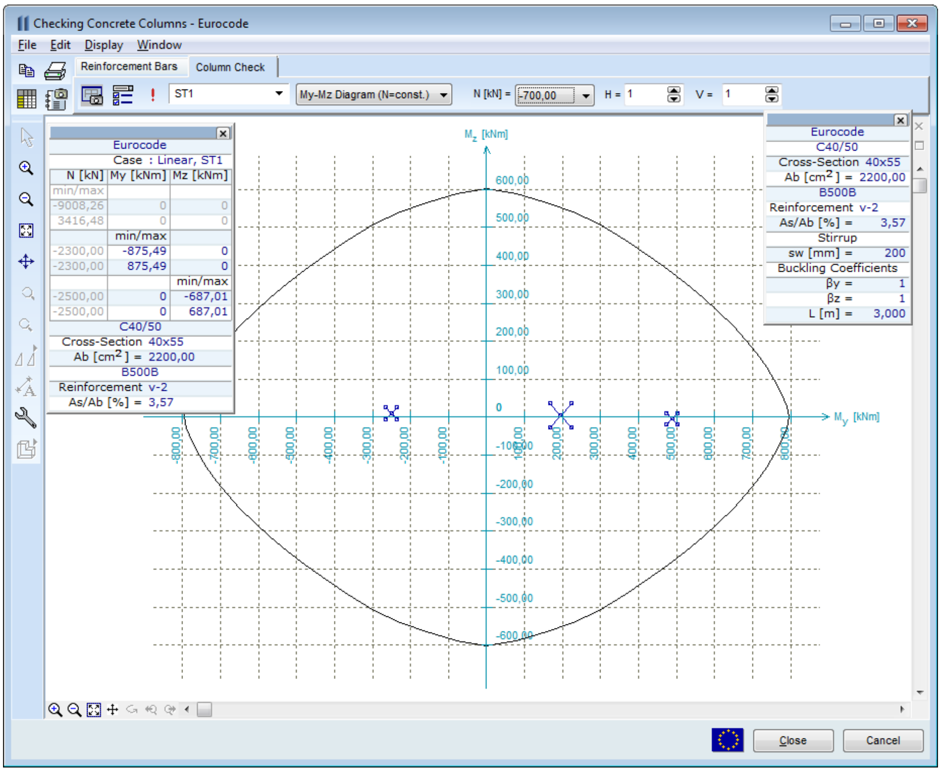


This display mode can be used with cross-sections that are symmetric. You can display the design values of the internal forces, by enabling the *Write values to* check-box.

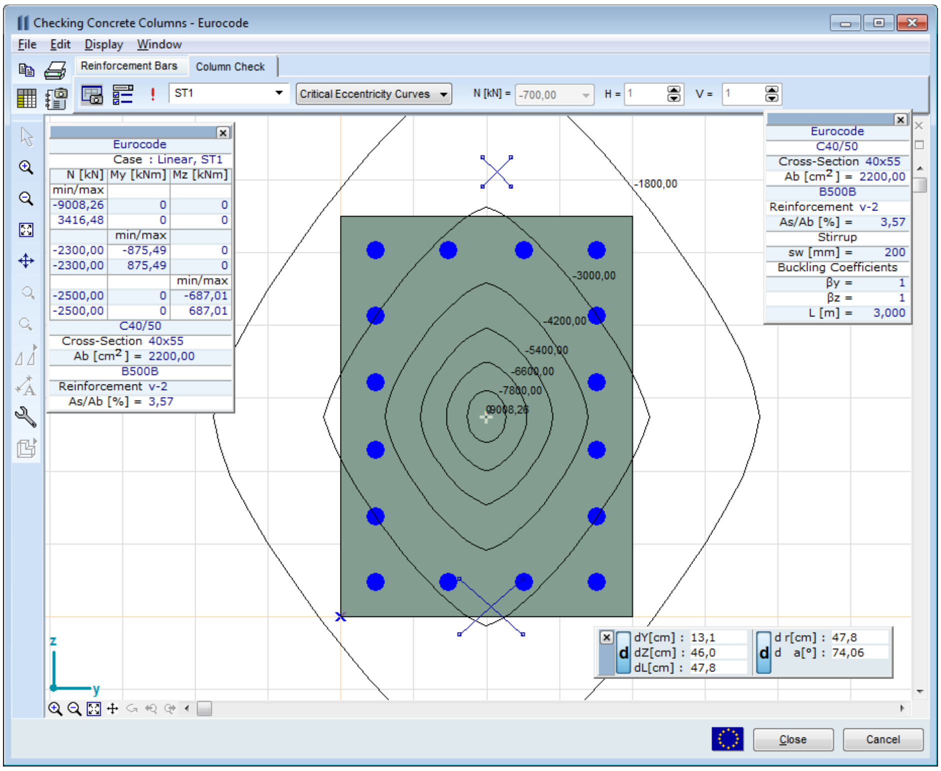
The design values of the internal forces are displayed as follows:

Blue rectangle: the design value N_{xd}-M_{yd}-M_{zd} is under the interaction surface.
x red cross: the design value N_{xd}-M_{yd}-M_{zd} is above the interaction surface.

My-Mz diagram Displays the M_y-M_z interaction diagram at a given N value.



Load eccentricity limit curves



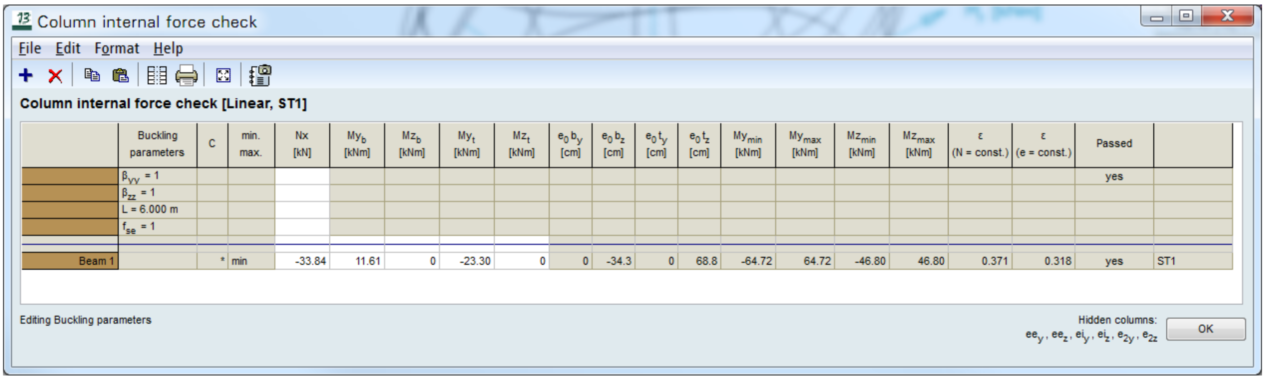
Displays the load eccentricity limit curves based on $\frac{M_{yRi}}{N_i}$ or $\frac{M_{zRi}}{N_i}$.

Blue rectangle: the design value N_{xd}-M_{yd}-M_{zd} is inside the load eccentricity limit curve.
x red cross: the design value N_{xd}-M_{yd}-M_{zd} is outside the load eccentricity limit curve.

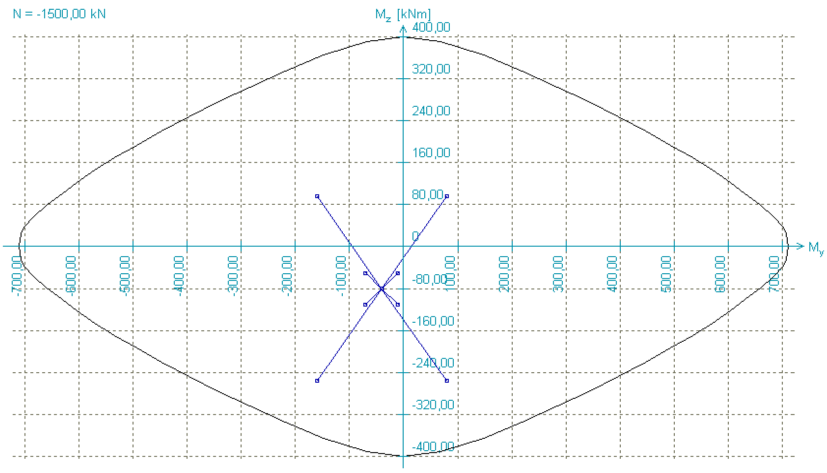
Internal forces The *Column internal force check* table contains the maximum normal forces and moments at the top and bottom end of the selected columns and different eccentricity values.

Additional columns displaying $My_{min}, My_{max}, Mz_{min}, Mz_{max}$ moment resistance maximums at the given Nx are also available.

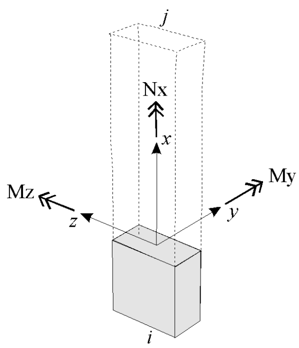
Efficiencies The program calculates two types of efficiency. The first one is *ε (N = const.),* the moment efficiency: this is defined on the *My-Mz* diagram as the ratio of the distance of design force point from the origin to the distance of the intersection point of the curve and the half line drawn from the origin through the same point from the origin. The second one is *ε (e = const.)* the efficiency 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.



	Buckling parameters	C	min. max.	Nx [kN]	My_b [kNm]	Mz_b [kNm]	My_t [kNm]	Mz_t [kNm]	$e_0 b_y$ [cm]	$e_0 b_z$ [cm]	$e_0 t_y$ [cm]	$e_0 t_z$ [cm]	My_{min} [kNm]	My_{max} [kNm]	Mz_{min} [kNm]	Mz_{max} [kNm]	ε (N = const.)	ε (e = const.)	Passed	
	β_{yy} = 1																		yes	
	β_{zz} = 1																			
	L = 6.000 m																			
	f_{se} = 1																			
Beam 1		*	min	-33.84	11.61	0	-23.30	0	0	-34.3	0	68.8	-64.72	64.72	-46.80	46.80	0.371	0.318	yes	ST1



On *N-M*$_R$ 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-M*$_R$ strength interaction diagrams and in the load eccentricity limit curves.Signs of the foces and moments are determined according to the picture.



Rebars thinner than 1/12 of the stirrup distance will be ignored for compression.

6.5.9.1. Check of reinforced columns according to Eurocode 2

Design moments in bending directions are $M_d = N_d \cdot e_d$
where N_d 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.

$e_0 = M_I / N_d$ initial eccentricity calculated from the first order force and moment.
If moments at the top and bottom end of the column are different, a substitute eccentricity will be determined:
e_e: If the moments at the column endpoints are different, an **equivalent eccentricity** is determined according to the following

- in a braced direction $e_e = \max\begin{Bmatrix} 0.6 \cdot e_a + 0.4 \cdot e_b \\ 0.4 \cdot e_a \end{Bmatrix}$ and $|e_a| \geq |e_b|$
  where e_a and e_b are the initial eccentricities at the ends of the column.
- in a non-braced direction $e_e = \max\begin{Bmatrix} |e_a| \\ |e_b| \end{Bmatrix}$, but with the sign of the eccentricity bigger in absolute value.

e_i : **increment due to inaccuracies** (imperfection)

$e_i = \alpha_h \Theta_0 \frac{l_0}{2}$, 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 mesh length.

e_2 : **second order increment of the eccentricity.**

$e_2 = \frac{1}{r}\frac{l_0^2}{\pi^2}$, where $\frac{1}{r} = K_r K_\varphi \frac{f_{yd}}{E_s \cdot 0.45 \cdot d'}$ if $\lambda \geq \lambda_{lim} = 20\frac{ABC}{\sqrt{n}}$ where $n = \frac{N_{Ed}}{A_c f_{cd}}$
$K_r = \min\left\{\frac{N_u' - N_{Ed}}{N_u' - N_{bal}}; 1.0\right\}$, $K_\varphi = \max\{1 + \beta\varphi_{ef}; 1.0\}$,
$\beta = 0.35 + \frac{f_{ck}}{200} - \frac{\lambda}{150}$, where f_{ck} is in N/mm^2,
$d' = \frac{h}{2} + i_s$, where i_s is the radius of inertia of rebars

Increments of eccentricities are determined in both bending planes. The program checks the following design situations:

At the middle of the column:

$\lambda_y/\lambda_z \leq 2$ and $\lambda_z/\lambda_y \leq 2$, furthermore

$$\frac{e_y/b_{eq}}{e_z/h_{eq}} \leq 0{,}2 \text{ or } \frac{e_z/h_{eq}}{e_y/b_{eq}} \leq 0{,}2$$

$$M_{dy,1} = N_d^* e_{ez}$$
$$M_{dz,1} = -N_d^*\left(e_{ey} \pm \left(e_{iy} + e_{2y}\right)\right)$$

$$M_{dy,2} = N_d^*\left(e_{ez} \pm (e_{iz} + e_{2z})\right)$$
$$M_{dz,2} = -N_d^* e_{ey}$$

otherwise

$$M_{dy} = N_d^*\left(e_{ez} \pm (e_{iz} + e_{2z})\right)$$
$$M_{dz} = -N_d^*\left(e_{ey} \pm \left(e_{iy} + e_{2y}\right)\right)$$

At the top and bottom of the column if the column is braced (non-sway):

$\lambda_y/\lambda_z \leq 2$ and $\lambda_z/\lambda_y \leq 2$, furthermore

$$\frac{e_y/b_{eq}}{e_z/h_{eq}} \leq 0{,}2 \text{ or } \frac{e_z/h_{eq}}{e_y/b_{eq}} \leq 0{,}2$$

$$M_{dy,1} = N_d^* e_{0z}$$
$$M_{dz,1} = -N_d^*\left(e_{0y} \pm e_{iy}\right)$$

$$M_{dy,2} = N_d^*(e_{0z} \pm e_{iz})$$
$$M_{dz,2} = -N_d^* e_{0y}$$

otherwise

$$M_{dy} = N_d^*(e_{0z} \pm e_{iz})$$
$$M_{dz} = -N_d^*\left(e_{0y} \pm e_{iy}\right)$$

At the top and bottom of the column if the column is not braced (sway):

$\lambda_y/\lambda_z \leq 2$ and $\lambda_z/\lambda_y \leq 2$, furthermore

$$\frac{e_y/b_{eq}}{e_z/h_{eq}} \leq 0{,}2 \text{ or } \frac{e_z/h_{eq}}{e_y/b_{eq}} \leq 0{,}2$$

$$M_{dy,1} = N_d^* e_{0z}$$

$$M_{dz,1} = -N_d^*\left(e_{0y} \pm \left(e_{iy} + e_{2y}\right)\right)$$

$$M_{dy,2} = N_d^*\left(e_{0z} \pm (e_{iz} + e_{2z})\right)$$

$$M_{dz,2} = -N_d^* e_{0y}$$

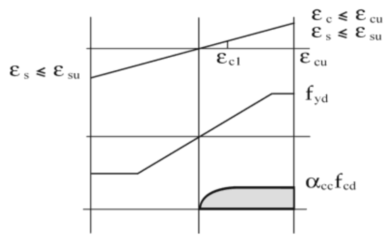
otherwise

$$M_{dy} = N_d^*\left(e_{0z} \pm (e_{iz} + e_{2z})\right)$$

$$M_{dz} = -N_d^*\left(e_{0y} \pm \left(e_{iy} + e_{2y}\right)\right)$$

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:

6.5.9.2. Check of reinforced columns according to DIN1045-1

Design moments in bending directions are $M_d = N_d \cdot e_d$
where N_d is the normal force in the column and $e_d = e_0 + e_a + e_2$ is the critical eccentricity in the given bending direction.
$e_0 = M_I/N_d$ initial eccentricity calculated from the first order force and moment.
If moments at the top and bottom end of the column are different, **a substitute eccentricity** will be determined:

- in a braced direction $e_e = \max\left\{\begin{matrix} 0.6 \cdot e_{02} + 0.4 \cdot e_{01} \\ 0.4 \cdot e_{02} \end{matrix}\right\}$ and $|e_{02}| \geq |e_{01}|$ where e_{02} and e_{01} are the initial eccentricities at the ends of the column.
- in a non-braced direction $e_e = \max\left\{\begin{matrix} |e_{02}| \\ |e_{01}| \end{matrix}\right\}$, but with the sign of the eccentricity bigger in absolute value.

e_a *:* **increment due to inaccuracies** (imperfection)

$e_a = \alpha_{a1}\frac{l_0}{2}$, where l_0 is the buckling length, $\alpha_{a1} = \frac{1}{100\sqrt{l}} \leq \frac{1}{200}$, where *l* is the mesh length.

$$\lambda_{max} = \max\left\{25; \frac{16}{\sqrt{\frac{N_d}{A_c f_{cd}}}}\right\}$$

If $\lambda \geq \lambda_{max}$ second order increment of eccentricity has to be taken into account, where λ is the column slimness calculated from the concrete cross-section.

***e_2* : second order increment of the eccentricity.**

$e_2 = K_1 \frac{1}{r}\frac{l_0^2}{10}$, where $\frac{1}{r} = K_2 \frac{2\cdot f_{yd}}{E_s\cdot 0.9\cdot d}$

$K_1 = \min\left\{\frac{\lambda}{10} - 2.5; 1.0\right\}$, $K_2 = \frac{N_{ud}-N_d}{N_{ud}-N_{bal}} \leq 1.0$

d is the effective height of the cross-section

Increments of eccentricities are determined in both bending planes. The program checks the following design situations:

At the middle of the column:

$$\frac{e_y/b}{e_z/h} \leq 0,2 \text{ or } \frac{e_z/h}{e_y/b} \leq 0,2$$

$$M_{dy,1} = N_d^* e_{ez}$$
$$M_{dz,1} = -N_d^*\left(e_{ey} \pm \left(e_{iy} + e_{2y}\right)\right)$$

$$M_{dy,2} = N_d^*\left(e_{ez} \pm \left(e_{iz} + e_{2z}\right)\right)$$
$$M_{dz,2} = -N_d^* e_{ey}$$

otherwise

$$M_{dy} = N_d^*\left(e_{ez} \pm \left(e_{az} + e_{2z}\right)\right)$$
$$M_{dz} = -N_d^*\left(e_{ey} \pm \left(e_{ay} + e_{2y}\right)\right)$$

At the top and bottom of the column if the column is braced (non-sway):

$$\frac{e_y/b}{e_z/h} \leq 0,2 \text{ or } \frac{e_z/h}{e_y/b} \leq 0,2$$

$$M_{dy,1} = N_d^* e_{0z}$$
$$M_{dz,1} = -N_d^*\left(e_{0y} \pm e_{ay}\right)$$

$$M_{dy,2} = N_d^*\left(e_{0z} \pm e_{az}\right)$$
$$M_{dz,2} = -N_d^* e_{0y}$$

otherwise

$$M_{dy} = N_d^*\left(e_{0z} \pm e_{az}\right)$$
$$M_{dz} = -N_d^*\left(e_{0y} \pm e_{ay}\right)$$

At the top and bottom of the column if the column is not braced (sway):

$$\frac{e_y/b}{e_z/h} \leq 0,2 \text{ or } \frac{e_z/h}{e_y/b} \leq 0,2$$

$$M_{dy,1} = N_d^* e_{0z}$$
$$M_{dz,1} = -N_d^*\left(e_{0y} \pm \left(e_{ay} + e_{2y}\right)\right)$$

$$M_{dy,2} = N_d^*\left(e_{0z} \pm \left(e_{az} + e_{2z}\right)\right)$$
$$M_{dz,2} = -N_d^* e_{0y}$$

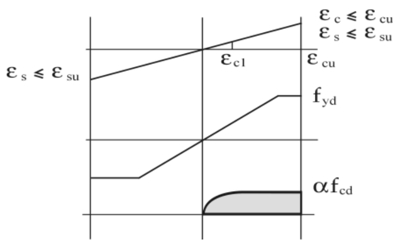
otherwise

$$M_{dy} = N_d^*\left(e_{0z} \pm \left(e_{az} + e_{2z}\right)\right)$$
$$M_{dz} = -N_d^*\left(e_{0y} \pm \left(e_{ay} + e_{2y}\right)\right)$$

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



6.5.9.3. Check of reinforced columns according to SIA 262

Design moments in bending directions are $M_d = N_d \cdot e_d$
where N_d is the normal force in the column and $e_d = e_{0d} + e_{1d} + e_{2d}$ is the critical eccentricity in the given bending direction.

e_{0d}: **increment due to inaccuracies (imperfection)**

$$e_{0d} = \max\left\{\alpha_i \frac{l_{cr}}{2}; \frac{d}{30}\right\}, \text{ where } \frac{1}{200} \geq \alpha_i = \frac{0.01}{\sqrt{l}} \geq \frac{1}{300}$$

l_{cr} is the buckling length, *l* is the actual length, *d* is the effective height of the cross-section.

$e_{1d} = \frac{M_{dI}}{N_d}$ **initial eccentricity** calculated from the first order force and moment.
If moments at the top and bottom end of the column are different, a substitute initial eccentricity will be determined:

- in a braced direction $e_e = \max\begin{Bmatrix} 0.6 \cdot e_a + 0.4 \cdot e_b \\ 0.4 \cdot e_a \end{Bmatrix}$ and $|e_a| \geq |e_b|$ where e_a and e_b are the initial eccentricities at the ends of the column.
- in a non-braced direction $e_e = \max\begin{Bmatrix} |e_a| \\ |e_b| \end{Bmatrix}$, but with the sign of the eccentricity bigger in absolute value.

e_2: **second order increment of the eccentricity.**

$$e_{2d} = \chi_d \frac{l_{cr}^2}{\pi^2}, \text{ where } \chi_d = \frac{2 f_{sd}}{E_s(d-d')}$$

Increments of eccentricities are determined in both bending planes. The program checks the following design situations (considering additional recommendations in Eurocode 2):

At the middle of the column:

$\lambda_y/\lambda_z \leq 2$ and $\lambda_z/\lambda_y \leq 2$, furthermore

$$\frac{e_y/b_{eq}}{e_z/h_{eq}} \leq 0{,}2 \text{ or } \frac{e_z/h_{eq}}{e_y/b_{eq}} \leq 0{,}2$$

$$M_{dy,1} = N_d^* e_{ez}$$
$$M_{dz,1} = -N_d^*\left(e_{ey} \pm (e_{0y} + e_{2y})\right)$$

$$M_{dy,2} = N_d^*\left(e_{ez} \pm (e_{0z} + e_{2z})\right)$$
$$M_{dz,2} = -N_d^* e_{ey}$$

otherwise

$$M_{dy} = N_d^*\left(e_{ez} \pm (e_{0z} + e_{2z})\right)$$
$$M_{dz} = -N_d^*\left(e_{ey} \pm (e_{0y} + e_{2y})\right)$$

At the top and bottom of the column if the column is braced (non-sway):

$\lambda_y/\lambda_z \leq 2$ and $\lambda_z/\lambda_y \leq 2$, furthermore

$$\frac{e_y/b_{eq}}{e_z/h_{eq}} \leq 0{,}2 \text{ or } \frac{e_z/h_{eq}}{e_y/b_{eq}} \leq 0{,}2$$

$$M_{dy,1} = N_d^* e_{1z}$$
$$M_{dz,1} = -N_d^*\left(e_{1y} \pm e_{0y}\right)$$

$$M_{dy,2} = N_d^*(e_{1z} \pm e_{0z})$$
$$M_{dz,2} = -N_d^* e_{1y}$$

otherwise

$$M_{dy} = N_d^*(e_{1z} \pm e_{0z})$$
$$M_{dz} = -N_d^*\left(e_{1y} \pm e_{0y}\right)$$

At the top and bottom of the column if the column is not braced (sway):

$\lambda_y/\lambda_z \leq 2$ and $\lambda_z/\lambda_y \leq 2$, furthermore

$$\frac{e_y/b_{eq}}{e_z/h_{eq}} \leq 0{,}2 \text{ or } \frac{e_z/h_{eq}}{e_y/b_{eq}} \leq 0{,}2$$

$$M_{dy,1} = N_d^* e_{1z}$$

$$M_{dz,1} = -N_d^*\left(e_{1y} \pm \left(e_{0y} + e_{2y}\right)\right)$$

$$M_{dy,2} = N_d^*\left(e_{1z} \pm (e_{0z} + e_{2z})\right)$$

$$M_{dz,2} = -N_d^* e_{1y}$$

otherwise

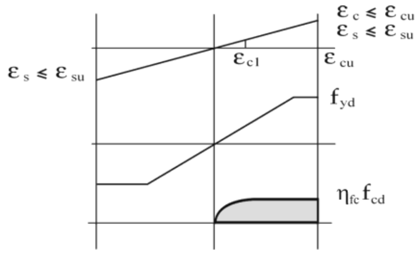
$$M_{dy} = N_d^*\left(e_{1z} \pm (e_{0z} + e_{2z})\right)$$

$$M_{dz} = -N_d^*\left(e_{1y} \pm \left(e_{0y} + e_{2y}\right)\right)$$

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):

$\varnothing < 8$, $s > 15\,\varnothing$, $s > a_{min}$, $s > 300$ mm

6.5.10. Beam reinforcement design – RC2 module

Design Codes

Eurocode 2:	*EN 1992-1-1:2004*
DIN:	*DIN 1045-1:2001-07*
SIA:	*SIA 262:2003*

☞ ***The beams are structural elements, with one dimension (the length) significantly greater than the dimensions of the cross section, loaded in bending and shear, and axial force is zero or of a small, negligible value.***

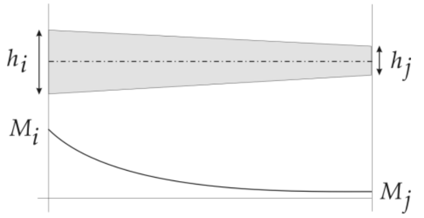
The beam reinforcement design module can be applied to beam structural elements modeled by beam or rib finite elements, that have the same material and constant or variable rectangular or T cross sections, assuming that the load is applied in the symmetry plane of the cross section.

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.

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**… *6.5.3 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.

Shear force is modified by $\Delta V = 2A_s f_{yd} \sin\alpha$, where A_s is the longitudinal tension reinforcement area, α is the angle between the extreme fiber and the centerline. Longitudinal reinforcement is assumed to be parallel to the extreme fiber.

☞ ***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.

6.5.10.1. 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).*
2. Determination of spacing of vertical stirrups considering shear forces about y or z axis (*Vy* or *Vz*) and the twisting moment *(Tx).*

The axial force is not taken into account. If the axial force cannot be neglected, the use of the column design module is recommended.
Bending and shear/twisting is analyzed separately, however the longitudinal tensile reinforcement is taken into account in the determination of the shear capacity.
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})*.

Longitudinal reinforcement against bending

The longitudinal reinforcement against bending is displayed in blue, compression reinforcement in red, the minimum reinforcement required by the design code in gray.

Stirrup spacing

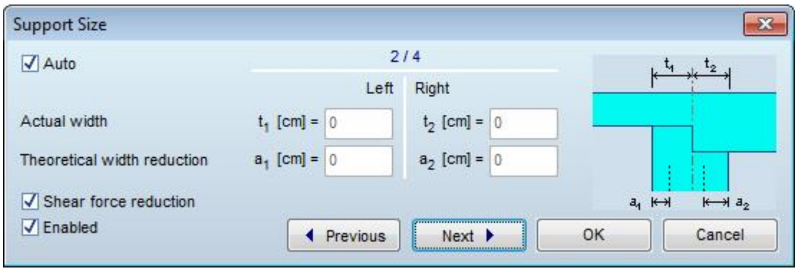
The allowable maximum stirrup spacing is displayed in black, the calculated spacing in blue, and the minimal spacing according to the design code in gray.

Longitudinal reinforcement against torsion

The longitudinal reinforcement against torsion is displayed in purple.

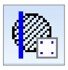
Setting support size

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



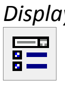
Checking *Auto* sets the detected width values. Uncheck it to set the *Actual width* and the *Theoretical width reduction* manually. It lets you specify the a_1 and a_2 segments on the side of the support that will be ignored in the calculations. The internal forces are linearly interpolated within the 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.

Beam parameters



See… *6.5.3 Beam reinforcement parameters (uniaxial bending)*

Display results



The display of diagrams and labels on each tab can be customized.

6.5.10.2. 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 efficiency (My_{Ed}/My_{Rd}), shear efficiency (Vz_{Ed}/Vz_{Rd}), torsion efficiency (Tz_{Ed}/Tz_{Rd}), relative deflection ($e_{z,rel}$), absolute deflection ($e_{z,abs}$).

Ultimate Limit State (ULS)

Bending Bendig moment resistance M_{Rd} is determined from the required (calculated) reinforcement. This version does not take into account the normal force. If normal force is considerable it is recommended to use the column reinforcement module. Bending moment efficiency (M_{Ed}/M_{Rd}) is also displayed as a diagram.

Shear The program calculates the shear resistance and shear efficiency from the actual stirrup parameters and distribution determining the shear resistance without shear reinforcement ($V_{Rd,c}$), the maximum shear force limited by the concrete compression struts ($V_{Rd,max}$), and the maximum shear force limited by the yield strength of the shear reinforcement. Shear efficiency (V_{Ed}/V_{Rd}) is also displayed as a diagram.

Shear and torsion The program calculates the shear and torsion resistance and shear and torsion efficiency from the actual stirrup parameters and distribution determining the shear resistance without shear / torsion reinforcement ($V_{Rd,c}$ and $T_{Rd,c}$), the maximum shear force / torsion moment limited by the concrete compression struts ($V_{Rd,max}$ and $T_{Rd,max}$), and the maximum shear force limited by the yield strength of the shear reinforcement.

The amount of stirrup area taken into account for torsion is: $A_{sw,T} = \frac{V_{Ed,i}}{V_{Ed,i}+V_{Ed}} \cdot A_{sw}$,

where A_{sw} is the total area of stirrups, $V_{Ed,i} = \frac{T_{Ed}}{2 \cdot A_k} \cdot z$ is the shear force from torsion, V_{Ed} is the design shear force

Service Limit State (SLS)

Cracking analysis After clicking on the *Check* 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 sombination. This cracking value can be higher than cracking calculated with rebars from the critical of envelope results.

Cracking

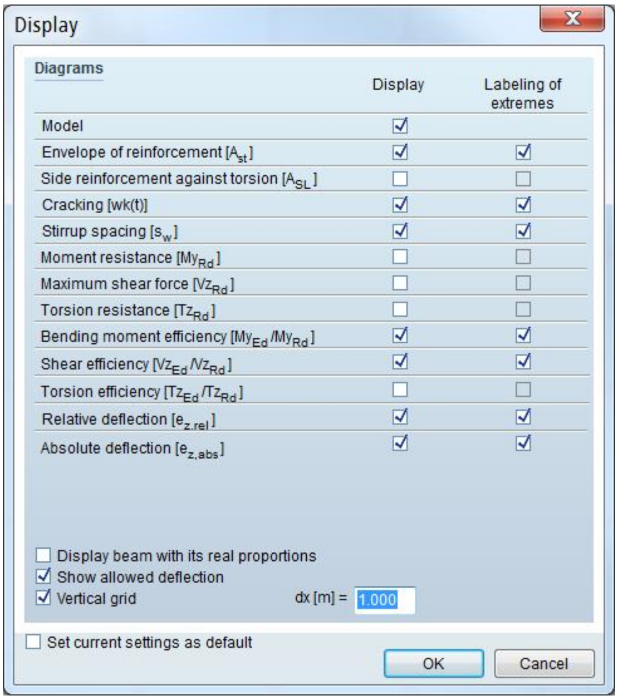
If *Parameters / Increase reinforcement according to limiting crack width* was checked when defining beam reinforcement parameters AxisVM will increase the number of rebars on the tension side until the calculated crack width falls below the limit, provided the total area of reinforcement does not exceed 4% ($A_s \leq 0.04 \cdot A_c$)

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.
The rebar scheme displays the number of rebars in the flange extension (outside of the web) in [square brackets]. Rebars in the web is displayed row by row from the outside in (round brackets).

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.***

6.5.10.3. Checking actual beam reinforcement

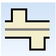
Changing actual beam reinforcement

On the *Actual beam reinforcement* tab it is possible to change the actual reinforcement assigned to finite elements. Select finite elements or remove the selection by clicking or dragging the mouse. To add 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 and bottom rebars in the selected elements. The – and + buttons decrease/increase these numbers.

The diagram of the actual reinforcement is filled with sky blue, the A_s reinforcement calculated for the current load case or combination can be seen behind it as a red diagram filled with pink.

Select all (Ctrl+A)

Selects all finite elements of the beam.

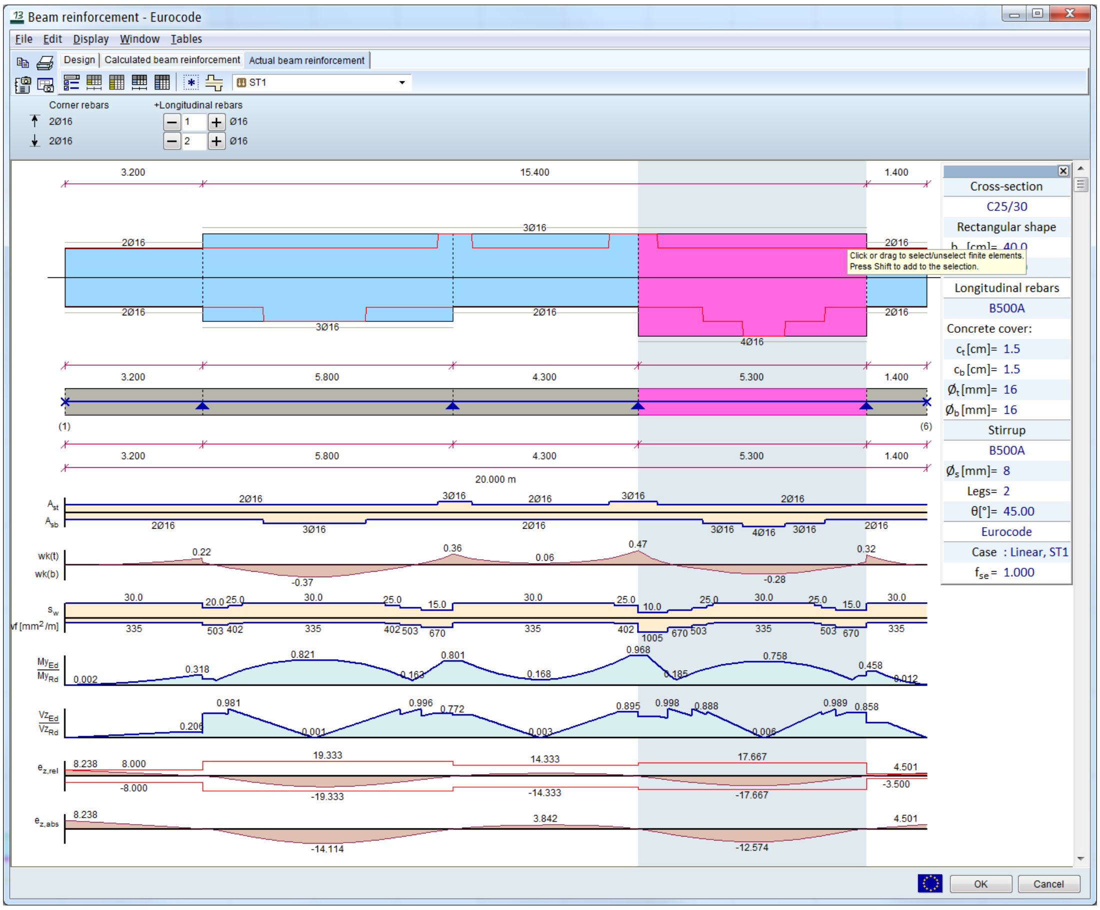


Apply calculated reinforcement

Sets the actual reinforcement according to the A_s values calculated for the current load case or combination over the entire beam.

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.

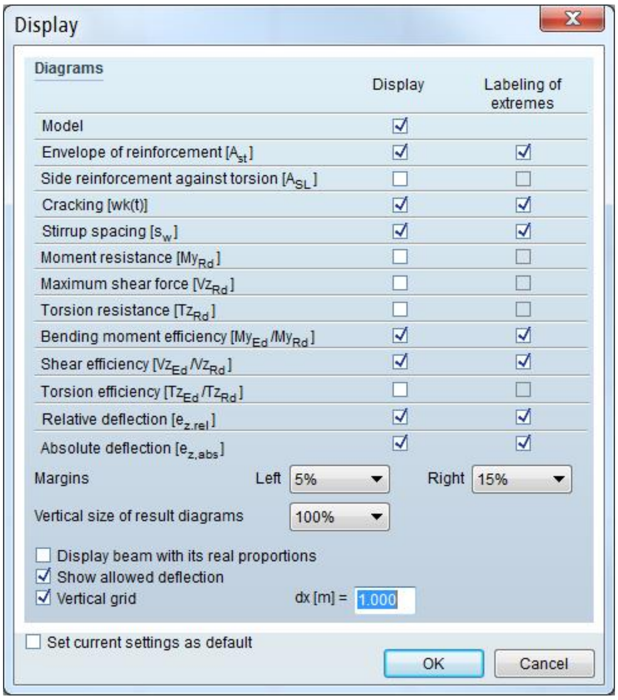


Result display

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.



6.5.10.4. Beam reinforcement according to Eurocode2

Symbols, material properties, partial factors	
f_{cd}	design value of the compressive strength of the concrete
f_{ctd}	design value of the yield strength of the concrete
α	= 0.85; a coefficient, that takes the sustained load and other unfavorable effects into account
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

Shear & torsion reinforcement design of stirrups

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 & torsion reinforcement assuming that shear crack inclination angle is 45°. 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

$$T_{Rd,c} = 2v\alpha_{cw} f_{ctd} t_{efi} A_k \sin\theta\cos\theta$$

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

$$A_{sl} = \frac{T_{Ed} u_k}{2A_k f_{yd} \tan\theta}$$

Spacing of shear & torsion stirrups is calculated from these formulas:

$V_{Rd,s} = \frac{A_{sw}}{s} z f_{ywd} \cot\theta$ and $V_{Rd,s} \geq V_{Ed} + V_{Edi}$,

$$s = \frac{A_{sw}}{V_{Ed} + V_{Edi}} z f_{ywd} \cot\theta$$

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.
If the user chooses the variable angle truss method, AxisVM determines the direction of the shear crack between 21,8° (ctgΘ=2,5) and 45° (ctgΘ=1) before the calculation of the reinforcement so that the exploitation of the inclined concrete compression beams reach its maximum (at most 100%). The shear crack inclination angle is increased in small steps to meet the requirement

$$\frac{V_{Ed}}{V_{Rd,max}} + \frac{T_{Ed}}{T_{Rd,max}} \leq 1$$

☞ ***The cross-section fails if critical shear force is higher than the shear resistance of the compressed concrete beams, i.e.:***

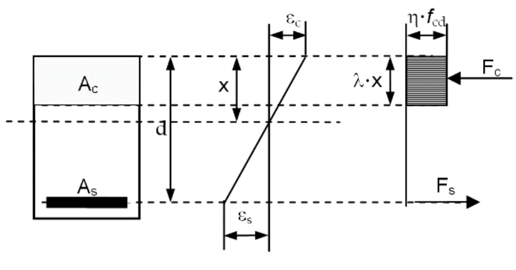
$$\frac{V_{Ed}}{V_{Rd,max}} + \frac{T_{Ed}}{T_{Rd,max}} > 1$$

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) $\rho_w = A_{sw}/s b_w$, 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

AxisVM calculates longitudinal reinforcement according to this figure:



Limit stress is assumed in the rebars. The depth of the compressed zone will be less than

$$x_0 = d \cdot \frac{\epsilon_{cu} - \epsilon_{c1}}{\epsilon_{s1} - \epsilon_{cu}}$$

If calculation results in a greater depth than x_0, a compression reinforcement is applied, but the sum of the area of reinforcement on the compression and on the tension side cannot exceed 4% of the concrete cross-section area.

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.

Minimum ($M_{min} \le 0$) and maximum ($M_{max} \ge 0$) values of the moment diagram and the corresponding reinforcement on tension and compression side is determined. Tension reinforcement is displayed in blue, compression reinforcement in red, the minimal tension reinforcement required by the design code appears in grey.

Compression reinforcement has to be considered even if tension reinforcement is the critical one, as longitudinal rebars thinner than 1/12 of the stirrup distance has to be ignored when determining the compression rebar diameter or the stirrup spacing.

Cracking is calculated according to *6.5.7.1 Calculation according to Eurocode 2*

6.5.10.5. Beam reinforcement according to DIN 1045-1

DIN 1045-1

Symbols, material properties, partial factors	
f_{cd}	design value of the compressive strength of the concrete
f_{ctm}	mean value of the tensile strength of the concrete
α	= 0.85; a coefficient, that takes the sustained load and other unfavorable effects into account
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

Shear & torsion reinforcement design of stirrups

The design is based on the following three values of design shear resistance:

$V_{Rd,ct}$ 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.
$V_{Rd,sy}$ Design shear resistance of the cross-section with shear reinforcement.

No shear reinforcement is required if $V_{Ed} \le V_{Rd,ct}$, DIN 1045-1 10.3.1 (2)
The cross-section does not fail if $V_{Ed} \le V_{Rd,max}$.
If $V_{Ed} > V_{Rd,ct}$ shear reinforcement should be applied DIN 1045-1 10.3.1 (3)
Stirrup spacing is determined to meet the requirement $V_{Ed} \le V_{Rd,sy}$.

For cross sections with shear reinforcement we can choose between the *regular* method (45° cracking) and *Variable Angle Truss* (VAT) method.
If the assumed compression trusses have reserve ($V_{Rd,max} > V_{Ed}$) according to the regular method, the VAT method will lead to considerable savings in shear reinforcement.
By changing the shear crack inclination angle the compressed concrete beams gets more load while shear reinforcement gets less.

The program is calculating the value

$$\cot\Theta = \frac{1.2 - 1.4\dfrac{\sigma_{cd}}{f_{cd}}A_{sw}}{1 - \dfrac{V_{Rd,c}}{V_{Ed}}}$$

In case of regular concrete: $0.58 \le \cot\Theta \le 3.0$
In case of light concrete: $0.58 \le \cot\Theta \le 2.0$ DIN 1045-1 10.3.4 (3)

The regular method assumes the angle of shear cracks to be 45°, so $\cot\Theta = 1$
$V_{Rd,sy} = \frac{A_{sw}}{s_w} f_{yd} z \cot\Theta$ DIN 1045-1 10.3.4 (7)
is the shear resistance due to the shear reinforcement.

If torsion is considerable, AxisVM also checks the following condition:

$$\left[\frac{T_{Ed}}{T_{Rd,max}}\right]^2 + \left[\frac{V_{Ed}}{V_{Rd,max}}\right]^2 \le 1$$ DIN 1045-1 10.4.2 (5)

No calculated shear & torsion reinforcement has to be applied if
$\boldsymbol{T_{Ed} \le V_{Ed} b_w / 4.5}$ and $\boldsymbol{V_{Ed}\left[1 + \frac{4.5 T_{Ed}}{V_{Ed} b_w}\right] \le V_{Rd,ct}}$ DIN 1045-1 10.4.1 (6)

Stirrup reinforcement from twisting moment

Resistant twisting moment on the basis of the failure of the compressed concrete bar:

$$T_{Rd,sy} = \frac{2A_{sw}}{s_w} f_{yd} A_k \cot\Theta$$

The stirrup distance:

$$\boldsymbol{s_w = \frac{2A_{sw}}{T_{Ed}} f_{yd} A_k \cot\Theta}$$

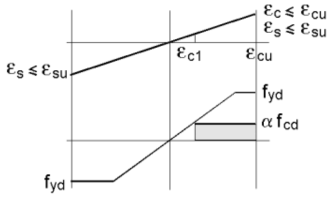
Longitudinal reinforcement is calculated from twisting moment $\boldsymbol{T_{Rd,sy} = \frac{2A_{sl}}{u_k} f_{yd} A_k \tan\Theta}$ **,**
so $\boldsymbol{A_{sl} = T_{Ed} u_k / 2 f_{yd} A_k \tan\Theta}$, which should be placed evenly along the cross-section contour.

The actual stirrup distance is taken into account form the summary of the torsion stirrup distance and the shear stirrup distance**:**

$$s_w = \frac{1}{\frac{1}{s_{w,V}} + \frac{1}{s_{w,T}}}$$

Longitudinal Beam Reinforcement based on DIN1045-1

σ,ε diagrams



The limit stress is developing in the reinforcement. The depth of the compressive concrete zone will exceed

$$x_0 = d \cdot \frac{\epsilon_{c2u} - \epsilon_{c2}}{\epsilon_{s1} - \epsilon_{c2u}}$$

where $\epsilon_{s1} = f_{yd}/E_s$.

If from the calculation a greater height than x_0 is obtained, compressive steel cross section is applied, but the sum of the compressive and tensile steel cross section cannot exceed 8% of the concrete cross section.

The software calculates for each load case and cross section the lower and upper reinforcement, and the value of the moment shifting.
Due to oblique cracks the tension reinforcement is designed for a tension force greater than calculated from *M / z*.
This is taken into account by design codes by shifting the moment diagram (DIN 1045-1 13.2.2)
Minimum $(M_{min} \le 0)$ and maximum $(M_{max} \ge 0)$ values of the moment diagram, and the corresponding tension and compression reinforcements are determined. On the reinforcement diagram the tension reinforcement is displayed in blue, the compressive in red, and the minimal tension reinforcement according to the design code in grey.
The compression reinforcement is necessary even if the tension reinforcement is the critical, because at the determination of the compression reinforcement diameters and stirrup spacing is taken into account that only the 1/12 of the stirrup spacing or longitudinal rebars with greater diameter are included.

Construction rules considered in the program

Ratio of stirrup reinforcement: $\rho_w = \frac{A_{sw}}{s} b_w$
From the above expression: $s_{max,1} = A_{sw}/\rho_w b_w$, where $\rho_w = 0.16 f_{ctm}/f_{yk}$
Minimal value of ρ_w is may calculated from Table 29. in DIN 1045-1 13.1.3
The s_{max} stirrup distance is taking into account Table 31. in DIN 1045-1 13.2.1
The maximum stirrup distance from twisting moments is $u_k / 8$.

☞ ***The software sends warning message and does not draw any reinforcement diagram in the following cases:***

Message ***The cross section is not acceptable for shear/torsion***

Event Any of the following conditions is not satisfied:

$V_{Rd,max} > V_{Ed}$ or $\left[\frac{T_{Ed}}{T_{Rd,max}}\right]^2 + \left[\frac{V_{Ed}}{V_{Rd,max}}\right]^2 \le 1$

Solution Increase the cross section of the concrete, or/and the concrete grade.

Message ***The cross section is not acceptable for bending (As + As2 > 0.08 * Ac)***

Event The cross sectional area of the longitudinal reinforcement is greater than 8% of the concrete cross section

Solution Increase the cross section of the concrete, or/and the concrete grade, or/and the steel grade.

6.5.10.6. Beam reinforcement according to SIA 262:2003

SIA 262:2003

Symbols, material properties, partial factors	
f_{cd}	design value of the compressive strength of the concrete
f_{ct}	design value of the yield 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	= 0.6; reduction factor for compressive strength of the concrete in a cracked zone

Shear & torsion reinforcement design of stirrups

The shear reinforcement design is based on three values of the shear resistance:

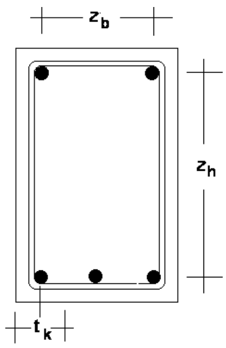
V_{Rd} The shear resistance of the cross section without shear reinforcement.
$V_{Rd,c}$ The maximum shear force that can be transmitted without the failure of the assumed compression bars.
$V_{Rd,s}$ The shear resistance of the cross section with the shear reinforcement.

No shear reinforcement is required if $V_d \leq V_{Rd}$

$V_{Rd} = k_d \tau_{cd} d b_w$, where $k_d = \frac{1}{1+k_v d}$, d in m, $k_v = 2.5$

The conrete cross-section does not fail if $V_{Rd,c} \geq V_d$

$V_{Rd,c} = b_w z k_c f_{cd} \sin\alpha \cos\alpha$



If $V_d \geq V_{Rd,c}$, shear reinforcement should be designed.
The stirrup distance is determined from the expression $V_{Rd,s} = \frac{A_{sw}}{s} z f_{sd} \cot\alpha$
Stirrup spacing is $s = \frac{A_{sw}}{V_d} z f_{sd}\, cot\, \alpha$
Longitudinal force from shear: $F_{td} = V_{Rd} \cot\alpha$
Additional longitudinal reinforcement: $\Delta A_{sl} = V_{Rd} \cot\alpha / f_{sd}$
which should be placed ½ to the tension zone, ½ to the compression zone.
Shear force from torsion: $V_{d,i} = \frac{T_d}{2A_k} z_i$
Shear force in a vertical fiber: $V_{d,h} = T_d/2z_b$
Shear force in the horizontal fiber: $V_{d,b} = T_d/2z_h$
The program checks the following expression $\frac{V_d}{V_{Rd,c}} + \frac{V_{di}}{V_{Rd,ci}} \leq 1$,
where $V_{Rd,ci} = t_k z_h k_c f_{cd} \sin\alpha \cos\alpha$
Stirrup distance from torsion: $s = A_{sw} \frac{2z_h z_b}{T_d} f_{sd} \cot\alpha$
Longitudinal reinforcement from torsion:

$$A_{slT} = \frac{\sum V_{d,i} \cot\alpha}{f_{sd}} = \frac{\frac{T_d}{z_h z_b}(z_h + z_b)\cot\alpha}{f_{sd}}$$

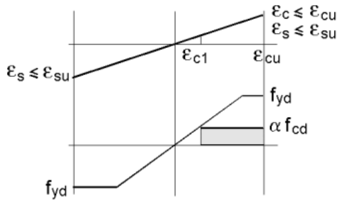
which should be placed evenly along the cross-section contour.

The actual stirrup distance is taken into account from the summary of the torsion stirrup distance and the shear stirrup distance**:**

$$s_w = \frac{1}{\frac{1}{s_{w,V}} + \frac{1}{s_{w,T}}}$$

Beam Longitudinal Reinforcement based on SIA 262:2003

σ,ε diagrams



The limit stress is developing in the reinforcement. The depth of the compressive concrete zone will exceed

$$x_0 = d \cdot \frac{\epsilon_{c2u} - \epsilon_{c2}}{\epsilon_{s1} - \epsilon_{c2u}}$$

where $\epsilon_{s1} = f_{yd}/E_s$.

If from the calculation a greater height than x_0 is obtained, compressive steel cross section is applied, but the sum of the compressive and tensile steel cross section cannot exceed 8% of the concrete cross section.

The software calculates for each load case and cross section the lower and upper reinforcement, and the value of the moment shifting.

Due to oblique cracks the tension reinforcement is designed for a tension force greater than calculated from *M / z.*

This is taken into account by shifting the moment diagram.

Minimum $(M_{min} \leq 0)$ and maximum $(M_{max} \geq 0)$ values of the moment diagram, and the corresponding tension and compression reinforcements are determined. On the reinforcement diagram the tension reinforcement is displayed in blue, the compressive in red, and the minimal tension reinforcement according to the design code in grey.

The compression reinforcement is necessary even if the tension reinforcement is the critical, because at the determination of the compression reinforcement diameters and stirrup spacing is taken into account that only the 1/12 of the stirrup spacing or longitudinal rebars with greater diameter are included.

Construction rules considered in the program

Maximum of the stirrup distance:

$$s_{max} = \frac{A_{sw} f_{yk}}{0.2 \cdot b_w f_{ctm} \sin\alpha} \leq 400 \text{ mm}$$

☞ ***AxisVM sends a warning message and does not draw any reinforcement diagram in the following cases:***

Message ***The cross section is not acceptable for shear/torsion***

Event The efficiency of concrete cross-section is greater than 1.

Solution Increase the cross section of the concrete, or/and the concrete grade.

6.5.11. Punching analysis – RC3 module

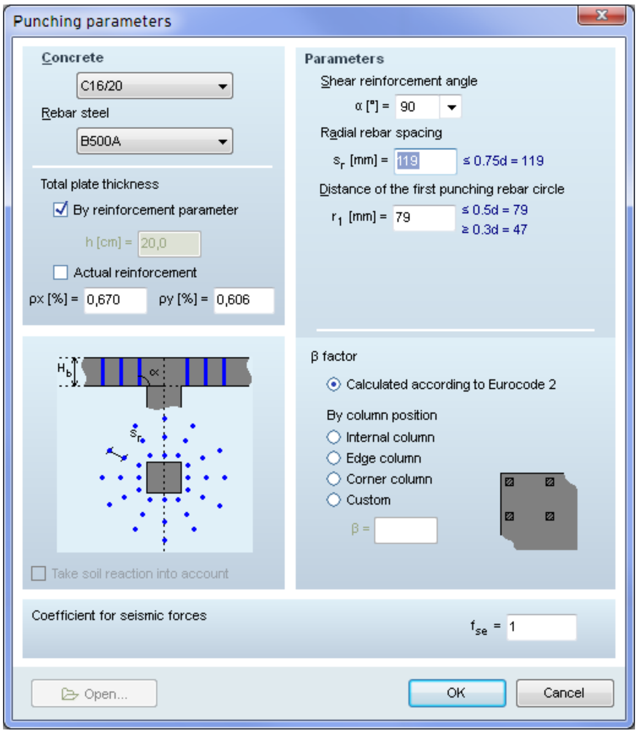
Punching shear control perimeters are determined based on the column cross-section and the effective plate thickness. Plate edges and holes are taken into account if they are closer to the column than six times the effective plate thickness. If column cross-section is concave a convex section is used instead.

Punching analysis can be performed based on the following design codes:

Design Codes

Eurocode 2: *EN 1992-1-1:2004*
DIN: *DIN 1045-1:2001-07*

In order to perform this analysis reinforcement parameters and actual reinforcement must be defined for the reinforced concrete plate.
After clicking the tool button 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).
The following parameters can be set:



Materials

Concrete, Rebar steel
Concrete and reinforcing steel grade used in calculation. These parameters are taken from the actual model by default and can be changed here.

Total plate thickness (h)
Plate thickness is taken from the actual model by default and can be changed here, 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.

Actual reinforcement
If this option is checked the ρx, ρy reinforcement ratios are calculated from the actual reinforcement. If left unchecked these ratios must be specified.

Parameters

Shear reinforcement angle
Angle between the plate and and the punching shear rebars (45°-90°).

Radial rebar spacing
Radial rebar spacing is the difference between the radii of two neighbouring rebar circles. The **OK** button is not available until basic design criteria are met:
MSZ: $t \leq 0.85\,h(1+\cot\alpha)$; EC2: $s_r = 0.75 \cdot d$; DIN: $s_w = 0.75 \cdot d$;

Distance of the first punching rebar circle
Distance of the first punching rebar circle from the convex edge of the column

β factor (Eurocode2 and DIN)

<table>
<tr><td>Calculated based on Eurocode</td><td colspan="3">$1 + k\dfrac{M_{Ed}}{V_{Ed}} \cdot \dfrac{u_1}{W_1}$</td></tr>
<tr><td rowspan="4">Approximate value by column position*</td><td></td><td>Eurocode</td><td>DIN</td></tr>
<tr><td>Internal column</td><td>1,15</td><td>1,05</td></tr>
<tr><td>Edge column</td><td colspan="2">1,4</td></tr>
<tr><td>Corner column</td><td colspan="2">1,5</td></tr>
<tr><td>Custom</td><td colspan="3">user-specified value</td></tr>
</table>

*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.

You can see coefficient of seimic forces at *4.10.23 Seismic loads – SE1 module.*

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 fulfillment of this requirement is not checked. If this requirement is not met, the user should choose a smaller diameter or place additional rebars.

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 H2 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 H1 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.

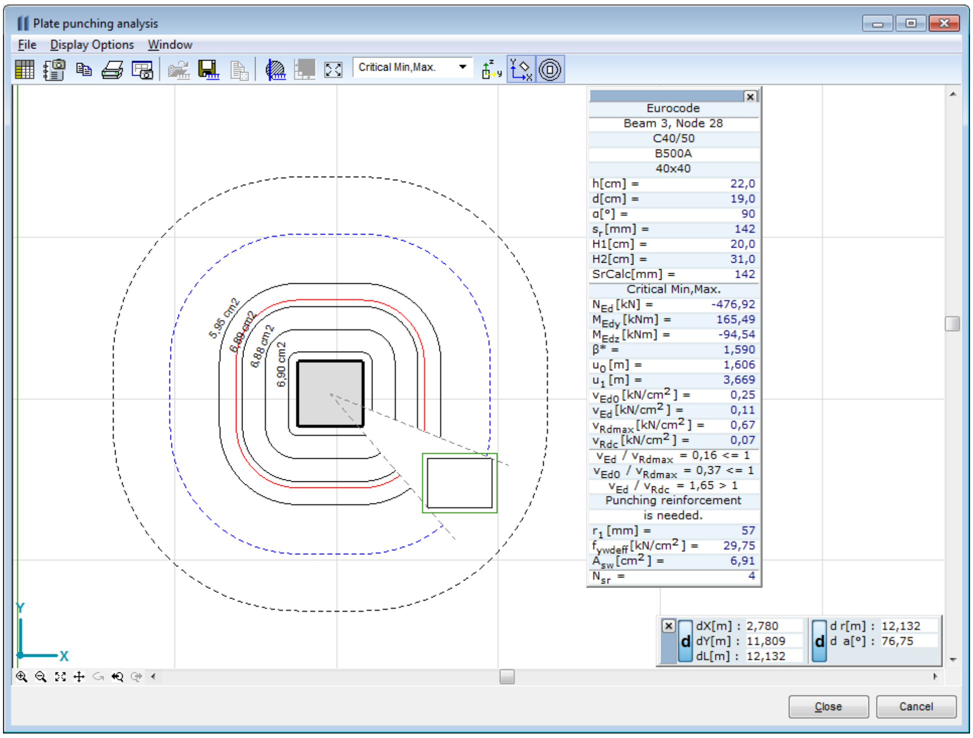
Saves the current punching parameters under a name. You can load back the saved parameters with the button *Loading*... on *Punching parameters* dialog.

Punching parameters dialog.

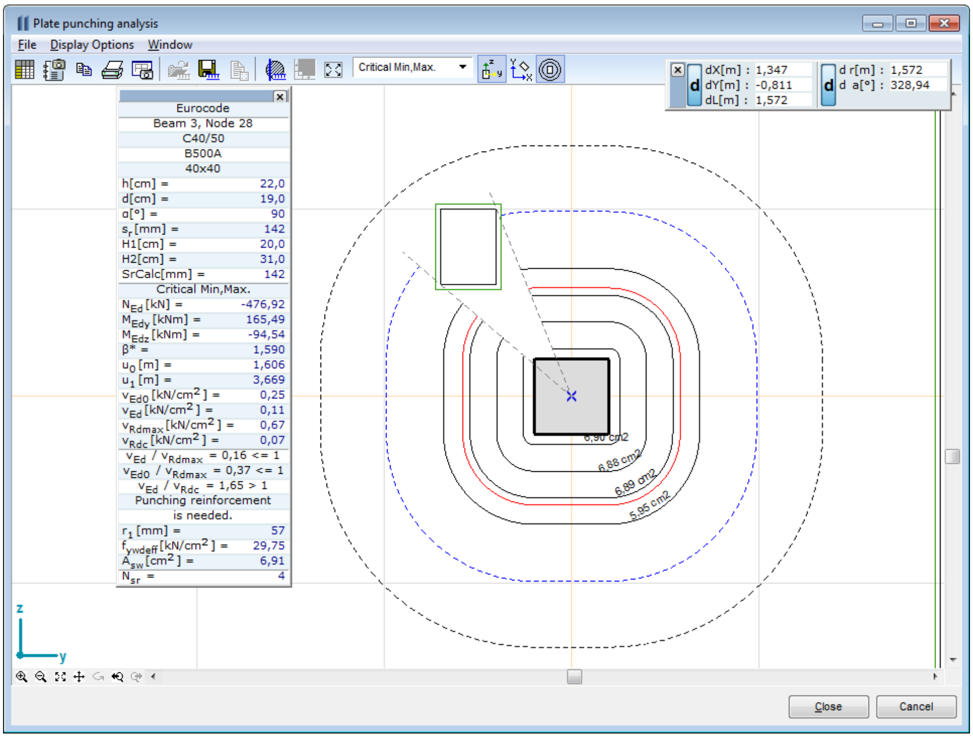
Inflates the plate boundary so that the entire column cross section is within the boundary.

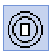
Fits the diagram to the window.

Column local coordinates are used.



Global coordinates are used.





Turns on and off the display of rebar circles.

To follow design calculations in detail click on the *Design calculations* button. **See…** also *6.6 Steel design*

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.

6.5.11.1. Punching analysis according to Eurocode2

The required punching reinforcement is calculated based on the following principles:
The column-plate 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:

$$v_{Ed} \leq v_{Rd,max} \text{ and } v_{Ed} \leq v_{Rd,cs}$$

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:

$$v_{Rd,c} = C_{Rd,c} k (100 \rho_1 f_{ck})^{1/3} + k_1 \sigma_{cp} \geq v_{min} + k_1 \sigma_{cp}$$

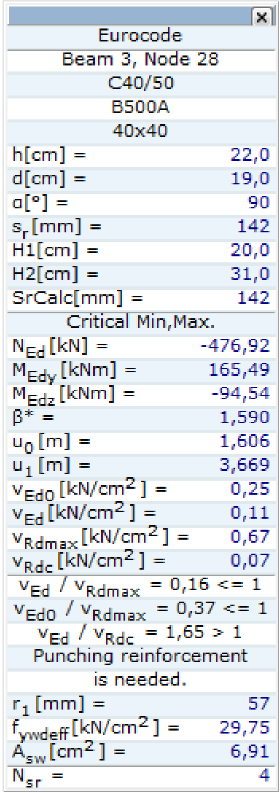
If $v_{Ed} > v_{Rd,c}$ then the required punching reinforcement is determined along the critical perimeter

$$v_{Rd,cs} = 0.75 \cdot v_{Rd,c} + 1.5 \cdot \frac{d}{s_r} \cdot \frac{A_{sw} f_{ywd,ef}}{u_1 d} \sin\alpha\,; \quad v_{Ed} \leq v_{Rd,cs}$$

The reinforcemert for each perimeter and the perimeter where no punching reinforcement is needed is calculated based on the formula:

$$v_{Ed} = \beta \frac{V_{Ed}}{u_i d} \leq v_{Rd,c}$$

Info window Under the design code, element identifier and materials the following parameters are displayed.



Eurocode	
Beam 3, Node 28	
C40/50	
B500A	
40x40	
h[cm] =	22,0
d[cm] =	19,0
α[°] =	90
s_r[mm] =	142
H1[cm] =	20,0
H2[cm] =	31,0
SrCalc[mm] =	142
Critical Min,Max.	
N_{Ed}[kN] =	-476,92
M_{Edy}[kNm] =	165,49
M_{Edz}[kNm] =	-94,54
β* =	1,590
u_0 [m] =	1,606
u_1 [m] =	3,669
v_{Ed0} [kN/cm^2] =	0,25
v_{Ed} [kN/cm^2] =	0,11
v_{Rdmax} [kN/cm^2] =	0,67
v_{Rdc} [kN/cm^2] =	0,07
v_{Ed} / v_{Rdmax} = 0,16 <= 1	
v_{Ed0} / v_{Rdmax} = 0,37 <= 1	
v_{Ed} / v_{Rdc} = 1,65 > 1	
Punching reinforcement is needed.	
r_1 [mm] =	57
f_{ywdeff}[kN/cm^2] =	29,75
A_{sw}[cm^2] =	6,91
N_{sr} =	4

h: plate thickness
d: effective plate thickness
α angle between the plate and the punching reinforcement
s_r: distance of reinforcement circles
H1: minimum plate thickness required with punching reinforcement
H2: minimum plate thickness required without punching reinforcement
N_{Ed}: design value of the punching force
M_{Edx}, M_{Edz} design value of the moment
β:* calculated excentricity factor
u_0: control perimeter at the column perimeter
u_1: critical control perimeter at *2d*
v_{Ed0}: shear stress along the u_0 perimeter
v_{Ed}: shear stress along the u_1 perimeter
v_{Rdmax}: maximum of shear stress
v_{Rdc}: shear stress without reinforcement
v_{Ed}/v_{Rdmax} : efficiency on the critical control perimeter
v_{Ed0}/v_{Rdmax} : efficiency on the u_0 perimeter
v_{Ed}/v_{Rdc} : efficiency (tension in concrete)
r_1: distance between the first rebar circle and the convex column edge
f_{ywdeff}: tension in the punching reinforcement
A_{sw}: punching reinforcement area on the critical control perimeter
N_{sr}: number of reinforcement circles

6.5.11.2. Punching analysis according to DIN 1045-1

The required punching reinforcement is calculated according to the following principles:
The column-plate 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: $v_{sd} \leq v_{Rd}$
The design value of the shear stress is $v_{sd} = \beta \frac{V_{sd}}{u \cdot d}$, where β is a factor expressing additional stress due to eccentric forces.
DIN 1045-1 assumes that the critical section is at a distance of *1,5d* from the edge of the cross-section.

Design value of the punching resistance of the connection without punching shear reinforcement is determined using the formula

$$v_{Rd} = f\left(v_{Rd,ct}, v_{Rd,cta}, v_{Rd,max}, v_{Rd,sy}\right)$$
$$v_{Rd,ct} = \left(0.14 \cdot \eta_1 \kappa (100 \cdot \rho_1 f_{ck})^{1/3} + 0.12\sigma_{cd}\right) \cdot d$$
$$v_{Rd,cta} = \kappa_a v_{Rd,ct}$$

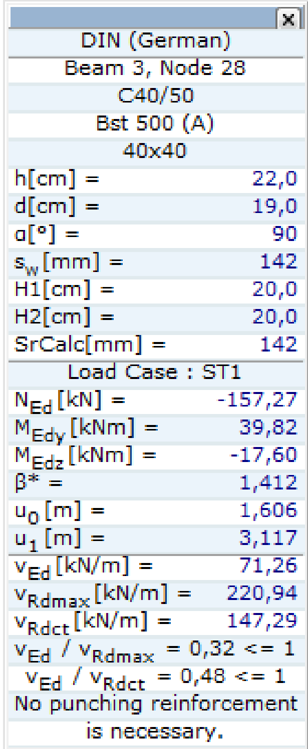
The design value of the maximum punching shear resistance is $v_{Rd,max} = 1.7 \cdot v_{Rd,ct}$
On the first perimeter at a distance of $r_0 = 0.5 \cdot d$ from the cross-section edge the required amount of punching shear $v_{Rd,sy0} = v_{Rd,c} + \frac{\kappa_s A_{sw0} f_{yd}}{u_0}$
Design value of the punching resistance of the connection with punching shear reinforcement is

$$v_{Rd,sy} = v_{Rd,c} + \frac{\kappa_s A_{sw} f_{yd} d}{u_i s_w}$$

If $v_{sd} > V_{Rd,ct}$, the required amount of punching shear reinforcement is calculated along the critical perimeter using the requirement $v_{sd} \leq v_{Rd,sy}$.

Info window

Under the design code, element identifier and materials the following parameters are displayed.



h: plate thickness
d: effective plate thickness
α: angle between the plate and the punching reinforcement
sw: distance of reinforcement circles
H1: minimum plate thickness required with punching reinforcement
H2: minimum plate thickness required without punching reinforcement
N_{Ed}: design value of the punching force
M_{Edx}, M_{Edz} design value of the moment
β: excentricity factor
u_0: control perimeter at the column perimeter
u_1: critical control perimeter at *2d*
v_{Ed}: shear stress along the u_1 perimeter
v_{Rdmax}: maximum of shear stress
v_{Rdct}: shear stress without reinforcement
v_{Ed}/v_{Rdmax} : efficiency on the critical control perimeter
v_{Ed}/v_{Rdct} : efficiency (tension in concrete)
κ_s correction factor: $1+\sqrt{\frac{200}{d}} \leq 2$
r_1: distance between the first rebar circle and the convex column edge
A_{sw}: punching reinforcement area on the critical control perimeter
N_{sr}: number of reinforcement circles

Warnings and error messages

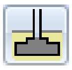
Message ***Compression force in plate is too high.***

Event The applied force is so high that the concrete plate fails irrespectively of the reinforcement.

Solution The most efficient solution is to increase plate thickness.
The critical punching area can be extended by increasing plate thickness and/or column size (reducing the design value of the specific shear force this way).
Choose a higher grade concrete.

6.5.12. Footing design – RC4 module

6.5.12.1. Pad footing design



AxisVM can determine the necessary size and reinforcement of rectangular pad foundations (with or without pedestal), and can check the footing against sliding and punching according to Eurocode7 and MSz. 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.

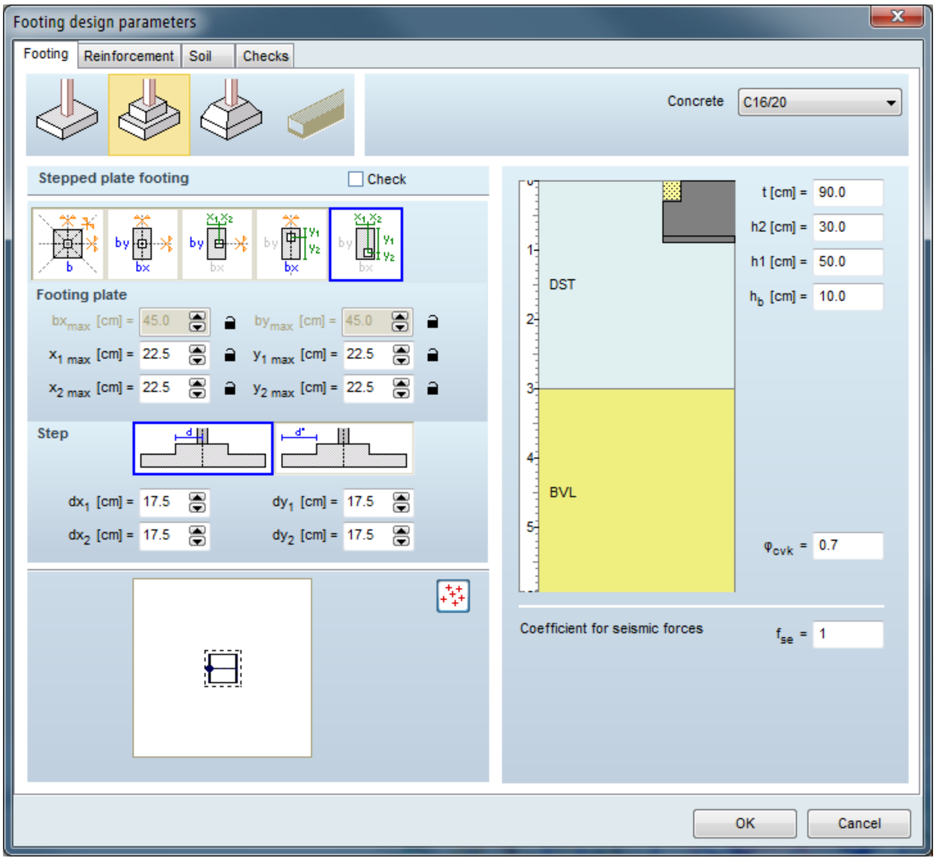
☞ ***The coordinate system used in footing calculations is the coordinate system of the support. However eccentricites are calculated from the center of the footing.***

Footing design parameters

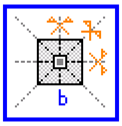
Click the *Footing design* icon and select one or more nodal supports with a vertical or slanted column. (If supports have been already selected, the dialog is displayed at the first click).
Footing desgn parameters have to be specified in a dialog.

Footing

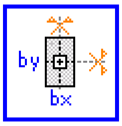
At the *Footing* tab select the footing type (simple plate / stepped / sloped) and set the geometry parameters and the friction coefficient between the footing and the blind concrete.



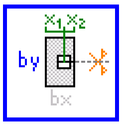
Symmetry of footing



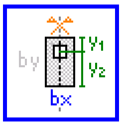
Square footing
b is the side length,
the column is concentric,
value or upper limit of *b* must be entered



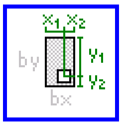
Rectangular footing
bx and *by* are the sides,
the column is concentric,
value or upper limit of *bx* and *by* must be entered



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

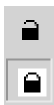


Single eccentric rectangular footing
the column is eccentric in *y* direction, concentric in *x* direction
y_1 and y_2 are the distance of the column axis from the edges of the footing
value or upper limit of y_1, y_2 and *bx* must be entered



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

If the lock button beside the edit field is down (closed), the entered value is given (it is checked). If the lock icon is up (open) the entered value is the upper limit (it is determined by the program). If *Check* is turned on, all values will be closed and cannot be opened until *Check* is turned off.

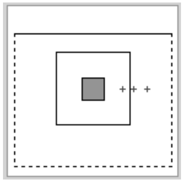


For stepped and sloped footings:
dx_1 and dx_2 are the distance of the edges of the step or the upper base of the frustum from the column axis in x direction. dy_1 and dy_2 are the distance of the edges of the step or the upper base of the frustum from the column axis in y direction. These are always given values.

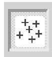
Footing parameters:

Concrete	material of the footing
t	foundation depth (distance between the bottom of the base plate and the 0 level)
h_2	step height (height of the step or the frustum)
h_1	base plate thickness
h_b	blind concrete thickness
φ_{cvk}	friction coefficient between the footing and the blind concrete

You can see coefficient of seimic forces at *4.10.23 Seismic loads – SE1 module.*



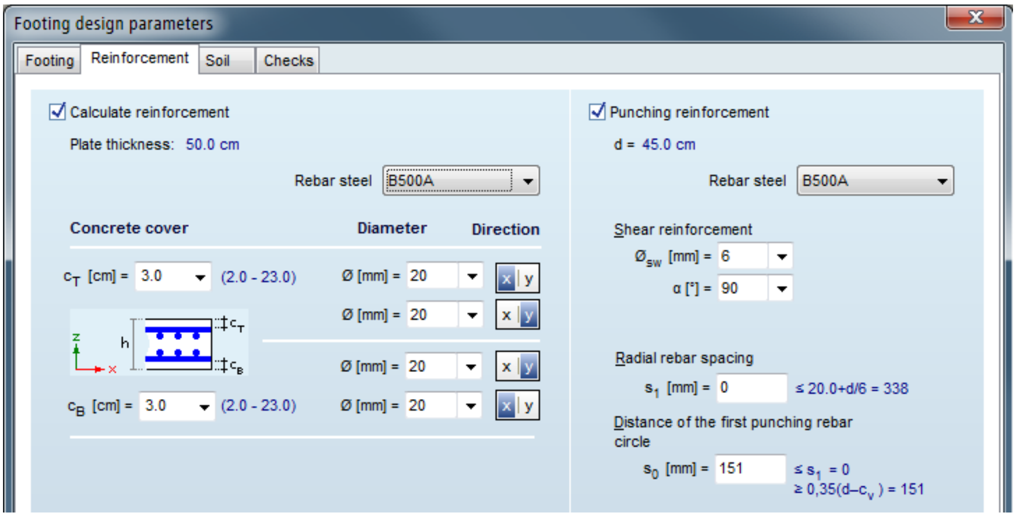
Under the edit fields the footing and the column is displayed in top view. Given sizes are drawn as continuous lines, upper limits as dashed lines. The forces appear as red crosses placed according to their eccentricities. This diagram is for orientation purposes only because the actual eccentricities are calculated taking into account the self weight of the footing and the backfill reducing the eccentricity.



If the 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 weigth of the backfill, footing and blind concrete.

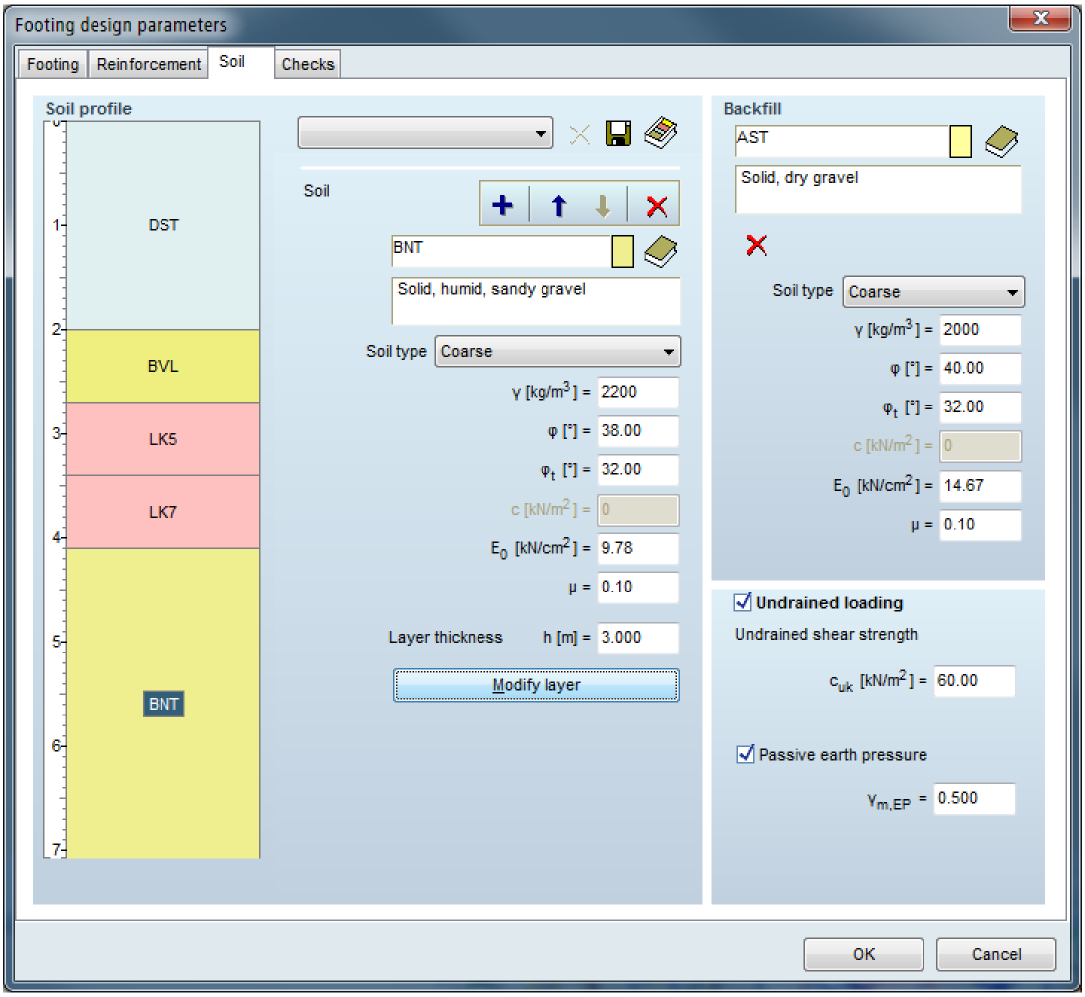
Reinforcement

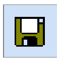
On the *Reinforcement* tab reinforcement calculations can be activated.
Rebar steel grade, c_T and c_B top and bottom concrete covers can be entered (i.e. the least distance between the surface of embedded reinforcement and the outer surface of the concrete). The top two rows under *Diameter* and *Direction* 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…** *6.5.11 Punching analysis – RC3 module*



Soil

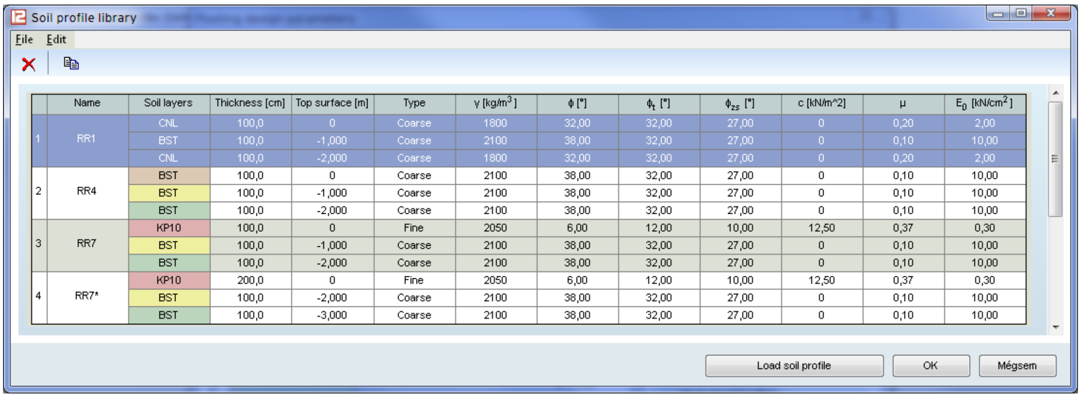
At the *Soil* tab you can specify the soil profile and the properties of the backfill. Soil profiles can be saved under a name and can be reloaded.
Properties of the selected layer is displayed in the *Soil* group box. Properties of the backfill is displayed in the *Backfill* group box.
Soil layer properties can be changed. These changes can be applied to the soil layer clicking the *Modify layer* button. Layer name and description can be modified. Layer color can be changed clicking the small color rectangle beside the name. Soil library icon is placed beside the color rectangle. Clicking this icon a soil library is displayed with predefined layer properties.





Saves the soil profile under a name. This way you can reload the same soil profile for other footings in the model.
If *Save a copy to the soil profile library* is checked the soil profile is also saved to a library. This way you can reload the same soil profile in different models.

Opens the soil profile library.



	Name	Soil layers	Thickness [cm]	Top surface [m]	Type	γ [kg/m³]	ϕ [°]	$ϕ_t$ [°]	$ϕ_{zs}$ [°]	c [kN/m^2]	μ	E_0 [kN/cm²]
1	RR1	CNL	100,0	0	Coarse	1800	32,00	32,00	27,00	0	0,20	2,00
		BST	100,0	-1,000	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00
		CNL	100,0	-2,000	Coarse	1800	32,00	32,00	27,00	0	0,20	2,00
2	RR4	BST	100,0	0	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00
		BST	100,0	-1,000	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00
		BST	100,0	-2,000	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00
3	RR7	KP10	100,0	0	Fine	2050	6,00	12,00	10,00	12,50	0,37	0,30
		BST	100,0	-1,000	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00
		BST	100,0	-2,000	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00
4	RR7*	KP10	200,0	0	Fine	2050	6,00	12,00	10,00	12,50	0,37	0,30
		BST	100,0	-2,000	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00
		BST	100,0	-3,000	Coarse	2100	38,00	32,00	27,00	0	0,10	10,00

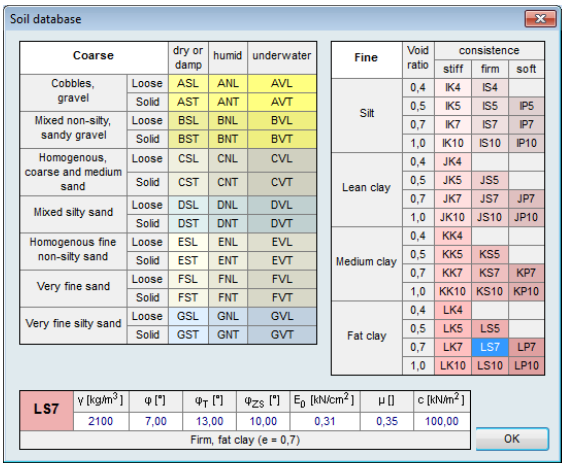
Deletes the selected soil profile

Soil layers have the following properties:

Soil type	coarse, coarse underwater or fine
Thickness	layer thickness
Top surface	Position of the top surface relative to the ground level
γ [kg/m³]	mass density
φ [°]	internal angle of friction
$φ_t$ [°]	Angle of friction between the soil and concrete
E_0 [N/mm²]	Young modulus of the soil
μ []	Poisson coefficient of the soil
c [kN/m²]	cohesion (only for fine soils)

Soil database

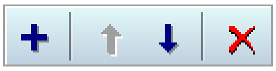
Clicking the *Soil database* icon two tables are displayed. After selecting a soil and clicking the **OK** button (or double clicking the soil) properties of the selected soil are copied to the *Soil* or *Backfill* group box.



Coarse		dry or damp	humid	underwater
Cobbles, gravel	Loose	ASL	ANL	AVL
	Solid	AST	ANT	AVT
Mixed non-silty, sandy gravel	Loose	BSL	BNL	BVL
	Solid	BST	BNT	BVT
Homogenous, coarse and medium sand	Loose	CSL	CNL	CVL
	Solid	CST	CNT	CVT
Mixed silty sand	Loose	DSL	DNL	DVL
	Solid	DST	DNT	DVT
Homogenous fine non-silty sand	Loose	ESL	ENL	EVL
	Solid	EST	ENT	EVT
Very fine sand	Loose	FSL	FNL	FVL
	Solid	FST	FNT	FVT
Very fine silty sand	Loose	GSL	GNL	GVL
	Solid	GST	GNT	GVT

Fine	Void ratio	consistence		
		stiff	firm	soft
Silt	0,4	IK4	IS4	
	0,5	IK5	IS5	IP5
	0,7	IK7	IS7	IP7
	1,0	IK10	IS10	IP10
Lean clay	0,4	JK4		
	0,5	JK5	JS5	
	0,7	JK7	JS7	JP7
	1,0	JK10	JS10	JP10
Medium clay	0,4	KK4		
	0,5	KK5	KS5	
	0,7	KK7	KS7	KP7
	1,0	KK10	KS10	KP10
Fat clay	0,4	LK4		
	0,5	LK5	LS5	
	0,7	LK7	LS7	LP7
	1,0	LK10	LS10	LP10

LS7	γ [kg/m³]	φ [°]	$φ_T$ [°]	$φ_{ZS}$ [°]	E_0 [kN/cm²]	μ []	c [kN/m²]
	2100	7,00	13,00	10,00	0,31	0,35	100,00
Firm, fat clay (e = 0,7)							



The function available on the *Soil* toolbox are: *Add new soil layer, Move up, Move down, Delete.*

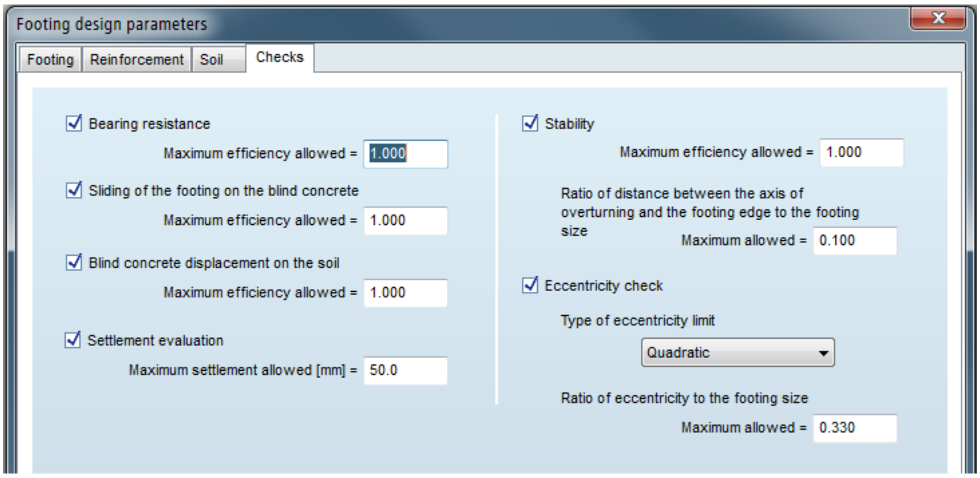
Add new soil layer	Adds a new soil layer with the properties and layer thickness set in the group box. The new layer always gets to the bottom of the soil profile.
Move up	Moves the selected soil layer up within the soil profile.
Move down	Moves the selected soil layer down within the soil profile.
Delete	Deletes the selected soil layer from the soil profile.

Modify soil layer Name, color, description and physical properties of the selected layer can be edited. Click on this button to apply changes to the selected layer.

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 deteremined 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 Required checks and allowed maximum efficiencies can be selected on the *Checks* tab.
If *Settlement evaluation* is activated the program checks whether the settlement is greater than the limit specified here but does **not** increase footing size.
Stability check allows setting the maximum allowed eccentricity ratio to ensure that the soil will be compressed under the footing to avoid overturning.



Bearing resistance The size of the footing is increased until the efficiency for soil rupture falls below the allowed maximum:

$$\lambda_{R,v} = \frac{V_d}{R_{V,d}} \leq \lambda_{R,v,lim}$$

Warnings and errors: If the bigger size of the footing exceeds 10 times the thickness a warning appears.

Sliding check The module determines if the design stress caused by horizontal force is under the sliding resistance between 1) the soil and the blind concrete, 2) the blind concrete and the foundation calculated from the effective area. $\tau_{Ed} \leq \tau_{Rd}$ and $\tau_{Ed2} \leq \tau_{Rd2}$.

Analysis of multi-layer soil If the soil has multiple layers the program calculates the efficiency from the stress at the top of the soil layer and the bearing resistance assuming that the stress spreads in 45°. Footing size is increased until efficiency for soil rupture falls below the limit.

$$B_i' = B' + 2(z_i - D) \text{ and } L_i' = L' + 2(z_i - D) \rightarrow A_i' = B_i' \cdot L_i'$$

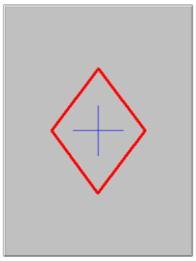
$$q_{E,d,i} = \frac{V_d}{A_i'} + (q_i' - q')$$

Stability AxisVM calculates the moment of actions around the axes of overtuning, sums up the stabilizing and destabilizing moments then checks the following:

$$\Delta_{EQU} = \left|\frac{M_{dst}}{M_{stb}}\right| \leq \Delta_{EQU,lim}$$

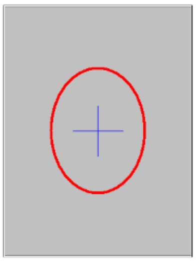
Eccentricity check Footing size is increased until the eccentricity factor (γ_{ecc}) calculated from load combination eccentricities falls below the allowed limit: $\gamma_{ecc} \leq \gamma_{ecc,lim}$.
$\gamma_{ecc} = 0.5$ means that the the bounding rectangle of the eccentricity limit curve has the 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),



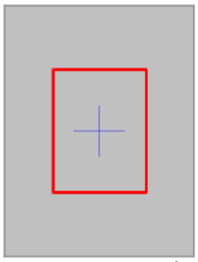
$$\gamma_{ecc} = \left|\frac{e_x}{b_x}\right| + \left|\frac{e_y}{b_y}\right|$$

Quadratic (ellipse)



$$\gamma_{ecc} = \sqrt{\left(\frac{e_x}{b_x}\right)^2 + \left(\frac{e_y}{b_y}\right)^2}$$

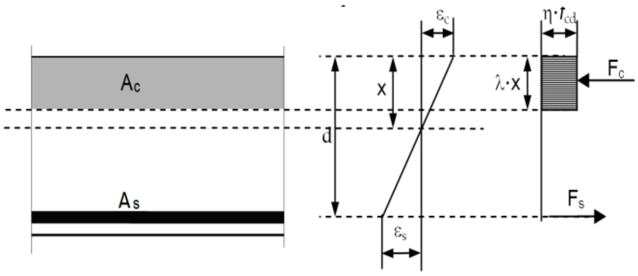
Axis of rotation (rectangle)



$$\gamma_{ecc} = \max\left(\frac{|e_x|}{b_x}; \frac{|e_y|}{b_y}\right)$$

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 ($A_s > 0.04A_c$).

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 1997-1:2004, 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 $q_{Rd} = s_\gamma \cdot \gamma' \cdot B' \cdot N_\gamma \cdot i_\gamma \cdot b_\gamma \cdot 0.5 + s_q \cdot q \cdot N_q \cdot i_q \cdot b_q + s_c \cdot c \cdot N_c \cdot i_c \cdot b_c$

Sliding check calculates if the footing meets the following criterion between the footing and the blind concrete and between the blind concrete and the soil: $H_d \leq R_d + R_{p,d}$
where H_d is the design value of the horizontal force, R_d is the design shear resistance, $R_{p,d}$ is the passive soil resistance at the side of the footing.

Design shear resistance is obtained from the formula $R_d = V_d \tan\delta_d$, where V_d is the design vertical action, δ_d is the design angle of friction: $\delta_d = \arctan\left(\frac{\tan\varphi}{\gamma_\varphi}\right)$, where φ is the angle of interface friction, γ_φ is the partial factor of shearing resistance, prescribed by the design approach.

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 $v_{Ed} \leq v_{Rd}$

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 $v_{Ed} \leq v_{Rd,c}$, 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 efficiency 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*):

$$\sigma_z = \frac{4\sigma_0}{2\pi}\left\{\arctan\left[\frac{b}{z}\cdot\frac{a(a^2+b^2)-2az(R-z)}{(a^2+b^2)(R-z)-z(R-z)^2}\right]+\frac{bz}{b^2+z^2}\cdot\frac{a(R^2+z^2)}{(a^2+z^2)R}\right\}$$

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:

$$h_{hi} = h_i \cdot \left(\frac{E_{si}}{E_{s0}}\cdot\frac{\rho_0}{\rho_i}\right)^{2/5}$$

where

h_{hi} is the effective thickness of the soil layer *i*

h_i is the thickness of the soil layer *i*

E_{s0} is the Young modulus of the the base layer

E_{s1} is the Young modulus of the soil layer *i*

ρ_0 is the density of the base soil layer

ρ_i is the density 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:

$$\Delta h_i = h_i\frac{\sigma_{ai}}{E_{si}};\ \ \sigma_{ai} = \frac{\sigma_{i-1}+\sigma_i}{2}$$

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_{si}: 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:

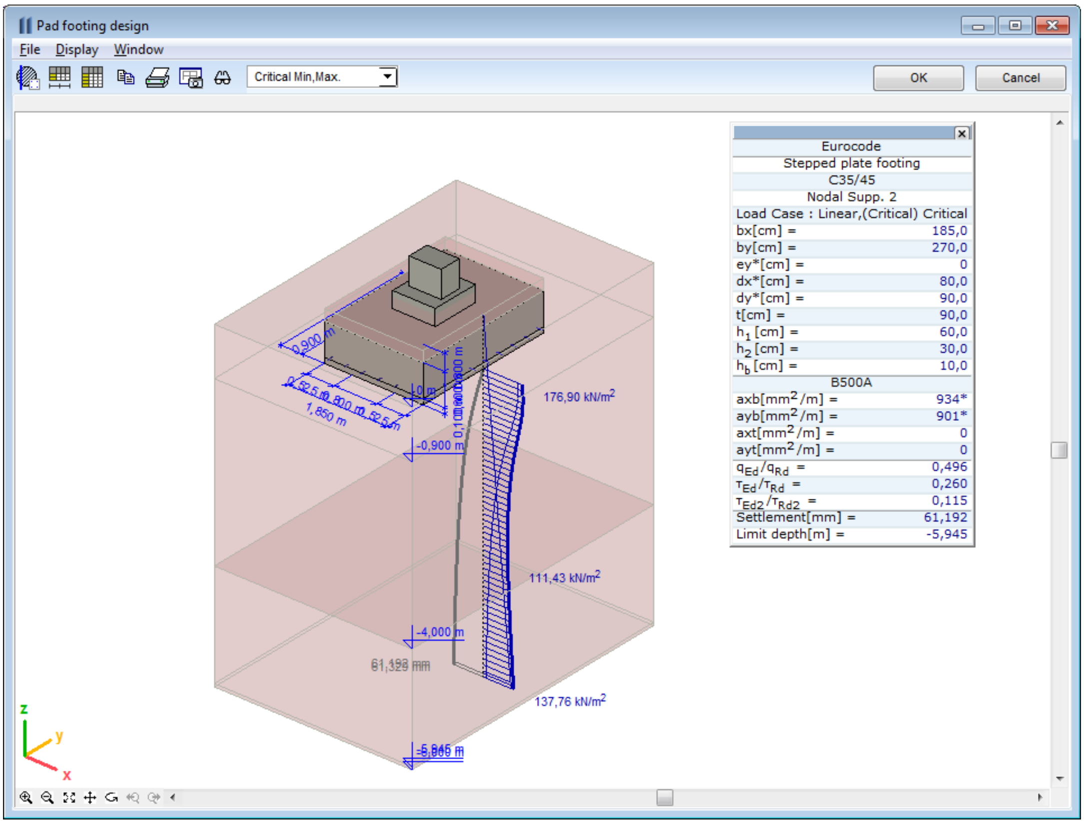
$$s_m = \sum_{i=0}^{m}\Delta h_i$$

AxisVM calculates the limit depth, where $\sigma = 0.1\cdot\sigma_{ob}$ (i.e. the extra stress caused by loading falls under the 10% 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*.

Results

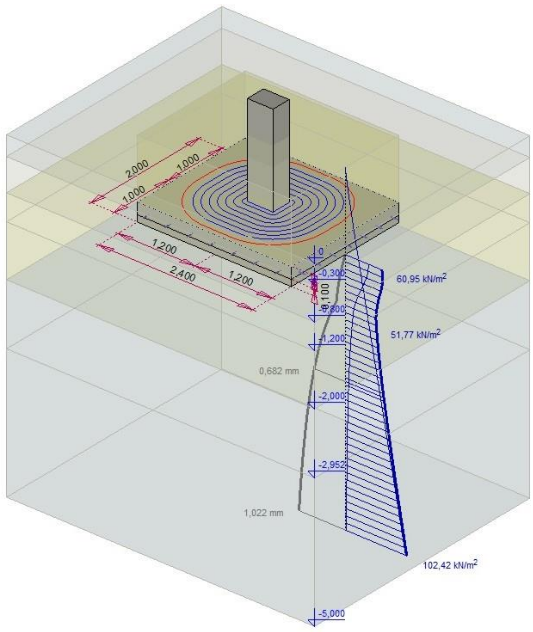
The designed foundation will be displayed in top view with soil layers, punching circles and places dimension lines automatically. The 3D model can be zoomed in and out, shifted and rotated just like the main model.

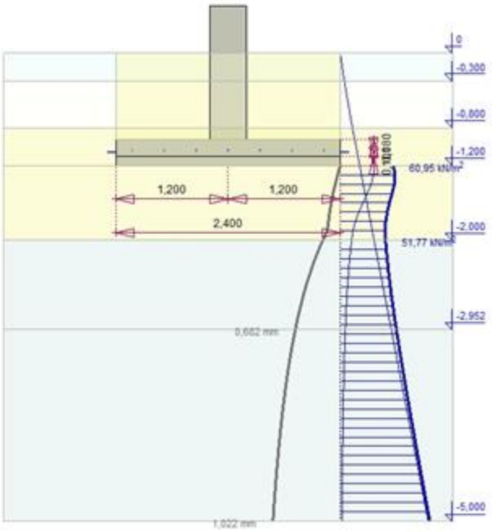


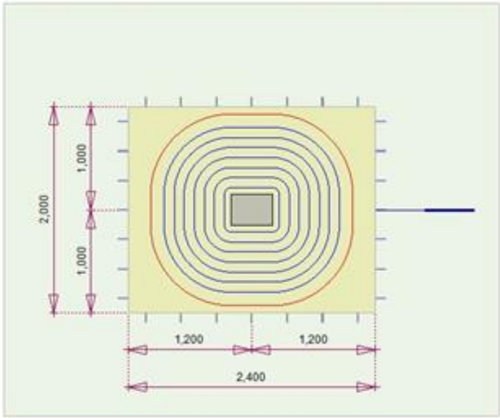
If the display of settlement is activated (see *Display parameters)* a thick blue diagram plots the total soil stress against depth. Thin diagrams show the stress due to loading and the self-weight of the soil. The first one is decreasing, the second one is increasing with depth. Horizontal lines show the sublayers. The gray 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 fo 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.

Rx, Ry, Rz, Rxx, Ryy, Rzz	support forces
q_{Ed}	design bearing pressure
q_{Rd}	design bearing resistance
q_{Ed}/q_{Rd}	soil utilization factor
Axb	local *x* direction bottom reinforcement (if calculated)
ayb	local *y* direction bottom reinforcement (if calculated)
axt	local *x* direction top reinforcement (if calculated)
ayt	local *y* direction top reinforcement (if calculated)
τ_{Ed}/τ_{Rd}	efficiency based on footing displacement relative to the blind concrete
τ_{Ed2}/τ_{Rd2}	efficiency based on blind concrete displacement relative to the soil
v_{Ed}/v_{Rd}	efficiency based on punching (for simple plate footings)
Settlement	predicted settlement of the footing
bx, by	footing base plate size in *x* and *y* direction
dx, dy**	pedestal (step or frustum) size in *x* and *y* direction
ex, ey**	eccentricity of the pedestal's center of gravity in *x* and *y* direction

Detailed internal forces

Displays the data in the table of *Footing internal forces* and the following results:

Design approach	design approach used to calculate the results of the line
c_x, c_y	*x* and *y* size of the effective rectangle
e_x, e_y	eccentricity of action in *x* and *y* direction
Rebars xb	rebar scheme in bottom *x* direction (if calculated)
Rebars yb	rebar scheme in bottom *y* direction (if calculated)
Rebars xt	rebar scheme in top *x* direction (if calculated)
Rebars yt	rebar scheme in top *y* direction (if calculated)
τ_{Ed}	design shear stress between the footing and the blind concrete
τ_{Rd}	design shear resistance between the footing and the blind concrete
τ_{Ed2}	design shear stress between the soil and the blind concrete
τ_{Rd2}	design shear resistance between the soil and the blind concrete

V_{Rdc}	minimum shear design resistance without punching reinforcement
V_{Rdmax}	maximum shear design resistance without punching reinforcement
V_{Rdcs}	shear design resistance with punching reinforcement
u_1	length of the critical line
A_{sw}	shear reinforcement along the punching line
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 determinded at that point and is greater than 0.1, otherwise it is 0.1)
Limit depth	the depth where stress ratio is 0.1 (if limit depth is greater than the bottom of the layer structure a *?* is displayed)

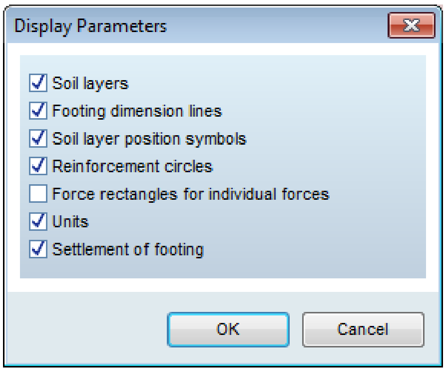
Copies image to the Clipboard

Prints image to the Clipboard

Saves the drawing into the Drawing Library

Display parameters

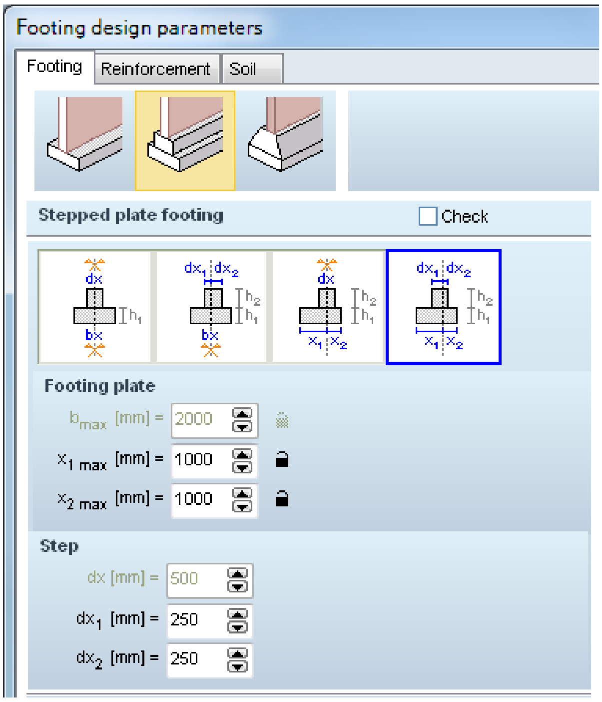
Turns on and off symbols of the drawing.



6.5.12.2. 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 and MSz. 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.



6.5.13. Design of voided slabs – CBX/ADK module

If the AxisVM configuration includes the COBIAX (CBX) module, it is possible to place void formers into slabs achieving weight reduction (and concrete reduction) and making larger spans available. For definition of COBIAX slabs **see** *4.9.5.2.COBIAX-domain – CBX module.*
For definition of AIRDECK slabs **see** *4.9.5.3 AIRDECK-domain – ADK module*

Design codes

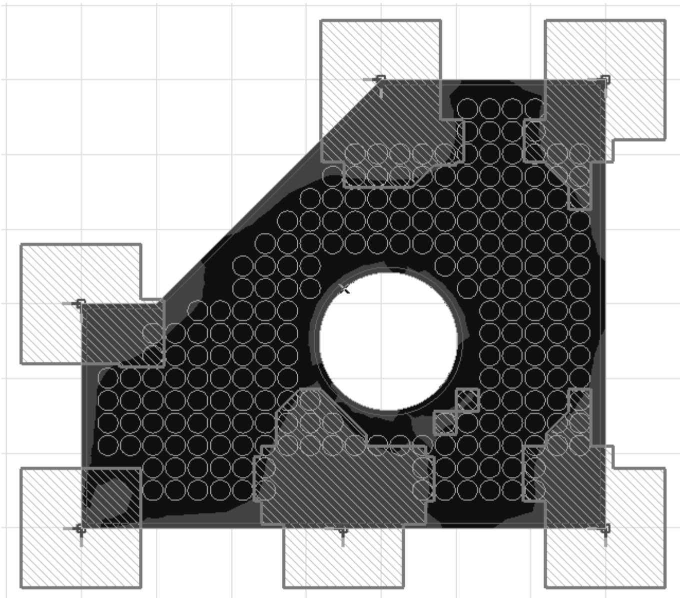
This design is available according to Eurocode, DIN 1045-1 and SIA (Swiss) design code and design of AIRDECK slab is available according to Eurocode.

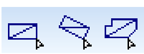
COBIAX/AIRDECK design must take into account that void formers reduce the stiffness and shear resistance of the slab. The effect of smaller bending stiffness can be seen in the results. Where shear forces would exceed the reduced shear resistance, placing of void formers must be avoided.
If the user defined the surface reinforcement parameters AxisVM calculates the design results used in reinforcement design. One of these design components is the difference between the actual shear force and the shear resistance. If actual reinforcement is also defined AxisVM calculates with the actual reinforcement.
Clicking on the Cobiax/Airdeck icon *vSz–vRd,c* will be displayed setting the color legend to show positive values (where shear force exceeds the resistance) in red and negative values in blue.
No void formers should be placed into the red zones. In other words, these must be converted to solid areas.

Defining solid areas

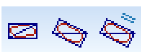
A toolbar appears to help solid area definition.
Existing solid areas and their polygon vertices can be moved.
Clicking on the *Update model* button converts solid areas into new domains without void formers. Due to changes in the model all resuls will be cleared.

Based on shear force isolines
AxisVM determines where to form solid areas based on the isolines of *vSz–vRd,c*.





The easiest way to create solid areas by hand is to draw rectangles, slanted rectangles or polygons.



The next three buttons are tools converting the *bounding rectangle* of an isoline into a solid area. The first one creates a rectangle parallel to global directions. The second one creates an optimized (smallest) rectangle. The third one creates a rectangle with two edges parallel to a given line.

Bounding circle of an isoline.

These three buttons works like the previous group but it is the *area* of the rectangle that will match the area within the isoline.

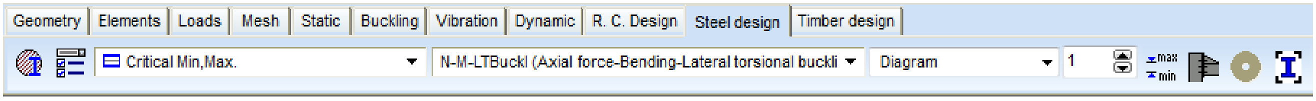
Clicking into the interior of domains converts them to solid areas. This tool is useful to convert domains created from solid areas back to solid areas and modify them. After clicking *Update model* the original domain will be updated processing changes in the outline of the solid area.

Deletes solid areas. Click the outline of the solid area to select it.

Deletes domains created from solid areas. Click the domain outline to select it. Deleting domains changes the model so existing results will be lost.

Update model replaces solid areas with domains without void formers. Running the analysis again it can be checked whether any void former falls into a red zone. If so, new solid areas must be added or existing areas (domains) must be converted to solid areas and extended to remove void formers from red zones. The cycle of running the analysis and checking the distribution must be repeated until all void formers are removed from red zones.

6.6. Steel design



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 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 fibre 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 positive

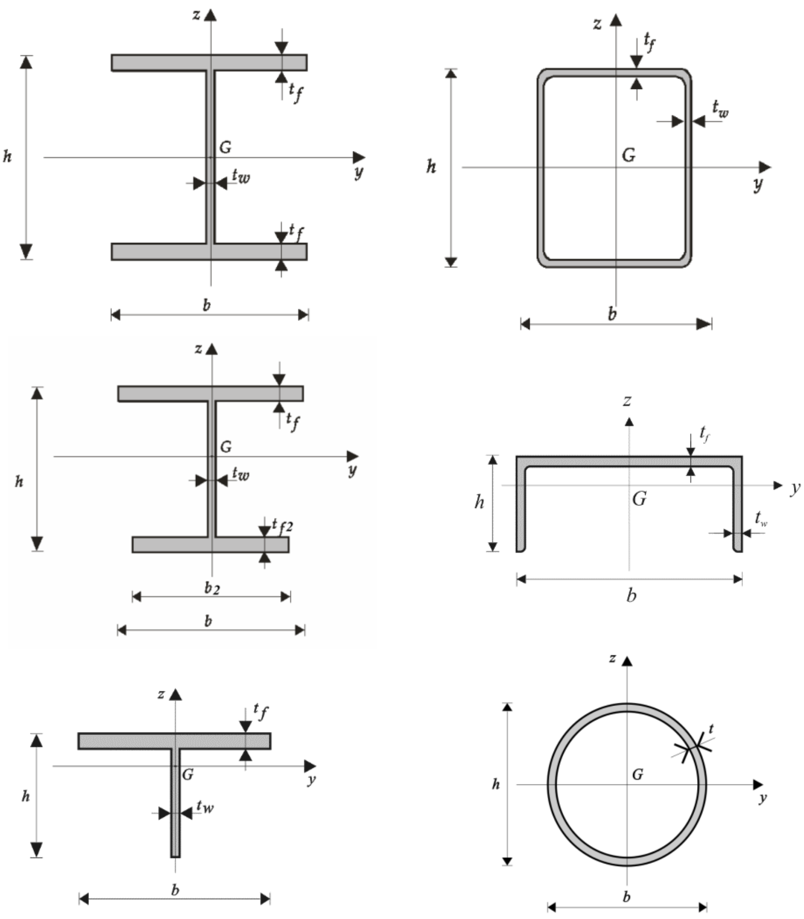
$W_{eff,(+),min}$ refers to those where the moment is negative

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 *TableBrowser*. 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.***

The principal axes of an arbitrary cross section have to be coincident with the local y and z axes.



Classes of Cross-Sections

The program is identifying the class of the cross-section based on EN 1993-1-1, Table 5.2, considering coexisting compression and bending.

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)
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 bending 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)

These information are 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).

$$\frac{N_{Ed}}{N_{Rk}/\gamma_{M_0}} + \frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rk}/\gamma_{M_0}} + \frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M_0}} \leq 1$$

$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$: it differs from zero only when the cross section is in class 4 and the original cross section is asymmetric to axis y.

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.

Plastic resistance check

For I, pipe and box shaped secions 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:

$$\left[\frac{M_{y,Ed}}{M_{N,y,Rd}}\right]^{\alpha} + \left[\frac{M_{z,Ed}}{M_{N,z,Rd}}\right]^{\beta} \leq 1$$

Compression-Bending-Buckling

The check is based on EN 1993-1-1, 6.3.3 (6.61) and (6.62):

$$\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M_1}} + k_{yy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rk}/\gamma_{M_1}} + k_{yz}\frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \leq 1$$

$$\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M_1}} + k_{zy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{M_{y,Rk}/\gamma_{M_1}} + k_{zz}\frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \leq 1$$

$$(\chi_{LT} = 1.0)$$

$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$ differs from zero only when the cross section is in class 4 and the original cross section is assymetric to axis y.

Axial Force-Bending-Lateral Torsional Buckling

When determining the lateral-torsional buckling resistance it is assumed that the cross section is constant and symmetric for the local z axis. It is also assumed the the loads act in the plane of symmetry, that is the plane of bending. 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 :

$$\frac{N_{Ed}}{\chi_y N_{Rk}/\gamma_{M_1}} + k_{yy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} M_{y,Rk}/\gamma_{M_1}} + k_{yz}\frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \le 1$$
$$\frac{N_{Ed}}{\chi_z N_{Rk}/\gamma_{M_1}} + k_{zy}\frac{M_{y,Ed} + \Delta M_{y,Ed}}{\chi_{LT} M_{y,Rk}/\gamma_{M_1}} + k_{zz}\frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M_1}} \le 1$$

$\Delta M_{y,Ed} = N_{Ed} \cdot e_{N,y}$: it differs from zero only when the cross section is in class 4 and the original cross section is assymetric to axis y.

χ_{LT} is calculated according to EN 1993-1-1 6.3.2.2 or 6.3.2.3.
The determination of the interaction factors of $k_{yy}, k_{yz}, k_{zy}, k_{zz}$ is based on EN 1993-1-1, Appendix B Method 2 (Tables B.1 and B.2).
The equivalent uniform moment factors C_{my}, C_{mz}, C_{mLT} are listed in Table B.3.
For tensile axial force, the check is performed using the effective moments based on ENV 1993-1-1, 5.5.3.

Shear /y The check is performed on the basis of EN 1993-1-1, 6.2.6.

$$\frac{V_{y,Ed}}{V_{c,y,Rd}} \le 1$$

Shear /z The check is performed on the basis of EN 1993-1-1, 6.2.6.

$$\frac{V_{zEd}}{\min(V_{czRd}, V_{bRd})} \le 1$$

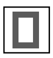
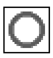
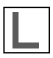
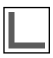
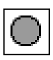
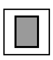
$V_{b,Rd} = V_{bw,Rd}$: The resistance is calculated with the contribution of the web but not the flanges.

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.

$$\frac{M_{Ed}}{M_{pl,Rd}} + \left(1 - \frac{M_{fRd}}{M_{pl,Rd}}\right) \cdot \left(2\frac{V_{Ed}}{V_{bw,Rd}} - 1\right)^2 \le 1$$

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		✓	✓	*in case of normal force (no bending)*	✓	✓	-	-
U		✓	✓	*if bending acts in the plane of symmetry*	✓	✓	-	-
C		✓	✓	*if bending acts in the plane of symmetry*	✓	✓	-	-
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 (*)		✓	✓	✓	✓	✓	✓	✓
2L		✓	✓	-	✓	✓	-	-
2L if a=0 (*)		✓	✓	✓	✓	✓	-	-
2U opened][		✓	✓	-	✓	✓	-	-
2U opened][if a=0 (*)		✓	✓	✓	✓	✓	✓	✓
2U closed []		✓	✓	-	✓	✓	-	-
2U closed [] if a=0 (*)		✓	✓	✓	✓	✓	✓	✓

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		-						
Arc		✓	✓	-	✓	✓	-	-
Half circle		✓	✓	-	✓	✓	-	
Reg. polygon shape		✓	✓	-	✓	✓	-	-
Wedged I		✓	✓	✓	✓	✓	✓	✓
Complex/ Other (**)		✓	✓	-	✓	✓	-	-

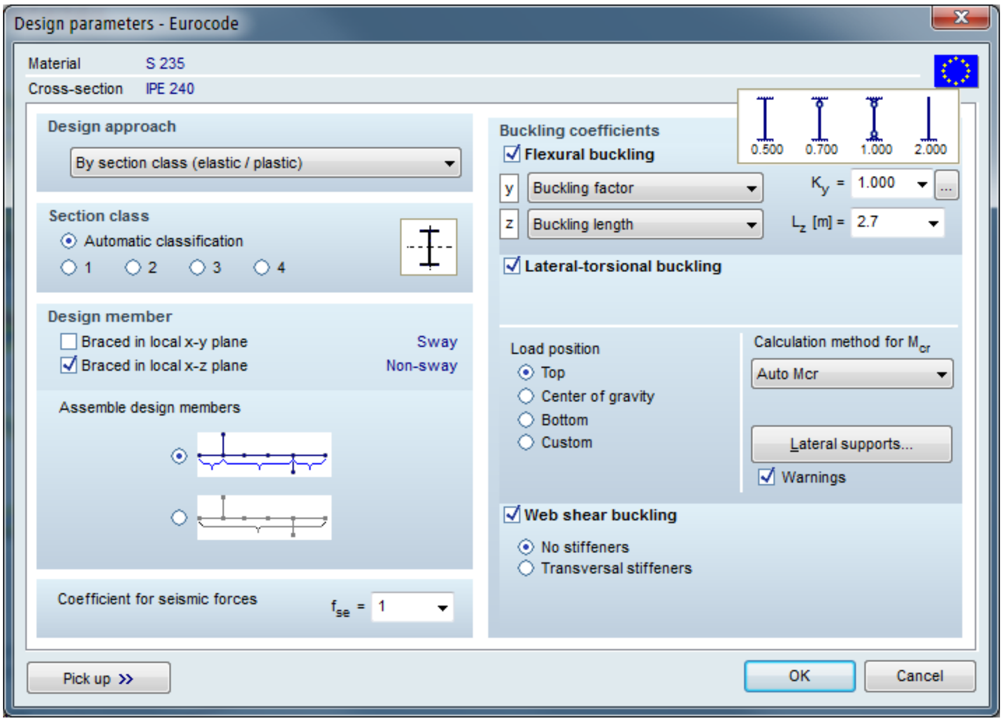
(*) For double-section types if the distance between the two sections is zero, the program will assume that the connection between the elements is continuous and will replace the two with one section (I, T or box). The connection needs to be calculated by the user.

(**) These sections are designed only if local coordinates are the same as principal directions.

☞ ***If the manufacturing process of the section is cold-formed or other, the member is not designed.***

Design Parameters

For the design based on Eurocode 3, the following design parameters should be defined and assigned to the structural members:



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.

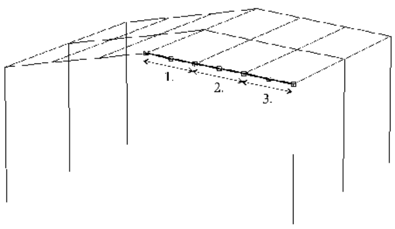
Design member bracing If the design member has a sway frame buckling mode in local *x-y* or *x-z* plane the respective bracing must be turned off. These settings affect the automatic flexural buckling calculation (AutoNcr), and in case of lateral torsional buckling the lateral supports for the AutoMcr method. Furthermore, they affect equivalent uniform moment factors *Cmy* and *Cmz* of the stability interaction check (EN 1993-1-1: 2005 Annex B: Method 2: Table B.3).

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 and local system orientation. Finite elements must be on the same line.

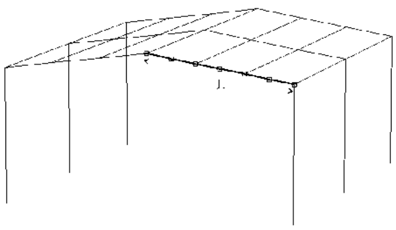
☞ ***Steel design members are not the same as the structural members (See... 3.2.13 Assemble structural members)***

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.



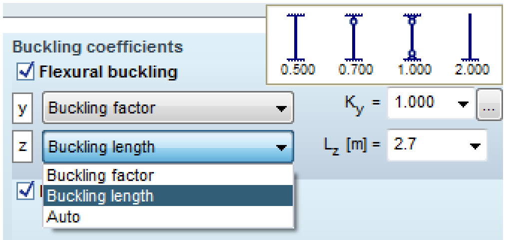
Factor for seismic forces **see...** *4.10.23 Seismic loads – SE1 module*

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.

Buckling (flexural)

To determine buckling resistance in *y* and *z* directions end support conditions must be defined. There are three ways to specify buckling behaviour.



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)!

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.

Auto. 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 structure 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 endpoints 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 endpoints 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 and rib elements. Design members are considered to have constant cross-section and effects from other structural parts (e.g. plates, springs, rigid bodies, line or surface supports) are ignored. Normal force along each element is considered to be constant and equal to its maximum value in the element.

Additional information on the automatic flexural buckling coefficient calculation tool is available from the menu *Help / AutoNcr Guide*.

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.

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

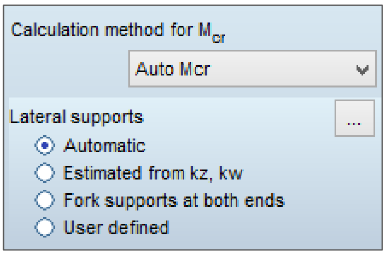
$$M_{cr} = C_1 \frac{\pi^2 E I_z}{(kL)^2}\left[\sqrt{\left(\frac{k}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(kL)^2 G I_t}{\pi^2 E I_z} + \left(C_2 z_g - C_3 z_j\right)^2} - \left(C_2 z_g - C_3 z_j\right)\right]$$

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 depends 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 $K_z > 1$.
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.

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 submodel 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]

Lateral supports



AxisVM determines the support conditions for the submodel automatically by default.
Alternatively the following methods may be chosen:
Estimated from kz, kw: the position of the supports are estimated similarly a sin version 12 of AxisVM;
Fork supports at both ends;
User defined: the user may edit/define the supports.

The four stiffness components of a lateral supports are R_y, R_{xx}, R_{zz}, R_w.
It the above mentioned option 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 $R_w = 0$.
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 submodel does not take into account the effect of nodal DOFs.

Without having proper supports the submodel 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 $k_z = k_w = 1$ 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.

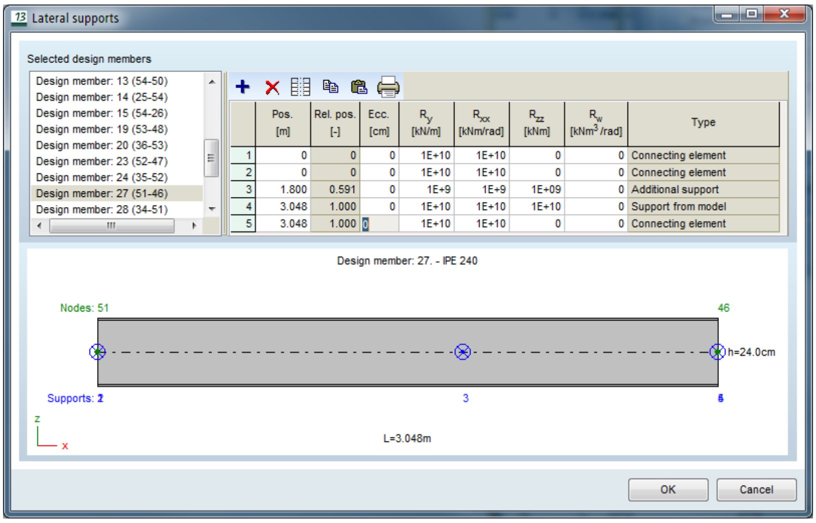
Lateral support conditions can be edited by clicking the … button.
A dialog appears with a table of lateral supports of the design members created from the selection. Design members are listed on the left. The table shows the lateral supports of the selected item.
The last item of the list is *Same supports 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.

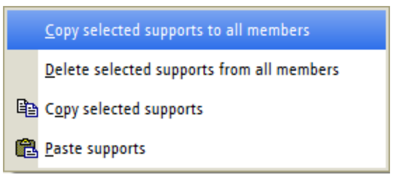


The table displays connecting elements even if the angle between their axis and the design member axis is greater than 15°, but only if the direction vector of the connecting element has a nonzero component in the local y axis of the design member (lateral support). So if the design member is a horizontal beam, vertical columns and other horizontal beams in line with the design member will not appear in the table.
Table fields allow entering numbers and certain parametric values as well:

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.

Right click on a list item on the left for additional operations

Right click on a table row for additional operations



☞ ***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.***

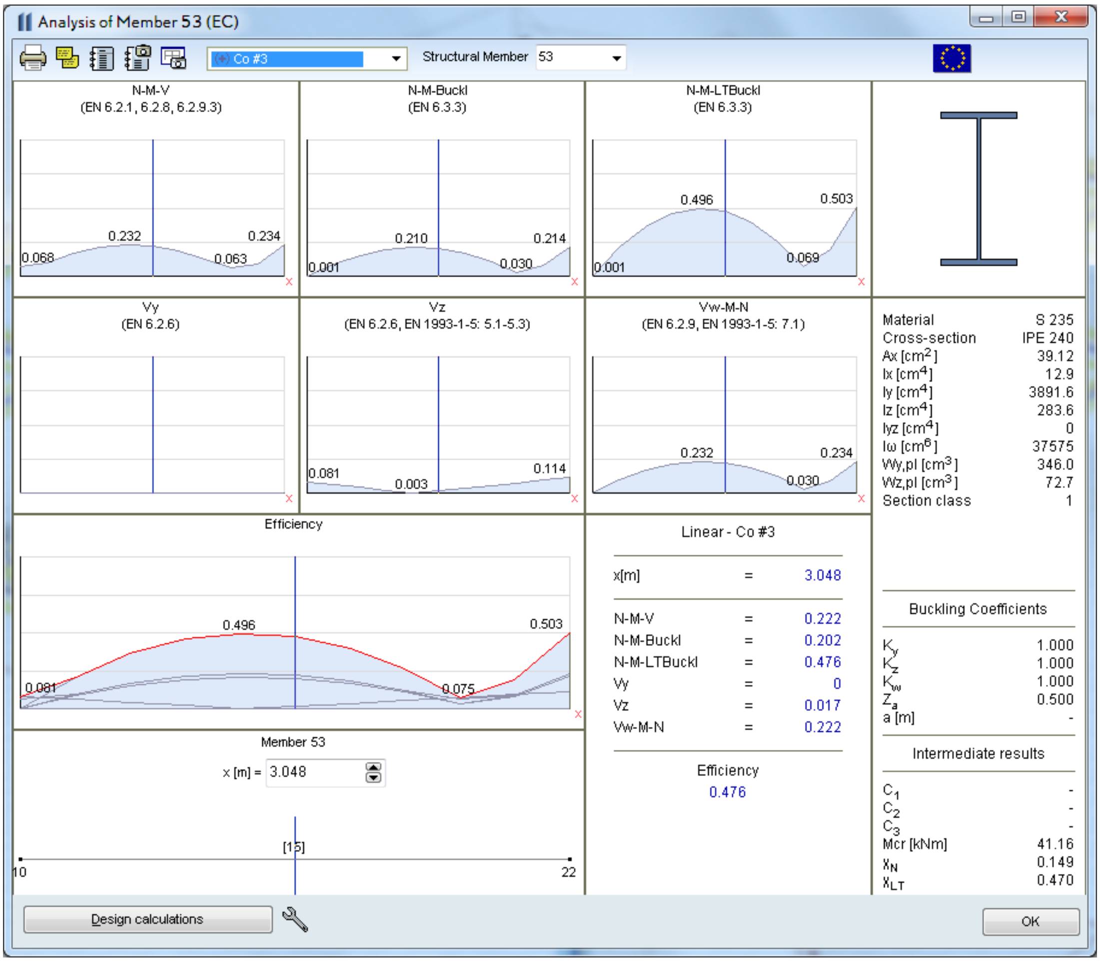
Version 12 estimated support conditions from k_z and k_w values. Models created with this version will appear with *Estimated from kz, kw* setting.
In order to help using and understanding the *AutoMcr* method and defining the lateral supports correctly, guidelines and examples can be found in the *AutoMcr 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 *4.10.23 Seismic loads – SE1 module.*

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.

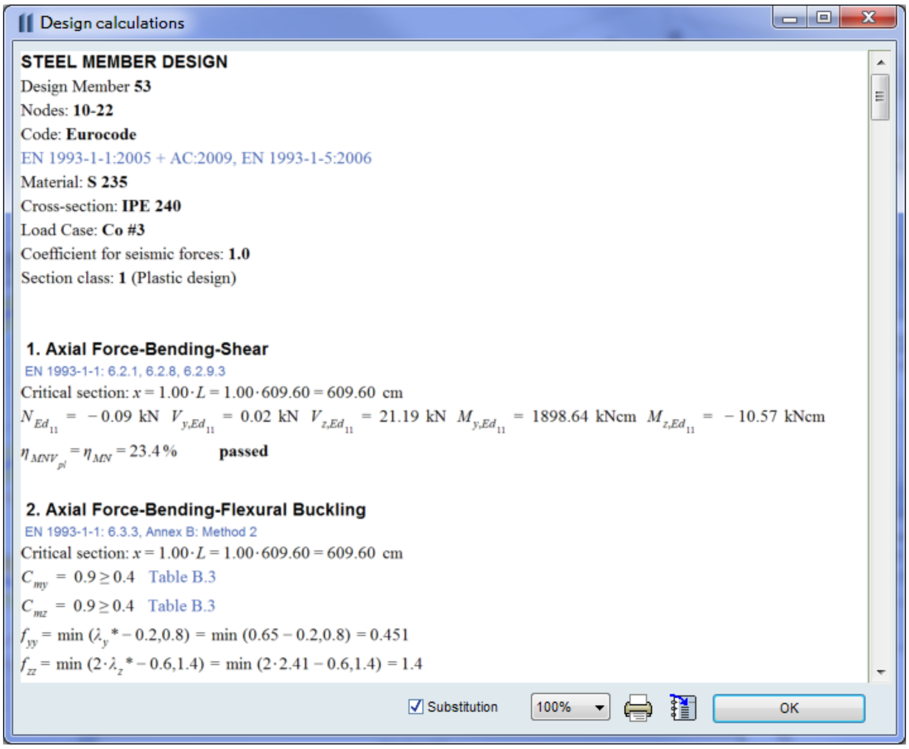
The report consists of the six basic interaction checks listed above and several partial results which make it easier to follow the calculations and provide useful details for cross-section optimization.

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*

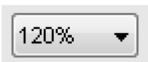
Clicking on the *Settings* icon beside the *Design calculations* button allows setting the basic units for force and length used in the design calculations. Important results also appear converted to standard AxisVM units (**see…** *3.3.8 Units and Formats*).

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.



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.

6.6.1.1. 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, 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 seach for the cross-section with an efficiency < 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 efficiency can also be set. Limiting the efficiency 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 and web buckling can be deactivated.

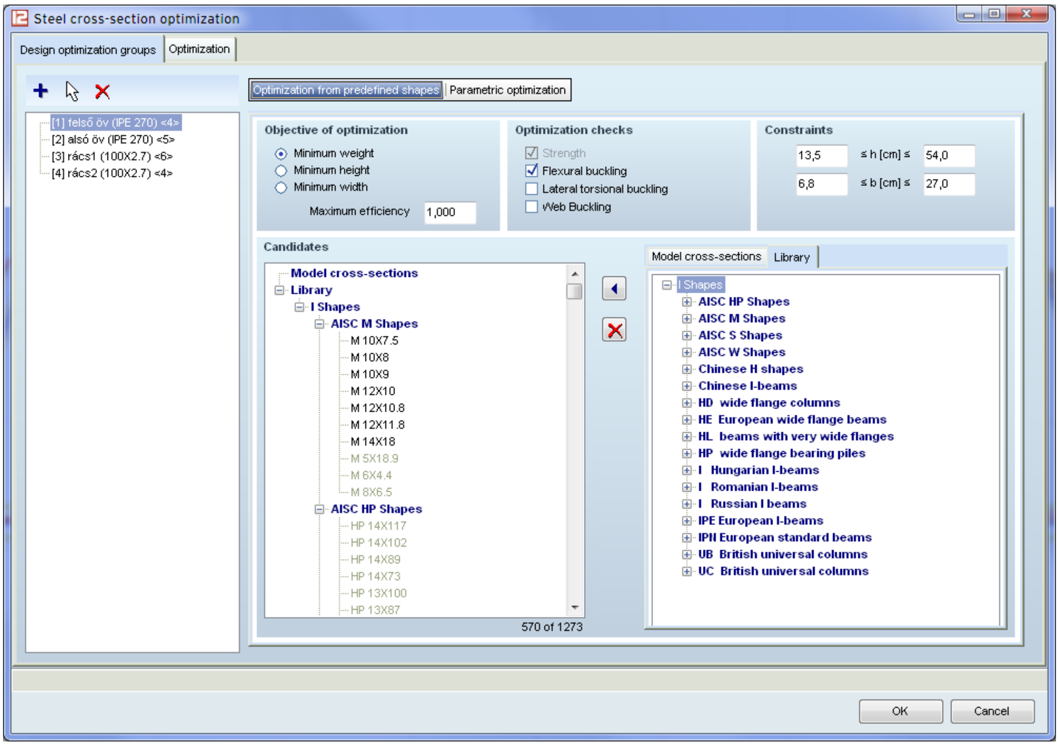
Optimization types

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.

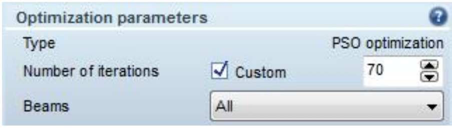
The range of candidates can be reduced by setting *Constraints*. Only cross-sections between the limits for height and width will be used as candidates (other sections will appear greyed).



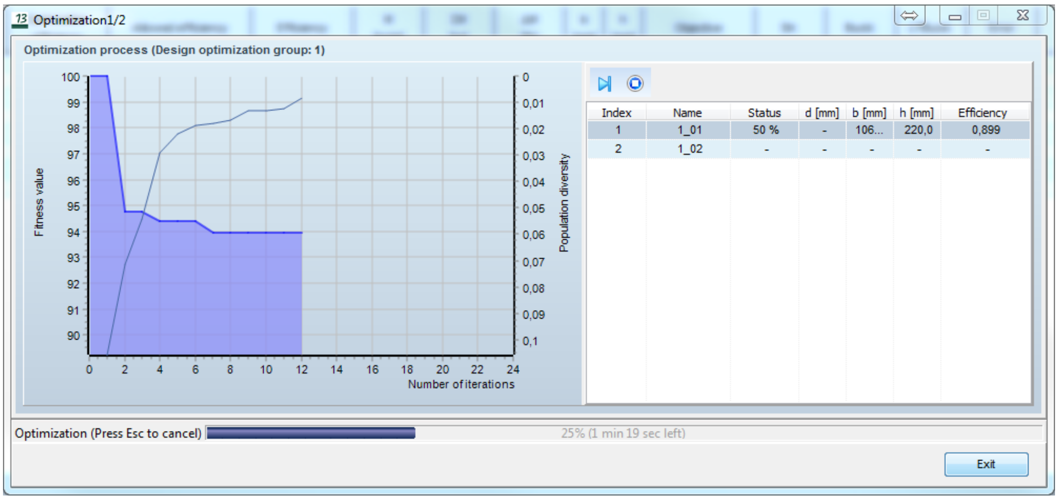
If a group contains more than one design member all members will be checked. Members are checked along their entire length. Not all candidates will be checked. The program analyses only those necessary for finding the global optimum.

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 efficiency and the efficiency 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.

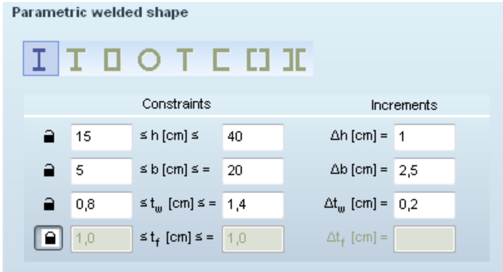
Stop the optimization. Stops the entire optimization. The current status of each group is considered to be the result. Same as pressing the **Esc** key.

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 efficiency 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 efficiency will be the highest efficiency 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 / optimized shape	Optimization efficiency	Efficiency	M [kg/m]	ΔM [%]	b [cm]	h [cm]	Target	Str.	Buckl.	LTBuckl.	Web Buckl.	Error	Method	Opt.	Replace
1	top chord	I 190	0,807	0,807	38,308		5,0	19,0	Weight	•	•			–		✓	✓
		I 150	**0,925**	**0,925**	**35,796**	**-7**	**5,0**	**15,0**						–	Parametric		
2	bottom chord	HFW500*200*4.5*8	0,830	0,830	42,387		20,0	50,0	Weight	•	•			–		✓	✓
		HFW500*200*4.5*8	**0,830**	**0,830**	**42,387**	**0**	**20,0**	**50,0**						–	Library		
3	diagonalsA	180X180X12,5	1,019	1,019	62,570		18,0	18,0	Weight	•	•	•		–		✓	✓
		200X200X12,5	**0,948**	**0,948**	**68,305**	**9**	**20,0**	**20,0**						–	Library		
4	diagonalsB	100X 60X 5,0	0,747	0,747	11,606		6,0	10,0	Weight	•	•	•		–		✓	✓
		100X60X4	**0,970**	**0,970**	**9,221**	**-21**	**6,0**	**10,0**						–	Library		

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 efficiency 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.

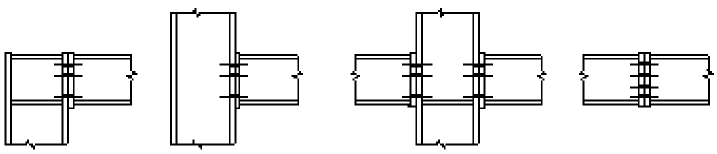
☞ ***Cross-section optimization may be a time-consuming task depending on the range of candidates and the size of groups so in case of a large search space (e.g. parametric optimization) it is recommended not to choose the slower AutoMcr method for lateral buckling calculations.***

6.6.2. Bolted joint design of steel beams – SD2 module

AxisVM calculates the moment-curvature diagram, the resistance moment and initial strength of steel column-beam bolted joints based on Eurocode3 (Part 1.8 Design of Joints).

Type of joints

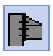
Beam to column or beam to beam joints.



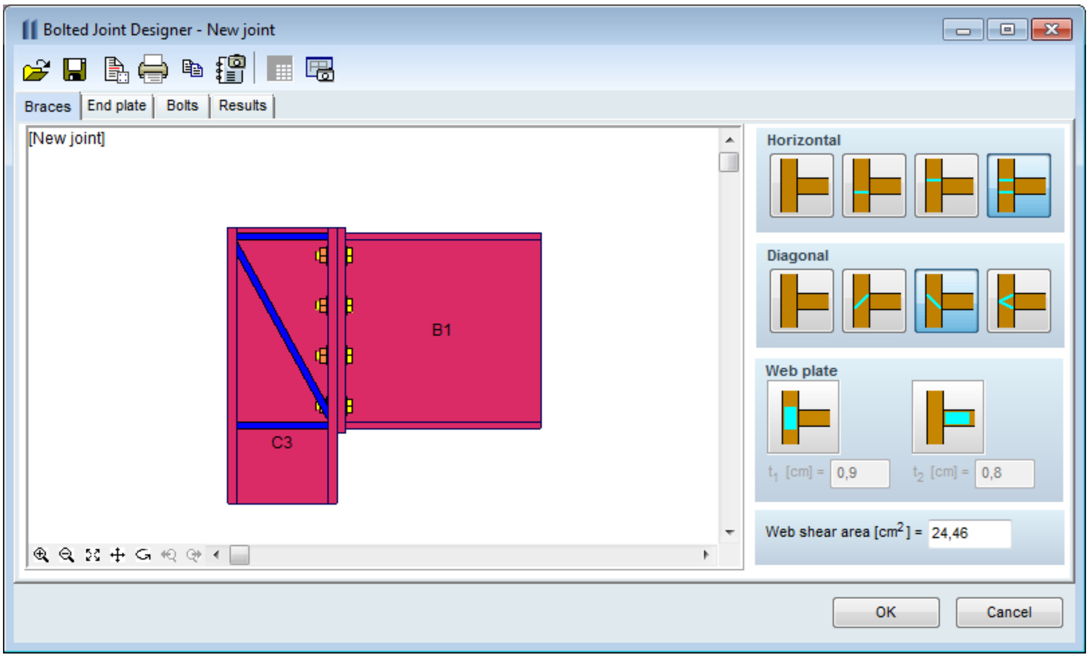
☞ ***Assumptions:***
- The beam and column cross-sections are rolled or welded I shapes.
- The beam end plate connect to the flange of the column.
- The pitch range of the beam is beetwen ± 30°.
- The cross-section class should be 1, 2 or 3.
- The normal force in the beam should be less than 0.05* $N_{pl,Rd}$
The program checks if these requirements are met.

The steps of the design

Select the beam and one of its end nodes.
(We can select several beams in one process if the selected beams have the same material and cross-section properties and connected columns also have the same material and cross-section properties.)



Click on the *Joint design* icon.
The Bolted Joint Designer will appear:

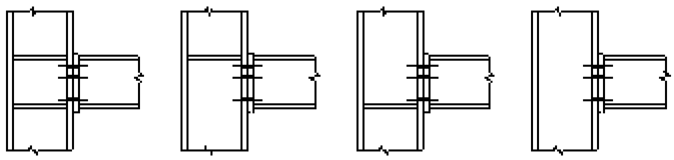


This dialog lets you assign the parameters of the joint in three steps.

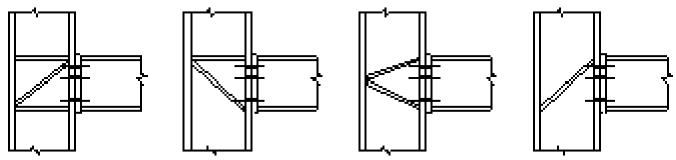
Bracings

We can assign horizontal, diagonal bracing plates and web thickening plates to increase the strength of the connection.

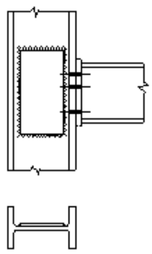
Horizontal bracings



Diagonal bracing



Web thickening plate

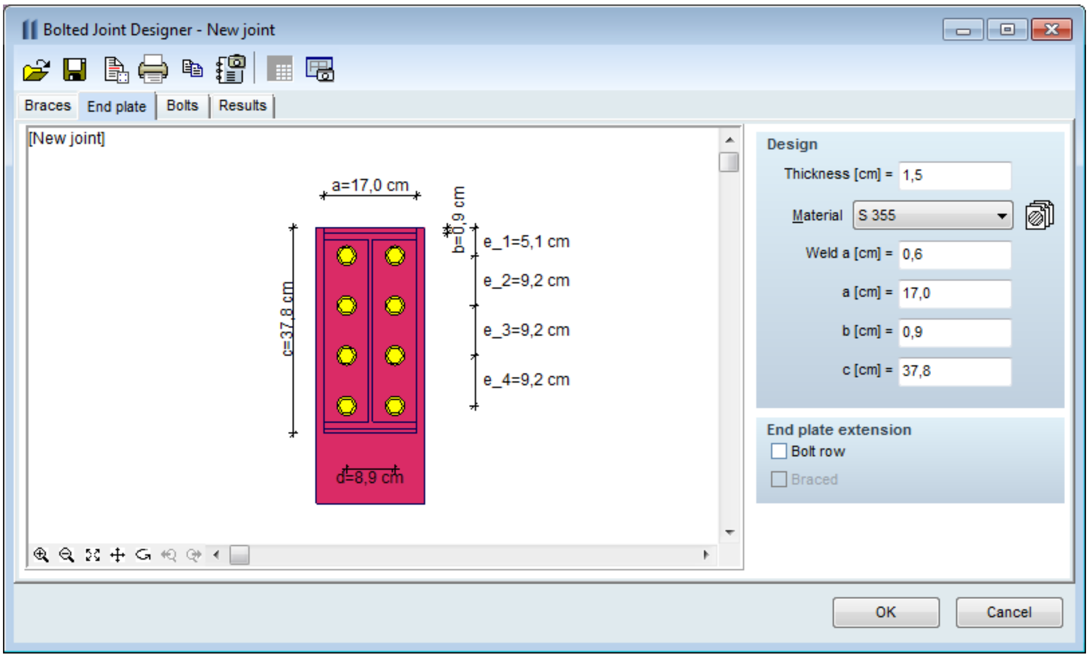


t1: thickening plate thickness on the column web
t2: thickening plate thickness on the beam web

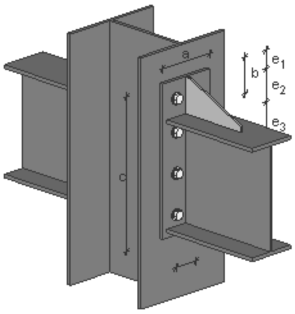
Web shear area

The program calculate the web shear area including the thickening plate area. If there is a hole in the web near to the connection you can decrease this value in the data field depending on the hole size.

End plate



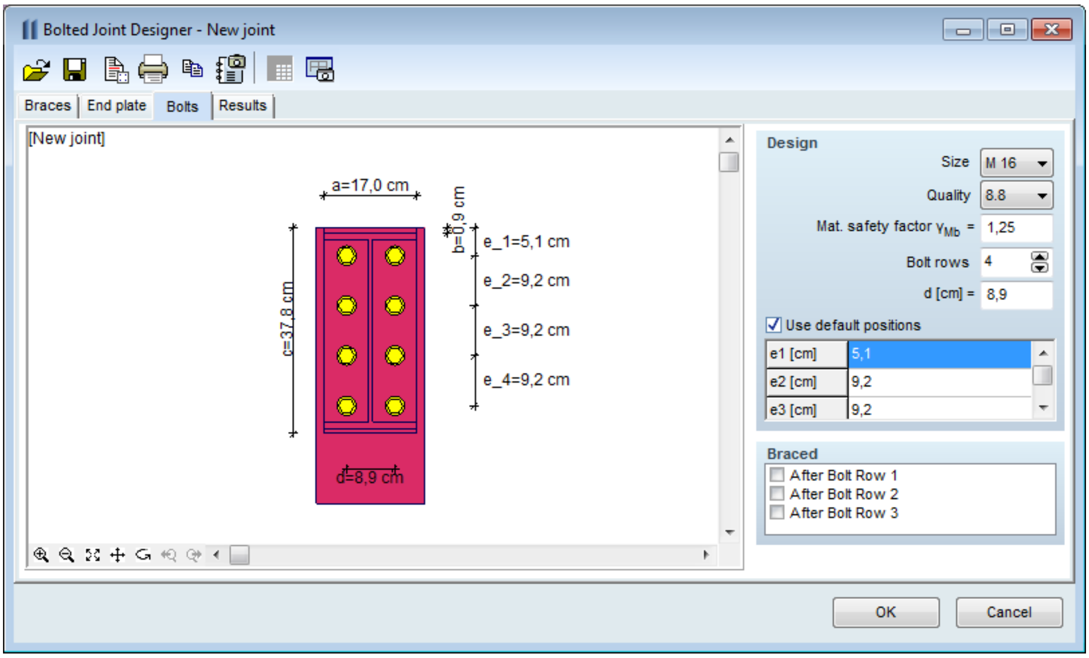
Parameters of the end plate:



Thickness
Material
Welding thickness
Width of the end plate (a)
Height of the end plate (c)
Distance between top flange of the beam and top of the end plate (b)
Bolts in the extension of the end plate

Bolt rows can be assigned to the tensile part of the end plate.

Bolts



The program places bolts in two columns symmetrical to the beam web. The same type of bolts is used in the connection.

Bolt parameters: *Size, Quality, Material safety factor, Number of bolt rows, distance of bolt columns (d)*

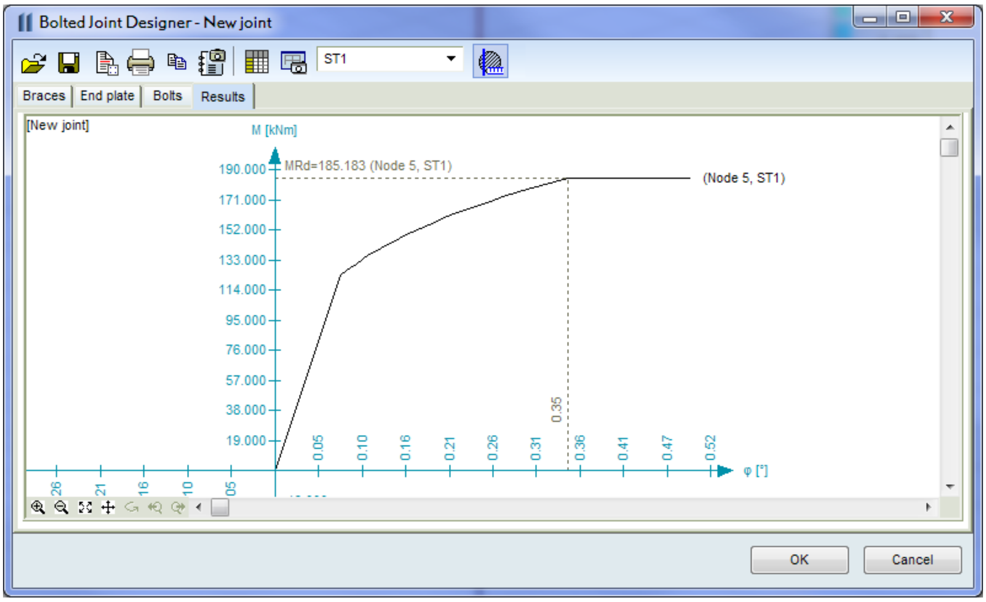
In case of automatic positioning of bolts the program places bolt rows in equal distances. The program checks the required minimal distances between bolts and from the edge of plates.
Turn off the option *Use default positions* to place the bolt rows individually.

☞ ***An error message will appear if the distances does not meet the requirements.***

Minimal bolt distances are checked based on EC2: *2.2 d* between bolts, *1.2 d* from edge of plate and in the direction perpendicular to the force.

Results

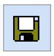
When we click on the *Results* tab AxisVM calculates the Moment-curvature diagram, the design resistant moment (M_{rD}) and the initial strength of the connection ($S_{j,init}$).



☞ ***A warning message will appear if the resistant moment is less than the design moment. The calculation method considers shear forces and normal forces together with the moments. As a consequence we can get different resistant moments (M_{rD}) for the same connection depending on the load cases (or combinations). Therefore AxisVM checks the M_{rD}-M_{sD} condition in all load cases.***

IconBar

Load the connection parameters.



Save the connection parameters. Saved parameters can be loaded and assigned to other beam-end joints later.

List of existing joints

Prints the displayed diagram. **See...** *3.1.10 Print*

Copies the diagram to the Clipboard

Saves the diagram to the Gallery

The result table contains the followings:
- node number
- beam number
- name of the load case or combination
- design moment (M_{sD})
- design resistant moment (M_{rD})
- a summary of the calculation results and intermediate results

Saving to Drawing Library

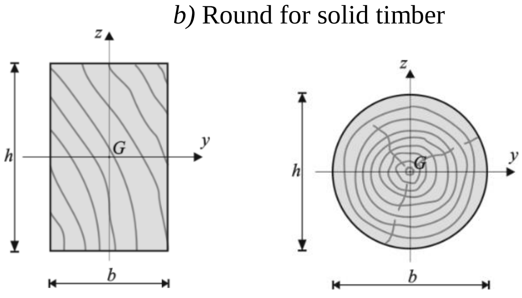
Additional parameters (f_{se} Coefficient for seismic forces, see *4.10.23 Seismic loads – SE1 module*)

6.7. Timber beam design – TD1 module

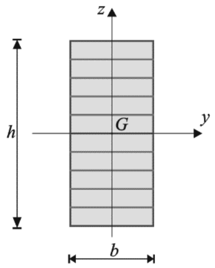
EUROCODE 5
(EN 1995-1-1:2004)

The timber beam design module can be applied to the following cross-sections and timber materials:

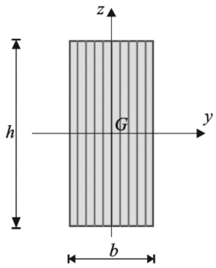
a) Rectangle for solid timber, Glued laminated timber (Glulam) and for Laminated veneer lumber (LVL)
b) Round for solid timber



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 perependicular to grain in *y* direction*	$f_{c,90,k,y}$
Compression strength perependicular 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 fv,k,z = fv,k,y = fv,k and fc90,k,z = fc90,k,y = fc90,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...** *4.9.8 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 is 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)

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 refe-rence 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

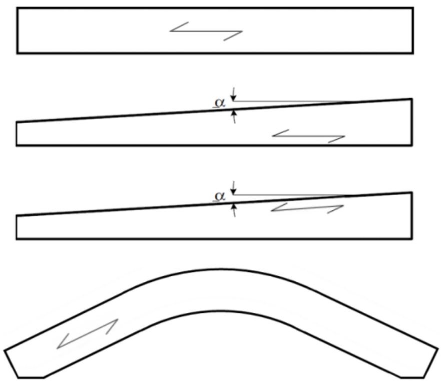
Analysis type	*Modulus (SLS)*	*Modulus (ULS)*
First order, linear elastic	$E_{mean,fin} = \frac{E_{mean}}{1 + k_{def}}$	$E_{mean,fin} = \frac{E_{mean}}{1 + \psi_2 k_{def}}$
	$G_{mean,fin} = \frac{G_{mean}}{1 + k_{def}}$	$G_{mean,fin} = \frac{G_{mean}}{1 + \psi_2 k_{def}}$
Second order, linear elastic	$E_d = E_{mean}/\gamma_M$	$E_d = E_{mean}/\gamma_M$
	$G_d = G_{mean}/\gamma_M$	$G_d = G_{mean}/\gamma_M$
Frequency	E_{mean}, G_{mean}	E_{mean}, G_{mean}

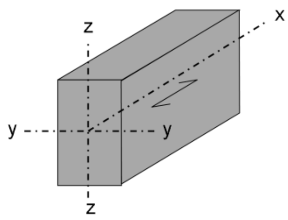
Conservative way $\psi_2 = 1$ is used.

Design assumptions

- There is no hole or other weaking in the beams.
- The cross-section constant (rectangle, round) or linear changing depth along the beam (tapered beam).
- The grain parallel to the beam x axis.
- In case of tapered beam the grain parallel one of the longitudinal edge.
- 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.





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 assumtions 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)

$$\frac{\sigma_{m,d}}{k_{crit} f_{m,d}} \leq 1$$

Compression and moment (EN 1995-1-1, 6.3.3)

$$\left(\frac{\sigma_{m,d}}{k_{crit} f_{m,d}}\right)^2 + \frac{\sigma_{cd}}{k_{c,z} f_{c,0,d}} \leq 1$$

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.

$$\frac{|\sigma_{mt,d}|}{k_{crit} f_{m,d}} \leq 1;$$

$$\sigma_{mt,d} = \frac{M_d}{W_y} + \frac{N_d}{A} < 0$$

where k_{crit} is the lateral buckling factor according to the following

$\lambda_{rel,m} \leq 0.75$	$k_{crit} = 1$
$0.75 < \lambda_{rel,m} \leq 1.4$	$k_{crit} = 1.56 - 0.75 \cdot \lambda_{rel,m}$
$1.4 < \lambda_{rel,m}$	$k_{crit} = 1/\lambda_{rel,m}^2$

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/NA:2010.

Shear(y) , Shear(z) and torsion

$$\max\left[\frac{\tau_{v,y,d}}{f_{v,d}}; \frac{\tau_{v,z,d}}{f_{v,d}}; \frac{\tau_{tor,d}}{k_{shape} f_{v,d}} + \left(\frac{\tau_{v,y,d}}{f_{v,d}}\right)^2 + \left(\frac{\tau_{v,z,d}}{f_{v,d}}\right)^2\right] \leq 1$$

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*)

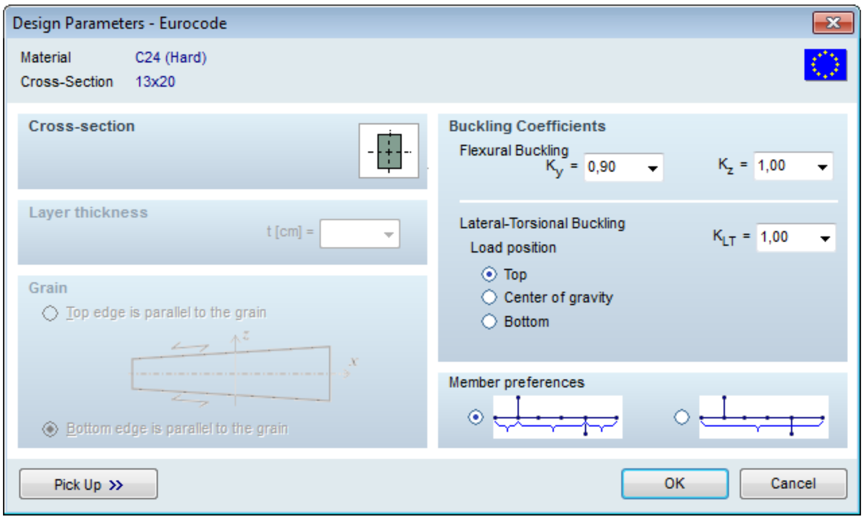
$$\frac{\tau_d}{f_{v,d}} + \frac{\sigma_{t,90,d}}{k_{dis} k_{vol} f_{t,90,d}} \leq 1$$

k_{dis} is a factor which takes into account the effect of the stress distribution in the apex zone ($k_{dis} =$ 1.4 for curved beams)

k_{vol} is a volume factor $k_{vol} = [V_0/V]^{0.2}$

Design Parameters

For the design based on Eurocode 5, the following design parameters should be defined and assigned to the design members:



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 paralel with the top edge or with the bottom edge. The top edge lays in the +z direction of the cross-section.

Stability parameters

Buckling 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

Lateral torsional buckling K_{LT}: lateral buckling length factors corresponding to the z axis.

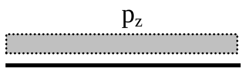
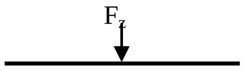
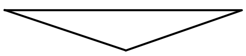
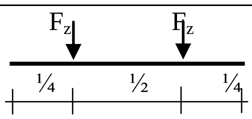
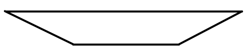
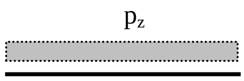
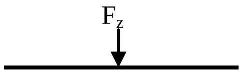
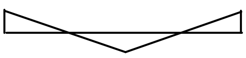
$$K_{LT} = l_{ef}/l$$

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 center of gravity, the program modify 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 $0.5h$

Common values of K_{LT} factor.
(Some of these values can be found in EN 1995-1-1, Table 6.1)

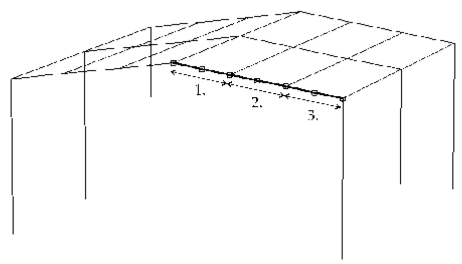
Loading type (direct load)	*My moment distribution between the lateral supports*	*Lateral support condition (in x-y plane)*
 p_z		0.9
 F_z		0.8
 F_z F_z ¼ ½ ¼		0.96
 p_z		0.42
 F_z		0.64

Loading type (no direct load)	*My moment distribution between the lateral supports*	*Lateral support condition (in x-y plane)*
M M		1.0
M M ½		0.76
M M=0		0.53
M M ½		0.37
M M		0.36

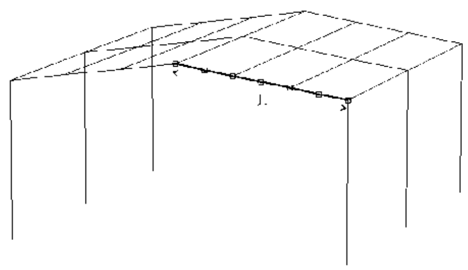
Loading type (cantilever)	*My moment distribution*	*Lateral support condition (in x-y plane)*
p_z		0.5
F_z		0.8

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.

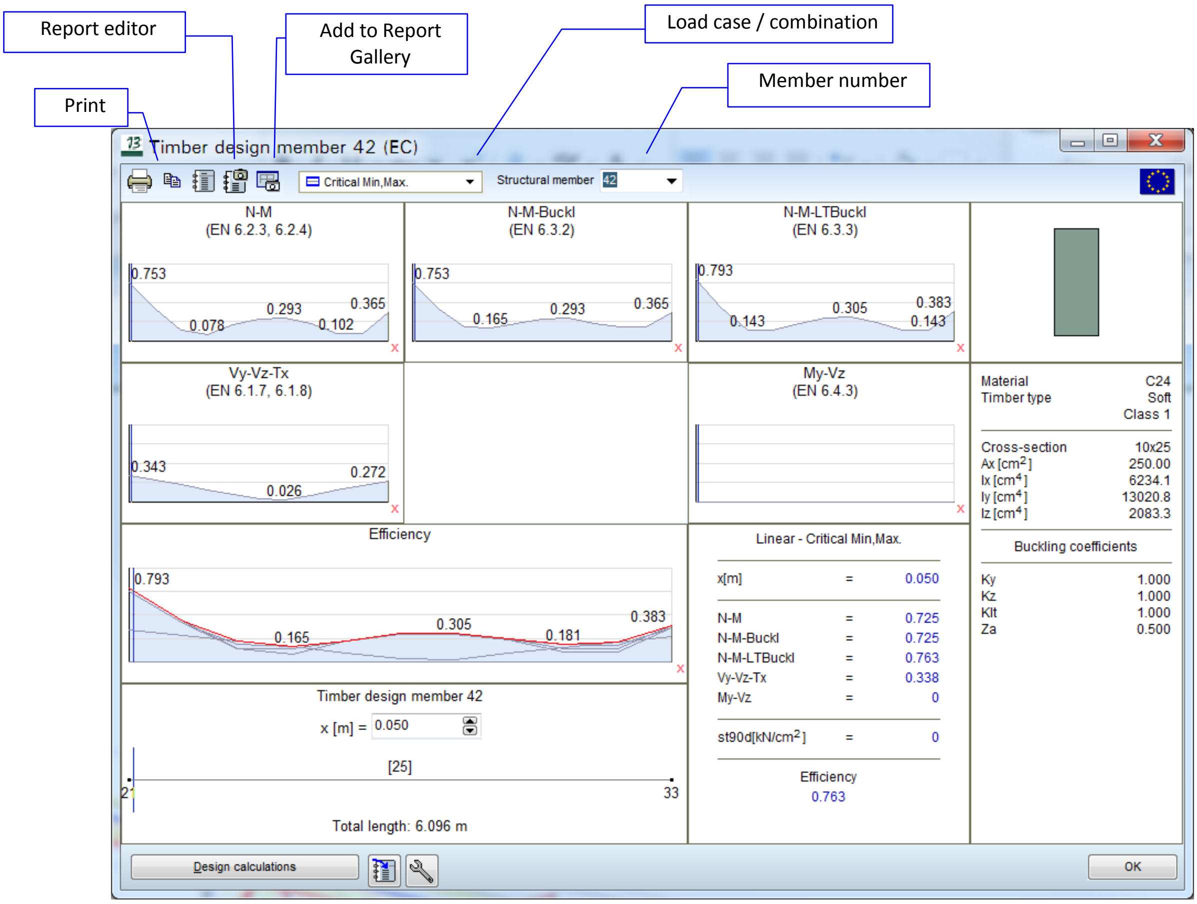


The finite elements in the selection set become only one design member irrespective of other finite elements connecting to its nodes.



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*). For further details see 6.6.1 Steel beam design according to Eurocode 3 – SD1 module

6.7.1. 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…** *6.6.1.1 Steel cross-section optimization*

6.8. Design of XLAM domains – XLM modul

Norms

There is no currently valid, overall design regulation for XLAM (CLT) panels, thus the following standards, pre-standards and normatives were taken as a basis of the design procedure: EN 1995-1-1, prEN 1995-1, DIN 1052:2004, DIN 1052:2008, GU 14.1.2008.

Material properties

The associated material properties can be selected from the different Glulam types of the material library (eg. GL24h), accordinng to the EN 1194.

If the user wishes, the material properties can be arbitrarily modified in the materal 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)	$f_{c,90,k}$
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

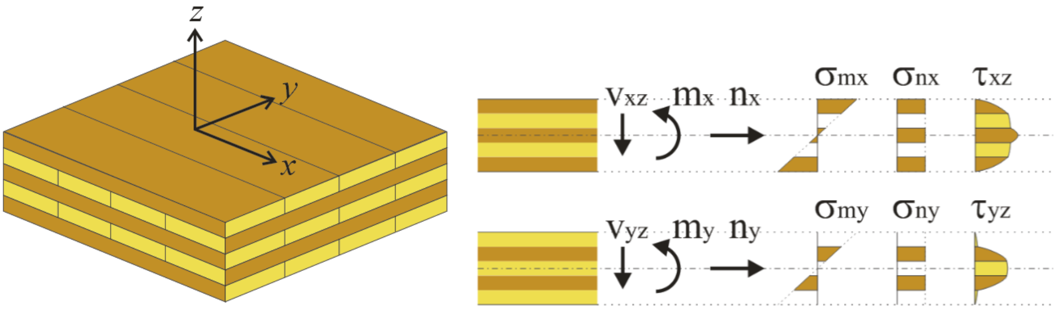
See in chapter *6.7 Timber beam design – TD1 module*

Load duration classes

See in chapter *6.7 Timber beam design – TD1 module*

Charasteristic 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 te 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 te 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, (occuring in the even or odd layers, depending on the user-defined orientation of te top layer).

σ_{ny} – relevant value of the normal stress in y direction due to normal forces, (occuring in the even or odd layers, depending on the user-defined orientation of te top layer).

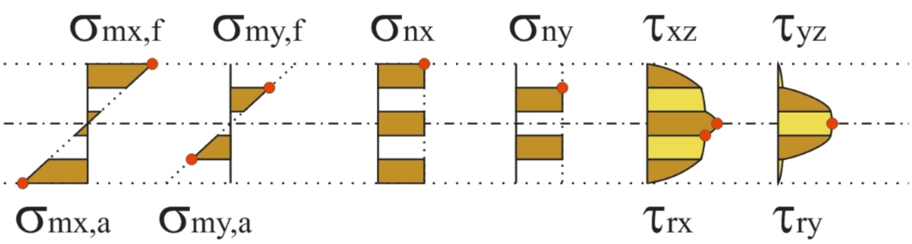
$\tau_{xz,max}$ – relevant shear stress acting in direction z on the plane whose normal is in direction x.

$\tau_{yz,max}$ – 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.

- Stresses if the orientation of the grains in the top layer coincides with the local x axis



Design value of strength

The design values of strength can be calculated according to the formula:
For $f_{c0d}, f_{c90d}, f_{vd}, f_{rd}$: $f_d = k_{mod} f_k / \gamma_M$
For $f_{md}, f_{t0d}, f_{t90d}$: $f_d = k_{sys} k_{mod} f_k / \gamma_M$
where,

k_{mod} modification factor for duration of load and moisture content (EN 1995-1-1, 3.1.3)
k_{sys} system strength factor
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 [N-M]
Shear /x [Vxz], Shear /y [Vyz]
Rolling shear /x [Vxz], Rolling shear /y [Vyz]

Normal force – Bending moment
The design value of the normal force may be tension or compression eather.
Tension and bending.

$$\frac{\sigma_{tx,max,d}}{f_{t,d}} + \frac{\sigma_{mx,max,d}}{f_{m,d}} \leq 1; \quad \frac{\sigma_{ty,max,d}}{f_{t,d}} + \frac{\sigma_{my,max,d}}{f_{m,d}} \leq 1$$

Compression and bending

$$\frac{\sigma_{cx,max,d}}{f_{t,d}} + \frac{\sigma_{mx,max,d}}{f_{m,d}} \leq 1; \quad \frac{\sigma_{cy,max,d}}{f_{t,d}} + \frac{\sigma_{my,max,d}}{f_{m,d}} \leq 1$$

Shear Shear in x and y direction.

$$\frac{\tau_{xz,max,d}}{f_{v,d}} \leq 1; \quad \frac{\tau_{yz,max,d}}{f_{v,d}} \leq 1$$

Rolling shear Rolling shear in x and y direction.

$$\frac{\tau_{rx,max,d}}{f_{r,d}} \leq 1; \quad \frac{\tau_{ry,max,d}}{f_{r,d}} \leq 1$$

7. AxisVM Viewer and Viewer Expert

AxisVM Viewer

AxisVM Viewer is a freely downloadable version of the program for viewing models without the possibility of making changes. Printing of drawings, tables or reports is not available.

This programs 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 it save the model 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.

8. 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.com.

9. Step by step input schemes

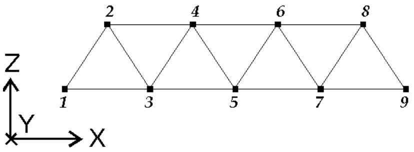
9.1. Plane truss model

Geometry

1.) Create the geometry (for example: in X-Z plane).
Set the X-Z view.

Draw the geometry.

⇨ **Polyline**



Elements

1.) Define truss elements.

⇨ **Truss**

Select the lines, which have the same cross-section and material, to define truss elements

2.) Loading material features from the material library

⇨ **Database** (Steel FE 430)

3.) Selecting cross-section from the database

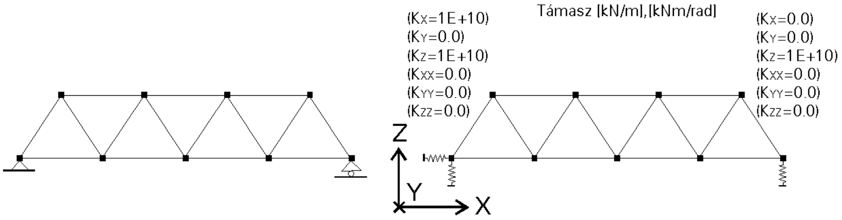
⇨ **Database** (∅76x7.0)

4.) Define support elements.

⇨ **Nodal support**

⇨ **Global**

⇨ Reference



Select the nodes, which have the same properties, to define support elements.

5.) Define the nodal degrees of freedom.

⇨ **Nodal DOF**

Select all nodes to define nodal degrees of freedom. Choose the *Truss girder in X-Z plane* from the list.

Loads

1.) Define load cases and combinations.

⇨ **Load case and load group**

⇨ Combination

2.) Apply loads (nodal, thermal, fault in length, dead load).

⇨ **Nodal**

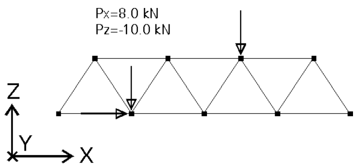
⇨ Truss

⇨ Truss

⇨ Truss

⇨ Truss

3.) Select the truss elements, which have the same load.



Static

Start a linear static analysis.

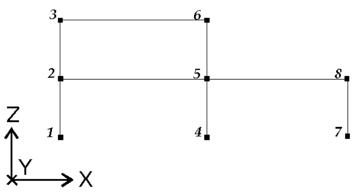
9.2. Plane frame model

Geometry

1.) Create the geometry (for example: in X-Z).
Set the X-Z view.

Draw the geometry.

⇨ **Polyline**



Elements

1.) Define beam elements.

⇨ **Beam**

Select the lines, which have the same cross-section and material, to define beam elements.

2.) Loading material features from the material library

⇨ **Database** (Steel FE 430)

3.) Selecting cross-section from the database

⇨ **Database** (∅76x7.0)

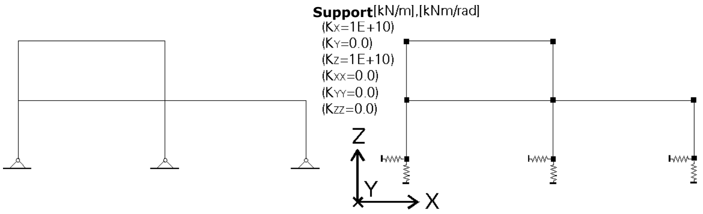
4.) Define support elements.

⇨ **Nodal support**

⇨ **Global**

⇨ Beam relative

⇨ Local



Select the nodes, which have the same properties, to define nodal support elements.

5. Define the nodal degrees of freedom.

⇨ **Nodal DOF**

Select all nodes to define degrees of freedom. Choose the *Frame in X-Z plane* from the list.

Loads

1) Define load cases and combinations.

⇨ **Load case and load group**

⇨ Combination

2.) Apply loads (nodal, distributed, temperature, fault in length, dead load).

⇨ Nodal

⇨ Beam

⇨ **Beam**

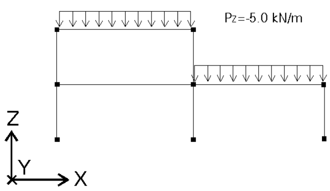
⇨ Beam

⇨ Beam

⇨ Beam

⇨ Beam

3.) Select the beam elements, which have the same load.



Static

Start a linear static analysis.

9.3. Plate model

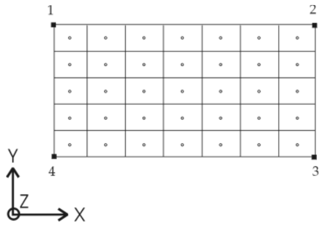
Geometry

1.) Create the geometry (for example: in X-Y plane).
Set the X-Y view.

Draw the element mesh.

⇨ **Quadrilateral**

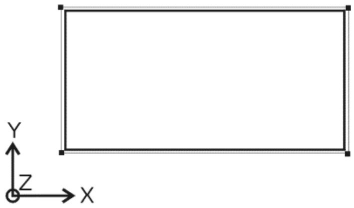
$n_{1-2}=7$
$n_{2-3}=5$



Elements

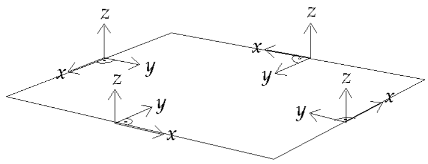
1.) Define domain.

⇨ **Plate** ⇨ **Material** ⇨ **Thickness**



2.) Define support elements.

⇨ Nodal support



⇨ **Line support**

⇨ **Edge relative**

⇨ Global

First, select the surface elements, and then select the supported edges, to define line support elements.
If you choose relative to edge support conditions, then the edge will represent the ***x*** direction, and the ***y*** direction will be perpendicular to the edge in the surface plane (according to the right-hand rule), and the ***z*** direction will be perpendicular to the surface plane.

3.) Define the nodal degrees of freedom.

⇨ **Nodal DOF**

Select all nodes to define degrees of freedom. Choose the *Plate in X-Y plane* from the list.

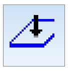
Loads

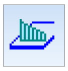
1.) Define load cases and combinations.

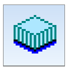
⇨ **Load case and load group**

⇨ Combination

2.) Apply loads (nodal, line, surface, dead load).

 ⇨ Nodal

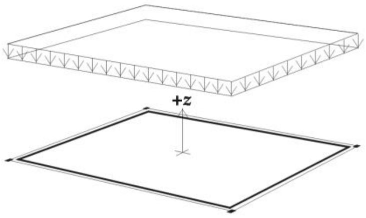
 ⇨ Plate

 ⇨ **Plate**

 ⇨ Plate

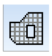
 ⇨ Plate

3.) Select domain, which have the same load.
The direction of distributed load is perpendicular to the plane of the surface, and the sign of the load is the same as of the local ***z*** axis of the plate (for example: p_z=-10.00 kN/m^2).

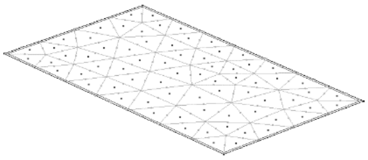


Elements

1.) Mesh generation



-select the domain
-set the avarage size of finite elements (for example.:0,5 m)



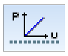
2.) Define the nodal degrees of freedom.

 ⇨ **Nodal DOF**

Select all nodes to define degrees of freedom. Choose the *Plate in X-Y plane* from the list.

Static

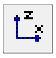
Start a linear static analysis.



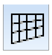
9.4. Membrane model

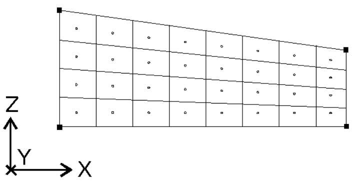
Geometry

1.) Create the geometry (for example: in X-Z plane).
Set the X-Z view.



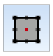
Draw the element mesh.

 ⇨ **Quads**



Elements

1.) Define membrane elements.

 ⇨ **Surface Elements**

⇨ **Membrane**

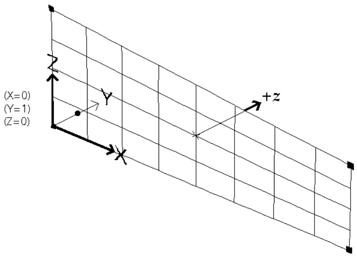
Select the quad/triangle surfaces, which have the same material, local directions and thickness, to define the membrane elements.

2.) Define material features (for example: selecting from the material library)

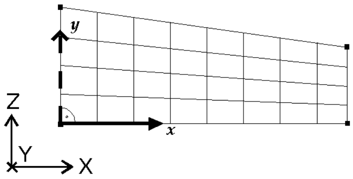
 ⇨ **Loading** (Concrete C20/25)

3.) Define the thickness (for example: 200 mm)

4.) The program automatically generates the local coordinate-system of finite elements

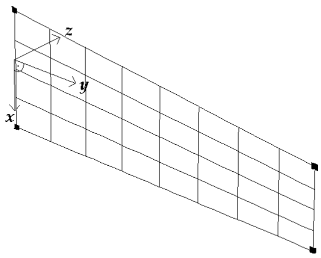


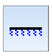
n_x, n_y, n_{xy} internal forces refer to the local x,y directions



5.) Define support elements.

 ⇨ Nodal support



 ⇨ **Line support**

⇨ **Edge relative**

⇨ Global

☞ ***You can also define surface supports (Winkler type elastic foundation).***

First, select the surface elements, and then select the supported edges, to define line support elements.
If you choose relative to edge support conditions, then the edge will represent the ***x*** direction, and the ***y*** direction will be perpendicular to the edge in the surface plane (according to the right-hand rule), and the ***z*** direction will be perpendicular to the surface plane.

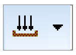
6.) Define the nodal degrees of freedom.

⇨ **Nodal DOF**

Select all nodes to define degrees of freedom. Choose the *Membrane in X-Z plane* from the list.

Loads

1.) Define load cases and combinations.

 ⇨ **Load case and load group**

 ⇨ Combination

2.) Apply loads (nodal, line, surface, dead load).

 ⇨ Nodal

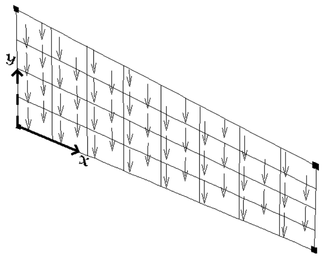
 ⇨ Membrane

 ⇨ **Membrane**

 ⇨ Membrane

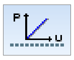
 ⇨ Membrane

Select the elements, which have the same load.
The direction of distributed load is determined in the local ***x-y*** direction of the membrane (for example: p_y = -10.00 kN/m^2).



Static

Start a linear static analysis.



9.5. Response spectrum analysis

Geometry

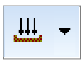
See... 9.1. - 9.4. Input Schemes.

Elements

See... 9.1. - 9.4. Input Schemes.

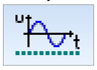
Loads/1

1.) Apply loads.

 ⇨ **Load**

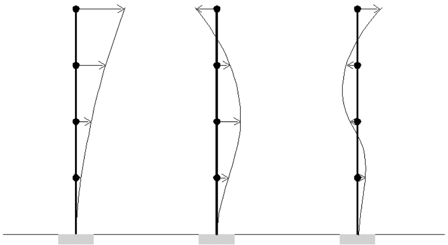
2.) Apply all the gravitational loads that you want to account as masses in the vibration analysis that precedes the static analysis.

Analysis/1



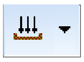
1.) Perform a vibration analysis. Vibration mode shapes for earthquake analysis are usually requested as 3 for in-plane structures and 9 for spatial structures are requested.

Include the gravitational load case described at Loads/1 point in the vibration analysis, and set the Convert loads to mass check-box enabled.

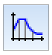


Loads/2

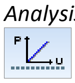
1.) Set a seismic load case.

 ⇨ **Load**

2.) Specify the parameters of the seismic loads.

 ⇨ **Seismic**

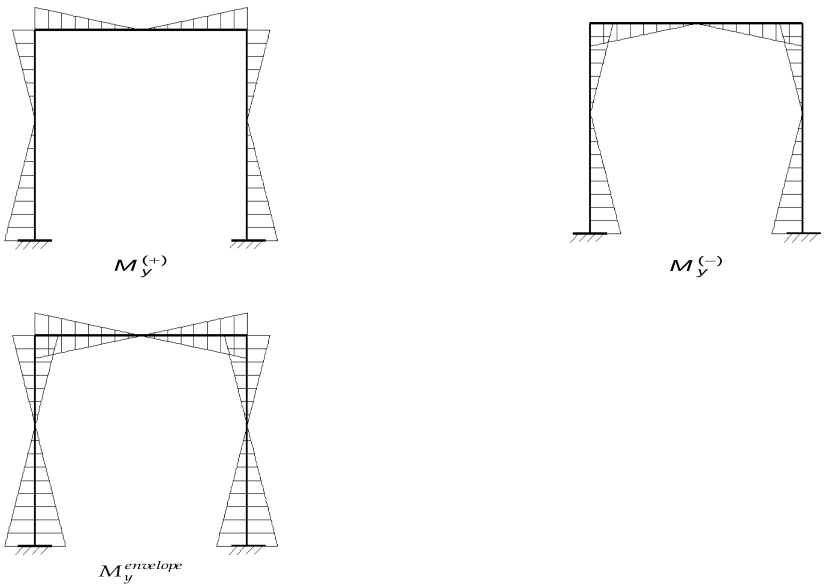
Analysis/2



1.) Start a linear static analysis.

2.) When generating the seismic type load cases, two are included. One **+** with values included as positives, and one **-** with values included as negatives. In addition the results corresponding to each vibration mode shape are provided (corresponding to load cases with 01, 02, ...n suffixes), that can be used in the generation of further combinations or of critical combinations.

See... *4.10.23 Seismic loads – SE1 module*

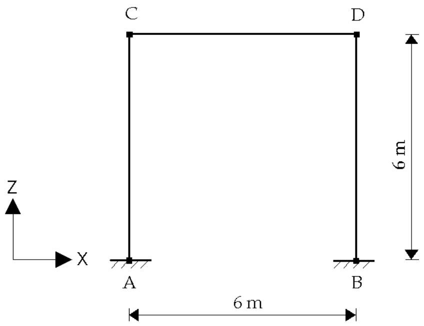


10. Examples

10.1. Linear static analysis of a steel plane frame

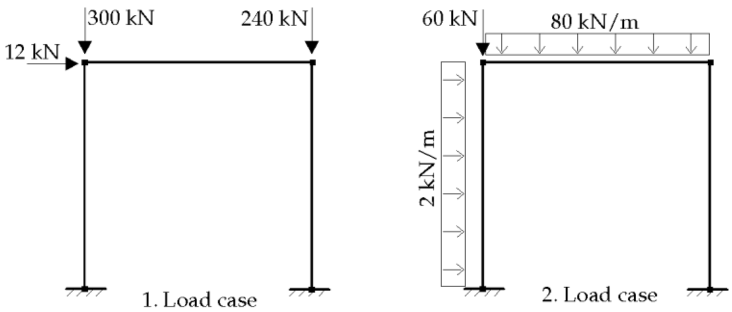
Input data **AK-ST-I.axs**

Geometry:



Material: Steel
Cross-section: I 240

Loads:



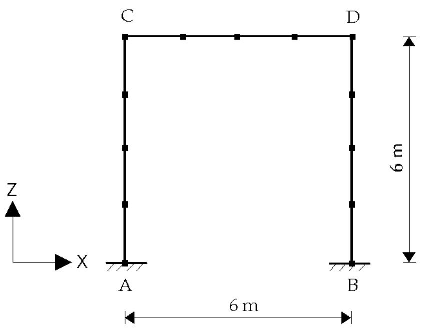
Results **AK-ST-I.axe**

Component		Analytic	AxisVM
1 Lc.	$e_x^{(C)}$ [mm]	17.51	17.51
	$M_y^{(A)}$ [kNm]	-20.52	-20.52
2 Lc.	$e_x^{(C)}$ [mm]	7.91	7.91
	$M_y^{(A)}$ [kNm]	63.09	63.09

10.2. Geometric nonlinear static analysis of a steel plane frame

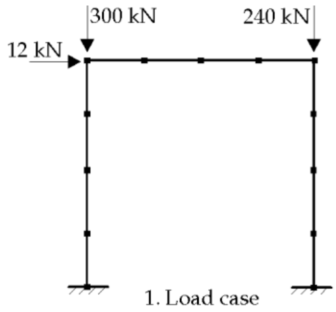
Input data **AK-ST-II.axs**

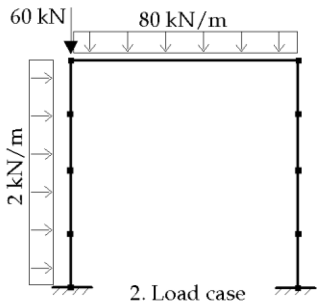
Geometry:



Material: Steel
Cross-section: I 240

Loads:





Results **AK-ST-II.axe**

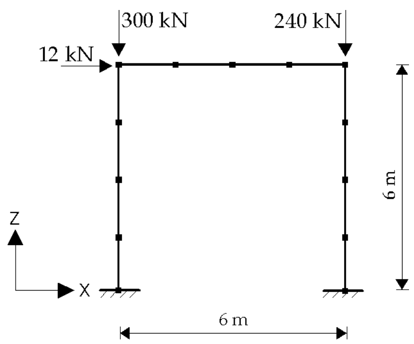
Component		With Stability Functions	AxisVM
1 Lc.	$e_x^{(C)}$ [mm]	20.72	20.58
	$M_y^{(A)}$ [kNm]	-23.47	-23.41
2 Lc.	$e_x^{(C)}$ [mm]	9.26	9.22
	$M_y^{(A)}$ [kNm]	66.13	66.25

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**

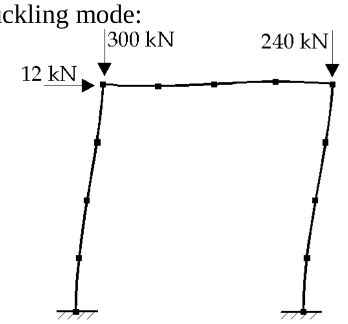
Geometry and loads:



Material: Steel
Cross-section: I 240

Results **AK-KI.axe**

Buckling mode:

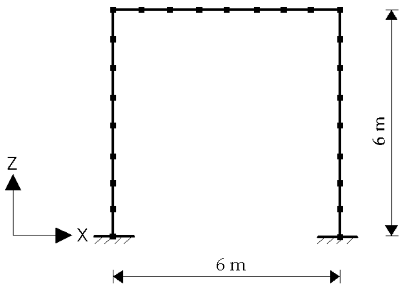


Critical load parameter	Cosmos/M™	AxisVM
n_{cr}	6.632	6.633

10.4. Vibration analysis (I-Order) of a steel plane frame

Input data **AK-RZ-I.axs**

Geometry:



Material: Steel
Cross-section: I 240

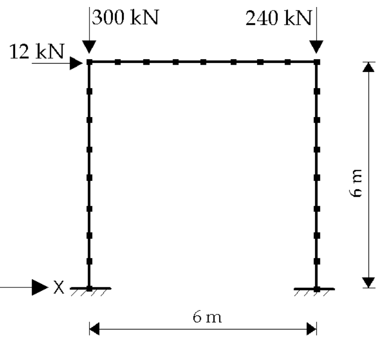
Results **AK-RZ-I.axe**

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:



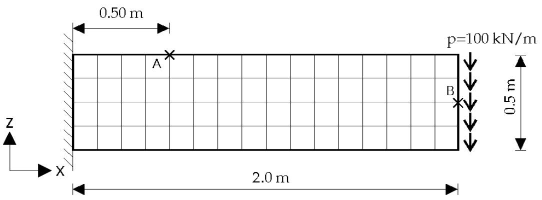
Material: Steel
Cross-section: I 240

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**



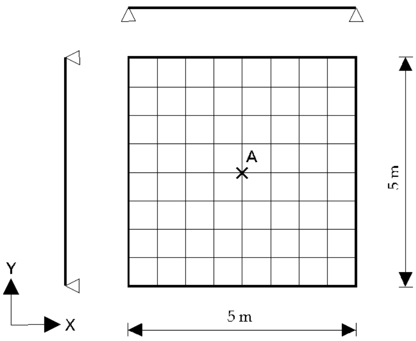
E=880 kN/cm^2
ν=0
t=0.10 m
p=100 kN/m
Mesh: 4x16

Results **VT1-ST-I.axe**

Component	Beam theory (shear deformations included)	AxisVM
$e_z^{(B)}$ [mm]	15.09	15.09
$n_x^{(A)}$ [kN/m]	1800.00	1799.86

10.7. Linear static analysis of a simply supported reinforced concrete plate

Input data **VL1-ST-I.axs**

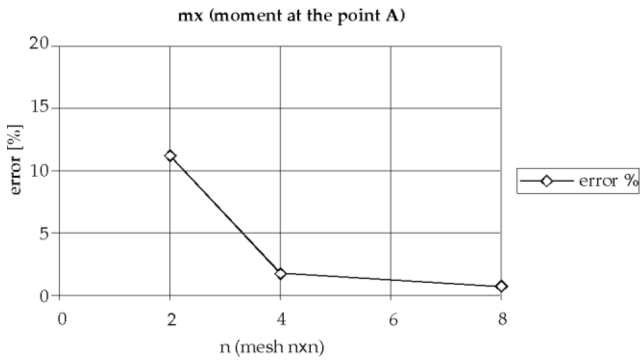


E=880 kN/cm^2
ν=0
t=0.15 m
p=50 kN/m^2
Mesh: 8x8

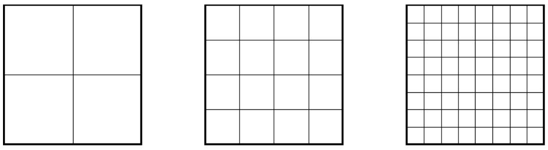
Results

Component	Analytic (shear deformations not included)	AxisVM (shear deformations included)
$e_z^{(A)}$ [mm]	51.46	51.46
$m_x^{(A)}$ [kNm/m]	46.11	46.31

Convergence analysis



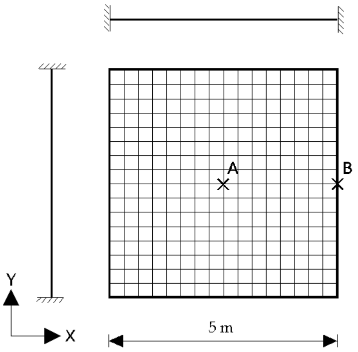
Meshes:



n=2 n=4 n=8

10.8. Linear static analysis of a clamped reinforced concrete plate

Input data **VL2-ST-I.axs**

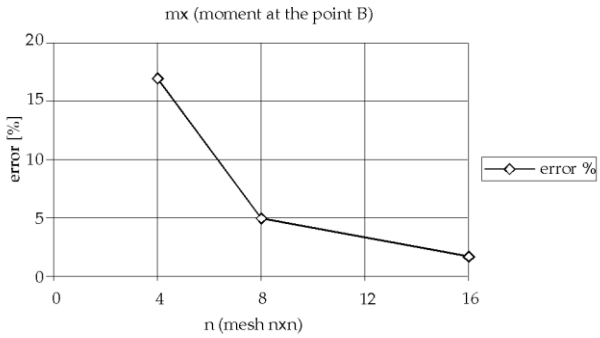


E=880 kN/cm2
ν=0
t=0.15 m
p=50 kN/m2
Mesh: 16x16

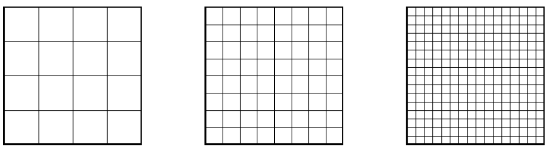
Results **VL2-ST-I.axe**

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

This page is intentionally left blank.

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. **Eurocode 2,** EN 1992-1-1:2004
22. **Eurocode 3,** EN 1993-1-1:2005
23. **Eurocode 3,** EN 1993-1-3:2006
24. **Eurocode 3,** EN 1993-1-5:2006
25. **Eurocode 5,** EN 1995-1-1:2004
26. **Eurocode 8,** EN 1998-1-1:2004
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

Notes

Notes