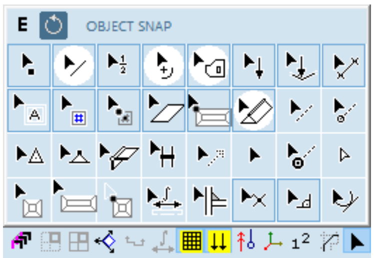
New features of AxisVM **X6 R2**

GENERAL FEATURES

- ### Control over which elements are detected by the cursor

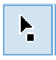
As a new element of the speed buttons, the *Object snap* function can be used to switch on and off certain objects to be detected or neglected by the cursor, irrespectively to the performed operation.



Each icon has three states:

 Deactivated snap function

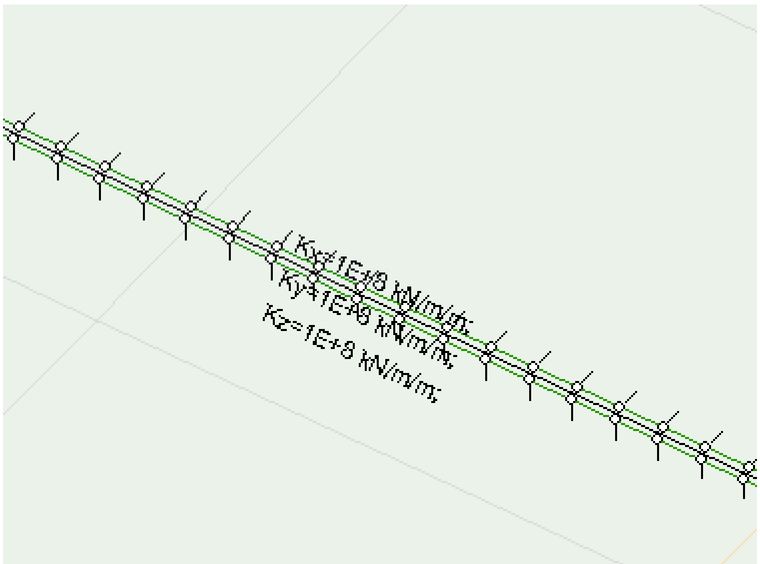
 Activated snap function

 Default snap depending on the current operation

Clicking on the *E* button turns on exclusive snap function, meaning only those elements will be detected, which are activated.

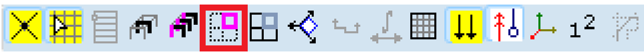
- ### Labels for edge hinge properties

As a new checkbox on the *Label* tab of *Display options* dialog, edge hinge stiffness properties along with the edge hinge symbols can be enabled and displayed on the model.



- ## Hiding selected elements

  As a new element on the speed button toolbar, and as a counterpart of the already existing *Show only selected elements*, *Hiding selected elements* will make selected elements invisible in the modeling area temporarily, until the function is turned off, giving you more control over model management and result evaluation.



- ## Python package for AxisVM

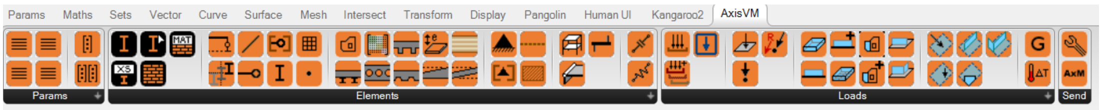
  Significantly expands the possibilities of the COM server by providing access to the python packages made by its huge 7 million user base. The library wraps the COM type library, and enhances it to have a more pythonic experience, while leaving the core behaviour on the table, so you don't have to worry about legacy code.

  COM server will serve as a bridge between Python Extension and AxisVM, by providing model and result information to Python side where it can be used and interpreted freely, or translates instruction sets from Python extension to interpreted by AxisVM.

LINKS WITH OTHER PROGRAMS

- ## New component package for AxisVM-Rhino/Grasshopper supporting all element and load types

  The Grasshopper-AxisVM connection provides the benefits of state of the art parametric design in AxisVM through the visual programming language and environment running in Rhinoceros 3D CAD application. Data transfer from Grasshopper to AxisVM is performed through the COM server, now covering the whole material and cross section library, all type of elements (node, line element, domains, releases, eccentricities, connection elements, supports etc.) and all type of load related objects (concentrated and line load, surface load, polyline load, load panel, thermal load, self weight, load cases, load groups and combinations).



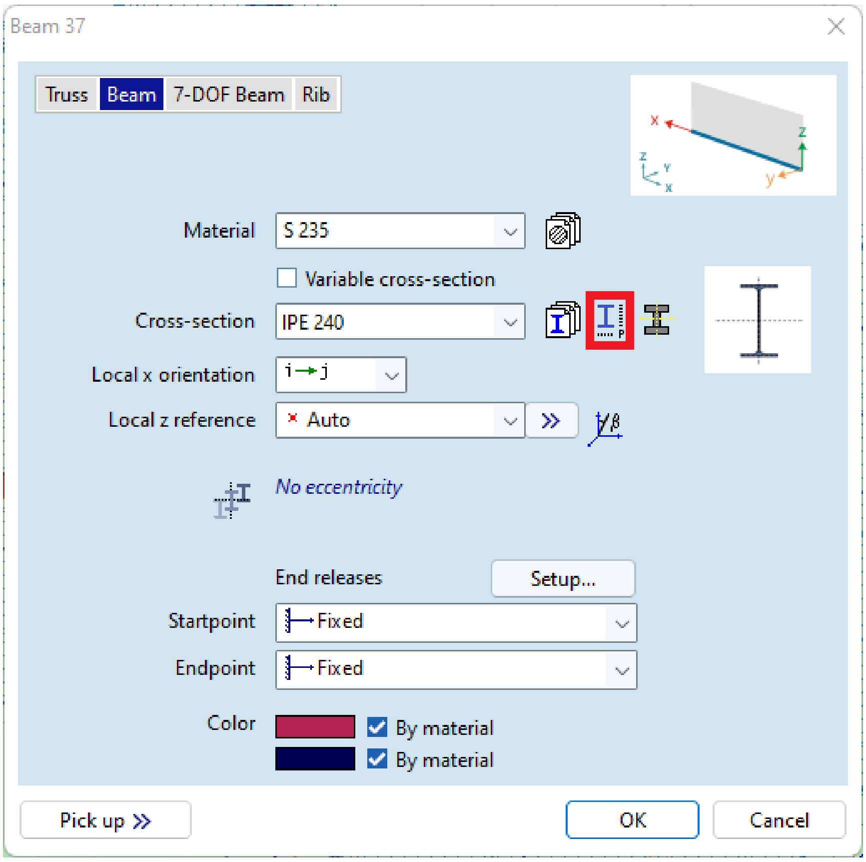
- ## Support of SAF 2.0 data transfer for import and export (**SAF** modul)

  Support of data transfer for SAF (Structural Analysis Format) file format based on its new 2.0 iteration, both on import and export side.

SAF

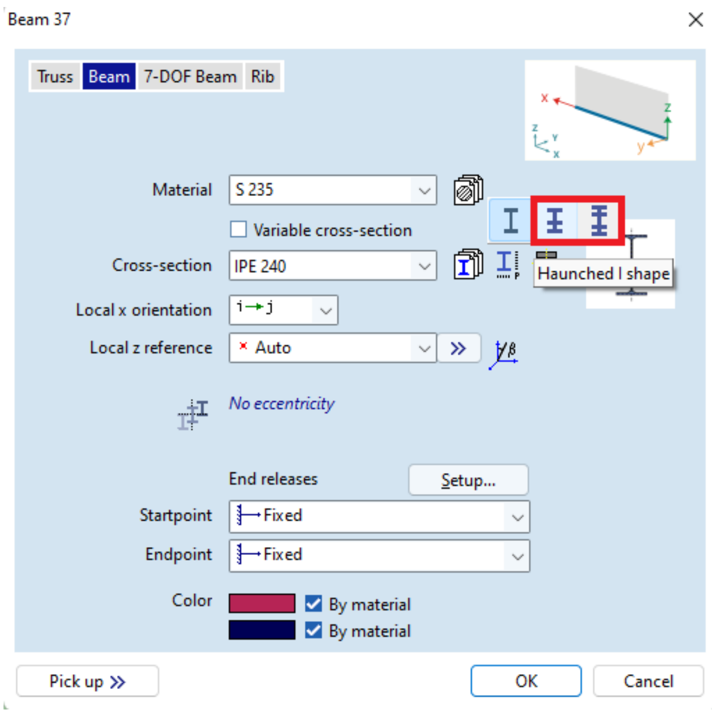
- Direct editing of cross-section parameters

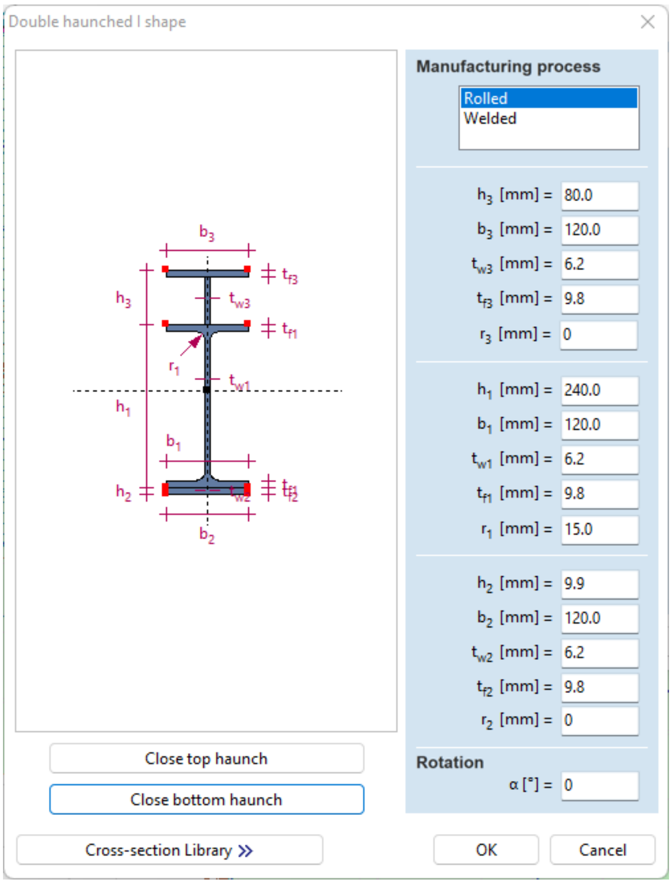
  Provides quick and easy access to the cross-section parameters dialog, without the need of leaving the line element dialog.

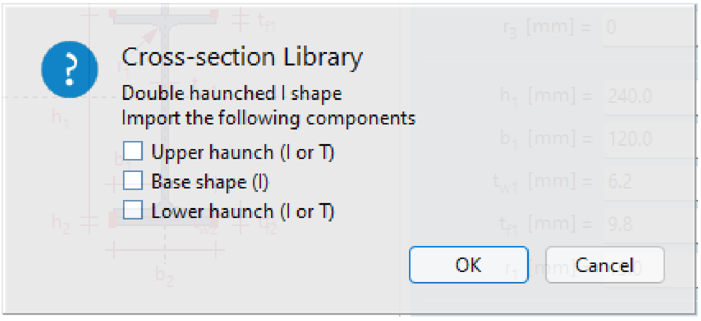


- Simple way to define haunched and double haunched I shapes

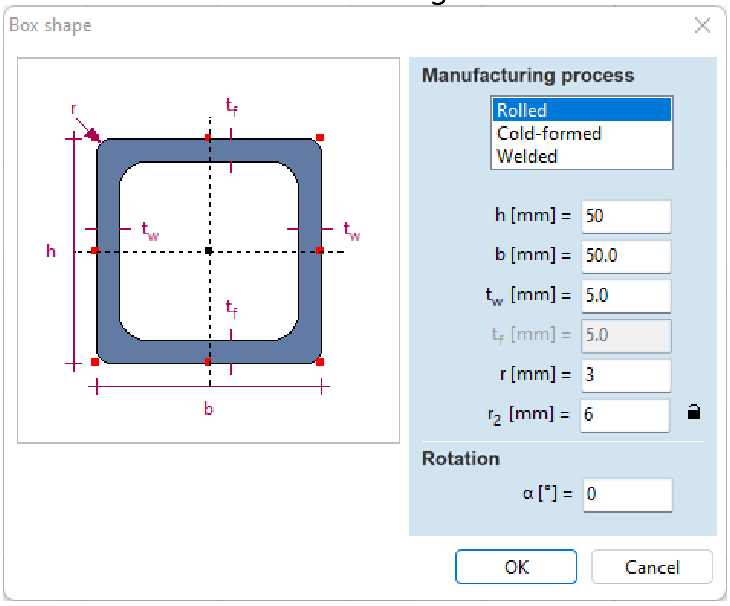
  In case of an I shaped cross section, the parametric editing of cross section icon offers the possibility to transform the shape into a haunched or double haunched type of section. The section editor for these cross sections itself has been added with the option to calculate the dimensions of the end section of a haunched shape, which is useful to create the end section of a haunched segment. This can be performed with the *Close haunch* button. Direct access for the *Cross section library* is also available from the dialogue, to add a library section to the base (I), or haunch (I or T) section.


Beam 37
Truss
Beam
7-DOF Beam
Rib
Material
S 235
Variable cross-section
Cross-section
IPE 240
Haunched I shape
Local x orientation
Local z reference
Auto
No eccentricity
End releases
Setup...
Startpoint
Fixed
Endpoint
Fixed
Color
By material
By material
Pick up
OK
Cancel


Double haunched I shape
Manufacturing process
Rolled
Welded
h3 [mm] = 80.0
b3 [mm] = 120.0
tw3 [mm] = 6.2
tf3 [mm] = 9.8
r3 [mm] = 0
h1 [mm] = 240.0
b1 [mm] = 120.0
tw1 [mm] = 6.2
tf1 [mm] = 9.8
r1 [mm] = 15.0
h2 [mm] = 9.9
b2 [mm] = 120.0
tw2 [mm] = 6.2
tf2 [mm] = 9.8
r2 [mm] = 0
Rotation
α [°] = 0
Close top haunch
Close bottom haunch
Cross-section Library
OK
Cancel



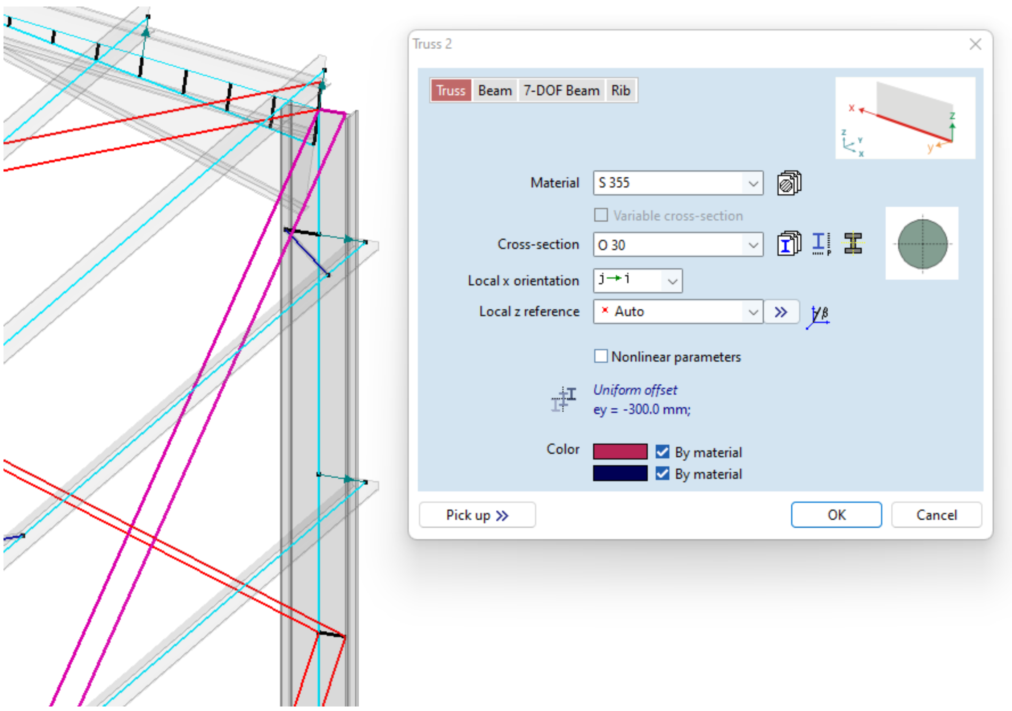
- Different values for inner and outer rolling radiuses for hollow sections



- ## Extended options for element eccentricities

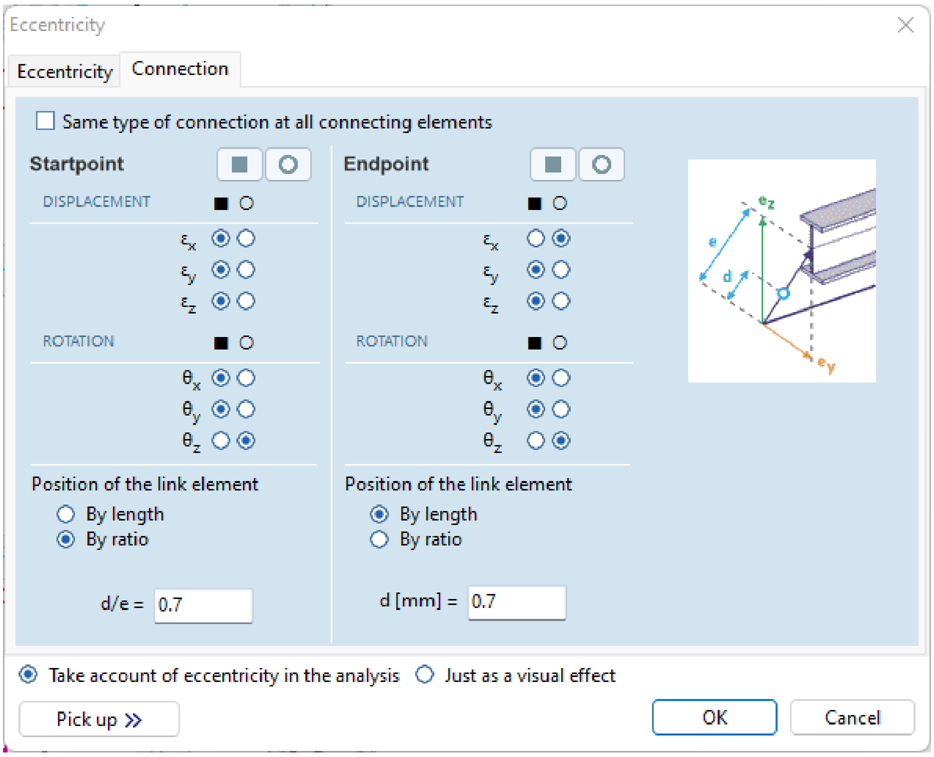
  Eccentricity settings are extended to include truss elements as well. Eccentricities in local y and z directions relative to the logical axis can be defined for every type of line elements now.

  X6R2 extends the eccentricity options with the decision if the effects of the eccentricities on analysis results should be taken into account, or if they should only act as a visual effect on the model, without affecting calculation results.



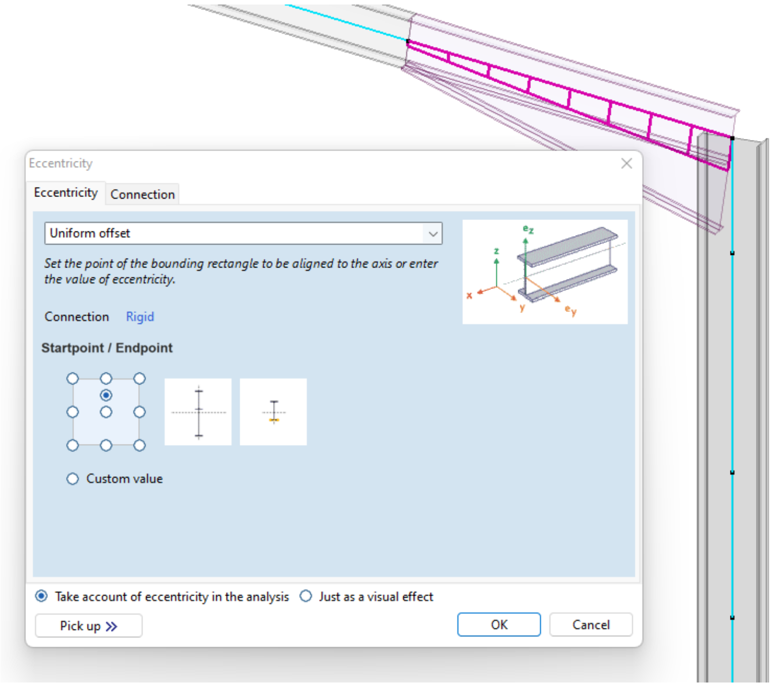
- ## Automatic creation of link elements between eccentrically connected elements

  Start, end and mid-point connections of eccentric members can be generated automatically, with control over release conditions on these points. These types of generated link elements will adapt to the changes in the connected members geometric parameters, such as position, and cross section changes.



- ## Eccentric fitting of haunched segments based on core section

  As a new option, in case of wedged and double-wedged I cross-sections, center of gravity of the core section can be used as a reference for the eccentricity settings. This can be helpful when defining a haunched segment, as usually the center of gravity of the core section has to be aligned.



- ## New ARBO/CRET element library for Aschwanden plugin

  Aschwanden is a Swiss company developing innovative, high-end solutions for reinforcement and for transmission of loads in reinforced concrete structures. AxisVM X6R2 will inlude two of their product types:

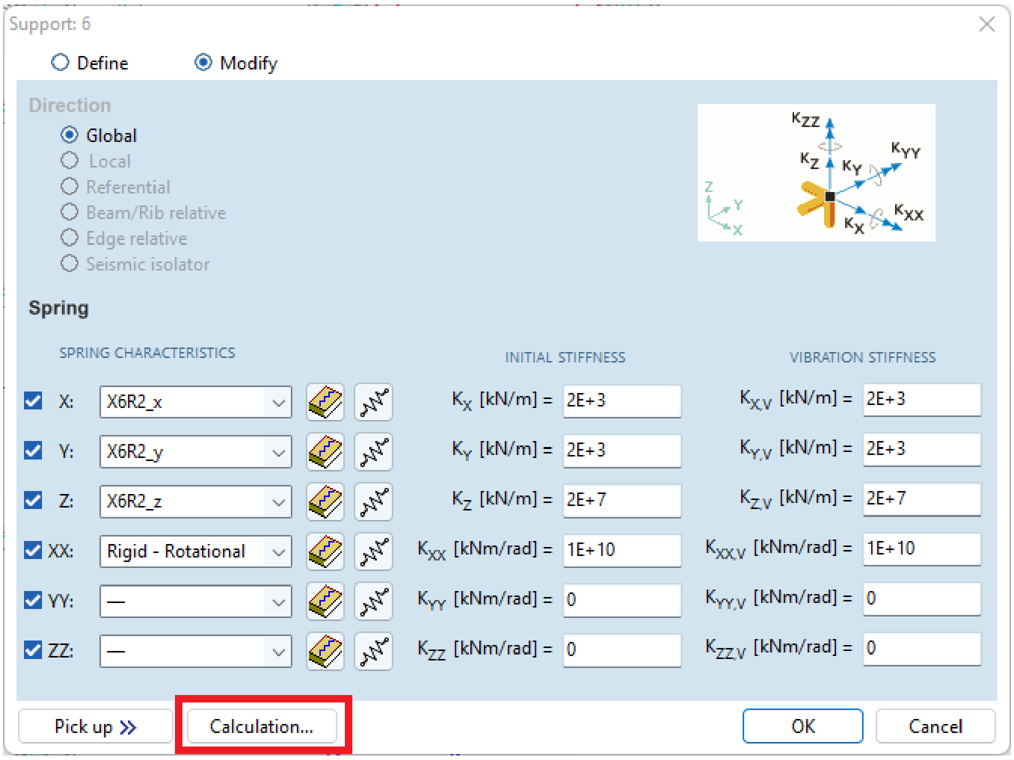
  ARBO cantilever slab connection element with thermal insulation

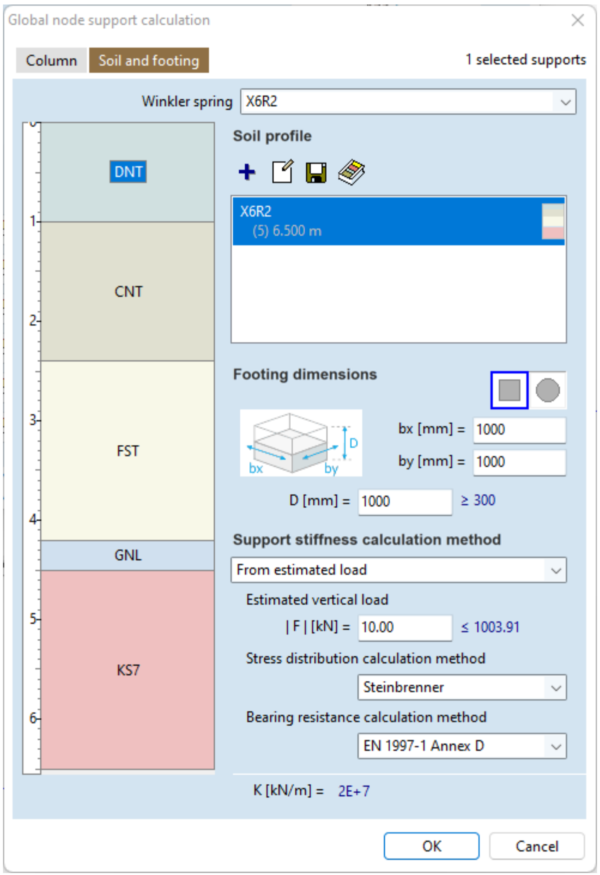
  CRET steel shear load connectors for movement joints

  AxisVM can calculate the number of required elements from the chosen type, and their utilization on domain-domain connections.

- ## Estimation of Winkler stiffness for supports

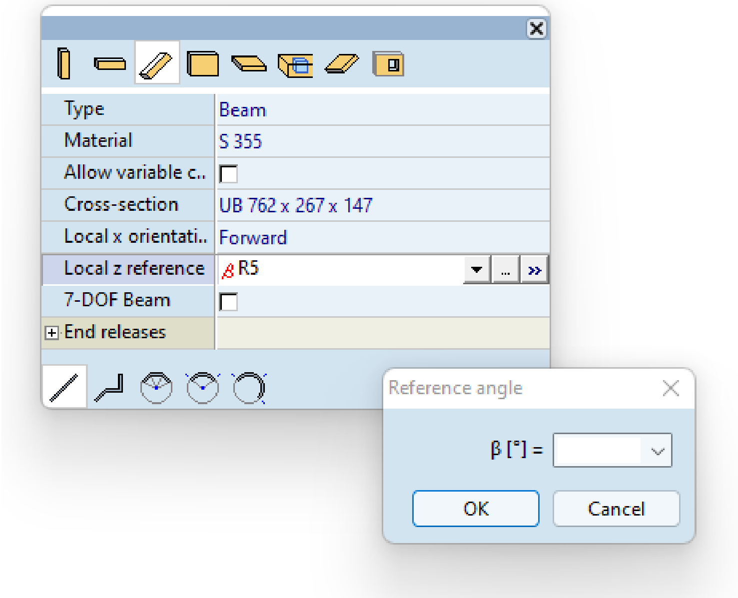
  Automatic estimation of Winkler stiffness for point, line and surface supports based on the underlaying soil parameters.



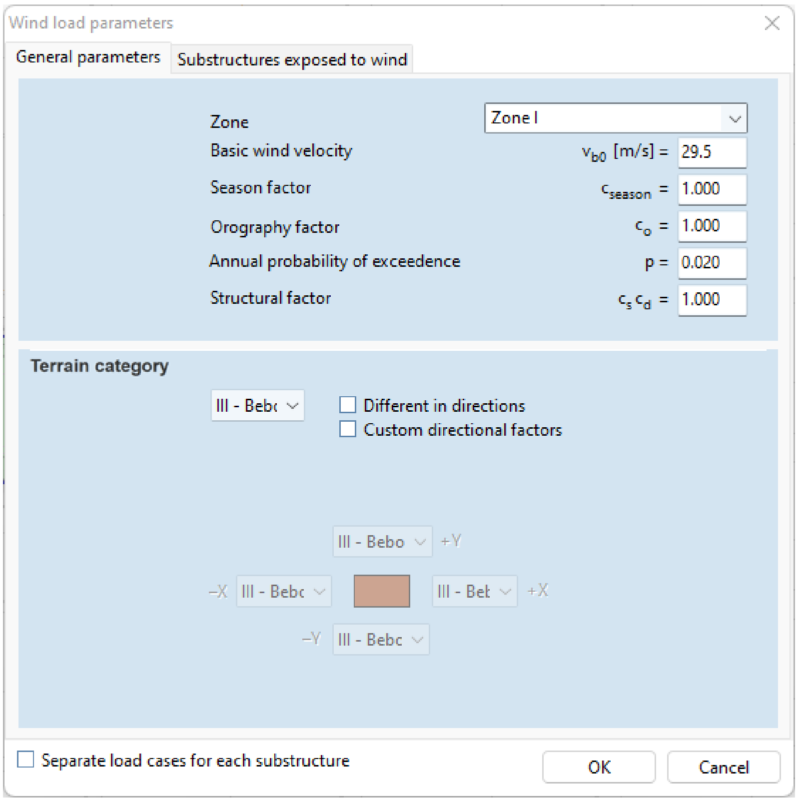


- ## Setting reference angle when drawing elements

  β reference angle is made available on the *Draw objects directly* dialog, therefore the alignment of the local Z axis of an element can be set this way when an element is being added to the model.



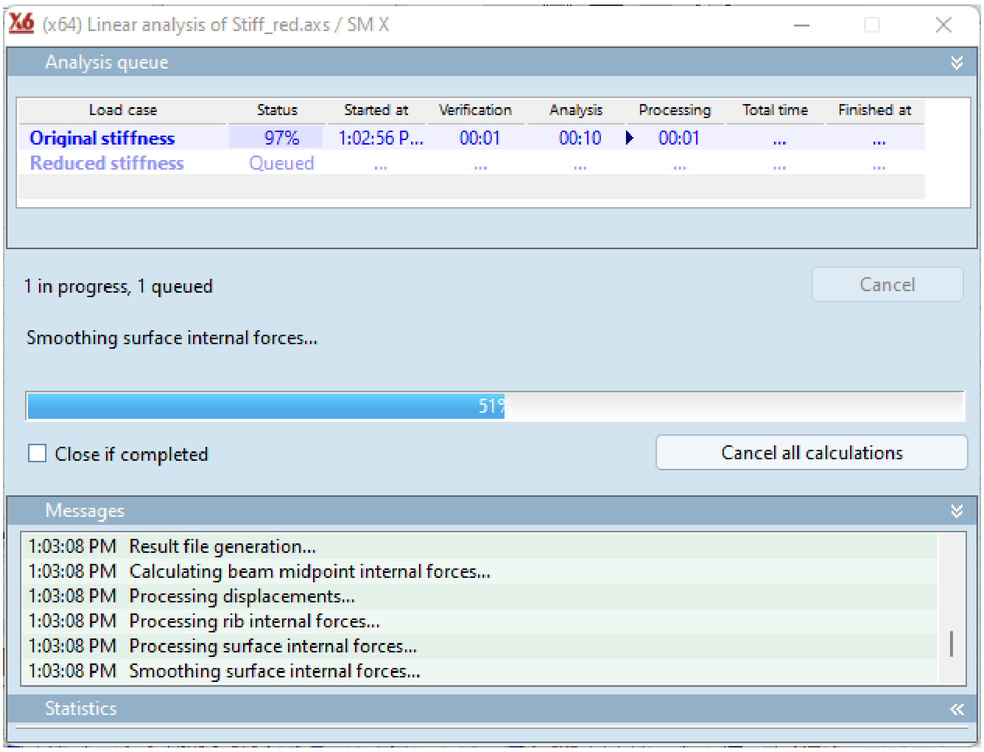
- SWG modul: Introduction of *Structural factor* in case of EC(NL)



ANALYSIS

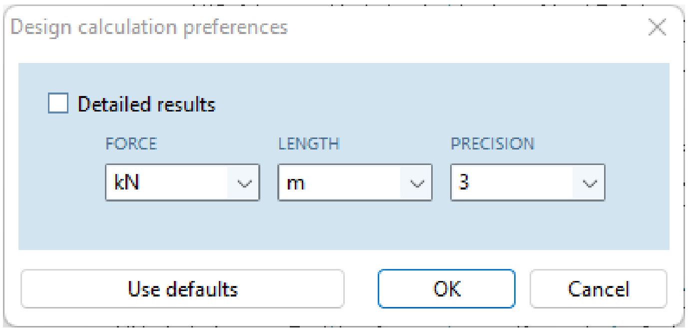
- Handling of reduced stiffnesses during analysis

  Stiffness reduction is optional in a vibration analysis. If the vibration analysis has been performed with stiffness reduction applied, and the response spectrum analysis used these modal shapes, AxisVM will perform linear analysis twice, first with stiffness reduction, then without it. For seismic load cases or combinations including a seismic load case the results obtained with reduced stiffness are displayed, while in other load cases or combinations the results obtained without reduced stiffness are displayed. The same method is applied when finding critical combinations or envelopes.



RESULTS AND REPORTS

- Detailed/simplified version of design calculation reports for footing, punching, RC beam and column design



- ## Calculation of normalized axial forces

  To ensure the ductile behaviour and energy absorption ability of reinforced concrete structures, it is often required to limit the normal forces in the elements. As a new result component, the *vd - Normalized axial force in reinforced concrete beams and columns* is calculated.

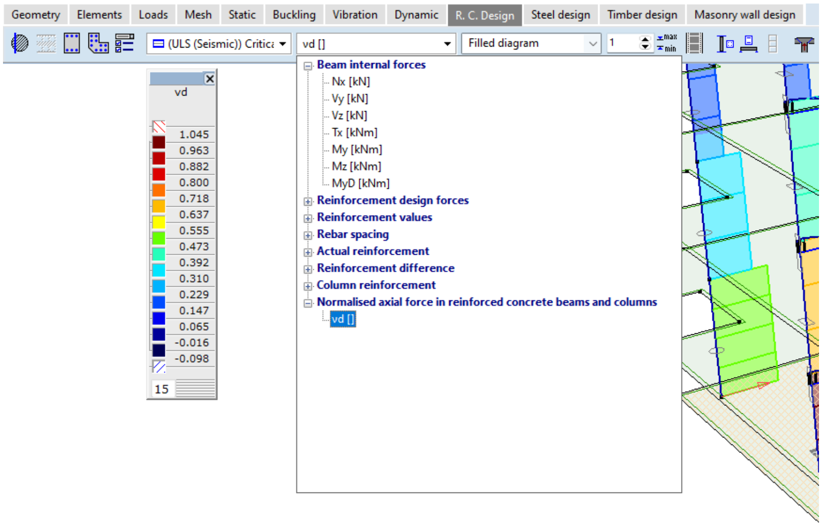
  $\nu = N/(b*h*fcd)$, where

  ν - normalized axial force

  N - axial force

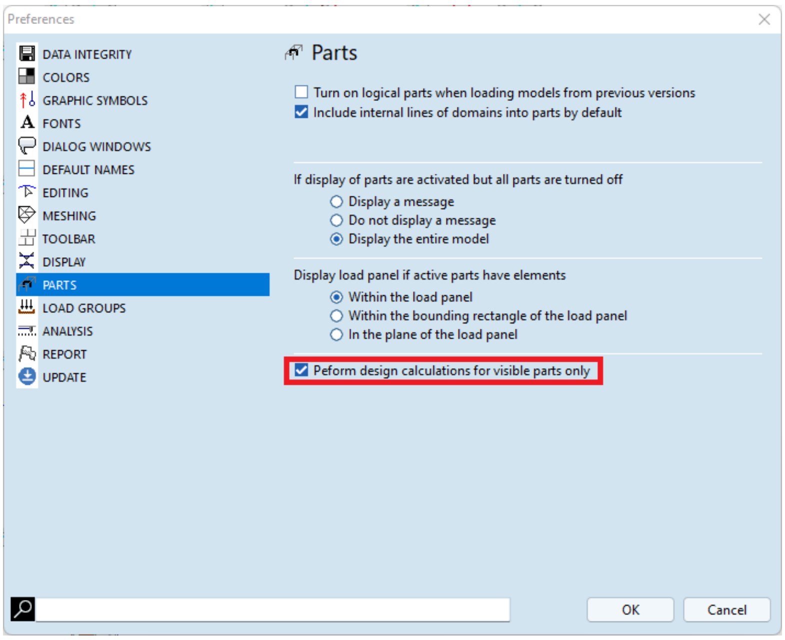
  b,h – cross-section dimensions

  fcd – design strength of concrete



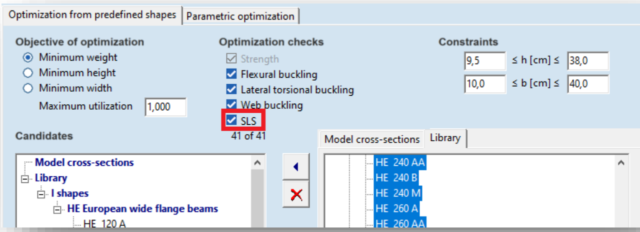
- ## Optimization of result evaluation process

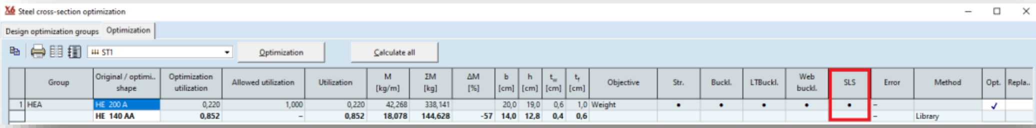
  Enabling the *Perform design calculations for visible parts only* under *Settings/Preferences/Parts*, calculation of critical stresses and internal forces will only be performed for the visible parts of the model. Using this option can help reducing the time of the design process when dealing with a specific part of the structure.



- Consideration of deflections in steel cross-section optimization (**SD9** module)

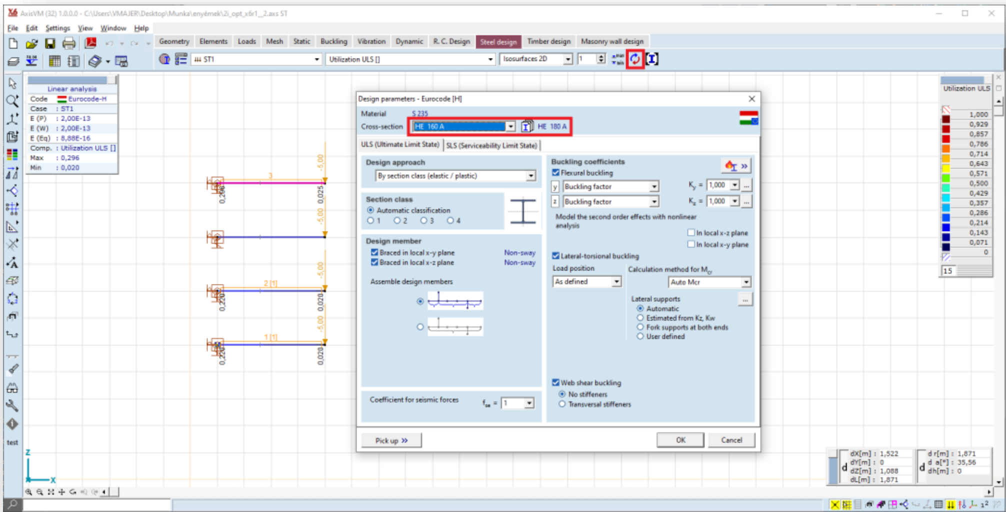
  Fitting into the existing workflow of ULS check optimization procedure, by clicking the *SLS* checkbox on the optimization dialogue, AxisVM will be able to use SLS utilization results as well from the Steel design module, and can offer cross sectional changes based on these at the end of the optimization process.





- Replace cross sections of steel design members (**SD1** module)

  Steel member design cross sections different from the original can be transferred to the model, by clicking the *Replace cross sections* button. Sections of design members, which have been changed, but not yet updated back to the model, are highlighted with dashed red contour.

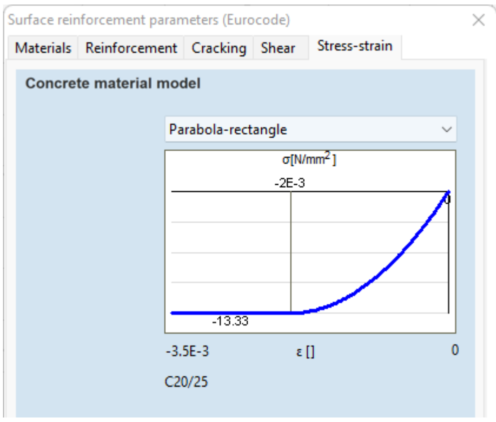


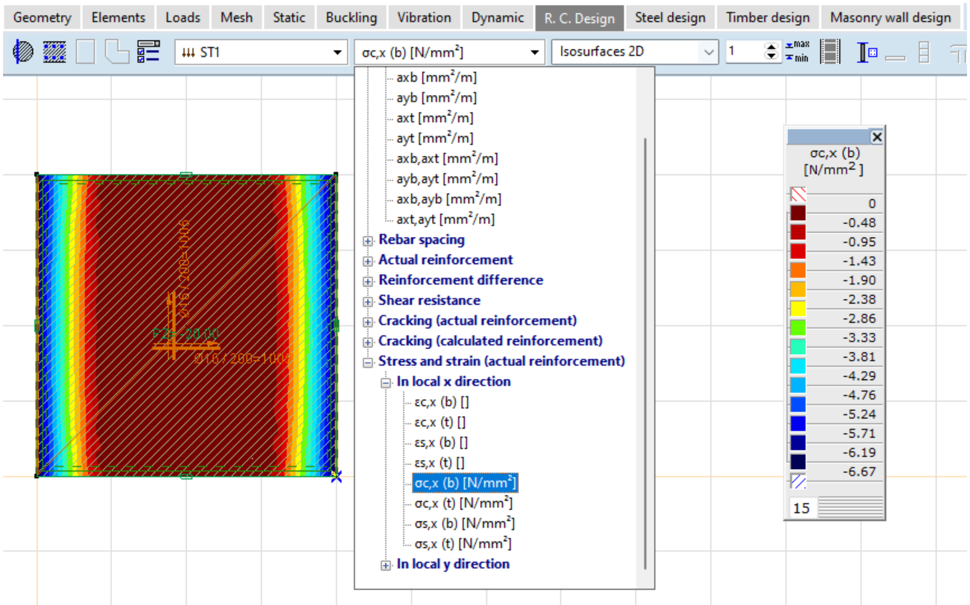
- Copy of lateral supports (**SD1** module)

  Lateral supports defined for a steel design member when using AutoMcr to perform lateral torsional buckling check can be copied with the *Pick up* button on the design parameters dialog.

- Visualization of stress strain results for concrete domains and rebars (**RC6** module)

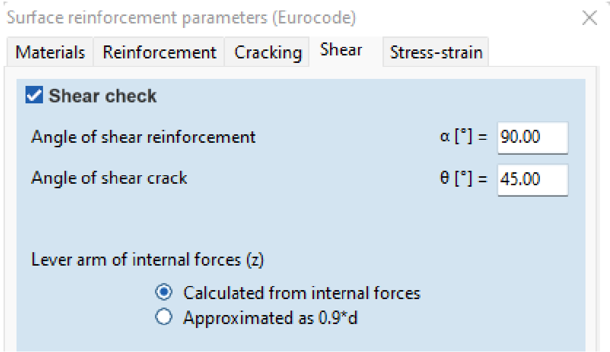
  Using the RC6 module, stress strain results can be obtained for the extreme layers of concrete domains, and for the rebars as well. (The function is available only if reinforcement is applied)





- ## Option to turn off shear check for reinforced concrete domains

  In the settings of the reinforcement parameters, the traditional beam-like shear check of the reinforced concrete domains can be optionally switched off.



- ## Fire design for reinforced concrete beams and columns (new **RC8-B** module)

  Using the RC8-B module, it is possible to perform fire design for truss, beam and rib elements made of reinforced concrete. User can choose from different standard fire curves according to the possibilities offered by the selected design code. The software calculates the temperature distribution within the reinforced concrete cross-section.

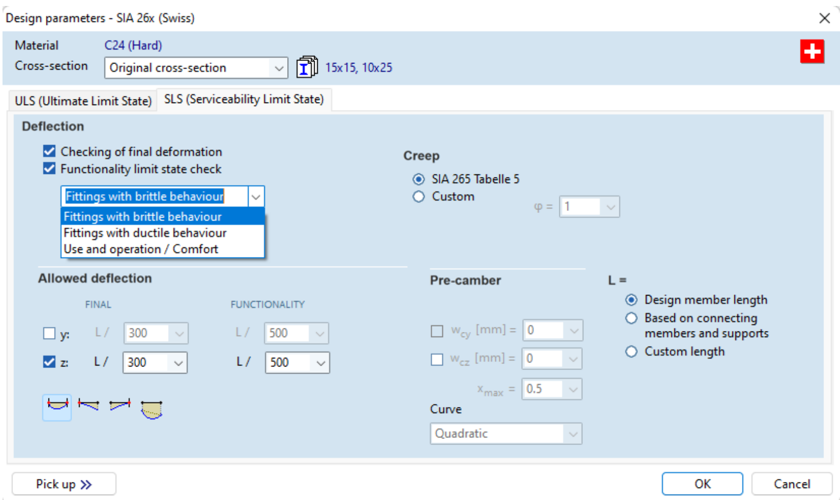
- ## Extended serviceability limit state checks according to SIA 260:2013 (TD1)

  Serviceability limit state checks for timber design members will be supplemented with the functionality check according to SIA 260:2013. Selecting functionality check, one of the following three types of checking can be requested:

  ‘Fittings with brittle behaviour’

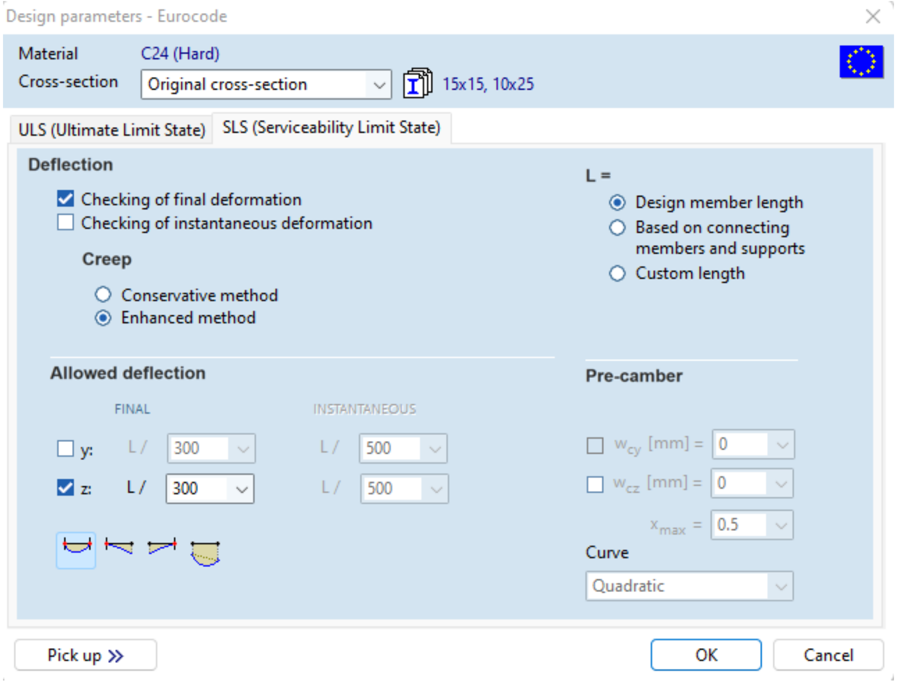
  ‘Fittings with ductile behaviour’

  ‘Use and operation’



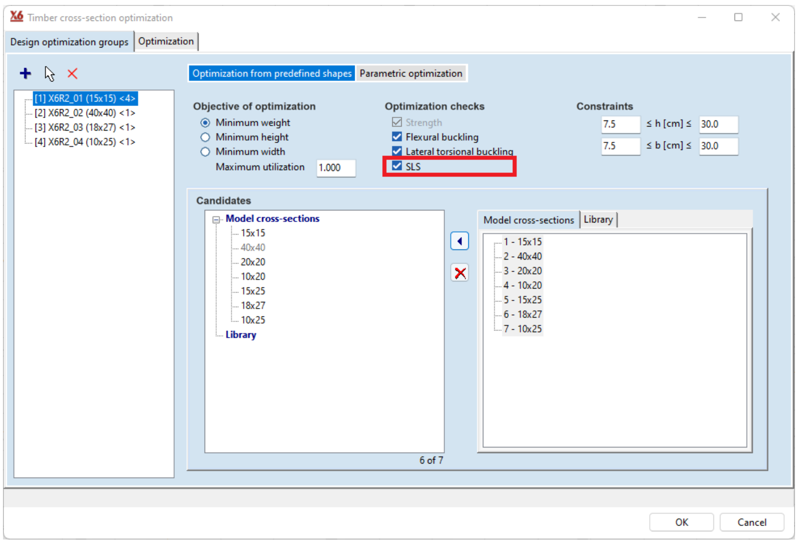
- ## Enhanced calculation of final deformation of design members (**TD1** module)

The calculation method of final deformation has been improved. In the previous versions, an approximate, conservative method was available. (The deflection was calculated considering a characteristic load combination) From the launch of X6R2, a precise method will also be available, where the final deflection is recalculated in the background for each timber element considering a quasi-permanent load combination.



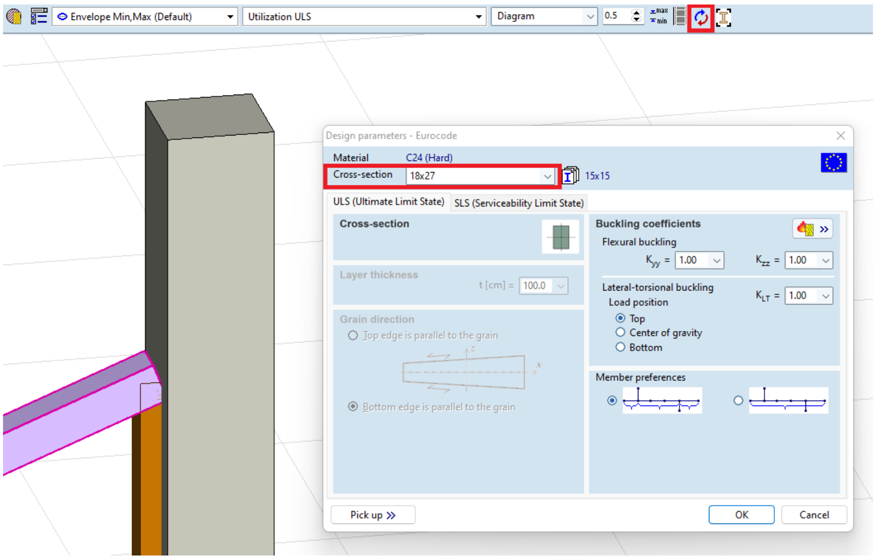
- ## Consideration of deflections in timber cross-section optimization (**SD9** module)

Fitting into the existing workflow of ULS check optimization procedure, by clicking the *SLS* checkbox on the optimization dialogue, AxisVM will be able to use SLS utilization results as well from the Timber design module, and can offer cross sectional changes based on these at the end of the optimization process.



- Replace cross sections of timber design members (**TD1** module)

Timber member design cross sections different from the original can easily be transferred to the model, by clicking the *Replace cross sections* button. Sections of design members, which have been changed, but not yet updated back to the model, are highlighted with dashed red contour.



- Shear check for masonry design according to SIA266:2015 (**MD1** module)

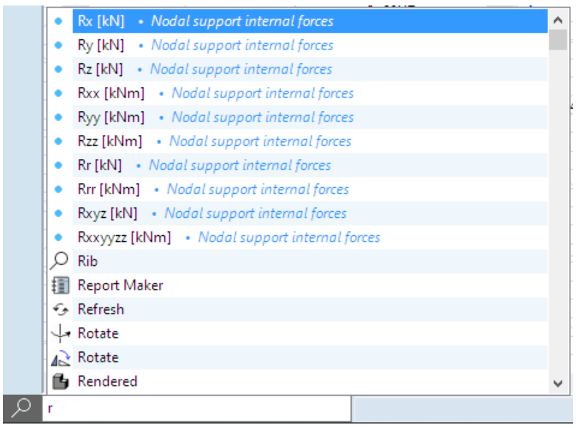
In the MD1 module, a new numerical method is implemented to find the maximum shear capacity in the case of the SIA standard. The algorithm considers the basic principles and boundary conditions of section 4.3.2.1 (SIA 266:2015) and solves the satisfying equation system by iterative method to find the possible stress field, which results in the maximum shear capacity.

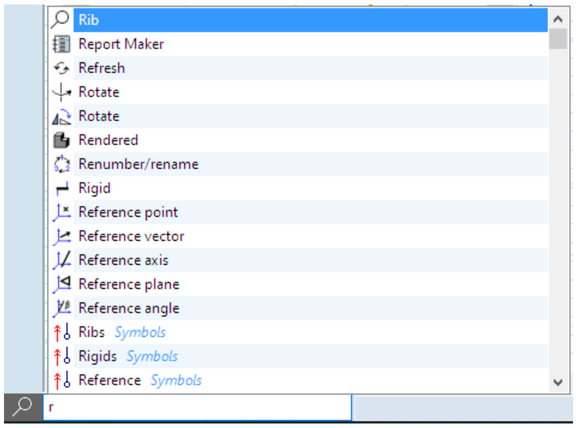
New features of AxisVM **X6 R1**

GENERAL FEATURES

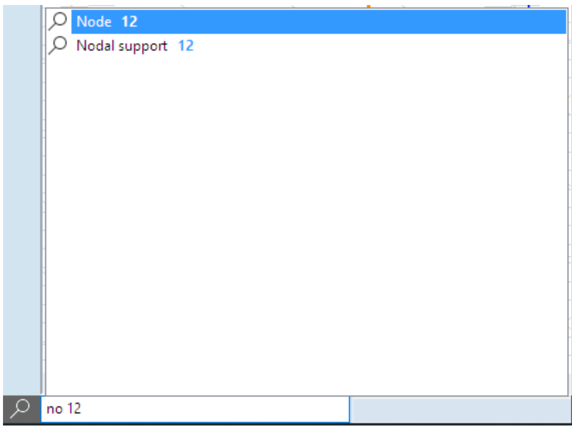
- Smart command line and search

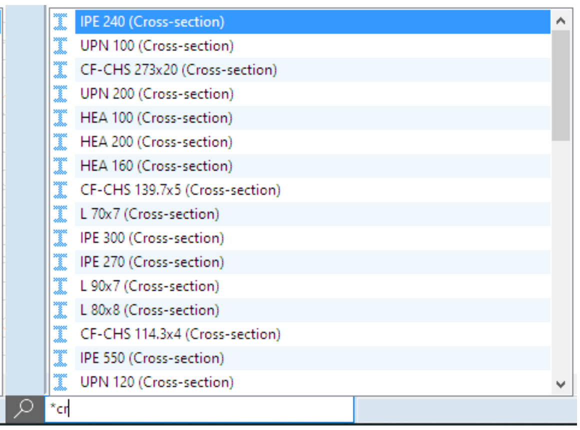
  Most of the commands and settings of AxisVM are accessible directly through a simple search function. By typing in letters of the command, a list of matches is displayed. The content of the list depends on the active tab of the main AxisVM window, e.g. the *Static* tab allows selecting result components, while the *Elements* tab does not. Load cases can be selected in both tabs.

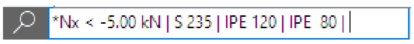




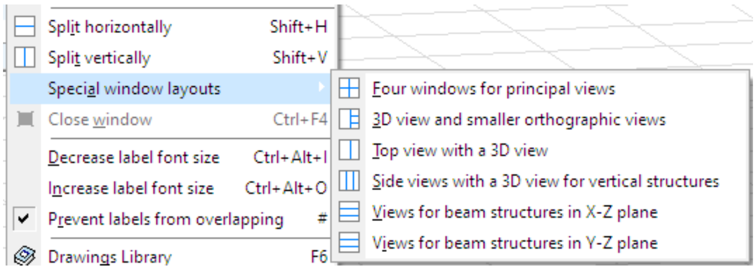
Additional functions are: finding an element by its number or name, selecting elements by property value (material, cross-section, length, eccentricity, domain thickness, domain area), by value of a result component, or by using multiple criteria.



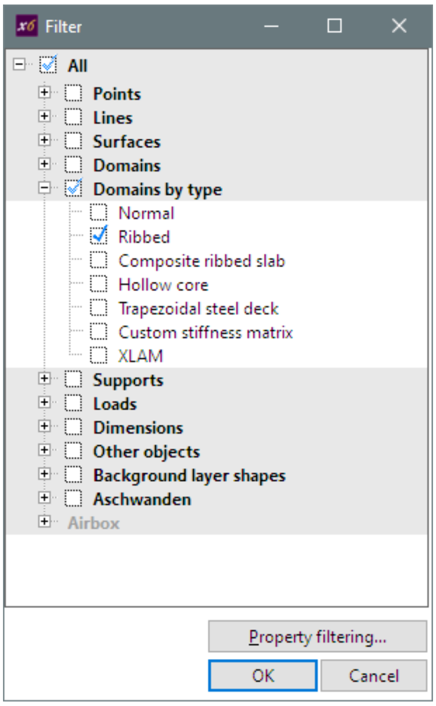




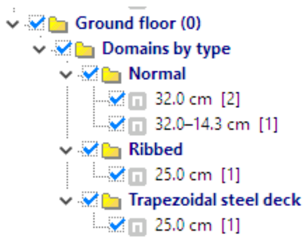
- ## Special window layouts in the Windows menu
  There are predefined layouts consisting of different views in each subwindow.



- ## Enhanced selection filter
  A selection can be filtered based on domain type (ribbed, hollow core, trapezoidal steel deck, XLAM, etc). The filter retains the last setting.



- ## Logical parts by type of special domains
  Logical parts are created based on the type of finite elements or architectural role of domains. X6 also creates logical parts based on the domain type.



- Mesh check selecting free edges
  This function selects edges connected to only one domain, making it easier to find meshing problems where meshes of two walls, or of a wall and of a slab, do not match.

- Customization of more graphic symbols
  Customizable color and size of beam/rib end releases and edge hinges.
  Customizable color and line thickness for logical and eccentric axes of beams and ribs.
  Customizable default font color and size for different labels (node number, material, cross-section, etc.)
- Copy/move virtual strips with the underlying elements
- Copy a system of structural gridlines
  It is easier to define slightly different systems of structural gridlines on each floor by copying an existing grid and then adjusting it.

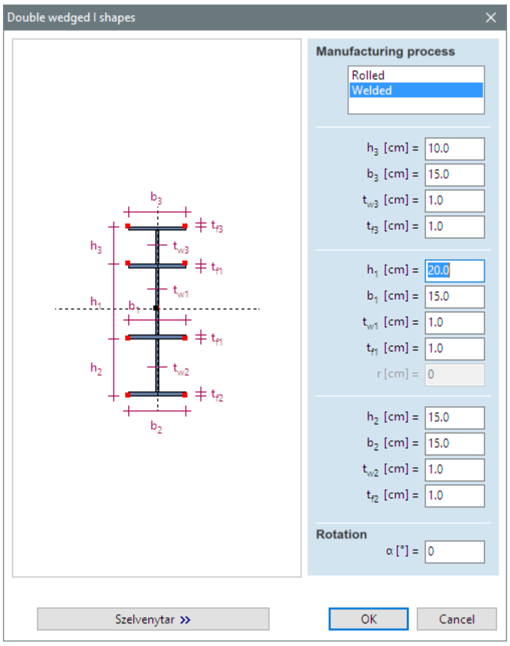
LINKS WITH OTHER PROGRAMS

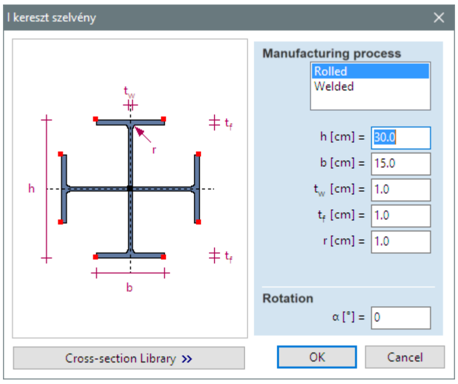
- New AxisVM components for Rhino/Grasshopper
  New plugin is available for parametric surface mesh generation .
- SAF interface imports and exports loads (SAF module)

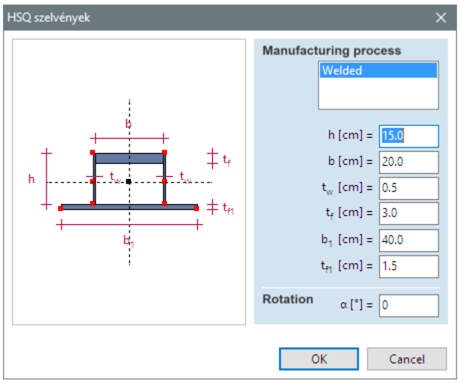
SAF

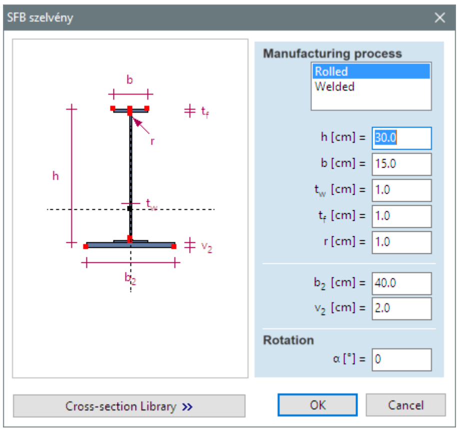
- Tekla interface imports and exports beam end releases, and nodal and line supports (TI module)

- Parametric double wedged I, crossed I, SFB, IFB, symmetric and asymmetric HSQ profiles



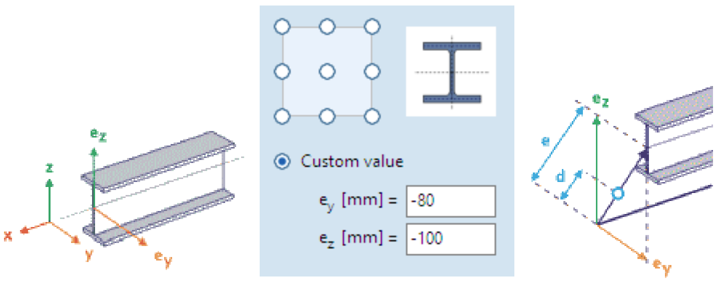


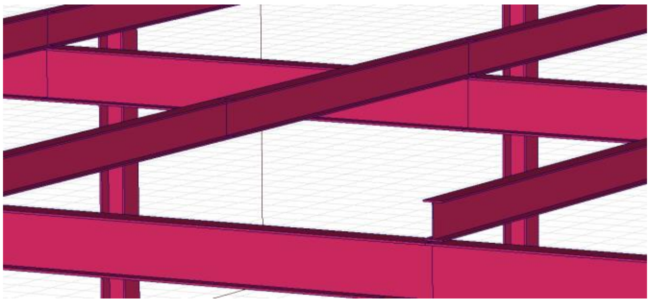




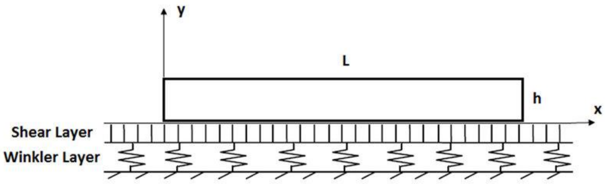
- Eccentric beams

  e_y and e_z eccentricities can be defined with an alignment point on the bounding envelope of the cross-section, or by entering custom values. Eccentricity of a beam placed on the top of other beams is calculated and updated automatically. Stiffness of the eccentric connection can also be controlled.





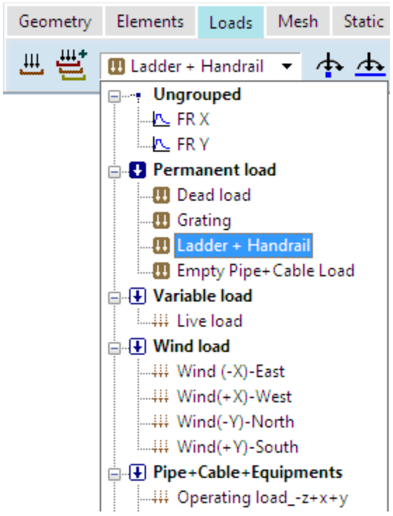
- Winkler-Pasternak elastic foundation
  This type of surface support also includes a shear layer.



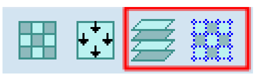
- New beam element with 7 degrees of freedom (new **7DOF** module)
- Copy nodal support stiffness values from an Excel table to AxisVM

LOADS

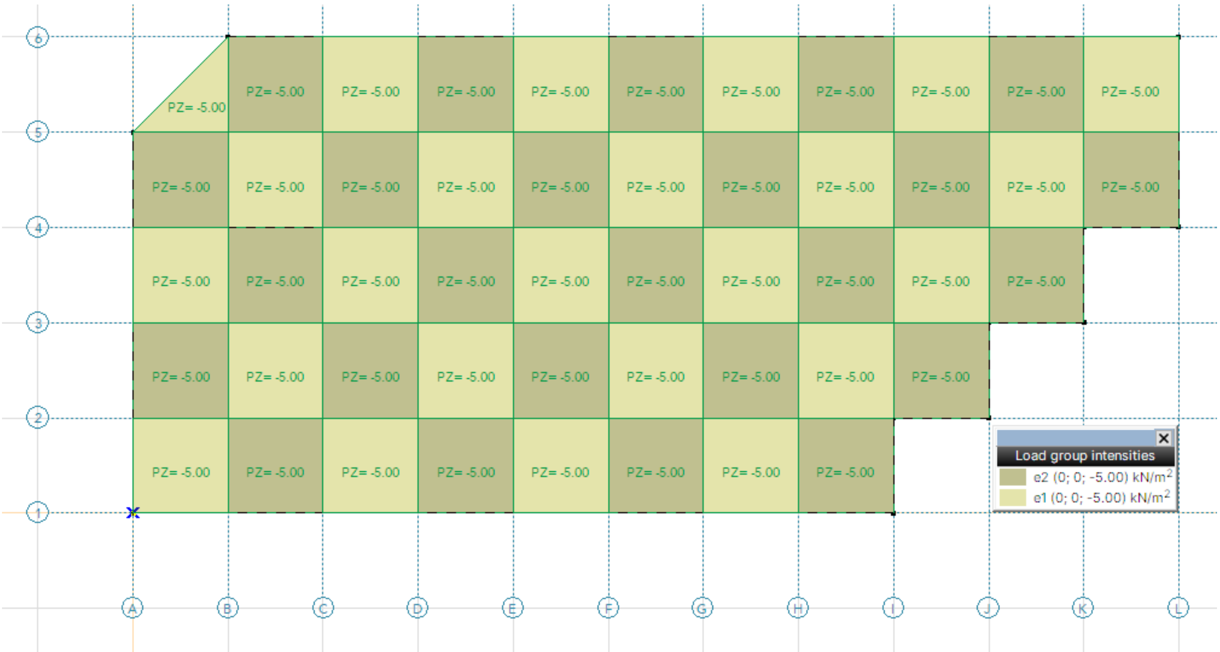
- Load cases can be selected from a structured dropdown list in the Loads tab



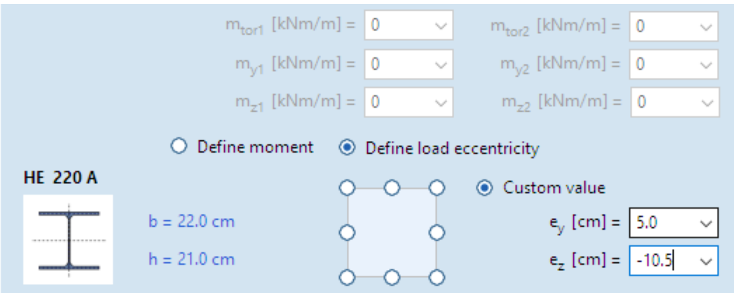
- Additional load split functions
  Split existing loads according to storeys or structural gridlines



- Display of color-coded surface loads of all load cases of a load group in a single view



- Generate wind loads from a CFD model (new **CFD** module)
  Export the model via an STL file for computational wind tunnel analysis and then import the nodal pressure values via a special file and convert them into static or dynamic loads.
- Display of color-coded snow and wind loads
- Applying eccentric concentrated or distributed loads on beams and ribs
  Instead of defining moments, loads can be created with an e_y and e_z eccentricity.



- Options to handle multiple loads
  Control what happens when loads with different or equal intensities are placed in the same position.
- Customized default color of surface and line loads

- Rules to generate custom combinations in the critical load group combinations table

Critical load group combinations

	PERM1	VAR1	VAR2	VAR3	VAR4	VAR5	VAR6
1	Active	Simultaneous (1)	Simultaneous (1)	Simultaneous (2)	Simultaneous (2)	Excluded	Active
2	Active	Simultaneous (1)	Simultaneous (1)	Simultaneous (2)	Simultaneous (2)	Active	Excluded
3	Active	Active	Active	Active	Active	Active	Active
4	Active	Simultaneous (1)	Simultaneous (1)	Leading (1)	Leading (1)	Active	Active
5	Active	Leading (1)	Leading (1)	Leading (2)	Leading (2)	Active	Active
6	Active	Simultaneous (1)	Simultaneous (1)	Leading (1)	Leading (1)	Leading (2)	Leading (2)
7	Active	Leading (1)	Leading (1)	Active	Active	Excluded	Excluded
8	Active	Leading (1)	Leading (1)	Excluded	Excluded	Active	Active
9	Active	Active	Active	Excluded	Excluded	Excluded	Excluded
10	Active	Excluded	Excluded	Active	Active	Excluded	Excluded
11	Active	Simultaneous (1)	Simultaneous (1)	Leading (1)	Leading (1)	Active	Excluded
12	Active	Simultaneous (1)	Simultaneous (1)	Leading (1)	Leading (1)	Excluded	Active

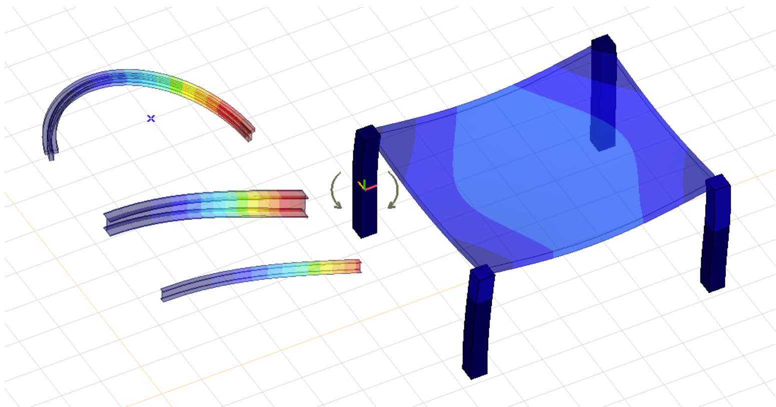
- Define the direction of spectra for seismic analysis
- Define moving loads on load panels
- Transparent display option for load panels
- Extend selection to loads with the same load component magnitude

ANALYSIS

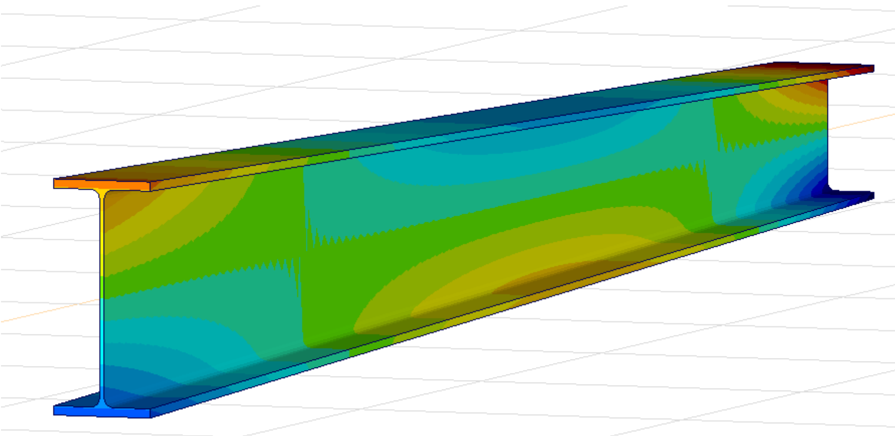
- Obtain imperfect shape from buckling shape(s) (New **IMP** module)
- Nonlinear analysis with reinforcement calculated from the ULS envelope of nonlinear results
  In earlier versions, only two options were available: *Actual reinforcement / Reinforcement calculated from critical internal forces*

RESULTS AND REPORTS

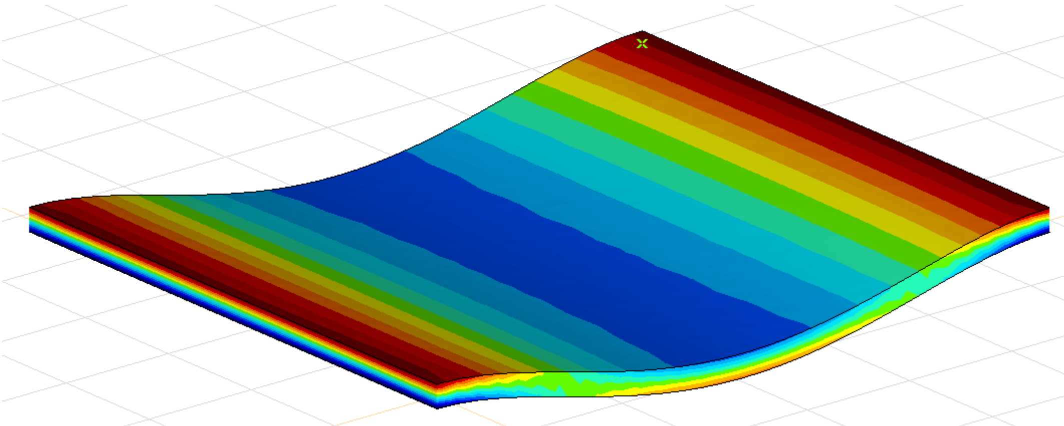
- Results display in rendered view



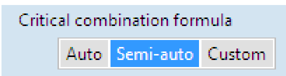
- Rendered view can display distribution of normal stress within the cross-section of a beam



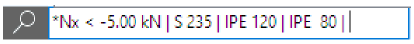
- Rendered view can display top and bottom components of surface stresses, reinforcement amounts, and crack width values simultaneously (**RC1** module)



- New automatic critical combination selection
  Semi-auto allows for choosing an SLS combination for results to be calculated from an SLS combination. The proper combination is selected automatically in other cases



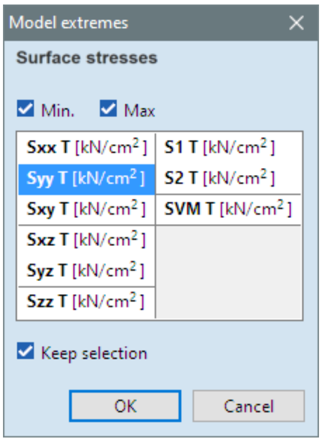
- Selection filtering by any result component
  See Smart command line above



- Bicolor (positive/negative) color display mode
  Setting two ranges for (min–0) and (0–max) in blue and red.

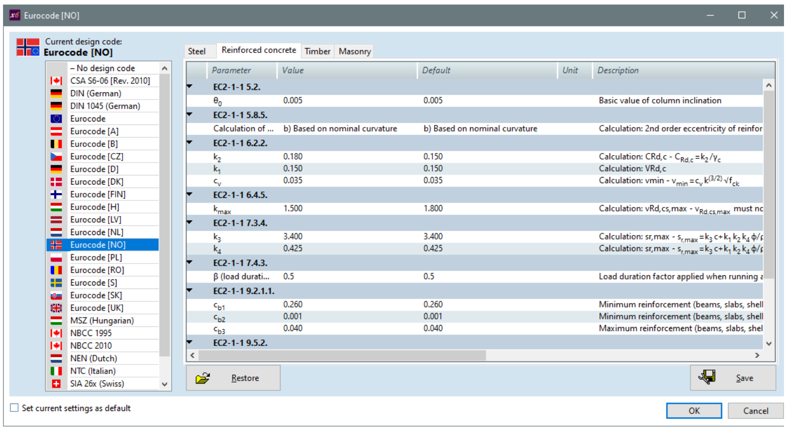
- New color legend option
  The new option does not display hatching and/or labels for values above or below the range.

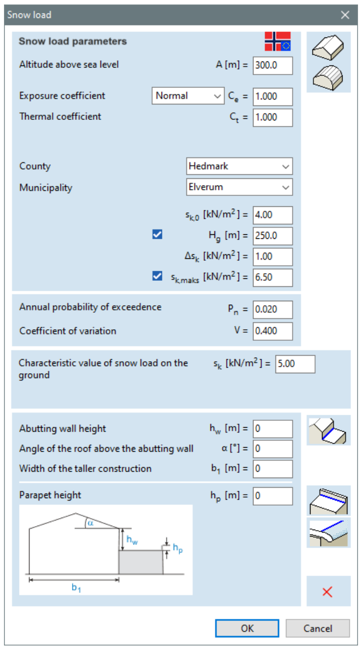
- New function for searching for minimum / maximum values
  Select which extreme (min. or max.) should be displayed. *Keep selection* helps when using the command *Show only selected elements* on the elements with extreme values.



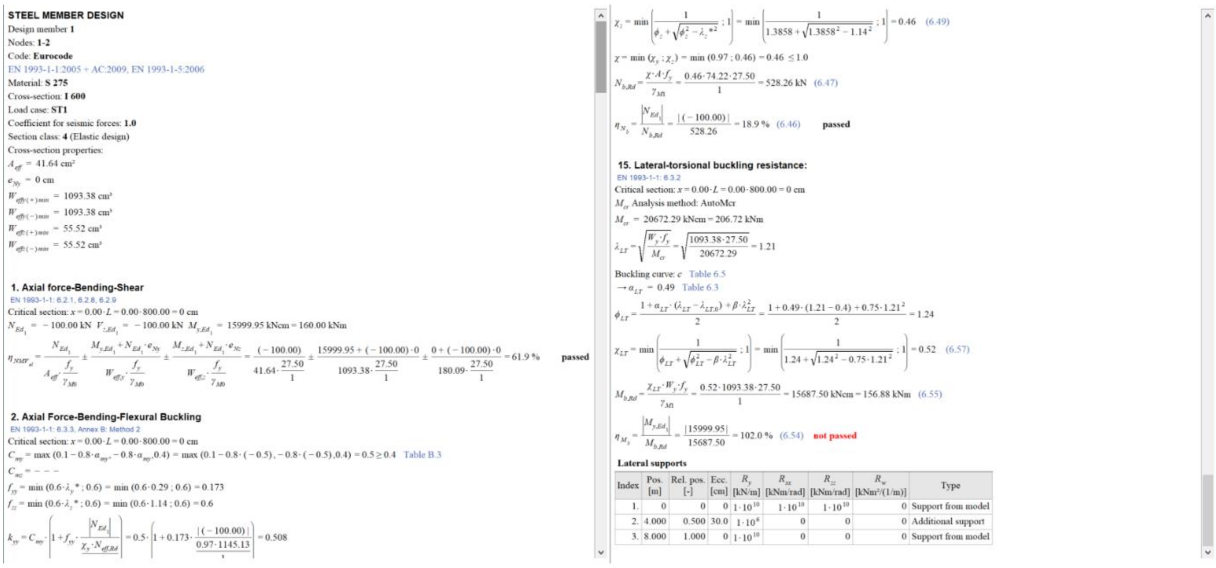
DESIGN

- Support of Norwegian NAD for Eurocode

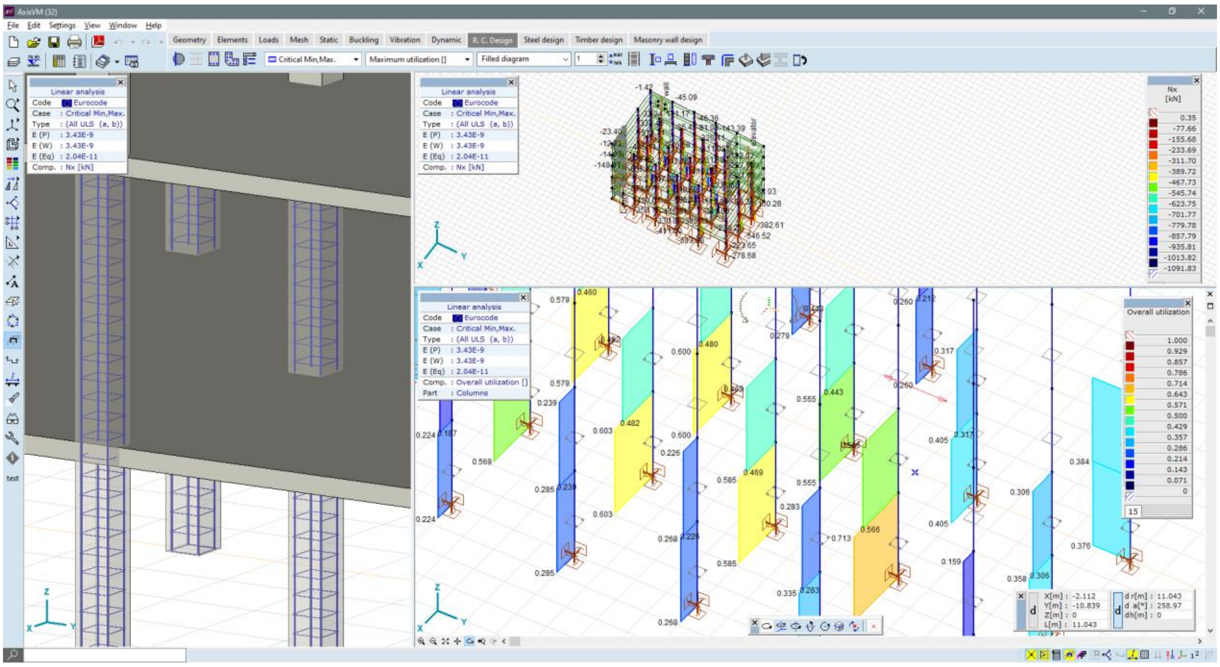




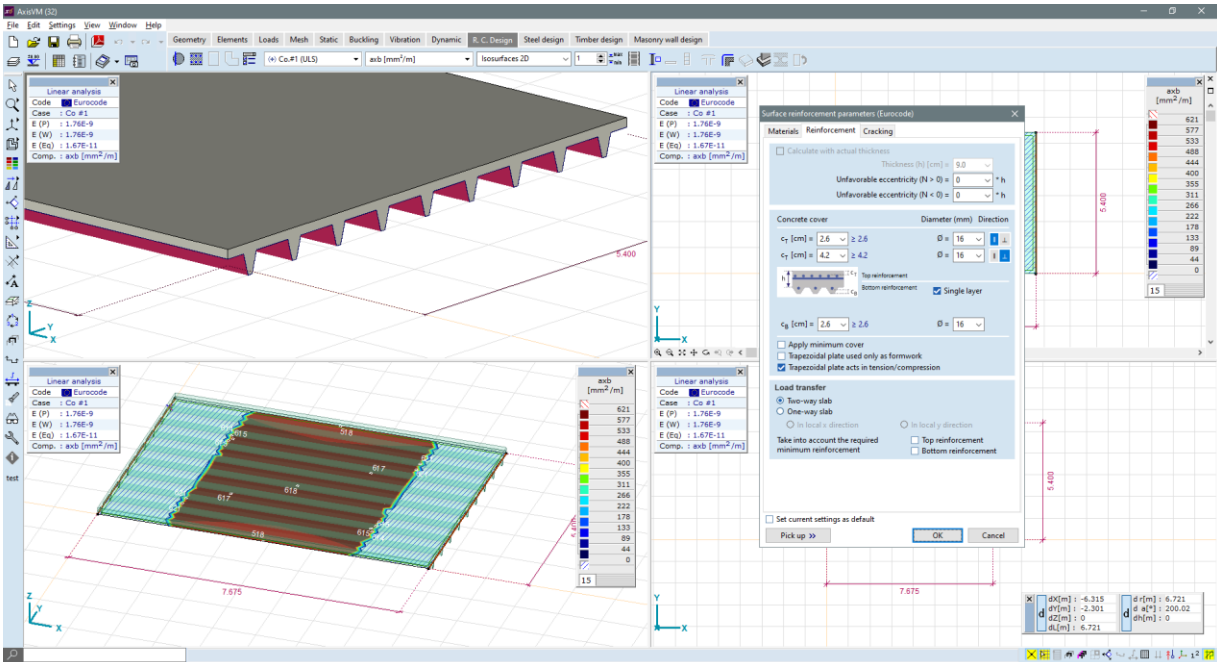
- Steel design calculation reports include effective cross-section parameters and lateral supports (**SD1** module)



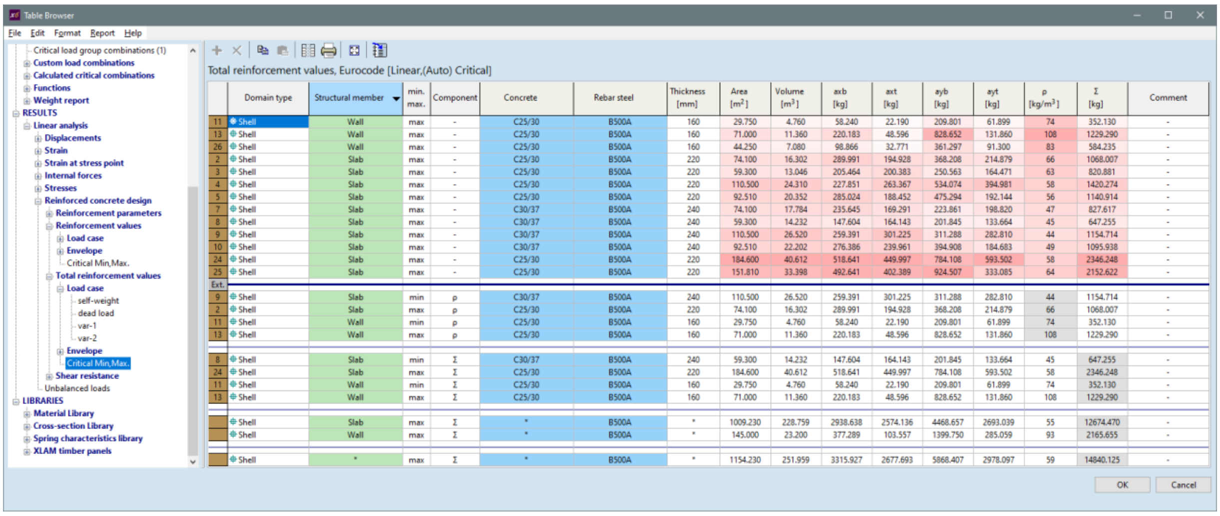
- Speed improvements in reinforced concrete beam design calculations by running multiple threads (**RC2** module)
- Utilization tables for reinforced concrete columns, reinforced concrete walls, and masonry walls in the Table Browser (**RC2**, **RC5**, **MD1** modules)
- Utilization result component and diagrams for reinforced concrete columns (**RC2** module)



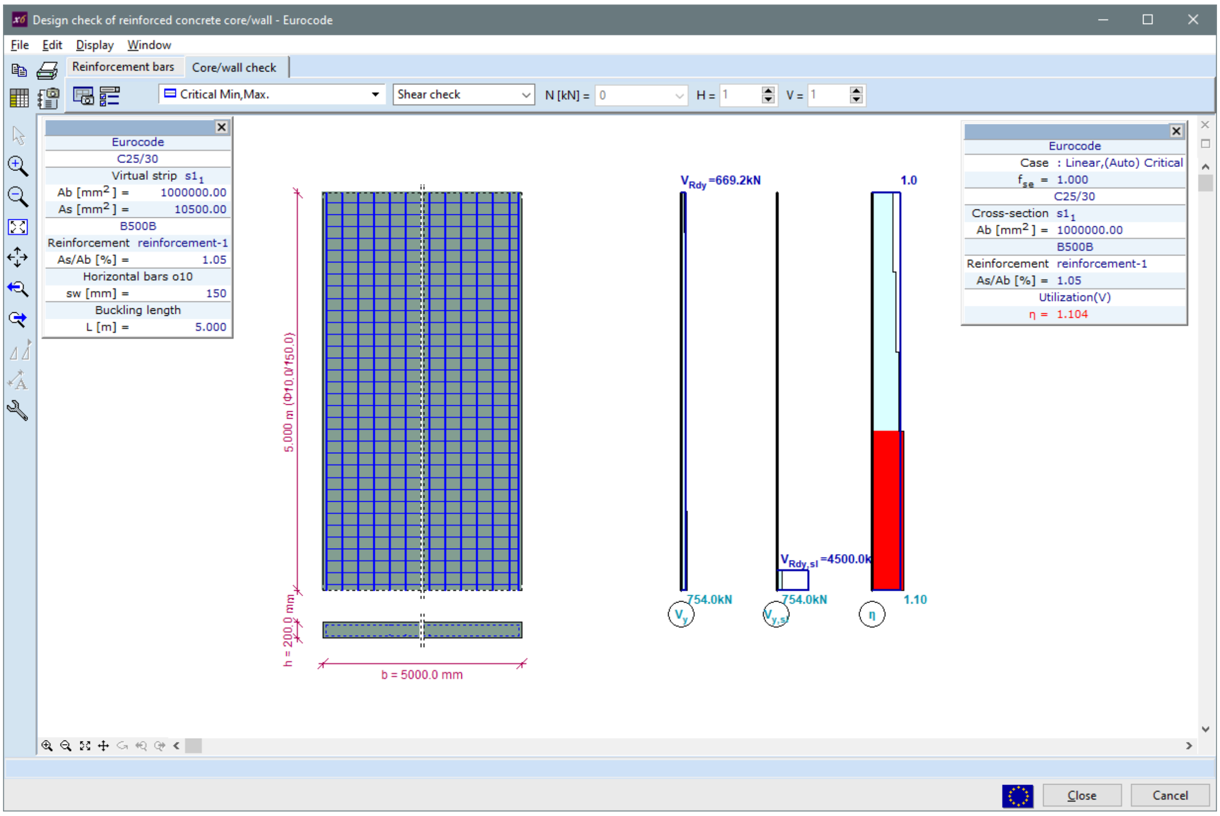
- Calculation of the required reinforcement for reinforced trapezoid steel decks. (**RC1** module)



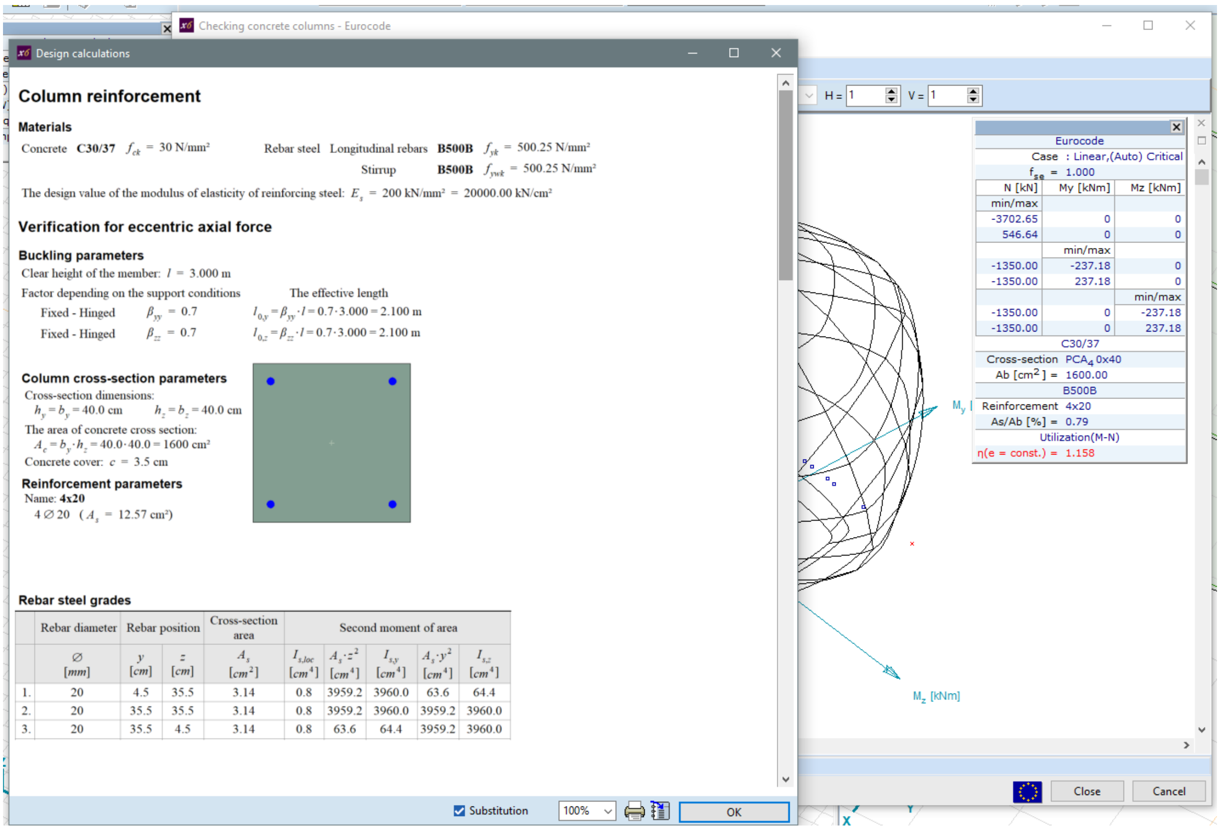
- New table for specific amount of reinforcement calculated for each domain (**RC1** module)



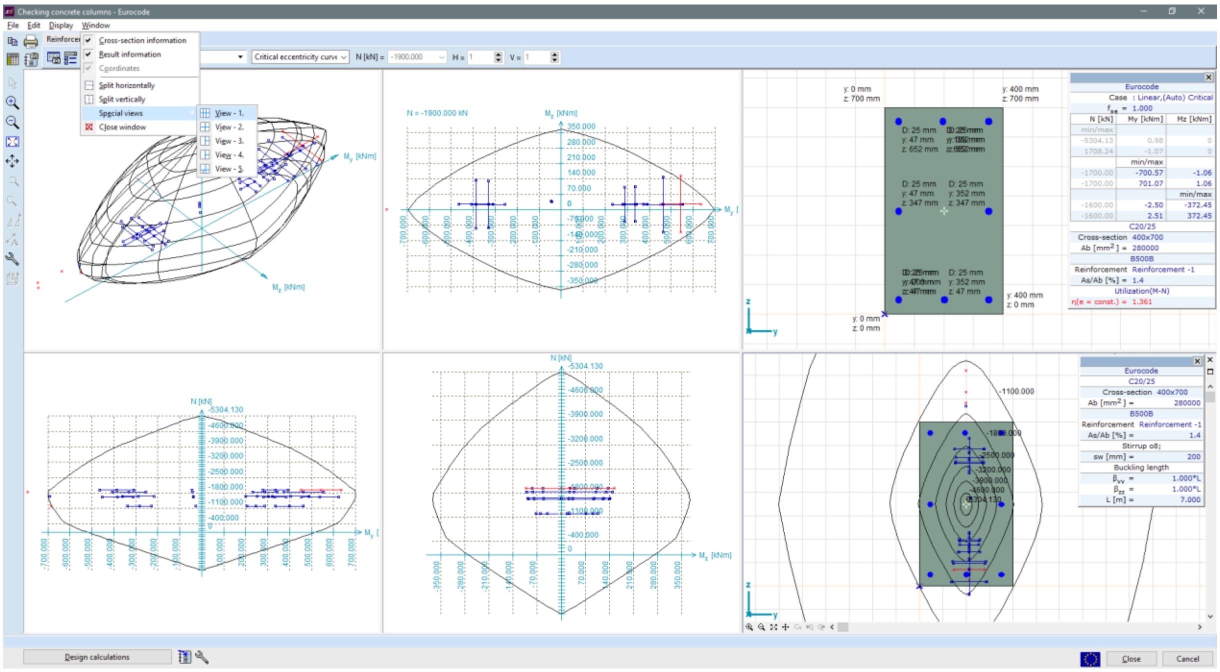
- Shear design of reinforced concrete walls (**RC5** module)



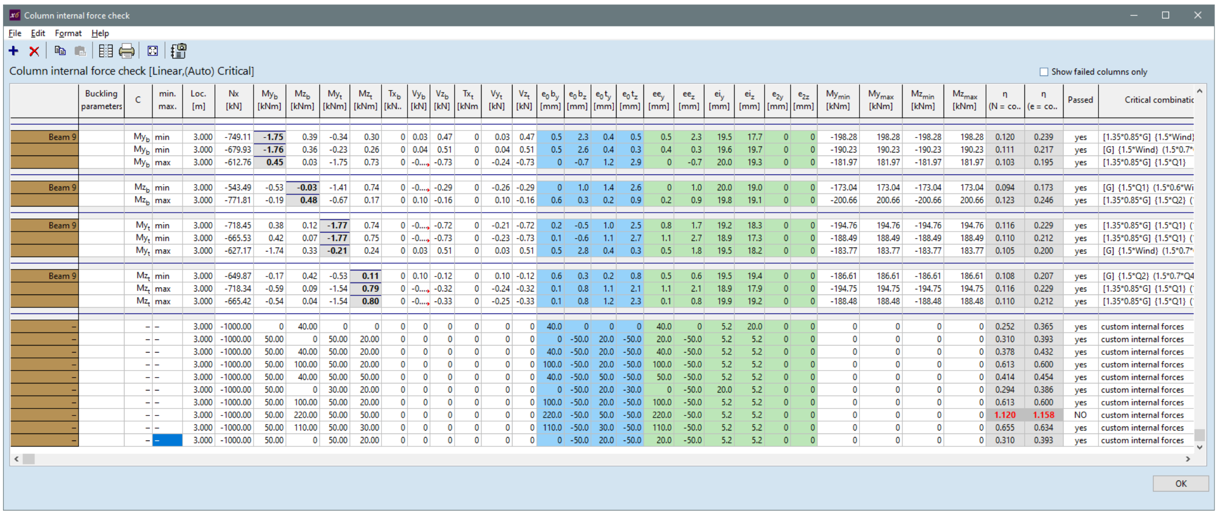
- Design calculation report for reinforced concrete columns (**RC2** module)



- Predefined window layouts for column reinforcement design (**RC2** module)



- Custom force values to design a reinforced concrete column are saved with the column (**RC2** module)
- Custom forces for column design can be imported via clipboard (**RC2** module)



February 8, 2021