GRAITEC MODELING THE FUTURE

What's New
in Advance Design 2024

Table of contents

1. Welcome to Advance Design 2024

GRAITEC is pleased to present the latest version of the leading structural analysis software – **Advance Design 2024**, part of the Graitec Advance suite.

GRAITEC has continuously strived to provide first-rate advancements for innovative software solutions to its valued customers, and the recent launch of its new and upgraded product range for 2024 is no exception, proving they are still on top of its game in terms of providing top-level Construction, AEC, and Building Design software solutions worldwide.

This version 2024 of Advance Design is enhanced with a lot of users-centric new functionalities with high-end benefits, and is articulated around a few main subjects:

- **New computing capabilities** directed towards **sustainable design**, including carbon emission estimations, cost estimations, but also displacement control nonlinear analysis.
- **Enhancing the analysis of timber structures acc. Eurocode 5** by increasing the scope of verification, including new Eurocode checks for bearing, notches, and compound sections, and the optimization of timber elements according to the deflection.
- **Enhance steel structure design capabilities** mainly by allowing design checks of cold form sections acc. American AISI code, web buckling analysis sections acc. Eurocode 3 and new welded truss tube connections.
- **Enhance concrete structure design capabilities** with new options and improvements to RC design modules, including new shapes of bars for RC Slab, new drawing capabilities, or transferring rib results to RC Beam.
- **Enhanced user experience and the comfort of program operation** by introducing improvements to facilitate everyday work, including templates for selections, and new commands for faster selecting, or filtering sections on databases.

Version 2024 of Advance Design also comes with a vast number of improvements and adjustments following the feedback received from thousands of users worldwide.

2. Advance Design - Quick list

This is a condensed/short list of new features for Advance Design 2024:

New computing capabilities

- **Carbon emission estimations** – Determination of CO2 emissions based on carbon factors entered for materials and/or elements. Results in graphical form and reports.
- **Cost estimations** – Determination of costs based on unit prices for materials and/or elements. Results in graphical form and reports.
- **Displacement control nonlinear analysis** – The ability to conduct nonlinear calculations using displacement increment steps, which allows to correctly analyze highly nonlinear problems with the post-peak behaviors and can easily manage the snap-through problems.
- **Defining initial constraints on linear elements per load case** – A new way of defining and considering initial constraints, making it easier to include initial constraints in combinations, as well as considering their impact on the rest of the structure.
- **A better formulation of drilling rotation in shell elements** – Thanks to this new formulation, shell elements are now capable of capturing torsional moments transmitted from perpendicular beam elements.

Enhance steel structure design capabilities

- **Verification of steel cold-formed sections according to AISI S100**– The possibility of performing the standard verification of cold-formed steel profiles according to the AISI S100-16 American standards.
- **Shear web buckling analysis for I sections according to EC3** – Verification of the stability of I-section webs (including the impact of transverse and longitudinal ribs) according to EN 1993-1-5.
- **User definition of LTB lengths for cold-formed sections for EC3** - Next step of improvements to cold-formed design acc. Eurocode 3 for analyzing buckling and lateral-torsional buckling, especially applied when the Advanced Stability analysis is not used.
- **Modeling welded tube connections** – Ability to define welded truss connections in the model for rectangular hollow sections, which can then be sent for analysis in Steel Connections design module.
- **Deflection in value in the shape sheets and diagrams** – When presenting the results of verification of deflection of steel elements in graphical form and in reports, deflection values can now be presented both in ratio form and as values in a unit of displacement (e.g. cm).

Enhancing the analysis of timber structures acc. Eurocode 5

- **Design of double rectangular sections acc. EC5** – Possibility of verification of timber beams and columns from compound profiles (consisting of two, three, or four rectangular section branches) according to EN 1995-1-1.
- **Verification of bearing pressure on the support acc. EC5** - Possibility of performing additional verification for timber elements - the check of the compression perpendicular to the grain according to support sizes, according to EN 1995-1-1 (6.1.5).

- **Verification of beams with a notch at the support acc. EC5** - Possibility of performing additional verification for timber elements - the check of the notch on an element ends, according to EN 1995-1-1 (6.5).
- **Optimization of timber elements according to the deflection (EC5)** – Ability to automatically select the optimal timber section due to the deflection condition according to EN 1995-1-1.

Enhanced user experience and the comfort of program operation

- **User templates for Selection** – The ability to save and easily select templates with object selection. The templates can contain both the current selection and the selection criteria.
- **Selecting identical elements** – A new option that allows you to quickly select objects that are identical to a previously selected one, with easy consideration of the range of properties being compared.
- **Possibility for filtering non-active elements** – A new filter in the selection by criteria window allows the selection of elements excluded from the creation of the analytical model.
- **Filtering profiles from databases** – Easily search for profiles from the library by filtering content by profile type or text from a name, as well as by separating cold-formed profiles.
- **Commands on the ribbon for adding stages** – A set of icons on the ribbon related to adding new construction stages and assigning elements to them.
- **Improved mesh generation for Pushover analysis** – When generating a calculation model with elements with defined plastic releases for pushover analysis, the finite element mesh at the ends of the elements is now regular, which significantly improves the mesh distribution in adjacent surface elements.
- **The next stage of unification of dialog windows** – Another set of dialog windows has been updated, giving them a unified appearance and components (Concrete settings, Steel settings, Timber settings, Design templates, Element selection, NL setting, Pushover settings)
- **Improvements to the selection window by criterion** – Several changes have been made to the Element selection window that, among other things, make it easier to choose a selection method.

Other novelties and small improvements

- **New database with cold-formed steel profiles for North America** – A new library of cold-formed steel profiles typical for the North American market.
- **New steel material database for North American cold-formed sections** - A new library of materials for cold-formed steel profiles according to the ASTM standard.
- **Spaced shafts parametric section** – Possibility to define parametrically profiles composed of two, three, or four identical rectangular shafts.
- **Improved combination creation with seismic cases for Canada** – During the automatic generation of combinations with seismic cases according to the NBC standard, seismic combinations are now considering both directions (EX+, EX-, EY+, EY-).
- **Defining Link at node on selection** – An additional way of quickly defining multiple links to node at the same time has been introduced, allowing to point to primary and secondary elements independently.

- **Possibility of material definition for supports** – The option to assign a material (concrete) to supports has been introduced. This makes it possible to both include foundations in the calculation of costs and CO2 emissions, as well as to send the concrete class to RC Footing module.
- **Dedicated system for master-slave links** – Master-slave links automatically generated (where the slab connects to the column or support) are now automatically placed in a dedicated system.
- **Improvements to the program defaults for North America** - Better customization of some of the program defaults when selecting a localization for the US or Canada.
- **Improvements in localization for Spain and Portugal** – The translations for Spanish and Portuguese languages have been improved and expanded on missing areas (like the Section editor).

Common improvements to all Design modules

- **Applying project templates when the module is already opened** - Ability to update settings from a template for a currently open project.
- **Display the steel ratio in the Info Panel** - Quickly available information on the steel ratio.
- **Small improvements** – Set of various small enhancements.

RC Beam design module

- **Import geometry and internal forces from Rib design –** Consideration of effective section and design internal forces when importing beams with rib option active.
- **Possibility to define the position of supports (columns/ walls)** - Column supports now distinguish whether they support a beam from below, above, or from both sides. This data is now imported from the Advance Design model and affects the drawings.
- **New support type "beam"** - New support type enables the correct generation of drawings when the support is another beam.
- **Ability to define secondary beams -** Possibility for generating better drawings by considering the position of the secondary beams approaching perpendicularly.
- **New diagrams for reinforcement for torsion** - The ability to display two new diagrams - the longitudinal reinforcement from torsion and transversal reinforcement for torsion.
- **A new chapter in the report on openings reinforcement** - In the detailed report for beams, if there are openings in the beam, there is now a new chapter on the reinforcement of openings.
- **Small improvements**
    - A better description of extreme dimensions in the drawing
    - Improvements to Reinforcement assumption dialog
    - Modification of the default Wmax value for France
    - Adjustment of waterproofing data entry for France
    - The additional coefficient for modification of the theoretical area of the longitudinal reinforcement

- Possibility to choose the way of distribution of reinforcement for torsion.
- Improvement in defining suspended loads with link distribution.
- Support conditions are added to the Design Assumptions dialog.

RC Column design module

- **Different upper beams geometry** - Possibility to individually specify the length and width of the top beams and to use that data for automatic determination of buckling lengths.
- **Possibility to disable the interaction curves checks** - A new option makes it possible to skip checking the interaction curves.
- **Minor improvements to ribbons** - Separation of icons for defining geometry and specifying section type.
- **Minor improvements in Info Panel** - Adding information in Info Panel on whether second-order effects have been included.
- **Improvement to transverse reinforcement on Edit dialog** - Quickly and precisely define and edit the distribution of transverse bars in the table with new definition methods.

RC Footing design module

- **Representation of the substitute footing in viewports** - A visual representation of the substitute footing when this analysis was activated.
- **Additional information on concrete cracking in Info Panel and report** - Greater control over calculations with additional concrete cracking information.
- **Auto correction of reinforcement when crack widths are too large** – A new option allows you to keep the cracking within the limit by automatically increasing the reinforcement.
- **Improved editing of the distribution of bars** - Changes in the Edit reinforcement dialog to facilitate the definition and modification of bar distributions on a pad.
- **Information in the report about the final reinforcement area** - Additional information in the report to better describe the final reinforcement area.
- **Improvements to seismic bearing capacity calculations for France** - Possibility to select the method for Nmax calculation: according to EN1998-5 or according to NF P94-261
- **A new method of determining soil pressure** - A new algorithm for determining active and passive earth pressure, following annex C of EN 1997-1.

RC Wall design module

- **Generation of drawing with schedules for bars and fabrics** – New drawing style added to allow automatic simultaneous generation of schedules for bars and fabrics.
- **Import and export forces using Excel file** - For Shear walls, it is possible to import and export resultant forces to an excel sheet.

• Masonry Wall design module

- **Loads representation in viewports** – To make it easier to view defined loads as well as imported internal forces, it is now possible to display loads and internal forces graphically.
- **Expanded report for RC6 calculations** - A new section has been added in the reports for verification of a wall subjected to in-plane bending according to the Romanian CR6 standard.
- **Possibility to choose the section for the stress diagram** – When displaying the results in the form of stress diagrams, it is now possible to select the position of the section (bottom/ middle/ top)
- **Saving the masonry databases data per model** – To facilitate the management of data from databases used by the Masonry Walls module, databases have been separated from the data of the current project. This allows convenient management of data used in each project.
- **Presentation of partial factors** - To make it possible to verify the determined partial factors for the current masonry section, they are now available in the Design Assumptions window.
- **Support depth representation in viewports** - Displaying the depth of the slab support on viewers to easily check the data.

RC Slab design module

- **New bar shapes** – Possibility to generate u-shaped and two bent bars on the edges of the slab.
- **Possibility to manage hooks for reinforcement zones** – Possibility for editing hooks separately for each reinforcement zone.
- **New options for defining reinforcement solution** - Possibility to define reinforcement coverage with reinforcement areas.
- **New methods for defining reinforcement zone graphically** - Faster modeling of reinforcement zones with additional graphic definition methods.
- **Set of improvements to drawings**
    - Possibility of drawing generation simultaneously or separately for each reinforcement direction
    - Improvements to bar annotations (better automatic generation to avoid collisions, the possibility to move individually the bar annotation, and the possibility to hide the text above the dimension line)
    - New drawing style for simultaneous generation of schedules for bars and fabrics
    - Better representation of openings
    - New option to hide grids in drawings.
- **Set of user improvements**
    - Allow the second layer to have a bigger diameter than the first one.
    - Displaying local axes for each slab
    - Possibility for disabling reinforcement layer by entering 0 coverage.
    - New option to generate one-way reinforcement zones.

Steel Connections design module

- **Welded truss tube connection** - New welded truss connection type with multiple diagonal configurations of square and rectangular tubes.
- **Improvement on the Info Panel for Shear plate joint** - A new column on the Info panel for the indication of the element in which the verification is made.
- **New option to ignore bending moment in the calculation for hinged beams** - More control over the type of forces used for vérifications.

3. New computing capabilities

A series of new features and improvements related to structural calculations and additional building analysis.

3.1. Carbon emission estimations

Calculation of CO2 emissions based on carbon factors entered for materials or elements, with results in graphical form and reports.

Following its engagement to build a sustainable future, Advance Design is now equipped with a new CO_2 emission calculator. The main purpose of this new tool is to facilitate the determination of embodied carbon values for structural elements. This new tool will help designers to better assess the environmental impact of the choices they make on structural elements and construction materials.

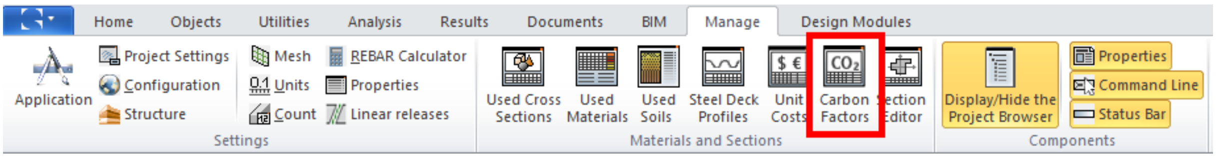
The fundamental principle of an **embodied carbon calculation** is typically to multiply the quantity of each material by a carbon factor (normally measured in $kgCO_2e$ per kg of material). As the embodied carbon associated with the production stage is the largest contributor to the embodied carbon of a structure, **production stage carbon factors** (carbon factors for the production stage - modules A1-A3), are used for calculations.

NOTE: *The mechanism for defining the data, the calculation, and the presentation of the results are identical to Cost estimations (more info about the Cost estimations you can find in the next paragraph).*

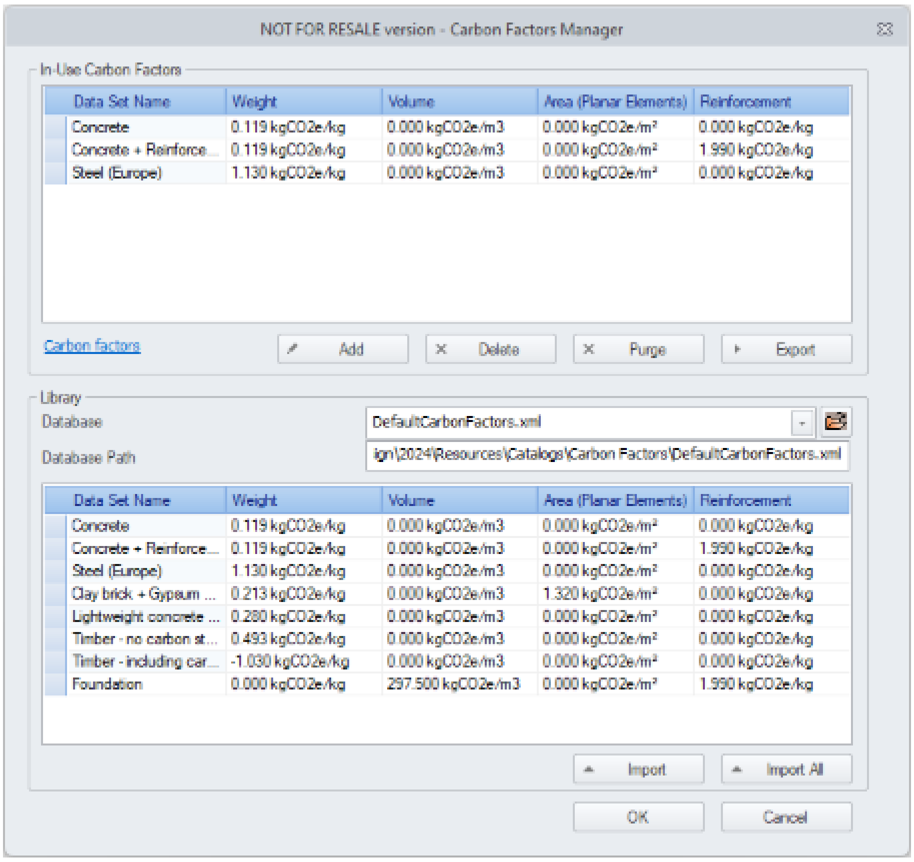
Defining carbon factors

The general procedure for defining data for CO_2 estimation is to prepare a list of data sets with specific carbon factors, and then assign them to individual materials or optionally to structural elements.

Preparing the list of data sets is done with the help of a dedicated Carbon Factor Manager. It can be opened from the *Materials and Sections* group in the *Manage* ribbon.



The Carbon Factor Manager window is divided into two parts. The upper part contains data sets added and used in the project, while the lower part shows the entries available in the selected database.



Carbon Factors Manger

In this top part, we can either import data from the database entry, or add a new data set using the *Add* button, and then we fill in the name of the data set, and CO_2 emission per unit of weight, volume, or area (for planar elements), and values corresponding to the weight of reinforcement.

Although in most cases we use carbon factors based on the weight of the material (such data is usually provided by manufacturers and other institutions), in special cases we can use factors based on volume or area. What is important, we can combine factors - for example, if for a wall or slab, we want to consider cladding elements, then in addition to the value according to weight we can add in the same position a value depending on the surface area. For concrete elements, we can simultaneously enter the carbon factor for reinforcement. This allows us to obtain a CO_2 value including the reinforcement, based on the quantity of reinforcement calculated during the concrete design analysis for that element.

In the bottom part of this manager, we can see data sets available on a selected library. Each user can independently prepare and manage the data content in the library.

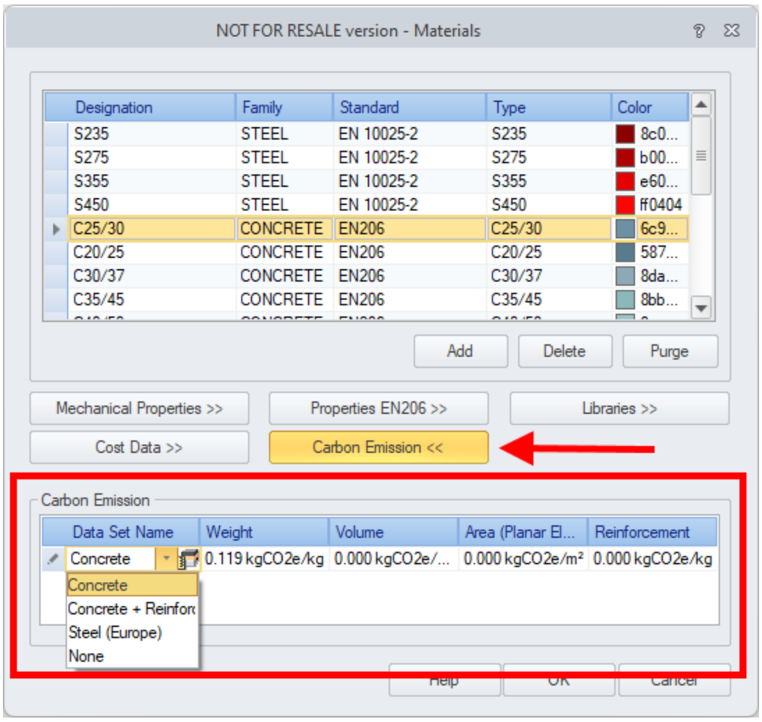
NOTE: *Advance Design provides a sample library with a list of examples of averaged values of the production stage carbon factors for selected materials. It is important to remember that the factors for many materials strongly depend on additional information, such as place and technology of production, use of recycling, type of ingredients (e.g. cement), etc. Therefore, for accurate estimation, adequate data provided by manufacturers should be used. Also note that for some localizations, the country's local regulations may dictate the use of other official data.*

Assigning carbon factors

There are two methods of assigning carbon factors: by assigning to materials or by assigning to elements (linear/planar/support/load area).

When Carbon factors are assigned to **Materials** (for example concrete C25/30) then such values are used for all objects from this material.

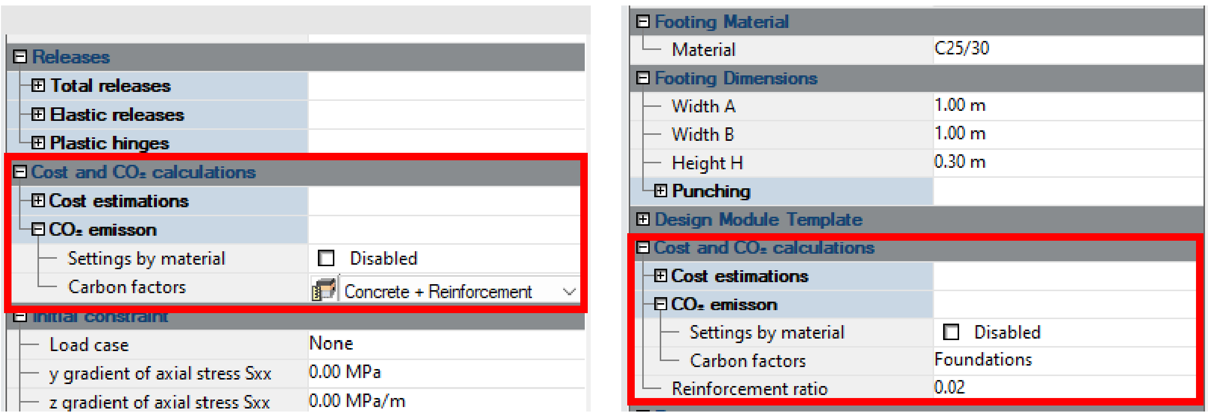
For this, the materials manager now includes a new Carbon Emission section where we can assign a specific CO_2 data set for each material by selecting from a drop-down list of available data sets.



Assigning a CO_2 emission data set to material

By default, each structural element inherits its material CO_2 emission. However, we can override this and impose another data set for selected elements. Assigning data sets with carbon factors to selected elements is done using new dedicated options available on the list of element properties.

Interestingly, it is also possible to determine carbon factors for foundations, although they are not separately defined or presented in the model. For this purpose, the parameters available in the properties of supports are used, including foundation dimensions and material. Note that in case of foundations, the reinforcement ratio that could be used for estimations needs to be set manually, while for other elements (beams, columns, slabs) it is calculated automatically during concrete design calculation analysis.

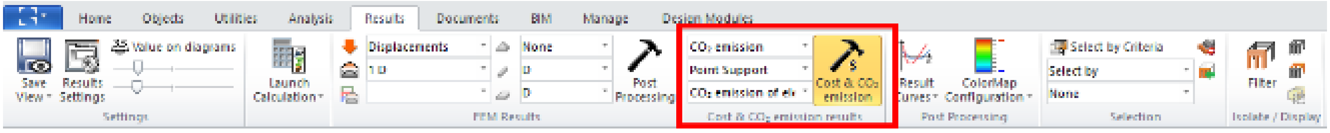


Overwriting default carbon factors of a linear element (left) and a support (right)

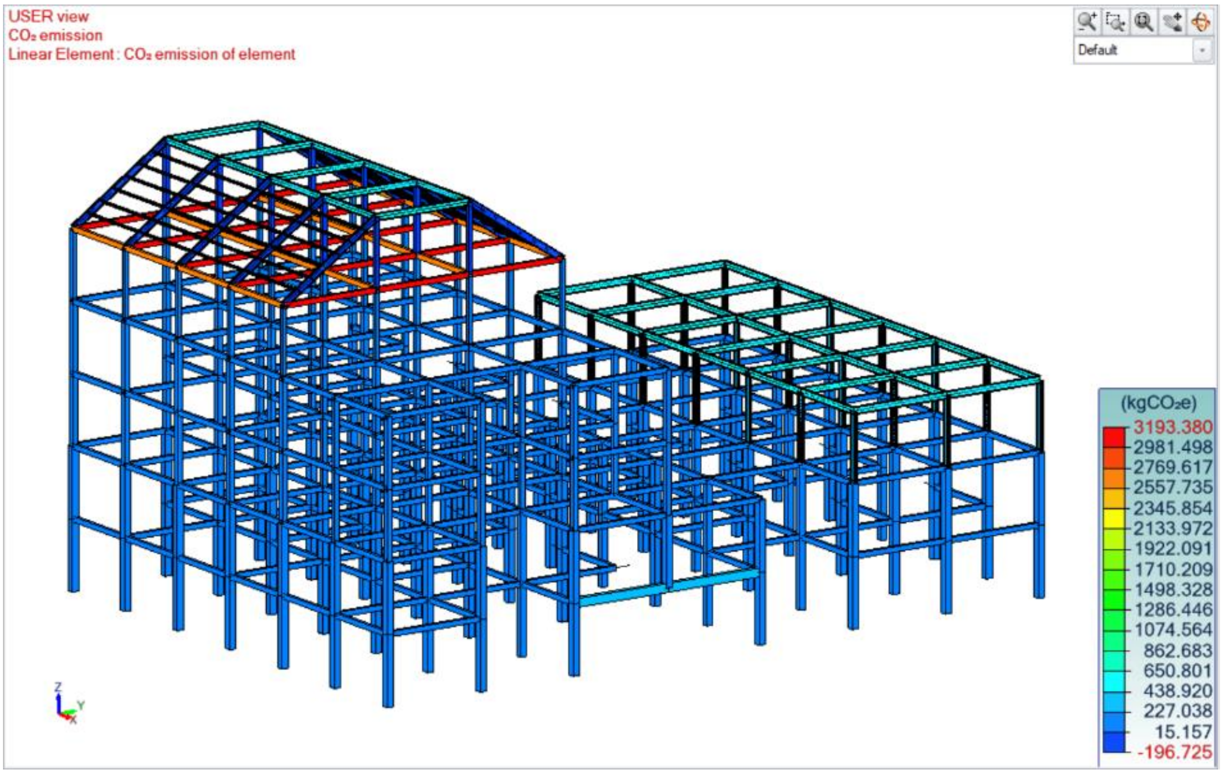
Results

The results of the calculations are available in graphical form and in the form of report tables.

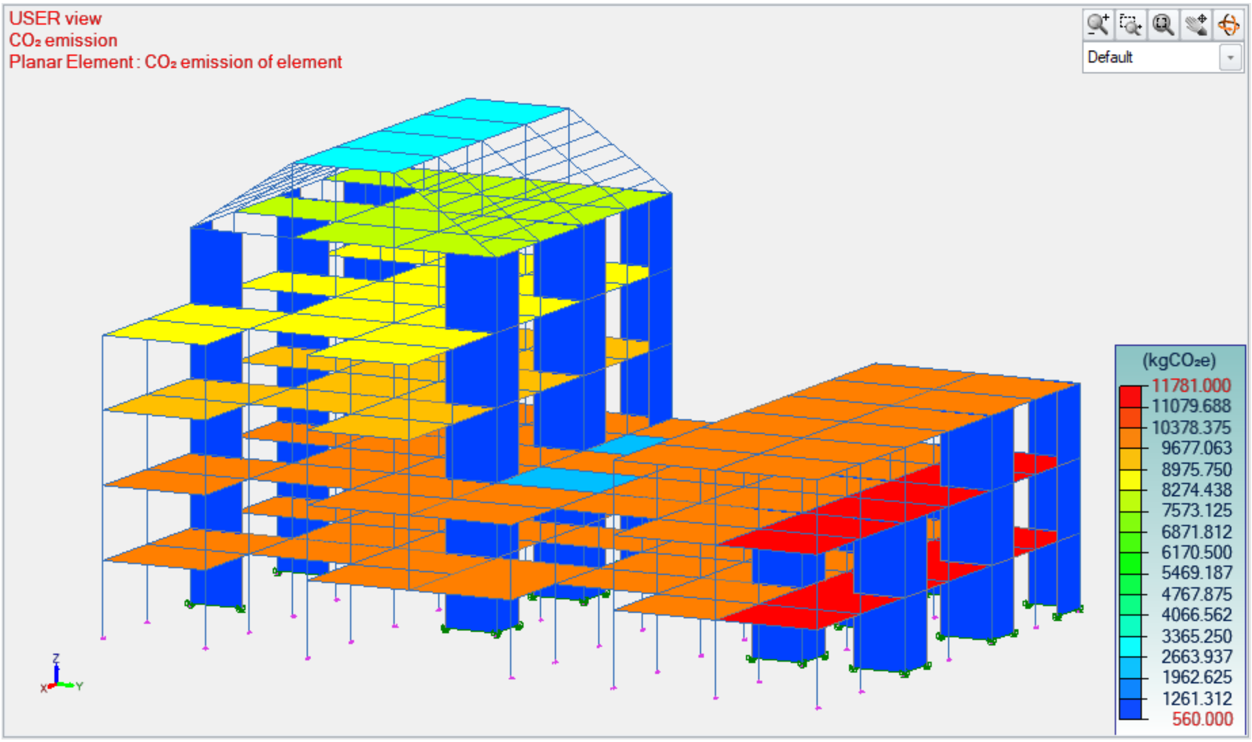
For the graphical results, on the *Results* ribbon, a new section for Cost and Carbon emission is used.



We can display the CO_2 emission results of point and linear supports, linear and planar elements, and load areas. The results values can be displayed per element, per unit of length (for linear elements), per unit of area (for planar elements), per unit of volume, and unit of weight.



CO_2 emission of linear elements displayed per element



CO_2 emission of planar elements displayed per element

For tables and reports, Advance Design can generate CO_2 emission tables. The tabulated results can be presented by material, by element type, and by system.

CO₂ emission estimation for materials						
Material	Emission/weight (kgCO₂e/kg)	Emission/volume (kgCO₂e/m³)	Emission/area (kgCO₂e/m²)	Volume (m³)	Weight (kg)	CO₂ emission (kgCO₂e)
C25/30	0.112	0.000	0.000	283.00	707500	79240.000
C25/30 (Foundations)	0.000	297.500	0.000	16.20	40500	9784.476
C30/37	0.119	0.000	0.000	904.20	2260500	268999.500
D27	-1.290	0.000	0.000	3.79	2313	-2984.373
S235	1.500	0.000	0.000	2.77	21747	32620.141
S355	1.130	0.000	0.000	10.20	80070	90479.100
			Total	1220.16	3112630	478138.844

Report table of CO_2 emission estimations by material

CO₂ emission estimation by element type			
Type	Volume (m³)	Weight (kg)	CO₂ emission (kgCO₂e)
Linear elements	191.96	542130	169170.868
Planar elements	992.20	2480500	293639.500
Foundations	36.00	90000	15328.476
Total	1220.16	3112630	478138.844

Report table of CO_2 emission estimations by element type

CO_2 emission estimation by system		
System No.	System name	CO_2 emission ($kgCO_2e$)
5	L1 - Columns	22107.539
6	L1 - Beams	9172.800
7	L1 - Slabs	168682.500
8	L2 - Columns	11424.000
9	L2 - Beams	30610.187
11	L3 - Columns	1400.000
12	L3 - Steel part	90479.100
13	L3 - Slabs	16065.000
14	L4 - Timber beams	-1967.250
15	L4 - Steel part	6961.615
16	L4 - Slabs	52836.000
17	Supports	15328.476
18	RC Walls	56056.000
19	L4 - Roof	-1017.123
20	Windwalls	0.000
	Total	478138.844

Report table of CO_2 emission estimations by system

3.2. Cost estimations

Calculation of costs based on unit prices for materials or elements, with results in graphical form and reports.

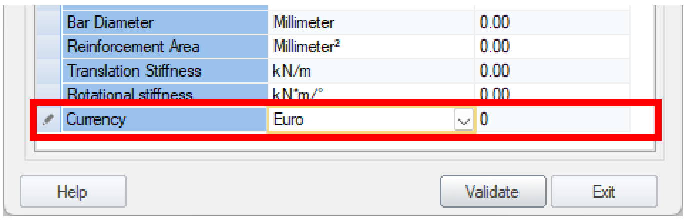
Advance Design is now equipped with a new cost calculator. This new tool will help designers to better assess the economic impact of the choices they make on structural elements and construction materials.

The principle of **cost estimations** is simple – we need to multiply the quantity of each material by a unit price. To do this, we need to define specific unit costs, and then assign them to individual materials or optionally to structural elements. Results of calculations in the form of report tables are available both in the descriptive and analytical models, while results in the form of graphs are available in the analytical model. Please note that the values from the reinforcement of concrete elements are only available if we have performed a reinforcement analysis on these elements.

NOTE: *The mechanism for defining the data, the calculation method, and the presentation of the results are identical to Carbon emission estimations (more info about the CO_2 estimations you can find in the previous paragraph).*

Possibility to set a currency unit

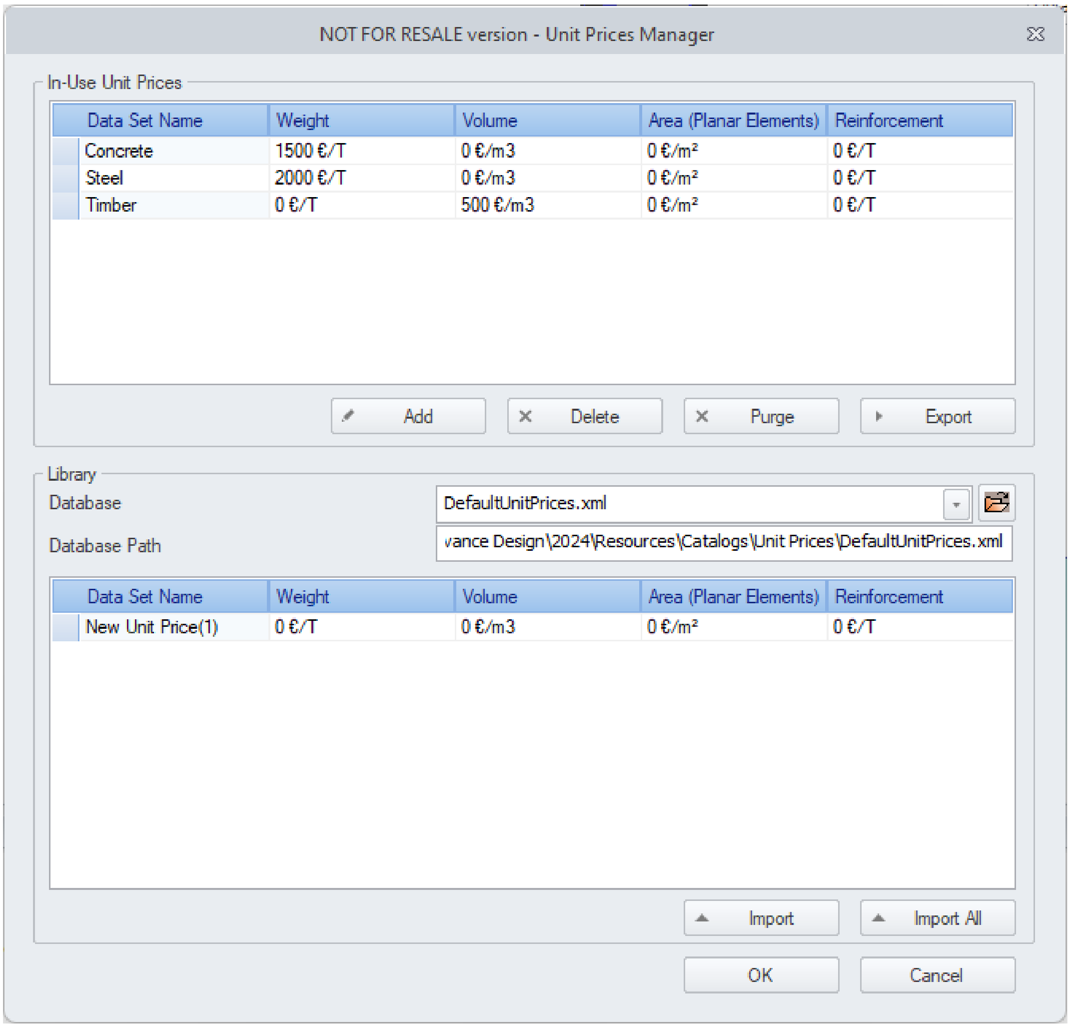
To be able to correctly set the currency used for cost estimations, a new unit-type, Currency is now available in the Unit Settings window.



Defining unit prices

Preparing the list of data sets with unit prices is done with the help of a dedicated Unit Prices Manager. It can be opened from the Materials *and Sections* group in the *Manage* ribbon.

The Unit Prices Manager window is divided into two parts. The upper part contains data sets added and used in the project, while the lower part shows the entries available in the selected database.



Unit Prices Manger

In this top part, we can either import data from the database entry or add a new data set using the *Add* button. When we add a new data set, we need to fill in the name, and the price per unit of weight, volume, or area (for planar elements), and optionally a value corresponding to the weight of reinforcement.

We can combine unit prices - for example, if for a wall we want to consider the cost of cladding elements, then in addition to the price per weight we can add in the same set a value depending on the surface area. For concrete elements, we can simultaneously enter the unit price for reinforcement, which allows us to obtain the cost of the reinforcement, based on the quantity of reinforcement calculated during the concrete design analysis.

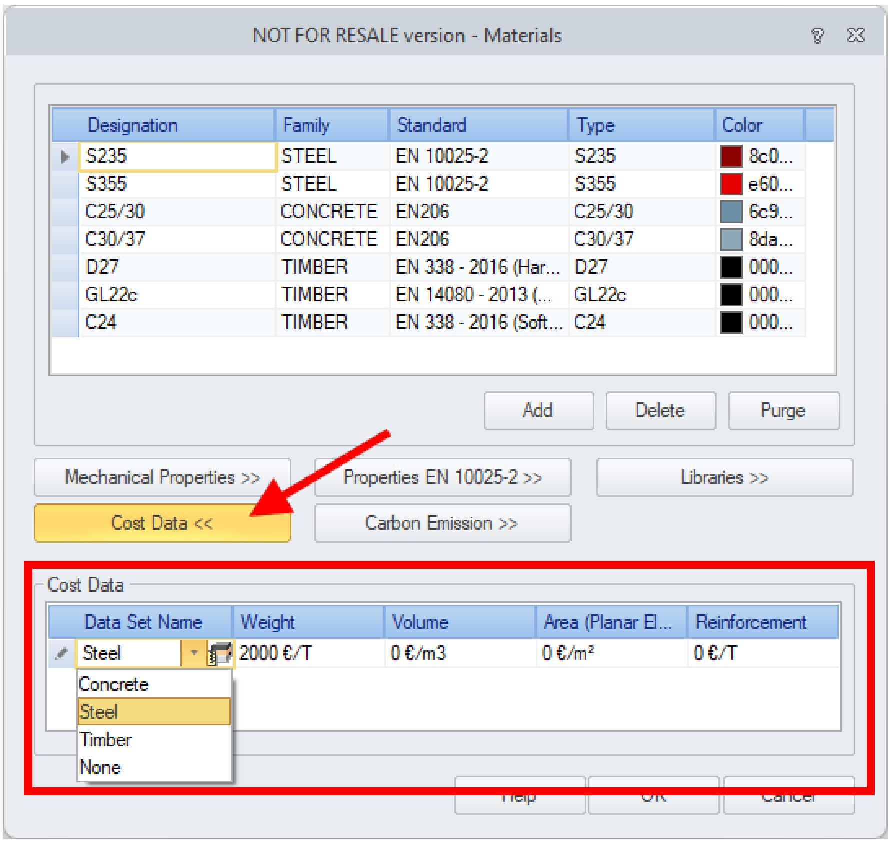
In the bottom part of this manager, we can see data sets available in a selected library. Each user can prepare and manage the data content in the library. Note that by default, the library with unit costs does not contain any values.

Assigning unit prices

There are two methods of assigning unit prices: by assigning to materials or by assigning to elements (linear/planar/support/load area).

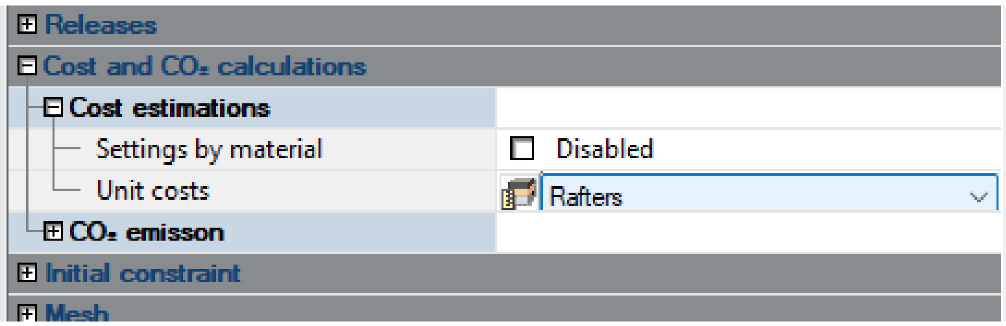
When unit prices are assigned to **Materials** then such values are used for all objects from this material.

For this, the materials manager now includes a new Cost Data section where we can assign a specific data set for each material by selecting from a drop-down list of available data sets.



Assigning a cost data set to a selected material

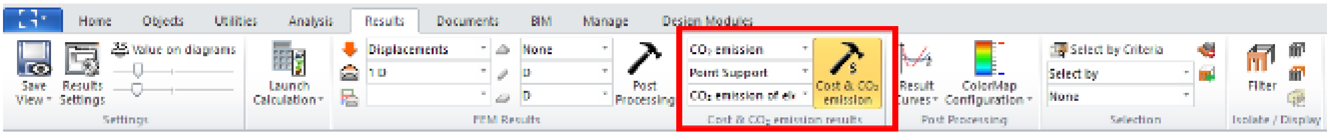
By default, each structural element inherits its material unit price. However, we can override this and impose other data sets for selected elements. Assigning data sets of unit prices to selected elements is done using new dedicated options available on the list of element properties. As it is available for the Carbon estimations, it is also possible to determine the cost for foundations, using the parameters available in the properties of support.



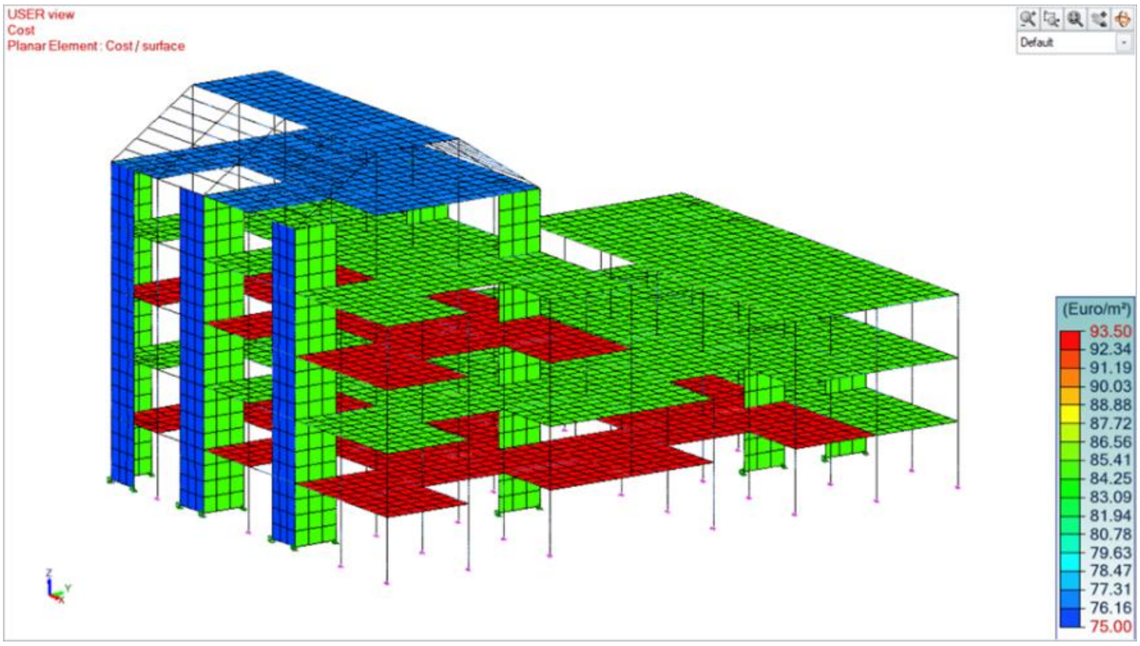
Overwriting default unit prices on the property list of a linear element

Results

The results of the calculations are available in graphical form and in the form of report tables. For the graphical results, on the *Results* ribbon, a new section for Cost and Carbon emission is used.



Similarly, as for carbon estimations, we can display the cost estimation results on point and linear supports, linear and planar elements, and load areas. The results values can be displayed per element, per unit of length (for linear elements), per unit of area (for planar elements), per unit of volume, and unit of weight.



Cost of planar elements displayed per surface unit

For tables and reports, Advance Design can generate cost estimation tables. The tabulated results can be presented by material, by element type, and by system.

Cost estimation for materials						
Material	Cost/weight (€/kg)	Cost/volume (€/m³)	Cost/area (€/m²)	Volume (m³)	Weight (kg)	Cost (€)
C25/30	0.15	0.00	0.00	299.20	748000	114621.90
C30/37	0.17	0.00	0.00	904.20	2260500	384285.00
D27	0.24	0.00	0.00	3.79	2313	555.23
S235	1.30	0.00	0.00	2.77	21747	28270.79
S355	1.45	0.00	0.00	10.20	80070	116101.50
			Total	1220.16	3112630	643834.42

Report table of cost estimations by material

Cost estimation by element type			
Type	Volume (m³)	Weight (kg)	Cost (€)
Linear elements	191.96	542130	210627.52
Planar elements	992.20	2480500	417285.00
Foundations	36.00	90000	15921.90
Total	1220.16	3112630	643834.42

Report table of cost estimations by element type

Cost estimation by system		
System No.	System name	Cost (€)
5	L1 - Columns	25660.08
6	L1 - Beams	12285.00
7	L1 - Slabs	240975.00
8	L2 - Columns	15300.00
9	L2 - Beams	32817.31
11	L3 - Columns	1875.00
12	L3 - Steel part	116101.50
13	L3 - Slabs	22950.00
14	L4 - Timber beams	366.00
15	L4 - Steel part	6033.40
16	L4 - Slabs	75480.00
17	Supports	15921.90
18	RC Walls	77880.00
19	L4 - Roof	189.23
20	Windwalls	0.00
	Total	643834.42

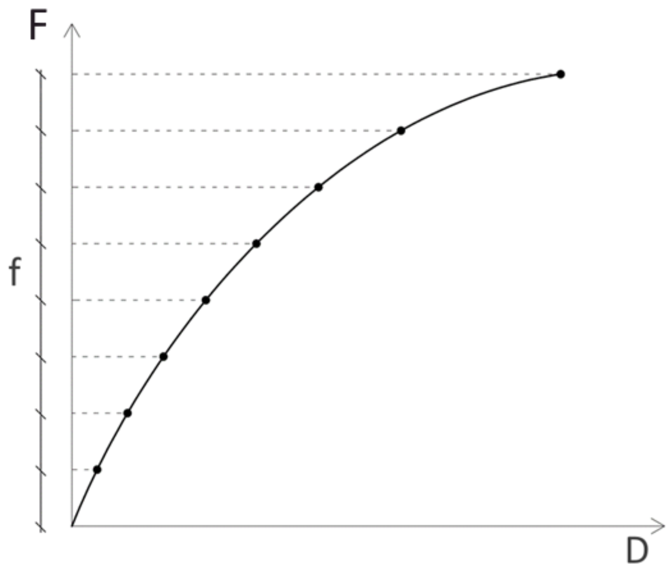
Report table of cost estimations by system

3.3. Displacement control nonlinear analysis

The ability to conduct nonlinear calculations using displacement increment steps, which allows to correctly analyze highly nonlinear problems with the post-peak behaviors and can easily manage the snap-through problems.

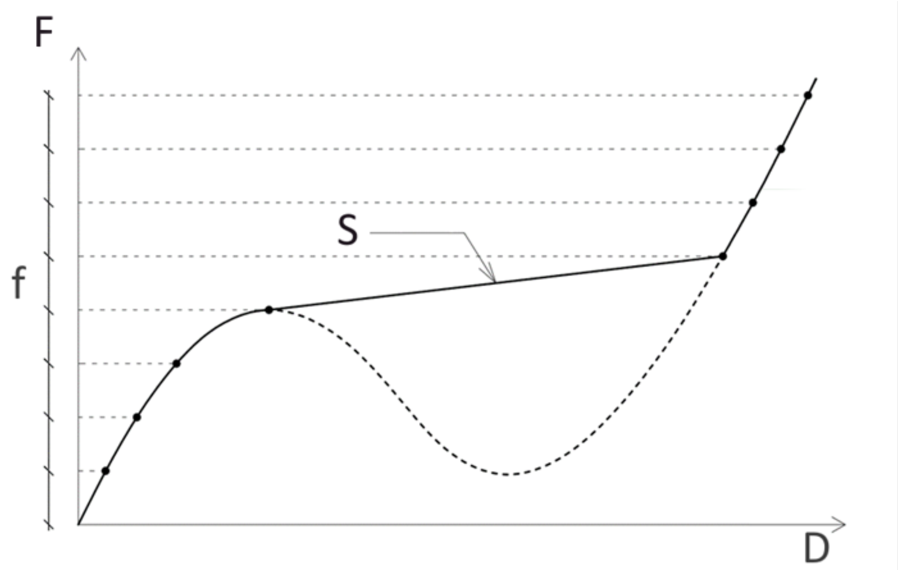
Technical background

Until present, Advance Design conducted nonlinear analysis by using a force control Newton Raphson approach. This numerical method is the classical and most used technique in nonlinear analysis. It consists of dividing the loads applied on the structure into equal force increments. Then, at each step of the analysis, one additional force increment is added, and the structure is analyzed nonlinearly. Hence the name force control.



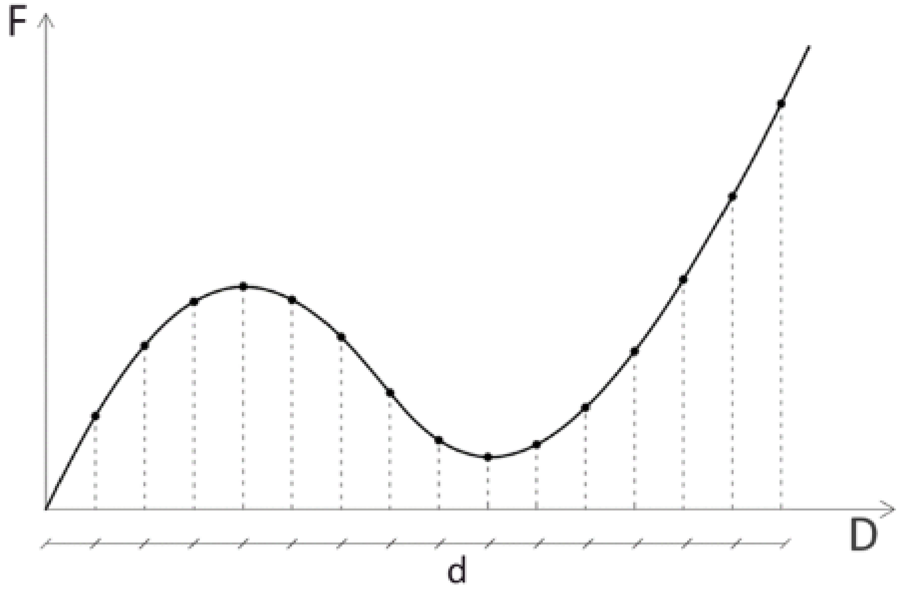
Example of force control nonlinear analysis
(F – Load, D – Displacement, f – Equal force increments)

The force control analysis has limitations. Since the applied loads on the structure are equally increased with every step, this technique cannot be used for structures that have a load drop (due to buckling, strength loss, or support failure …). The figure below presents the nonlinear force Vs displacement curve of a structure with load drop. Since it must always keep increasing the load, the force control solver exhibits a snap-through behavior and cannot capture the sagging part of the structural response (dashed line in the figure below).



Force control solver presenting a snap through (S) and not capturing the sagging part of the structural response

To face snap-through problems in nonlinear analysis, Advance Design is now equipped with a new displacement control solver. With this new approach, loads are applied on the structure and a target displacement is set for a chosen control node on the structure. This target displacement is divided into equal displacement increments. For each analysis step, the nonlinear solver will try to scale up/down all the loads applied on the structure to achieve the required displacement increment at the chosen control node (hence the name displacement control, refer to the next Figure). At each step, all the loads are scaled proportionally. In other words, the same scale factor is applied on all the loads. Thus, the initial load pattern distribution is always respected.



Displacement control solver capturing the entire structural response (F – Load, D – Displacement, d – Equal displacement increments)

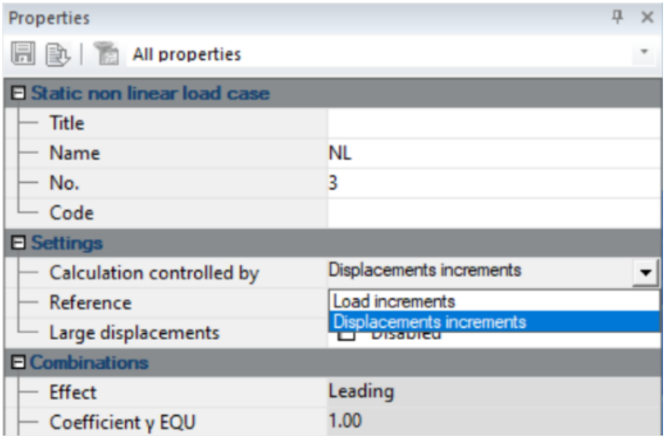
It is particularly important to note that displacement control analysis is different from imposed displacement loads. For an imposed displacement load on a node, a point load is automatically created on the concerned node so that it reaches the required imposed displacement. While for displacement control analysis, all the loads on the structure are proportionally scaled up/down to satisfy the target displacement of the control node (no automatic point load is created on the control node).

Now by controlling the nonlinear analysis with displacement increments, the loads on the structure can be increased or decreased without affecting the calculation procedure. Thus, structures with load drop can be completely studied using the displacement control nonlinear solver.

How to use displacement control analysis in Advance Design

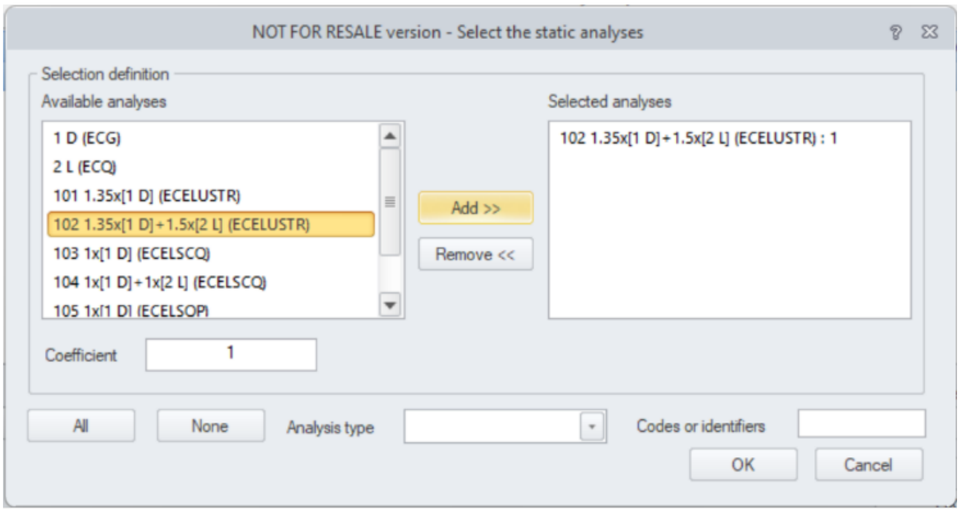
Setting up a displacement control nonlinear analysis in Advance Design is similar to creating the already existing force control Newton Raphson analysis. The user should conduct the following steps:

1. Place a point on the structure to serve as an indicator of the displacement control node position.
2. Create a nonlinear static analysis.
3. In the static nonlinear load case, the set calculation is controlled by displacement increments.



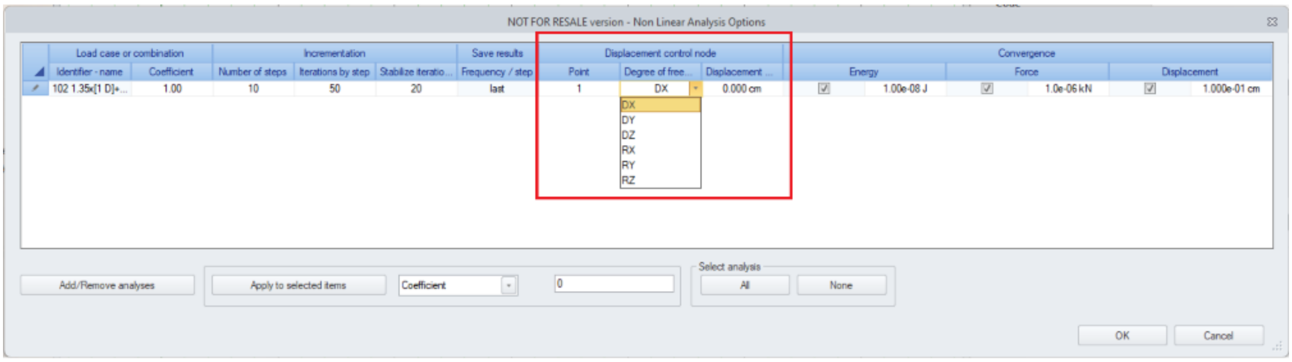
Selecting displacement control nonlinear analysis

4. In nonlinear analysis options, add the loads/combinations to be considered in the nonlinear.



Selecting Load cases/combinations for nonlinear analysis

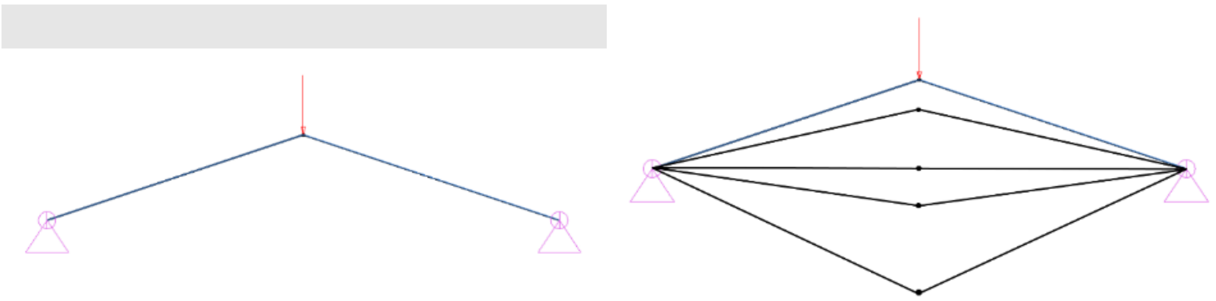
5. For each selected load case/combination set the parameters of the nonlinear analysis. The new analysis parameters for displacement control concern the displacement control node. The user indicates the control node number (ID number of point created in step 1), specifies on which global degree of freedom the controlled displacement/rotation is occurring, and then sets the target value of the corresponding displacement/rotation.



Nonlinear analysis parameters for displacement control

- **Application example 1: Von Mises truss snap-through buckling**

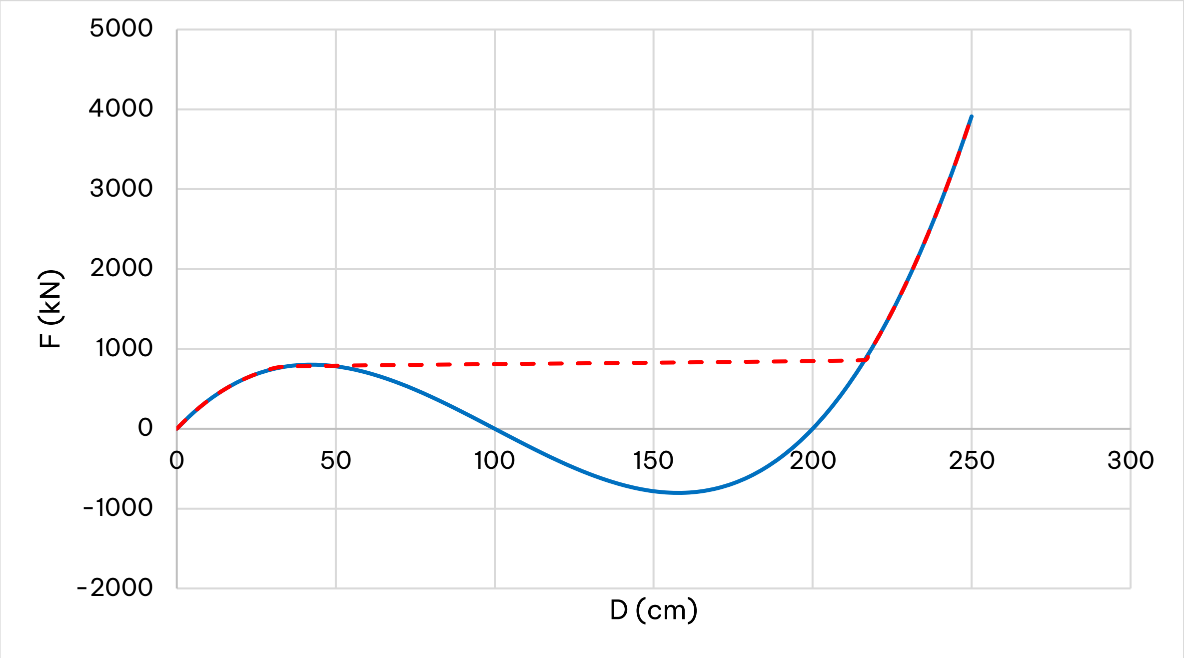
To start let us consider a textbook example of a simple structure made with 2 bars and a vertical load applied at their connecting node. This structure is called the Von Mises truss.



Von Mises truss model and its progressive deformation

The structure will deform progressively and a snap through buckling will occur as indicated in the figure above. An analytical solution (neglecting dynamic effects) exists for the Von Mises truss and can be used as a benchmark for Advance Design results.

For this purpose, the example was recalculated in Advance Design using both available methods of nonlinear calculations, and the following graph shows the resulting force Vs node displacement curve. The **red** dashed line presents the curve from the nonlinear **force control** analysis with large displacement. The **blue** line presents the curve from nonlinear **displacement control** analysis with large displacement, and it is identical to the curve from the analytical solution.

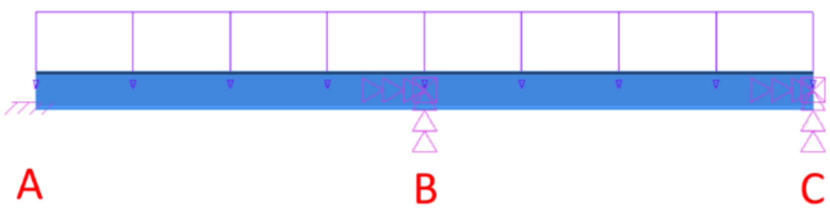


Force control analysis curve (dashed red) and Displacement control analysis curve (blue)

It is clear that the force control solver had a snap-through problem and could not calculate the sagging part of the force-displacement curve, while the displacement control analysis was capable of managing the snap-through buckling and captured accurately the entire structural response.

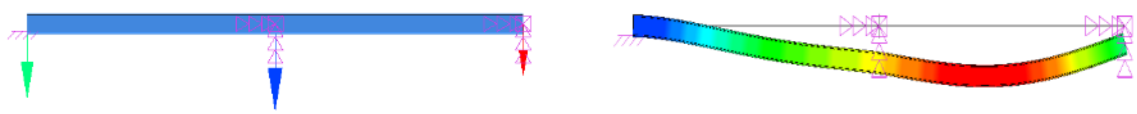
Application example 2: Support failure

Let us consider a uniformly loaded beam with fixed support (point A) and 2 vertical nonlinear spring supports (at points B and C). Each spring behaves linearly until a certain bearing capacity limit. Once this limit is reached, the spring will fail and drop all its support reactions.

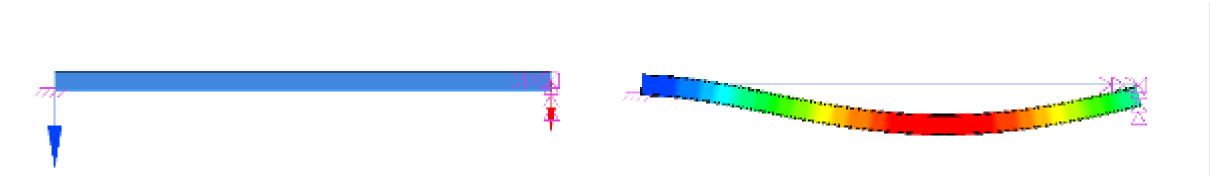


Beam on nonlinear springs (points B and C)

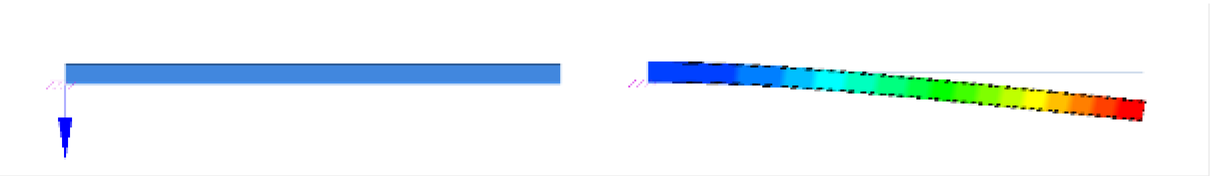
Using Advance Design, we want to conduct a step-by-step nonlinear analysis to study the progressive failure of springs and its effect on the structure. Both springs have the same linear stiffness. However, Spring at point C has double the bearing capacity. Thus, it is expected that spring at point B will fail first then spring at point C will follow later. That is, during successive iterations of nonlinear calculations, we expect to get 3 different systems: the first in which all supports work, the second in which the middle support has stopped working, and finally the third in which the right elastic support has also stopped working.



System 1 - All supports are active



System 2 - Failure of the middle support

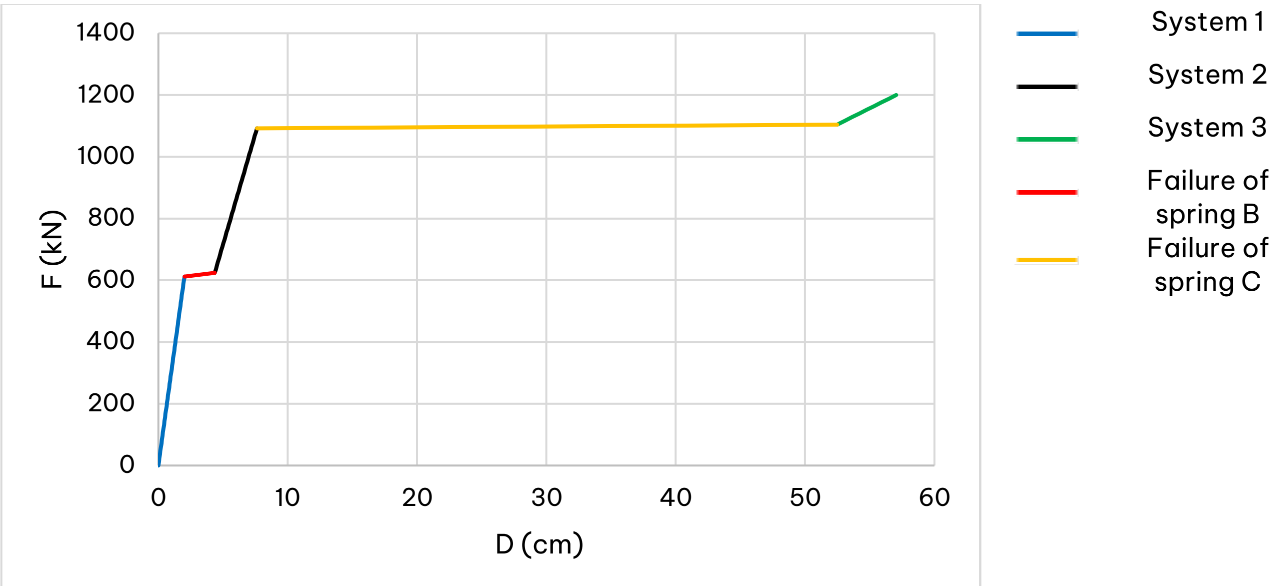


System 3 - Failure of the right support

The dynamic effect of spring support failure is neglected, and the midpoint of the beam (location of spring B) is considered for measuring vertical displacement.

The example was recalculated in Advance Design using both available methods of nonlinear calculations.

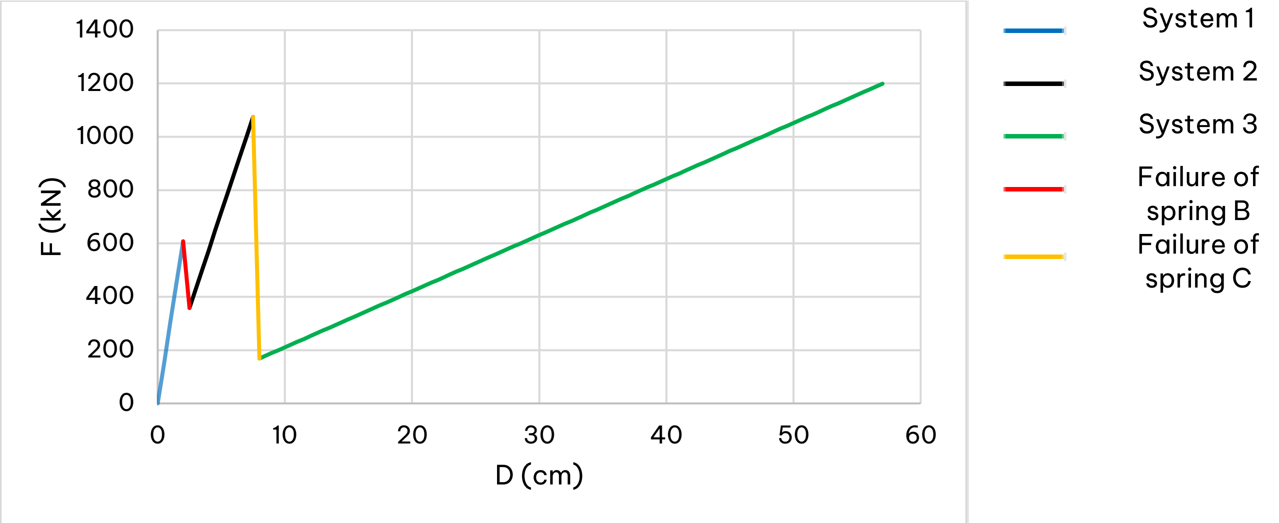
First, a nonlinear **force control** analysis is conducted in Advance Design. The figure below presents the curve of total force applied on the structure Vs measured displacement (the vertical deflection at the middle of the beam).



Force Vs displacement curve resulting from force control nonlinear analysis

It is remarkable that for the failure of springs B and C, the force control solver could not drop the load applied on the structure but instead, a snap-through behavior is observed.

Next, a nonlinear displacement control analysis is conducted in Advance Design. The figure below presents the curve of total force applied on the structure Vs measured displacement.



Force Vs displacement curve resulting from displacement control nonlinear analysis

During the failure of springs B and C, the displacement control solver was capable of dropping the load applied to the structure and avoiding snap-through problems.

Of course, this is just a specific example, but it shows that some projects can give different results in certain ranges, depending on the chosen method of nonlinear analysis. Thanks to the new solver, Advance Design can get correct analysis results for highly nonlinear problems with post-peak behaviors and can easily manage the snap-through problems.

3.4. Possibility for defining initial constraints on linear elements per load case

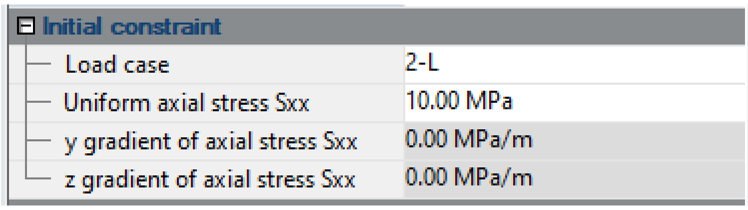
A new way of defining and considering initial constraints, making it easier to include initial constraints in combinations, as well as considering their impact on the rest of the structure.

In previous versions of Advance Design, initial constraints for line elements served to locally increase the element's internal forces/stresses for design calculation purposes. Since the main objective was to serve the design calculation of the element, this increase of internal forces/stresses affected only the element carrying them, and no interactions with other structural components were considered. In addition, initial constraints were defined as a global property of a given linear element, which caused problems if they were not to be included in all combinations.

With a new version, the influence of initial constraints for linear elements is studied on the entire structural system and it is considered during finite element analysis. For this, a linear element with initial constraints is trying to deform in such a way as to relieve this initial stress. Three scenarios are possible:

- The element is blocked and cannot deform - In this case, initial constraints cannot be relieved and are exhibited as internal stresses in the element.
- The element is connected to other structural components - In this case, the element will deform in a way to partially relieve the initial constraints. However, by doing so, the other structural components are getting stressed. An energy equilibrium is established between initial constraints relieve of the line element and stress increase in other structural components.
- The element is free to deform - In this case, the element will freely deform in a way to relieve the initial constraints.

Starting with Advance Design 2024, initial constraints of linear elements are linked to load cases. Previously, they were defined as the properties of a linear element, but now we have the option of choosing which load case it belongs to.



Initial constraint	
Load case	2-L
Uniform axial stress Sxx	10.00 MPa
y gradient of axial stress Sxx	0.00 MPa/m
z gradient of axial stress Sxx	0.00 MPa/m

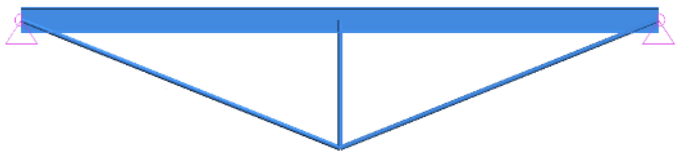
Defining initial constraints

Three types of initial constraints are available:

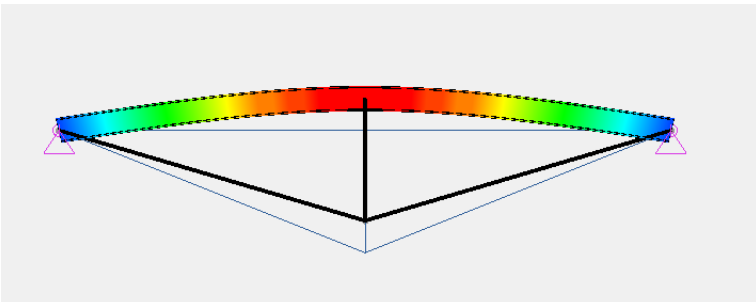
- Uniform axial stress Sxx - Initial Axial stress distributed uniformly all over the section.
- y gradient of axial stress Sxx - Initial axial stress variation per unit length of the section in the local y direction (this is similar to axial stress gradient generated by bending moment around local y direction).
- z gradient of axial stress Sxx - Initial axial stress variation per unit length of the section in the local z direction (this is similar to axial stress gradient generated by bending moment around local z direction).

Let us look at the following example that will help you understand how initial constraints work.

Let us consider a simple beam truss assembly with pre-tensioned bars via turn buckles – it is a single-span horizontal beam with two pre-tensioned bars and a vertical bar.



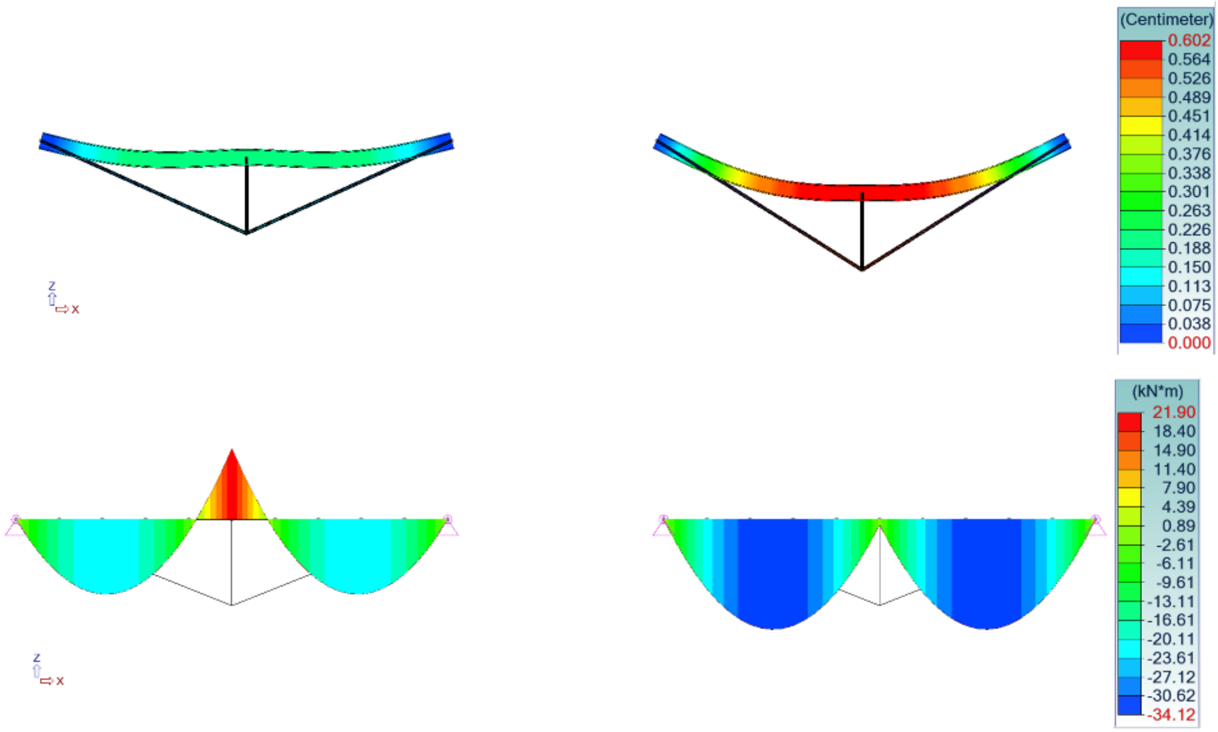
As a result, part of the stresses from these tendons were distributed through the vertical element to the entire system, causing the beam to bend upward. Using other words - under the effect of initial constraints (pretension), the beam will have an initial upward deflection.



Beam deflection due to pretension

When gravity loads are applied to the beam, this beam will present a lower deflection and smaller bending moment compared to a system without pre-tensioned bars.

In the comparison below you can see the deflections and bending moments in the beam for two cases - with pre-tensioned bars (left) and without pre-tensioned bars (right).



Deflections (top) and bending moments (bottom) for example with pre-tensioned bars (left) and without pre-tensioned bars (right)

3.5. A better formulation of drilling rotation in shell elements

Improved definition of shell finite elements, thanks to shell elements are now capable of capturing torsional moments transmitted from perpendicular beam elements.

The drilling rotation of a shell node is the rotation degree of freedom around the local axis perpendicular to the shell finite element.

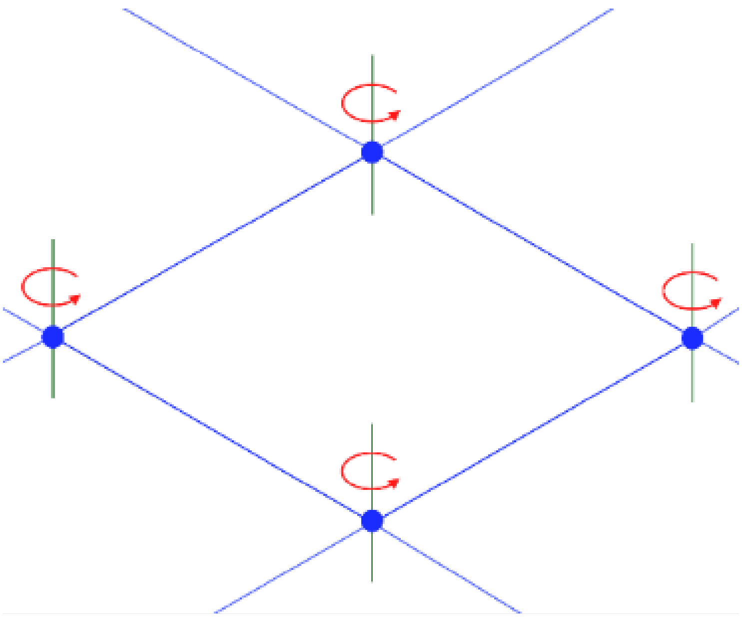
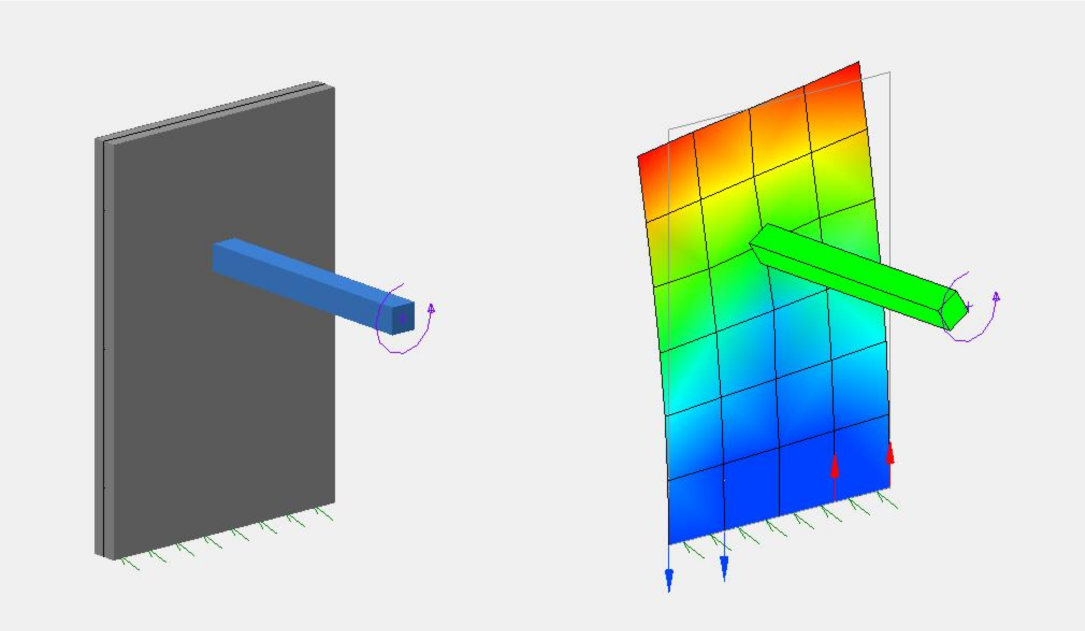


Illustration of drilling rotation in a Q4 shell element

Advance Design 2024 uses a new and better formulation for calculating this drilling rotation. This change slightly improves the accuracy of calculations and has a positive effect on the convergence of nonlinear calculations; however, the biggest benefit is that shell elements are now capable of capturing torsional moments transmitted from perpendicular beam elements.



Example of beam transmitting moment to shell element in Advance Design 2024

4. Enhance steel structure design capabilities

A series of novelties and improvements related to the verification and optimization of steel element structures.

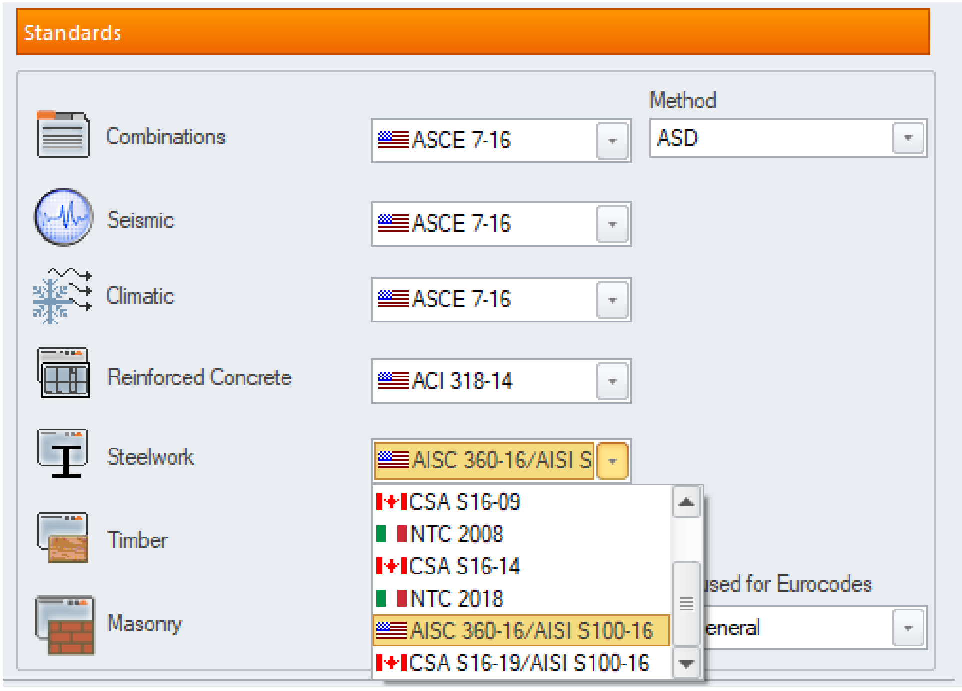
4.1. Verification of steel cold-formed sections according to AISI S100

The possibility of performing the standard verification of cold-formed steel profiles according to the AISI S100-16 American standard.

With Advance Design 2024, the possibility of standard verification for cold-formed profiles for North America has been introduced. The calculations are based on AISI S100-16 Specification (American Iron and Steel Institute - North American Specification for the Design of Cold-Formed Steel Structural members, 2016 Edition) and with the use of AISI Manual -- Cold-Formed Steel Design, 2017 Edition.

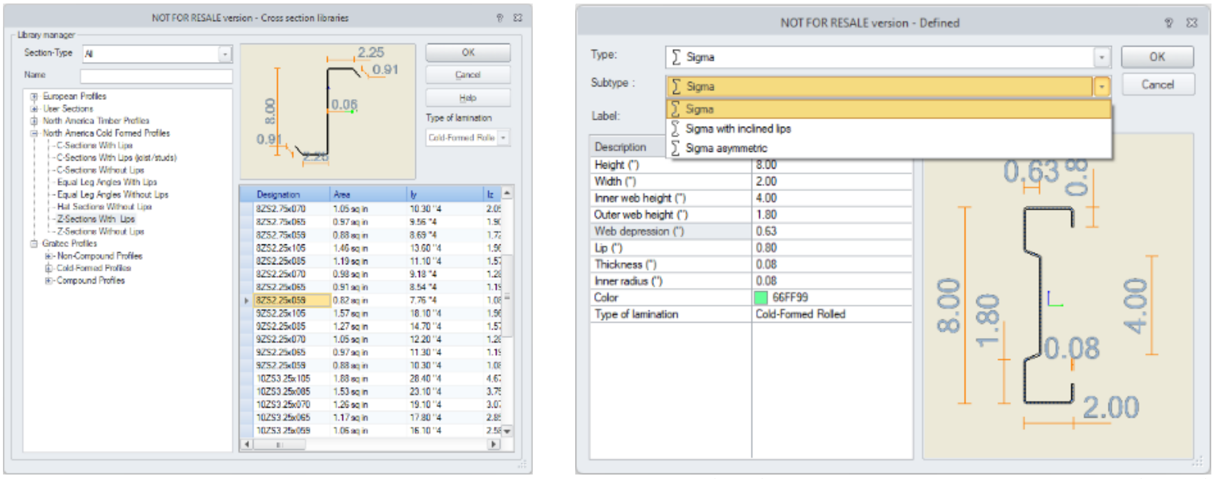
The verification process involves evaluating the structural capacity of cold-formed steel sections and ensuring they meet the necessary strength and stiffness requirements and includes bending, compression, and tension verifications.

Verifications according to these regulations are conducted if AISC 360-16/AISI S100-16 or CSA S16-19/AISI S100-16 is selected as the standard for steel verification in the project configuration:



Choosing the standard for the project

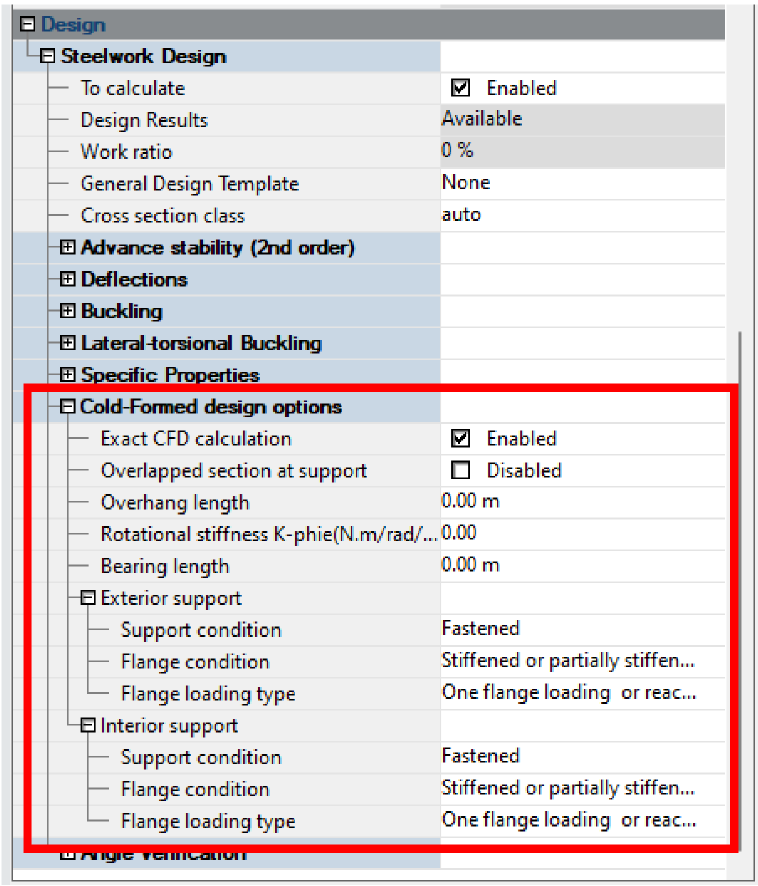
Cold-formed profiles can be analyzed, both those that are defined in the program's profile libraries (those that have the lamination type set as Cold - Formed) and parametric cold-formed profiles.



Example of a cold-formed profile from the database (left) and defined as parametric (right)

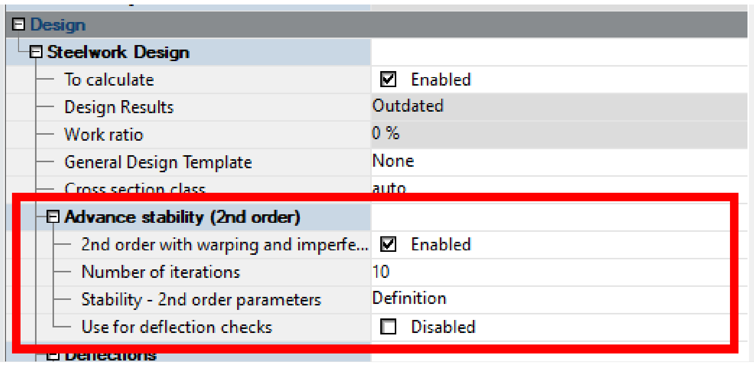
To facilitate the definition of cold-formed profiles for the North American market, new dedicated steel grades (ASTM Cold Formed library) have been added, as well as an additional new cold-formed steel profile library (acc. AISI Cold-Formed Steel Design Manual). For more information, see the separate dedicated paragraph in this document.

To be able to properly verify cold-formed profiles according to the AISI standard, the list of steel design properties has been expanded to include a new set of Cold-Formed design options.

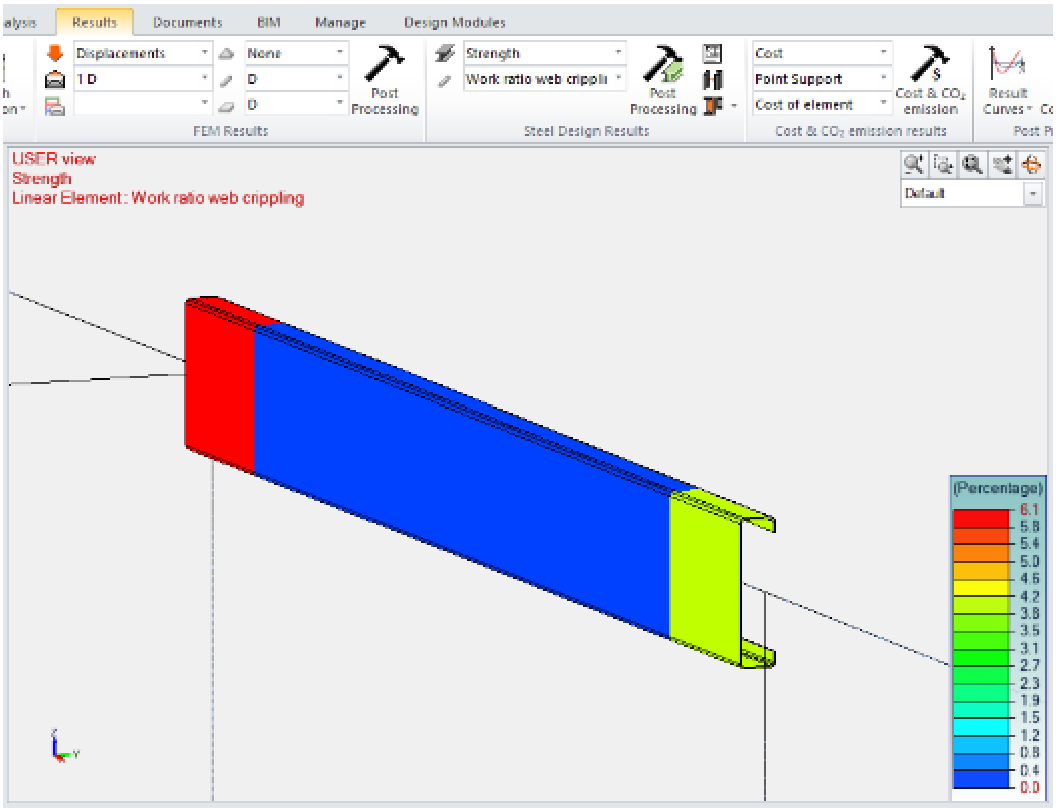


While most of the parameters are directly related to the provisions of the standard, the first of the options (*Exact CFD calculation*) is used to toggle whether the effective cross-section is computed for each load combination only on several points along the element (the option is off) or the analysis must be performed at multiple points between each finite element mesh node (the option is on). This has a direct impact on the calculation time.

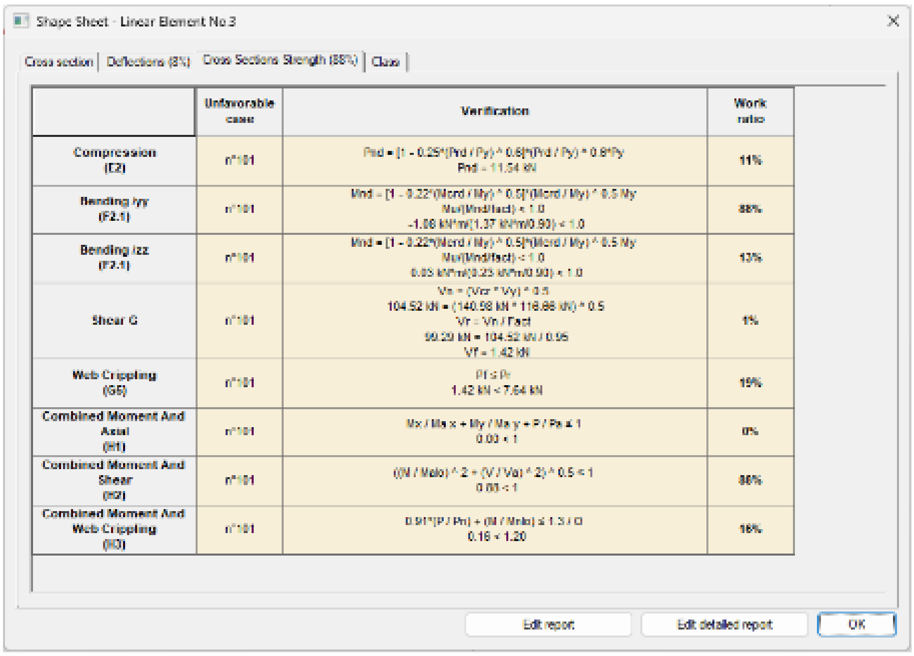
It should also be mentioned that with the latest version of Advance Design when analyzing steel profiles according to AISC 360-16/AISI S100-16, we can conduct verification considering 7 degrees of freedom (2nd order analysis including warping). For this purpose, the Advance Stability option is activated.



After the profile analysis, the calculation results are available for other types of steel profiles in graphical form and in the Shape Sheet window and related reports.

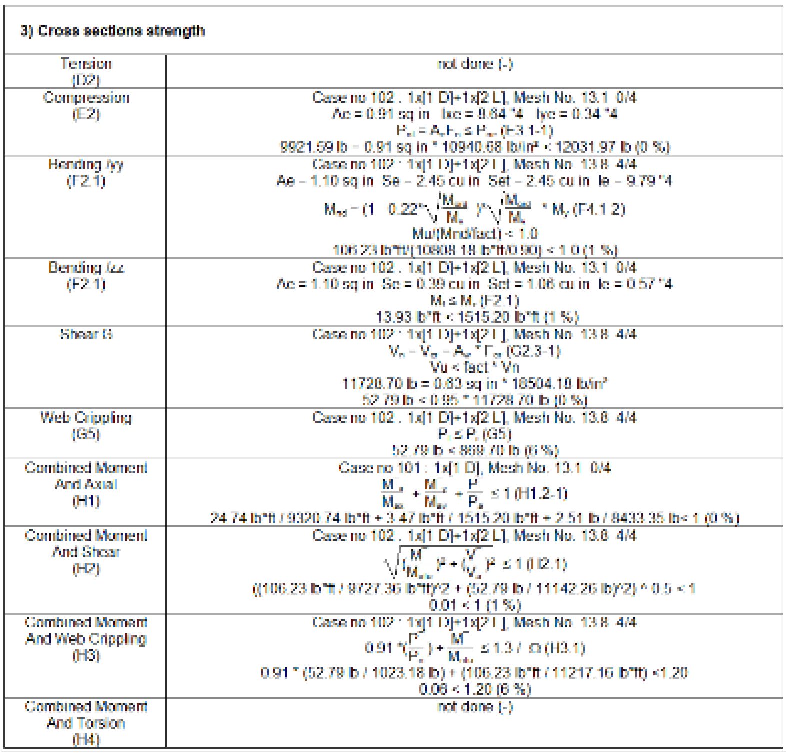


Results in graphical form for steel element - Work ratio for web crippling



	Unfavorable case	Verification	Work ratio
Compression (E2)	n°101	Pnd = [1 - 0.25*(Pnd / Py) ^ 0.6]*(Pnd / Py) * 0.6*Py Pnd = 11.54 kN	11%
Bending /yy (F2.1)	n°101	Mnd = [1 - 0.22*(Mcrd / My) ^ 0.5]*(Mcrd / My) ^ 0.5 My Mu/(Mnd/fact) < 1.0 -1.08 kN*m/(1.37 kN*m/0.90) < 1.0	88%
Bending /zz (F2.1)	n°101	Mnd = [1 - 0.22*(Mcrd / My) ^ 0.5]*(Mcrd / My) ^ 0.5 My Mu/(Mnd/fact) < 1.0 0.03 kN*m/(0.23 kN*m/0.90) < 1.0	13%
Shear G	n°101	Vn = (Vcr * Vy) ^ 0.5 104.52 kN = (140.98 kN * 116.88 kN) ^ 0.5 Vr = Vn / Fact 99.29 kN = 104.52 kN / 0.95 Vf = 1.42 kN	1%
Web Crippling (G5)	n°101	Pf ≤ Pr 1.42 kN < 7.64 kN	19%
Combined Moment And Axial (H1)	n°101	Mx / Ma x + My / Ma y + P / Pa ≤ 1 0.00 < 1	0%
Combined Moment And Shear (H2)	n°101	((M / Malo) ^ 2 + (V / Va) ^ 2) ^ 0.5 < 1 0.88 < 1	88%
Combined Moment And Web Crippling (H3)	n°101	0.91*(P / Pn) + (M / Mnlo) ≤ 1.3 / Ω 0.18 < 1.20	16%

Shape Sheet widow with a summary of performed verifications



3) Cross sections strength

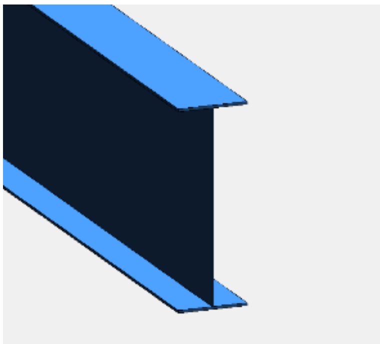
Tension (D2)	not done (-)
Compression (E2)	Case no 102 . 1x[1 D]+1x[2 L], Mesh No. 13.1 0/4 Ae = 0.91 sq in Ixe = 8.64 "4 Iye = 0.34 "4 $P_n = A_e F_n \le P_{no}$ (E3.1-1) 9921.59 lb = 0.91 sq in * 10940.68 lb/in² < 12031.97 lb (0 %)
Bending /yy (F2.1)	Case no 102 : 1x[1 D]+1x[2 L], Mesh No. 13.8 4/4 Ae = 1.10 sq in Se = 2.45 cu in Set = 2.45 cu in Ie = 9.79 "4 $M_{nd} = (1 - 0.22 \sqrt{\frac{M_{crd}}{M_y}}) \sqrt{\frac{M_{crd}}{M_y}} * M_y$ (F4.1.2) Mu/(Mnd/fact) < 1.0 106.23 lb*ft/(10808.18 lb*ft/0.90) < 1.0 (1 %)
Bending /zz (F2.1)	Case no 102 . 1x[1 D]+1x[2 L], Mesh No. 13.1 0/4 Ae = 1.10 sq in Se = 0.39 cu in Set = 1.06 cu in Ie = 0.57 "4 $M_r \le M_n$ (F2.1) 13.93 lb*ft < 1515.20 lb*ft (1 %)
Shear G	Case no 102 : 1x[1 D]+1x[2 L], Mesh No. 13.8 4/4 $V_n = V_{cr} = A_w * F_{cr}$ (G2.3-1) Vu < fact * Vn 11728.70 lb = 0.83 sq in * 18504.18 lb/in² 52.79 lb < 0.95 * 11728.70 lb (0 %)
Web Crippling (G5)	Case no 102 . 1x[1 D]+1x[2 L], Mesh No. 13.8 4/4 $P_r \le P_n$ (G5) 52.79 lb < 868.70 lb (6 %)
Combined Moment And Axial (H1)	Case no 101 : 1x[1 D], Mesh No. 13.1 0/4 $\frac{M_x}{M_{ax}} + \frac{M_y}{M_{ay}} + \frac{P}{P_a} \le 1$ (H1.2-1) 24.74 lb*ft / 9320.74 lb*ft + 3.47 lb*ft / 1515.20 lb*ft + 2.51 lb / 8433.35 lb< 1 (0 %)
Combined Moment And Shear (H2)	Case no 102 . 1x[1 D]+1x[2 L], Mesh No. 13.8 4/4 $\sqrt{(\frac{M}{M_{alo}})^2 + (\frac{V}{V_a})^2} \le 1$ (H2.1) ((106.23 lb*ft / 8727.36 lb*ft)^2 + (52.79 lb / 11142.26 lb)^2) ^ 0.5 < 1 0.01 < 1 (1 %)
Combined Moment And Web Crippling (H3)	Case no 102 : 1x[1 D]+1x[2 L], Mesh No. 13.8 4/4 $0.91 * (\frac{P}{P_n}) + \frac{M}{M_{nlo}} \le 1.3 / \Omega$ (H3.1) 0.91 * (52.79 lb / 1023.18 lb) + (106.23 lb*ft / 11217.16 lb*ft) <1.20 0.06 < 1.20 (6 %)
Combined Moment And Torsion (H4)	not done (-)

Shape Sheet report with details of performed verifications

4.2. Shear web buckling analysis for I sections (EN 1993-1-5)

Possibility for running a verification of the stability of I-section webs according to EN 1993-1-5.

Advance Design 2024 now checks webs for shear buckling according to EN 1993-1-5. This verification can be critical for deep I-sections, which can be used on many types of structures, including bridges.



Such beams usually carry heavy loads, and because bending is typically taken by flanges, the web is kept very thin, making it prone to plate buckling from the shear force, also known as shear buckling.

Shear buckling is mentioned in §6.2.6(6) from EN 1993-1-1, as a verification to perform when the width-to-thickness ratio of the web plate (h_w/t_w) exceeds a limit value.

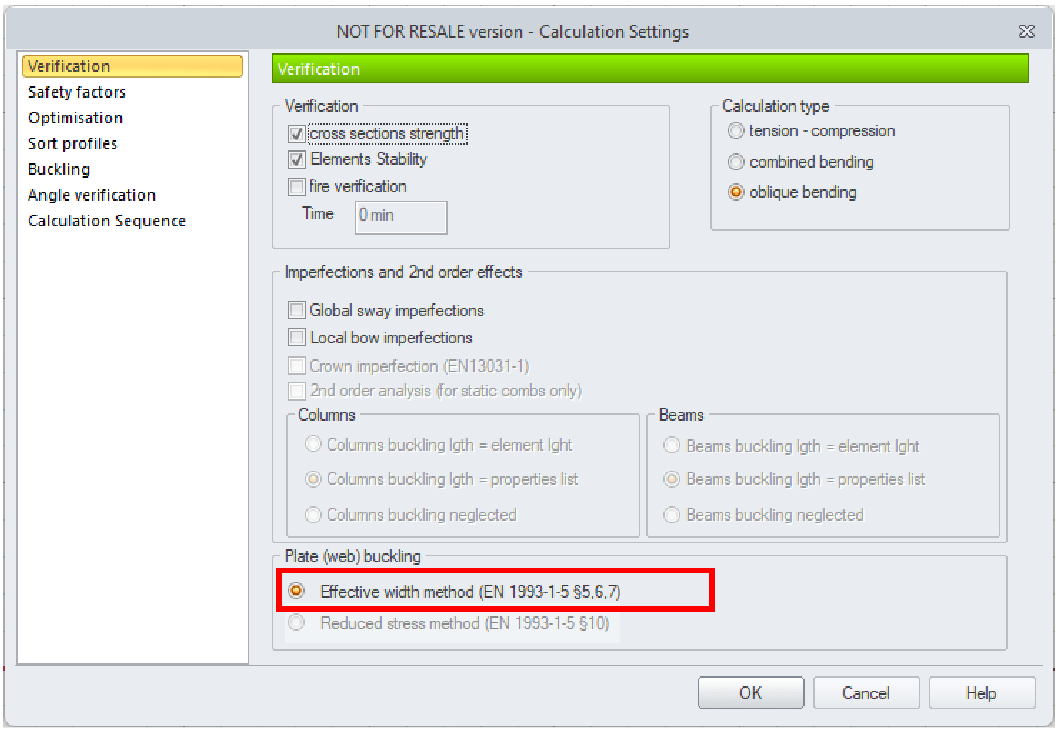
§5.1(2) from EN 1993-1-5 is more specific, introducing two limit values for web slenderness:

- For an unstiffened web.
- For a stiffened web.

The EN 1993-1-5 offers two methods when to account for buckling in steel plates:

- The effective width method, specified in sections 4 to 7.
- The reduced stress method is briefly covered in section 10.

The method implemented in Advance Design 2024 is the **effective width method**. The reduced stress method is not available yet, but it is planned to be implemented soon.



Theoretical background – the Effective width method

In the effective cross-section method, the plate-like behavior of the web is accounted for by reducing the gross width (b or h_w) to an effective width (b_{eff}), with a full f_y yield strength.

Note that the shear buckling verification of the web is performed with the gross web height (h_w) (not the effective height).

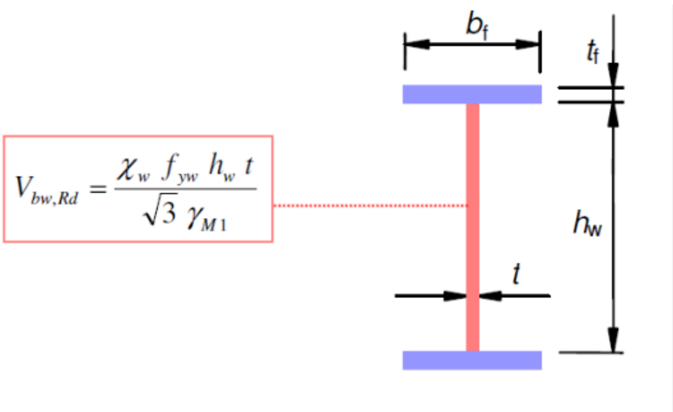
- **Verification**

The shear buckling verification consists of ensuring that the design shear force (V_{Ed}) does not exceed the design resistance for shear ($V_{b,Rd}$).

The design shear resistance ($V_{b,Rd}$) is estimated from:
- The contribution from the web ($V_{bw,Rd}$)
- The contribution from the flanges ($V_{bf,Rd}$)

$$V_{b,Rd} = V_{bw,Rd} + V_{bf,Rd} \leq \frac{\eta\, f_{yw}\, h_w\, t}{\sqrt{3}\, \gamma_{M1}} \qquad (5.1)$$

- **Contribution from the web**

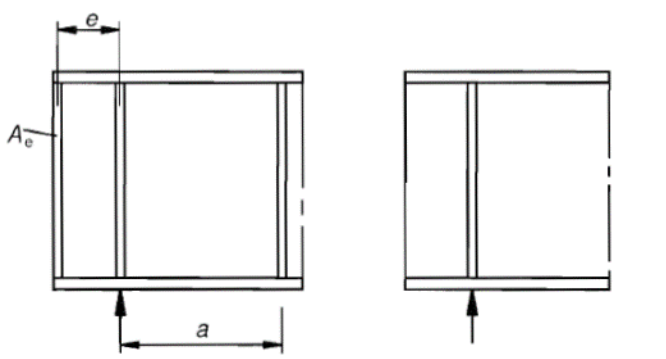


The contribution of shear resistance from the web

The contribution from the web requires the determination of a shear buckling factor (χ_w) (EN 1993-1-5, 5.1).

This shear buckling factor (χ_w) depends on:
- The relative slenderness of the web (λ_w)
- Whether the transverse stiffeners at the end of the web panel are **rigid** on **non-rigid**

NOTE: *In Advance Design 2024, the end stiffeners are considered* ***non-rigid.***

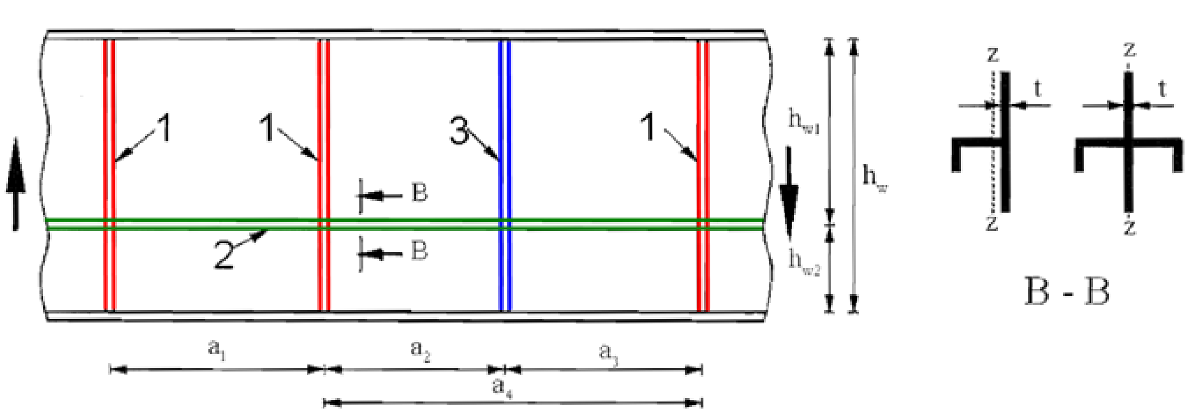


Rigid end post (left), non-rigid end post (right)

The relative web slenderness (λ_w) is given in eq. (5.3).

$$\bar{\lambda}_w = 0{,}76\sqrt{\frac{f_{yw}}{\tau_{cr}}} \quad (5.3)$$

It involves the stress and the shear buckling coefficient (k_τ), defined in Annex A from EN 1993-1-5. This shear buckling coefficient (k_τ) depends on the distance between transverse stiffeners as well as the inertia about the z-z axis of the longitudinal stiffeners:

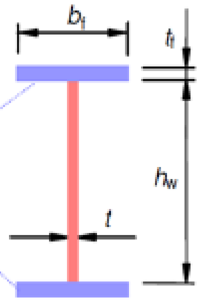


1 – Rigid transverse stiffeners, 2 – Longitudinal stiffeners, 3 – non-rigid transverse stiffener

NOTE: *In Advance Design 2024, with the effective width method only transverse stiffeners can be defined and they are assumed as non-rigid.*

Once web slenderness (λ_w) has been determined, the shear buckling factor (χ_w) is computed according to Table 5.1, with Advance Design considering the end posts as **non-rigid**.

- **Contribution from the flanges**

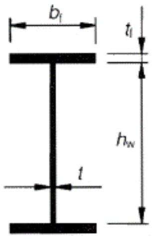


$$V_{bf,Rd} = \frac{b_f\, t_f^2\, f_{yf}}{c\, \gamma_{M1}}\left(1-\left(\frac{M_{Ed}}{M_{f,Rd}}\right)^2\right)$$

The contribution of shear resistances from the flanges

The flanges may only contribute if they have some remaining capacity after taking the bending moment (EN 1993-1-5, 5.4). For this, we need to calculate the moment of resistance of the cross-section consisting of the effective area of the flanges only ($M_{f,Rd}$) .

$$M_{f,Rd} = b_f \cdot t_f \cdot \left(h_w + t_f\right) \cdot \frac{f_y}{\gamma_{M0}}$$



Note that the presence of an axial force (N_{Ed}) may reduce the resisting moment of the flanges through the factor:

$$\left(1-\frac{N_{Ed}}{\frac{\left(A_{f1}+A_{f2}\right)f_{yf}}{\gamma_{M0}}}\right) \qquad (5.9)$$

We need also determine the c length for the flange's contribution:

$$c = a\left(0{,}25+\frac{1{,}6\,b_f t_f^2\, f_{yf}}{t\,h_w^2\, f_{yw}}\right)$$

Note that the contribution from the flanges, when it exists, is usually significantly lower than the contribution from the web.

- **Shear buckling verification**

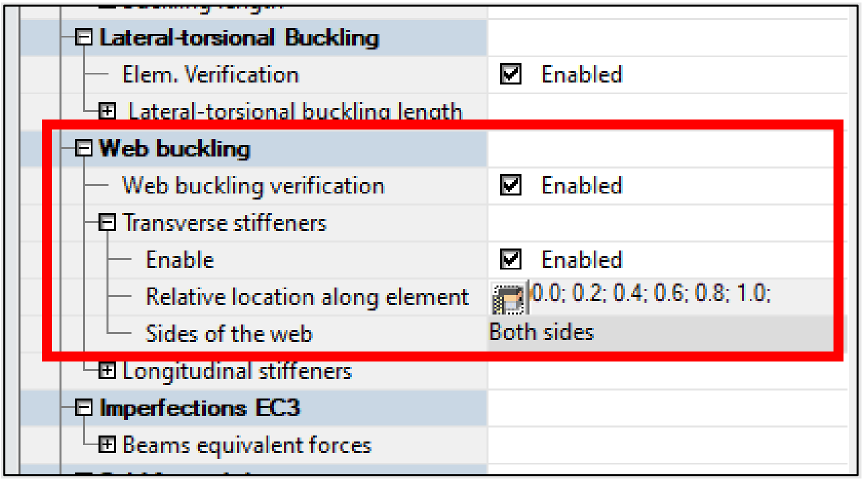
The shear buckling verification consists of ensuring the design shear force V_{Ed} does not exceed the design resistance for shear $V_{b,Rd}$ from web and flanges:

$$\eta_3 = \frac{V_{Ed}}{V_{b,Rd}} \leq 1{,}0 \qquad (5.10)$$

When η_3 exceeds 0,50, extra verification is performed to account for the interaction between shear force, bending moment, and axial force (EN 1993-1-5, 7.1).

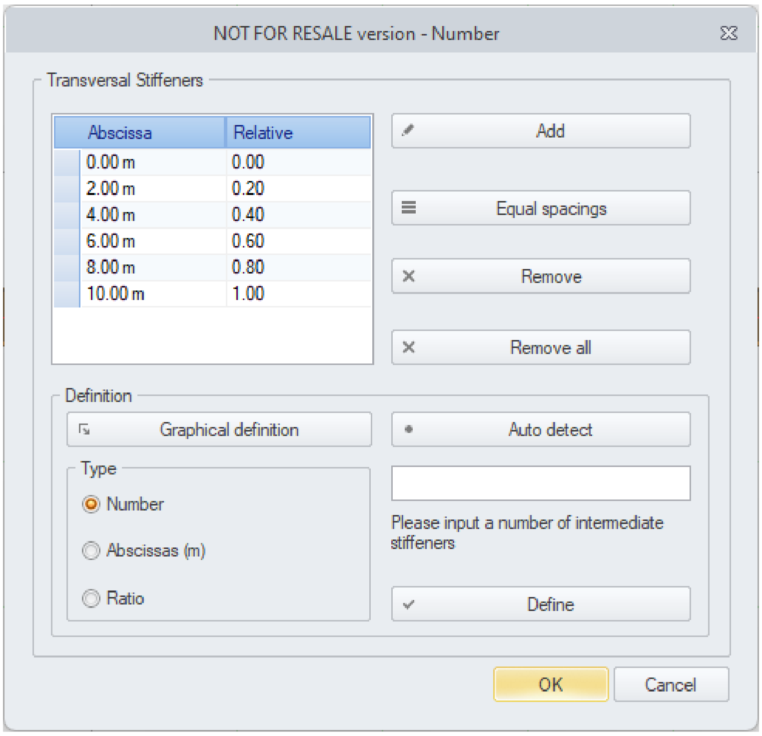
Definition and results in Advance Design

On Advance Design, a new 'Web buckling' section has been added to the steel design properties, for managing parameters related to the shear buckling verification.



Web shear buckling parameters (Effective width method)

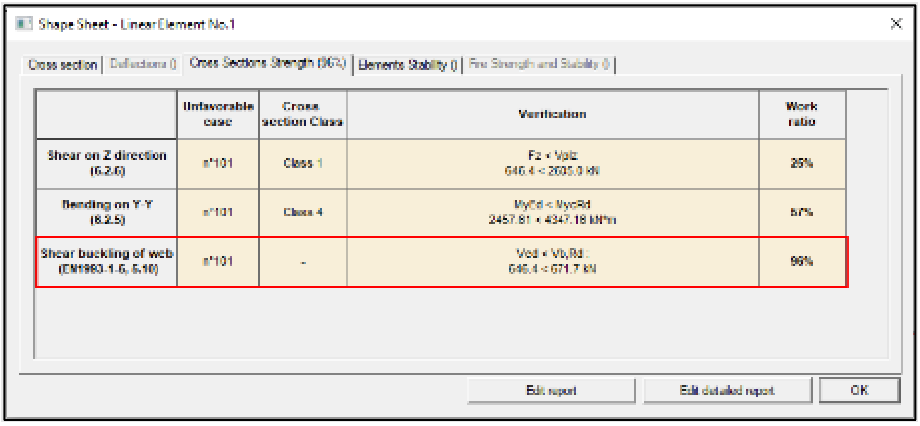
The *Web buckling verification* checkbox governs whether Advance Design performs the verification or not. The *Transverse stiffeners* parameters let us position the transverse stiffeners along the member, with various definition modes being available. For this, a dedicated dialog is opened.



Transverse stiffener definition

Advance Design allows many ways to define transversal stiffeners, including by equal distribution after entering their number, by entering their relative or absolute coordinates, but also by graphical definition (indicating points by the length of the element in the 3D view) or by using the command for automatic detection based on intersecting elements.

After calculation, the basic results from the web shear buckling verification are available in the shape sheet:



Shape sheet

The detailed version of the shape sheet report provides more information regarding:

- The shear buckling checks itself, mentioning the buckling coefficients ($k_{\tau,sB}$, and k_τ), the slenderness parameter (λ_w), the shear buckling factor (χ_w)...

Shear buckling of web (EN1993-1-5, 5.10)	Case no 101 : 1x[1 L], Mesh No. 1.1 0/4 Point x = 0.00 m Cross section : Class - $k_\tau = 5.34$ $k_{\tau,sB} = 0.00$ $\sigma_E = 3.04$ MPa $\bar{\lambda}_w = 2.89$ $\chi_w = 0.29$ $V_{bw,Rd} = 623.1$ kN $M_{f,Rd} = 37.61$ kN*m $k_{NEd} = 0.00$ c = 77.35 cm $V_{bf,Rd} = 48.6$ kN $\frac{h_w}{t} < \frac{31}{\eta} \varepsilon \sqrt{k_\tau}$: 250.00>59.70 $V_{Ed} < V_{b,Rd}$: 646.4 < 671.7 kN (96 %)

- The interaction between shear force, bending moment, and axial force (if relevant)

Interaction - shear, bending, axial force (EN1993-1-5, 7.1)	Case no 101 : 1x[1 L], Mesh No. 1.1 0/4 Point x = 0.00 m Cross section : Class - $\eta_1 = 0.00$ $\eta_3 = 0.96$ $M_{pl,Rd} = 5715.39$ kN*m $M_{f,Rd} = 3797.60$ kN*m $\eta_1 \leq \frac{M_{f,Rd}}{M_{pl,Rd}}$ Bending moment can be carried out by flanges alone. No need to satisfy eq. (7.1) : 0.00 < 1.00 (0 %)

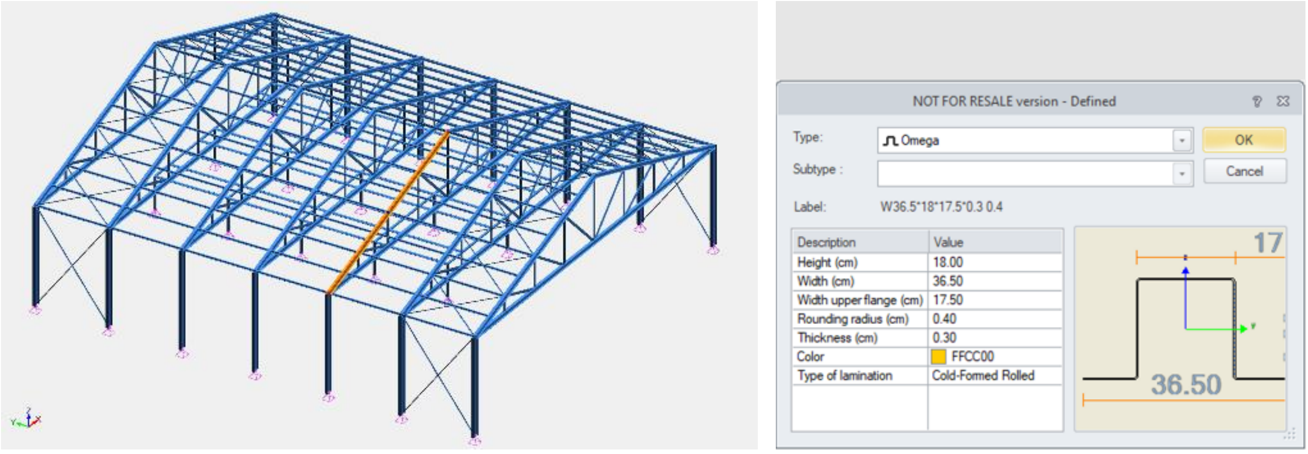
- The position of the stiffeners

Stiffeners	Transverse stiffeners - relative positions: 0.00; 0.20; 0.40; 0.60; 0.80; 1.00; Location: Both sides

4.3. User definition of buckling and LT buckling lengths for cold-formed sections for EC3

Next step of improvements to cold-formed design acc. Eurocode 3 for analyzing buckling and lateral-torsional buckling.

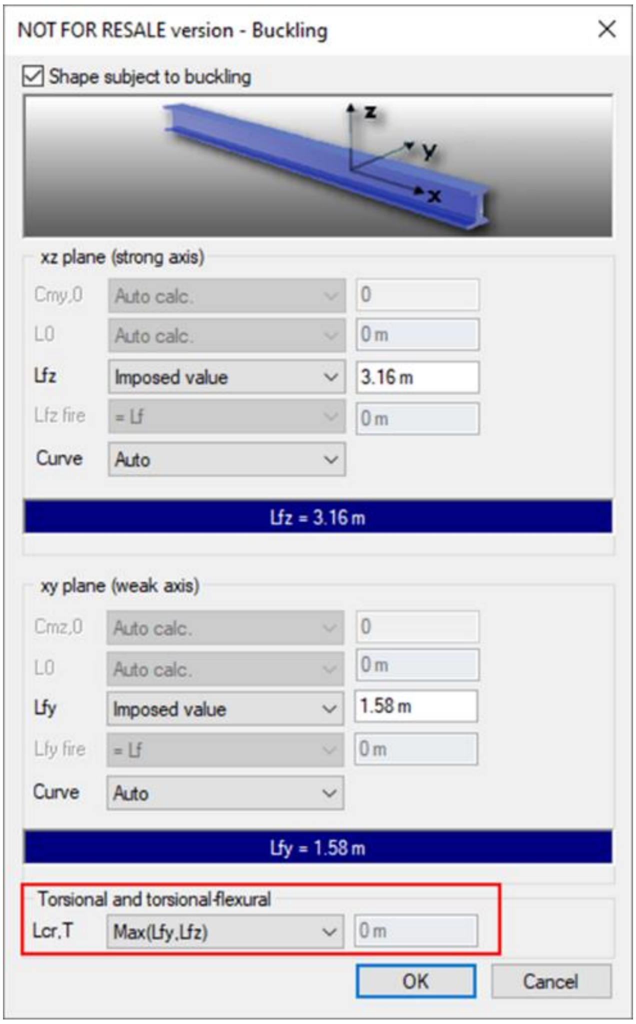
Advance Design 2024 now allows for a more accurate definition of buckling lengths and lateral-torsional buckling lengths on cold-formed structural components.



The hat section is used as a top chord of a lightweight steel structure

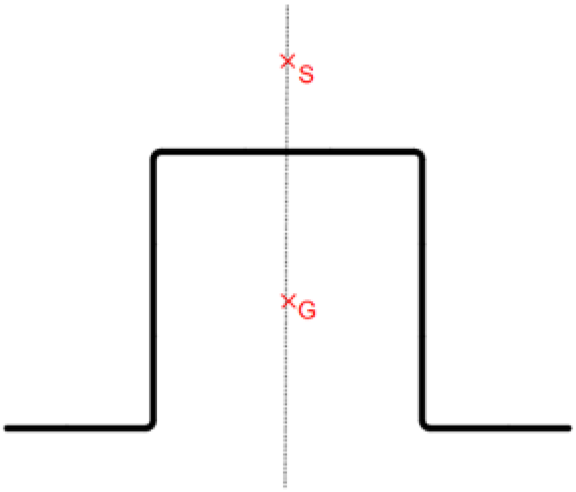
Buckling length definition

The buckling dialog now lets the users define the critical length for torsional buckling and torsional-flexural buckling, involved in the calculation of the Ncr,T, and Ncr,TF critical forces, respectively.



Buckling settings

Indeed, the complex shape of these open-walled sections, often asymmetrical, results in low torsional resistance, with a significant offset between their shear center and gravity center.



S – Shear center, G - Centroid

This makes them prone to **torsional buckling** (where compression causes a twist of the element) as well as **flexural-torsional buckling** (a combination of bending and twisting of the compressed member), as explained in §6.3.2 from EN1993-1- 3.

This new option ensures a precise estimation of the corresponding critical forces $N_{cr,T,}$ and $N_{cr,TF}$.

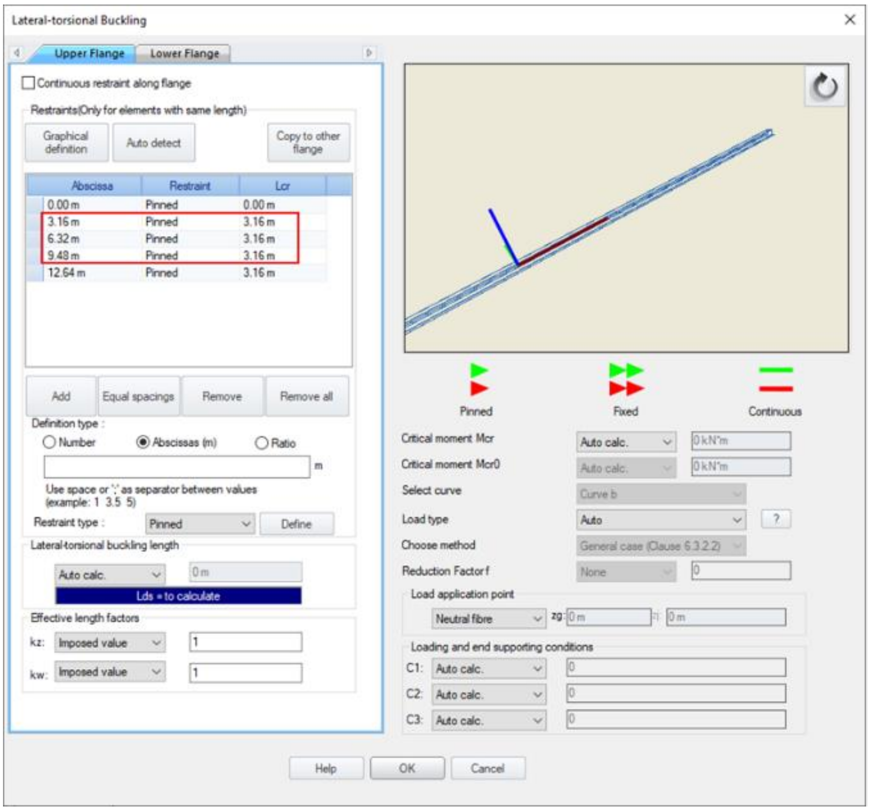
After calculation, the detailed shape sheet will mention:

- The critical lengths for torsional and torsional-flexural buckling ($L_{cr,T}$ and $L_{cr,TF}$)
- The corresponding critical forces ($N_{cr,T}$ and $N_{cr,TF}$)
- The corresponding buckling coefficients χ_T and χ_{TF}

Buckling (6.2.2)	Case no 101 : 1x[1 S], Mesh No. 2.1 0/4 Cross section : Class 4 $N_{Ed} < N_{b,Rd}$: 200.8 kN < 210.3 kN L_{fy} = 1.58 m, $\lambda_{bar,y}$ = 0.43, α_y = 0.34 L_{fz} = 3.16 m, $\lambda_{bar,z}$ = 0.16, α_z = 0.34 $L_{cr,TF}$ = 3.16 m, $N_{cr,TF}$ = 312.1 kN, $L_{cr,T}$ = 3.16 m , $N_{cr,T}$ = 312.1 kN $\chi = \min(\chi_y, \chi_z, \chi_T, \chi_{TF})$ = min(0.91, 1.00, 0.50, 0.50) $N_{Ed} < \frac{\chi * A_{eff} * f_{yb}}{\gamma_{M1}}$ (6.48 EN1993-1-1) 200.8 kN < $\frac{0.50 * 11.86 cm^2 * 355.00 MPa}{1.00}$ (95 %)

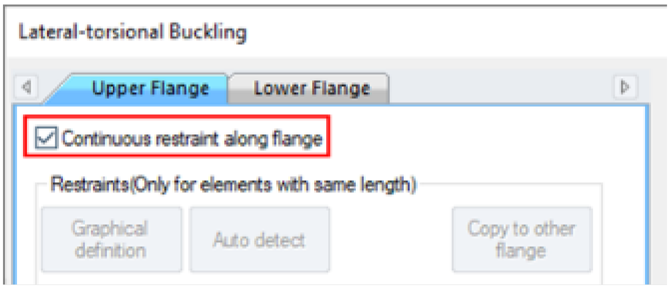
Lateral-torsional buckling length definition

The Lateral-torsional buckling dialog now lets the users define intermediate restraints on the top or bottom sides of cold-formed members.



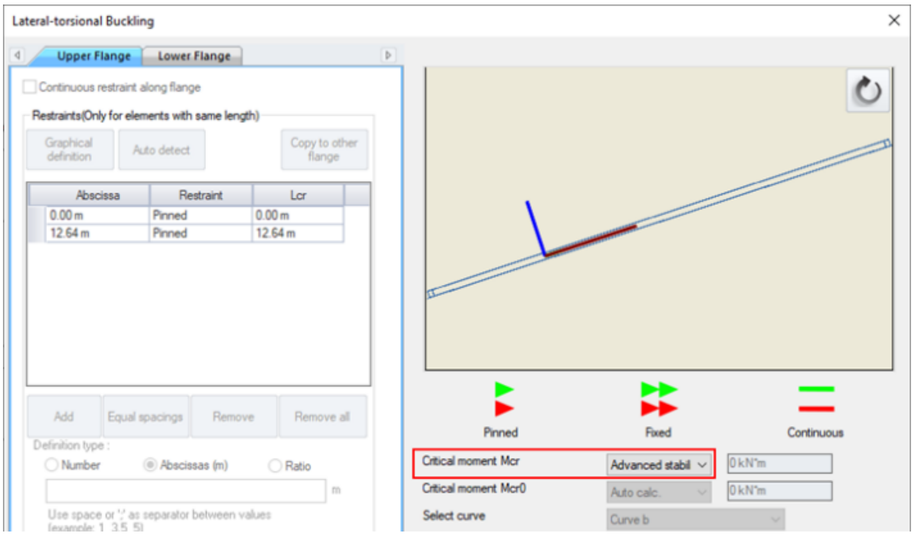
Lateral-torsional buckling settings

The users may even neglect the lateral-torsional buckling effects entirely by ticking the 'Continuous restraint along flange' box, which assumes the torsion effects have been avoided by other means (by using batten plates, by ensuring a fixed connection from a slab to the flange…).



Neglecting the lateral-torsional buckling effects

Alternatively, the users may rely on the Advanced stability engine to compute the critical moment (M_{cr}) in case of a complex section shape.



Option for selecting the method of Critical moment calculation

In this case, the $N_{cr,T}$ and $N_{cr,TF}$ critical forces will also result from the Advanced stability analysis.

After calculation, the detailed shape sheet will return the various parameters involved in the calculation of the critical moment (M_{cr}), conducted according to Annex I from EN 1999-1-1. Such parameters include:

- The cross-section monosymmetric factor (Ψ_f)
- The C_1, C_2 and C_3 coefficients
- The Wagner factor (z_j) (which reflects the asymmetry of the section)

Lateral-torsional Buckling (6.2.4)	Case no 101 : 1x[1 S], Mesh No. 2.2 4/4 Cross section : Class 4 $M_{y,Ed} < M_{by,Rd}$: 6.00 kN*m < 25.45 kN*m $\lvert M_{y,Ed} + \Delta M_{Ed}\rvert < \frac{\chi_{LT} * W_{eff,y} * f_{yb}}{\gamma_{M1}}$ $\lvert -6.00\ kN{*}m + 0.00\ kN{*}m\rvert < \frac{0.76 * 94.78\ cm3 * 355.00\ MPa}{1.00}$ Eccentricity favorable effects are ignored. k_z = 1.00, k_w = 1.00, k_{wt} = 1.18, ψ_f = 0.81, C1 = 1.13, C2 = 0.46, C3 = 0.53, Lds = 12.64 m, Ldi = 0.97 m, Lcr = 12.64 m, zg = 0.00 cm, zj = 18.20 cm, Mcr = 60.23 kN*m, $\lambda_{bar\ LT}$ = 0.75, χ_{LT} = 0.76 Mcr was computed analytically. (24 %)

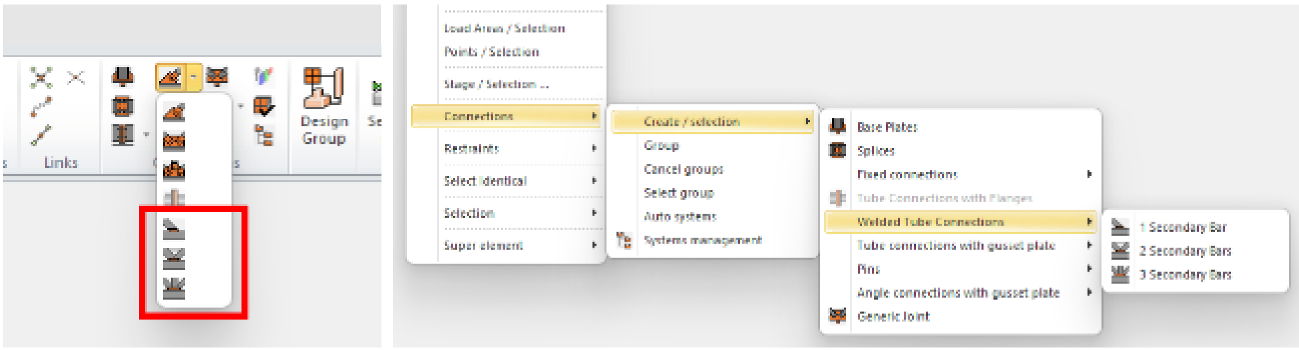
4.4. Modeling welded tube connections

Ability to define welded truss connections for rectangular hollow sections.

In the latest version of Advance Design, welded tube connections have been added to the set of many types of steel connections that can be created in the model and then sent for verification to the Steel Connection design module. These are 2d joints that allow the connection of welded rectangular and square steel tubes, i.e. connections typical of trusses.

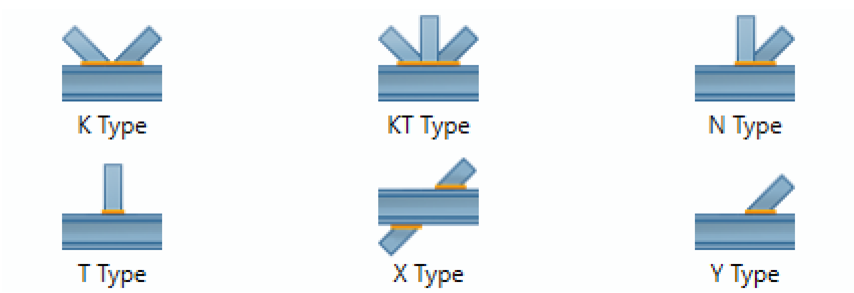
New connections defined on trusses in the Advance Design model

New connections can be defined using new commands from the ribbon or the context menu.

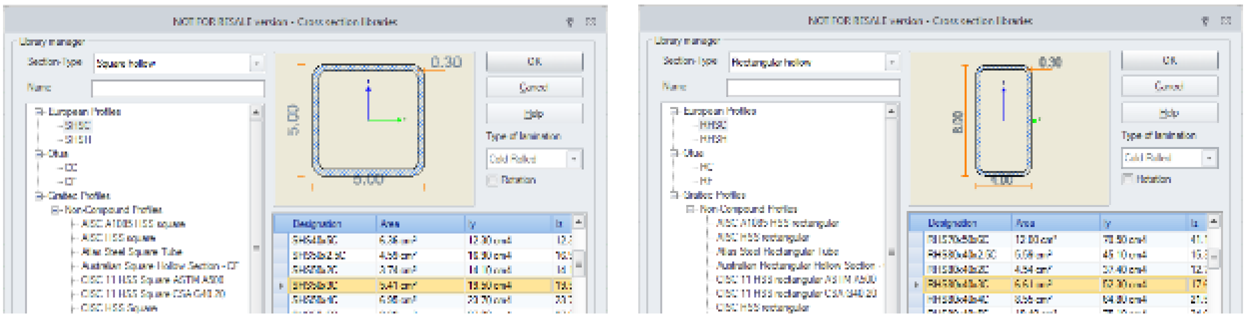


We can define 3 category of welded tube joints that differs with the number of secondary bars - with 1, 2 or 3 secondary bars. With them, we can model the most common typical joints supported by Eurocode 3 (T type, Y type, N type, K type, X type and KT type):

- 1 secondary bar:
  - T type
  - Y type
- 2 secondary bars:
  - K type
  - N type
  - X type
- 3 secondary bars
  - KT type

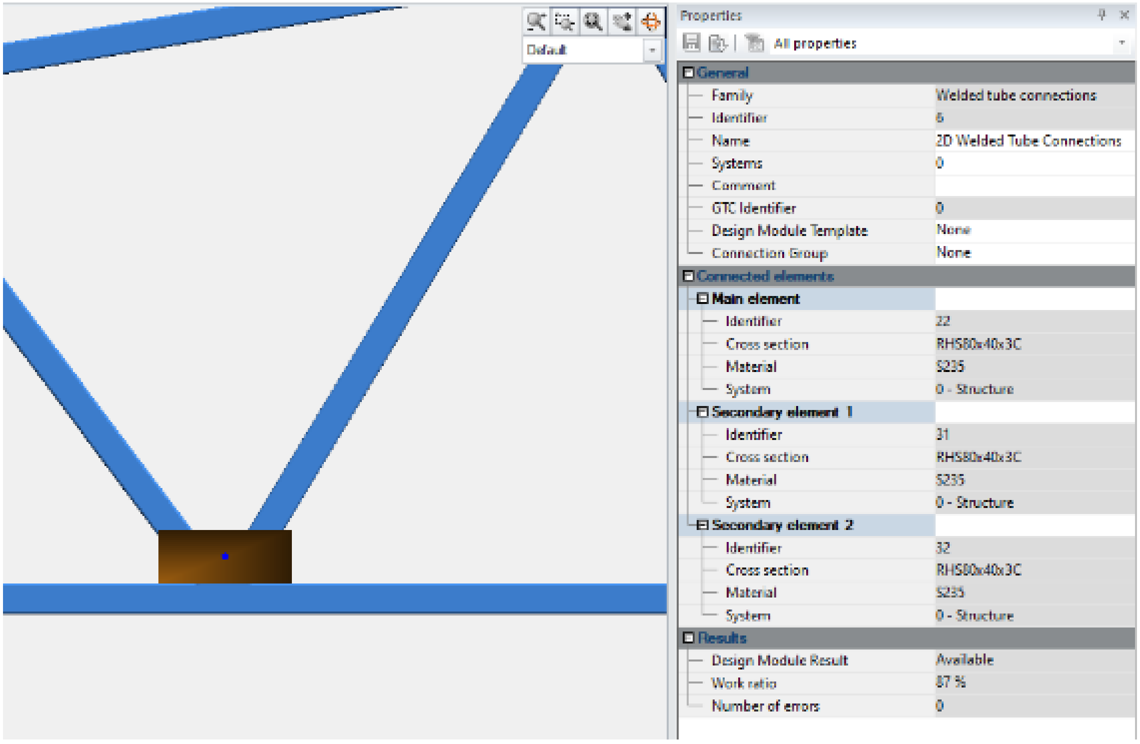


All these connections allow to join open steel profiles with a square or rectangular cross-section. Such sections can be used as main elements (chords) and as secondary elements (diagonals). In addition, for main elements (chords) it is possible to use I sections.



Square and rectangular open steel profiles on Advance Design databases

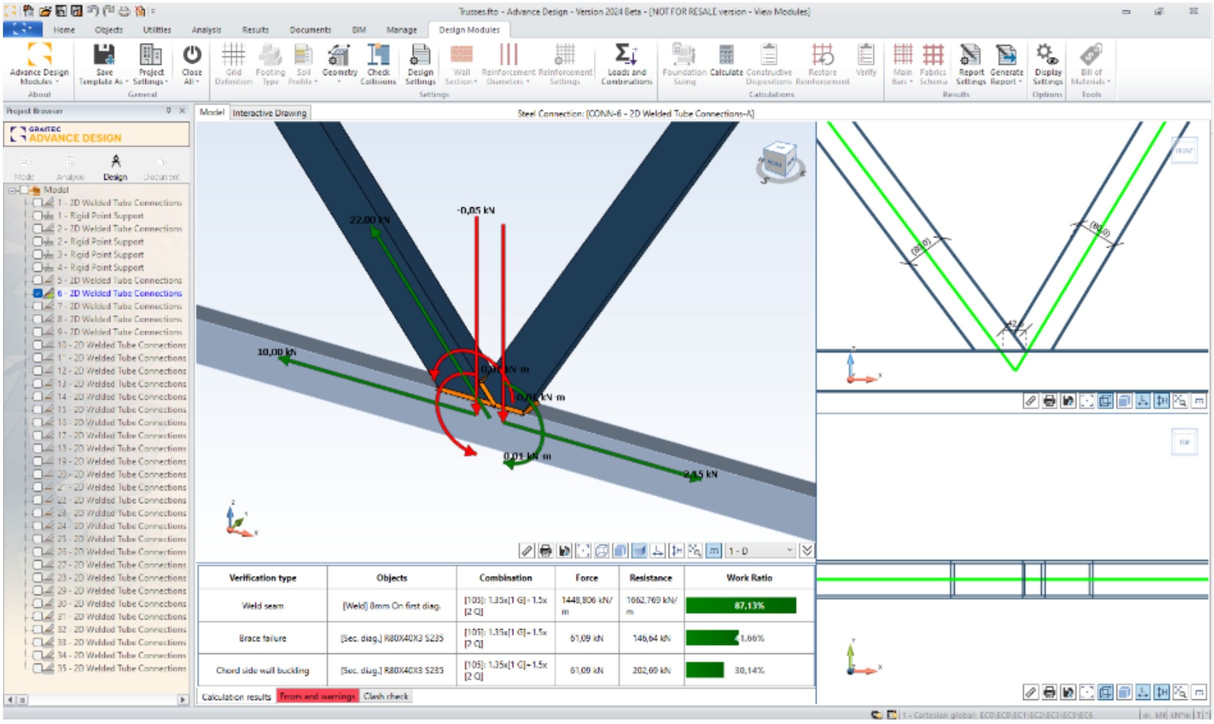
As in the case of other available connections, new connections can be defined individually or many at once, and groups can be created from them. After defining new connections we have their properties, in which we can check the most important parameters.



Properties of a selected new welded tube connection

New connections can be opened or exported to the Steel Connection design module for further analysis. Along with data on geometry, profiles, and materials, internal forces are also transferred. Further details, including welds' parameters, can then be defined in the Steel Connection design module and the load capacity analysis of these connections can be conducted.

Note: *More details about new connections in the Steel Connection design module you can find in a separate dedicated chapter.*



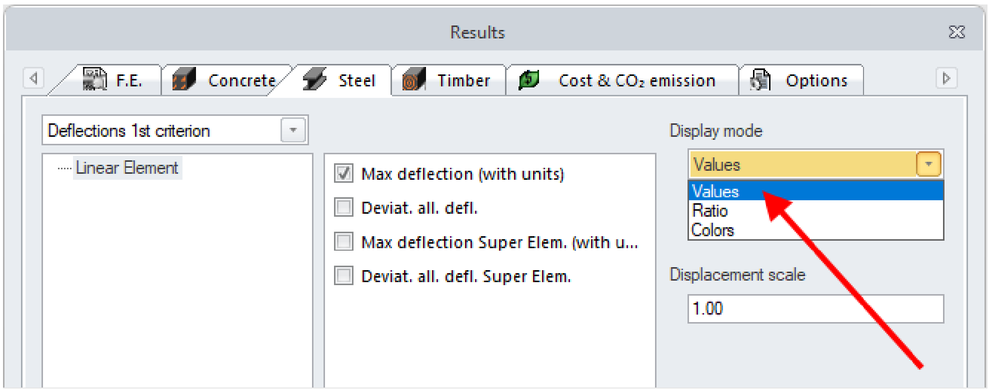
Welded truss connection opened on Steel Connection design module

4.5. Deflection in value (cm or mm) in the shape sheets and diagrams

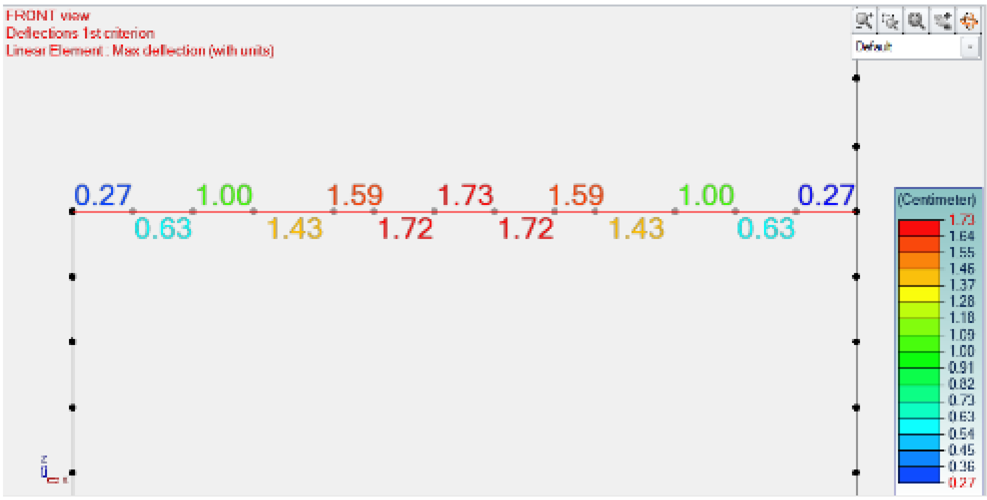
Easier to check results and create documentation for steel member deflection verification.

Previously, deflection values from the verification of steel elements were displayed only as a ratio (e.g.,1/574) to make comparison with limit values (e.g., 1/500) easier. Now the deflection can also be displayed in displacement values (e.g., cm).

For graphical presentation, the new display mode is available in the Result Settings window for Deflection results, as a new 'Values' item in the Display Mode list.

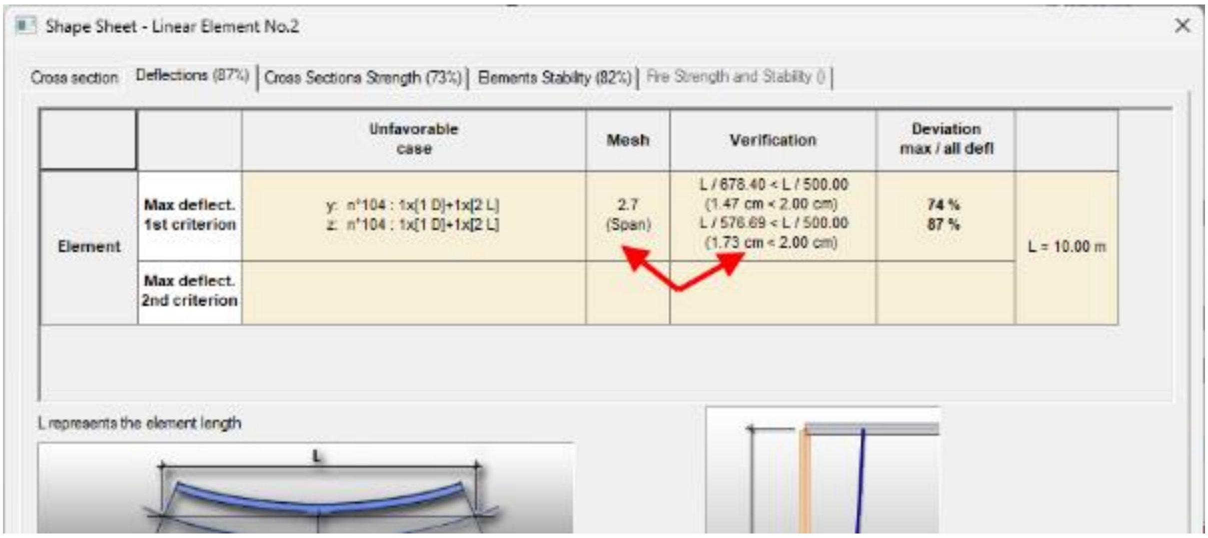


Result settings



Graphic results (max deflection in cm)

In the case of the Shape Sheet window and report, the values in the displacement unit are displayed just below the ratio values. In addition, the Mesh column displays information about the location of the verification point - whether it is in the span or the extremity of the element.



Shape Sheet

5. Enhancing the analysis of timber structures acc. Eurocode 5

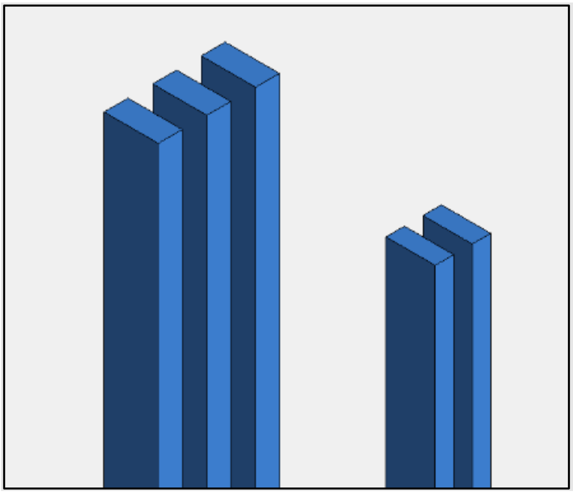
A series of novelties and improvements related to the verification and optimization of timber element structures according to Eurocode 5.

5.1. Design of spaced rectangular sections (EC5)

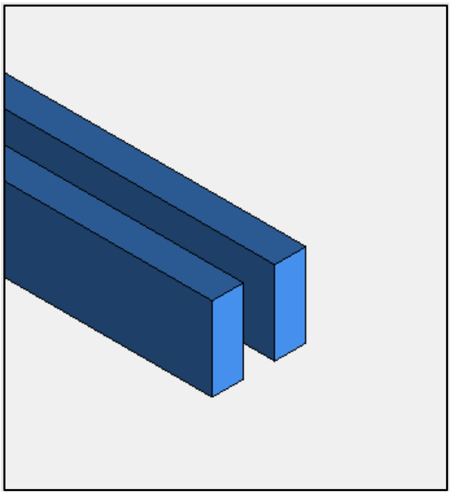
Ability to verify timber columns and beams with compound sections according to Eurocode 5.

Advance Design 2024 now enables the design of spaced members using Eurocode 5.

This includes:
- Spaced columns
- Spaced beams



Spaced columns



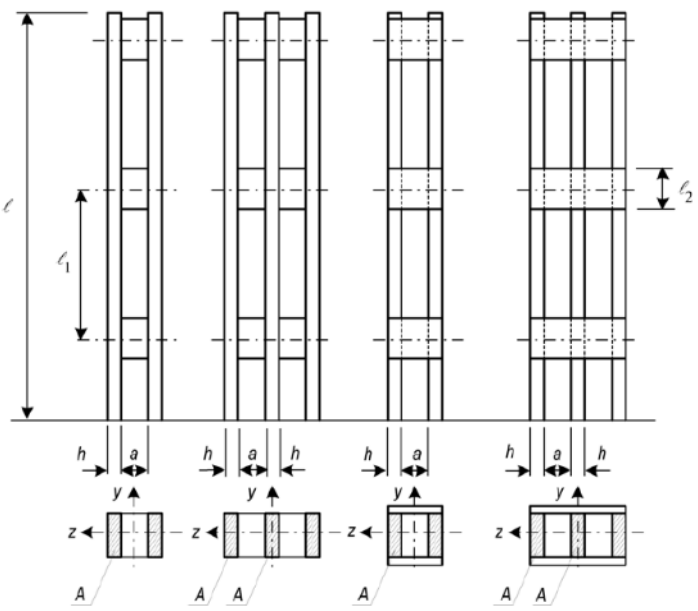
Spaced beams

Spaced shaft profiles are defined using a new parametric profile type *(see details in a separate paragraph on new parametric profiles).*

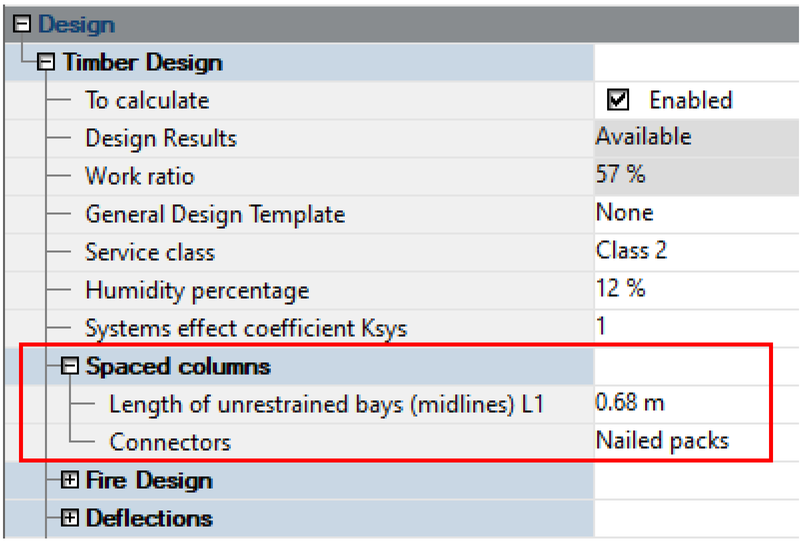
Spaced columns

A spaced column is a built-up column made of 2 to 4 identical sections (shafts) separated and connected by packs or gussets fixed by glue or mechanical fasteners. They are often used as compression chords or in frame construction due to their high load-carrying capacity.

Spaced columns are covered in Annex C.3 from EN1995-1-1.



A vertical element with a cross-section defined as a spaced shaft will be recognized as a column, which will give access to additional parameters in the timber design section, where the users can define the distance between connectors as well as their type.



Spaced columns are designed according to annex C from EN 1995-1-1 which is intended for simply supported members. This annex describes two checks related to spaced columns:

- Verification of the axial load-carrying capacity (§C.3.2)
- Verification of the load on fasteners, gussets, or packs (§C.3.3)

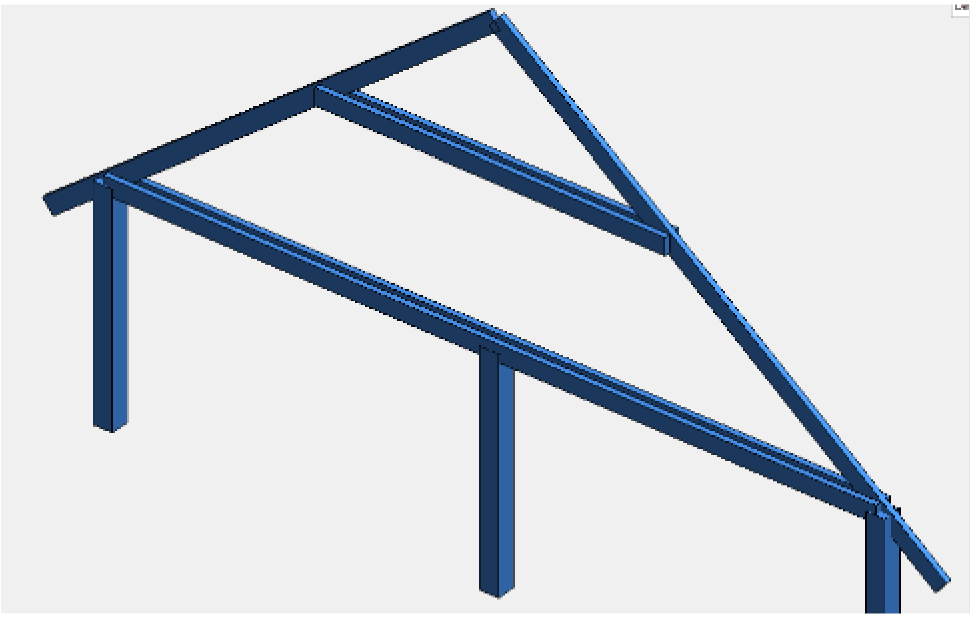
After calculations, the results of the analysis can be viewed on the 'Stability' tab of the Shape sheet window as well as in the report. The detailed version of the shape sheet additionally provides the intermediate terms involved in these verifications.

4) Elements stability	
Verification	Case no 103, Mesh No. 5.5 $\lambda_y = 86.6$ $\lambda_z = 64.1$ Lfy = 5.00 m Lfz = 5.00 m Ky=1.504 Kcy=0.475 Kz=1.056 Kcz=0.490 Km=0.700 Kcrit=0.994 $\lambda_{rel,y} = 1.378$ $\lambda_{rel,z} = 1.019$ $\lambda_{rel,m} = 0.755$ $\lambda_1 = 32.332$ $\eta = 3.000$ $\lambda_{ef} = 85.080$ Kc=0.475 Ldy = 5.00 m Ldz = 5.00 m *Work ratio Verification:* Case no 103, Mesh No. 5.5 6.23: $\sigma_{c0d} / (K_{cy} F_{c0d}) + \sigma_{myd} / F_{myd} + K_m \sigma_{mzd} / F_{mzd} \leq 1$ 0.11695 < 1 (12%) Case no 103, Mesh No. 5.5 6.24: $\sigma_{c0d} / (K_{cz} F_{c0d}) + K_m \sigma_{myd} / F_{myd} + \sigma_{mzd} / F_{mzd} \leq 1$ 0.11335 < 1 (11%) Case no -, Mesh No. -, 6.33: $\sigma_{md} / (K_{crit} F_{md}) \leq 1$ not done (-)
Axial Load-carrying capacity (C.3.2)	Case no 103, Mesh No. 5.5 (C.1) : $\sigma_{c.0.d} \leq k_c{*}f_{c.0.d}$ 0.96 MPa < 0.48 * 17.28 MPa : 0.96 < 8.21 MPa (12 %) Simply supported column under concentric axial load, as per Annex C from EN1995-1-1
Shear Force on connectors (C.3.3)	Case no 103, Mesh No. 5.5 (C.5): $V_d = \frac{f_{c,d}}{60 * k_c} = \frac{28.8\ kN}{60 * 0.48} = 1.0\ kN$ (C.13): $T_d = \frac{V_d{*}L_1}{a_1} = \frac{1.0\ kN * 700\ mm}{75\ mm} = 9.4\ kN$ $F_{v,Ed} = T_d = 9.4\ kN$

Shape sheet (detailed version)

Spaced beams

Spaced beams are commonly used as collar ties or rafter ties. Therefore, they are meant to resist the tension or compression forces caused by gravity loads and uplift loads.



Spaced beams as rafter tie and collar tie

The design of spaced beams according to EN 1995-1-1 is conducted in the same manner as for a usual rectangular section, but with adjusted parameters to account for the spacing between the rectangular shafts.

The following parameters are modified:

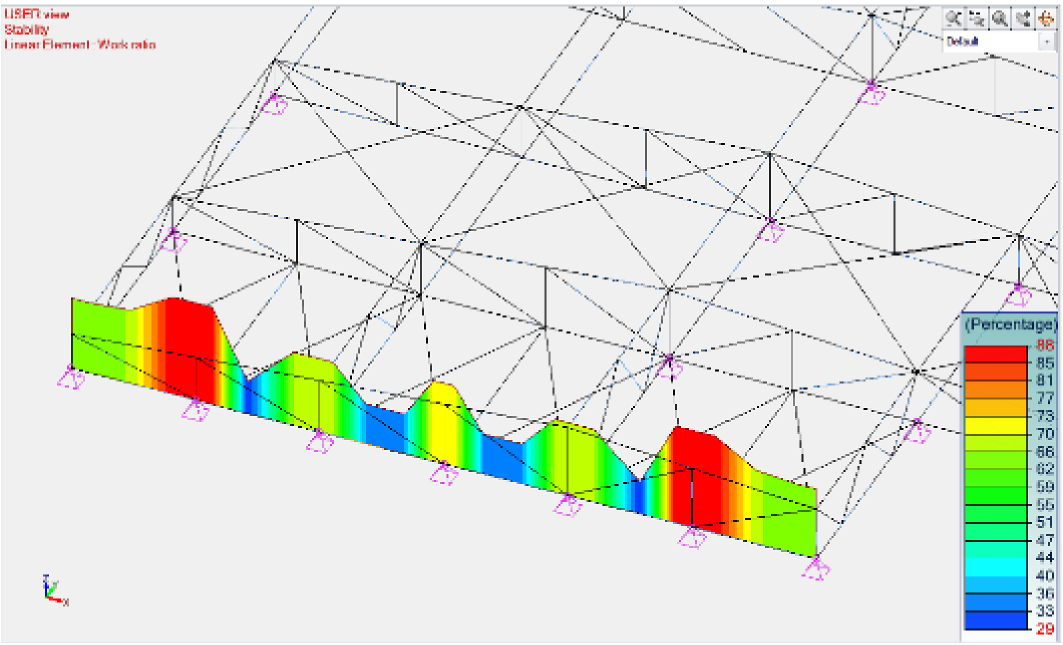
- *Depth factor k_h*
  For members in tension, depth factor k_h uses section width. For spaced beams width = b (width of a single rectangle)

- *Critical bending stress $\sigma_{m,crit}$*
  It is used for lateral-torsional buckling effects. It is calculated using simplified (6.32) formula, with b and h of a single rectangle.

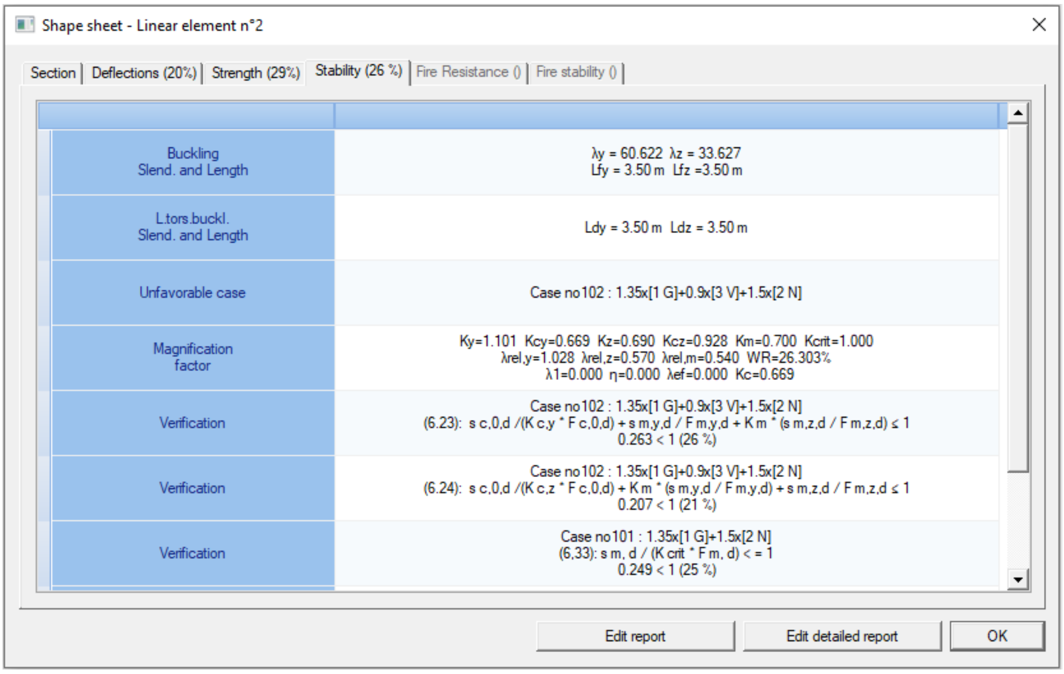
- *Shape of the cross-section factor k_{shape}*
  k_{shape} is used in the torsion check. It is calculated based on the b dimension of a single rectangle.

- *Effective width b_{ef}*
  For shear verification, the effective width b_{ef} is computed with width = 2b (width of the two rectangles).

The results of the timber design are available graphically or through the shape sheet:



Stability work ratio (%)

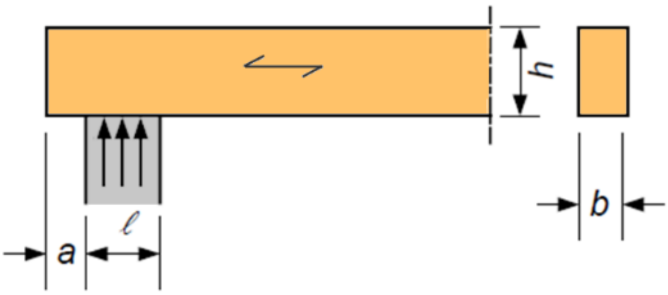


Shape sheet

5.2. Verification of bearing pressure on the support acc. EC5

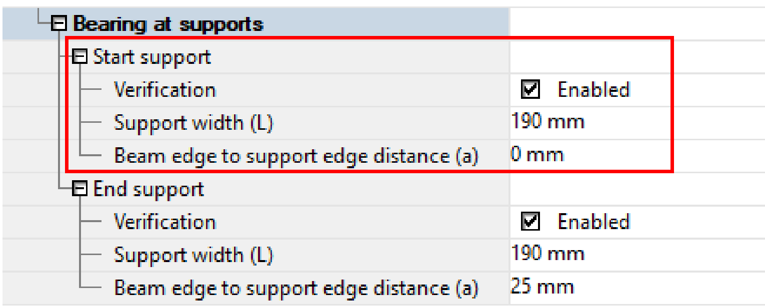
Possibility of performing additional verification for timber elements according to Eurocode - the check of the compression perpendicular to the grain according to support sizes.

Advance Design 2024 now checks the compression perpendicular to the grain on the end supports of a beam, according to §6.1.5 from EN 1995-1-1.



Member on discrete support

A new 'Bearing at supports' section has been added to the timber design properties for users to define the actual contact length (L) as well as a potential overhang (a).



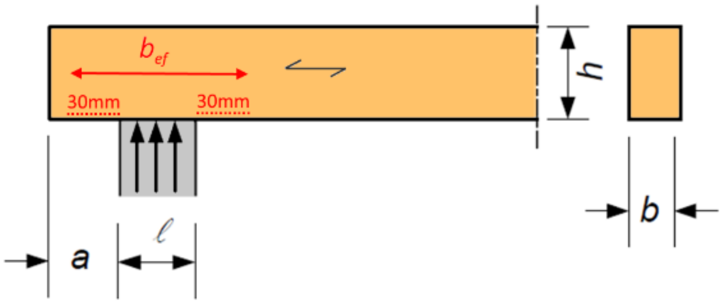
The verification is intended to ensure that, in the effective contact area, the compressive stress perpendicular to the grain ($\sigma_{c,90,d}$) does not exceed the compressive strength ($f_{c,90,d}$) magnified by a $k_{c,90}$ factor.

$$\sigma_{c,90,d} \leq k_{c,90} f_{c,90,d}$$

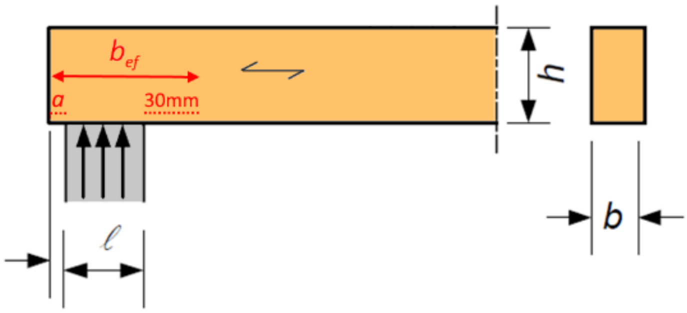
The compressive stress perpendicular to the grain depends on the compressive load on the end support ($F_{c,90,d}$) and the effective contact area (A_{ef}),

$$\sigma_{c,90,d} = \frac{F_{c,90,d}}{A_{ef}}$$

The effective contact area (A_{ef}) is determined by extending the actual contact length (i.e. support width) by a 30mm distance on each side, without exceeding the length of the overhang (a).



Effective contact width if long overhang (a ≥ 30mm)



Effective contact width if short overhang (a < 30mm)

The $k_{c,90}$ magnification factor accounts for the fact that the compression capacity is larger than the pure material strength perpendicular to the grain when a beam is subjected to concentrated pressure over a small surface.

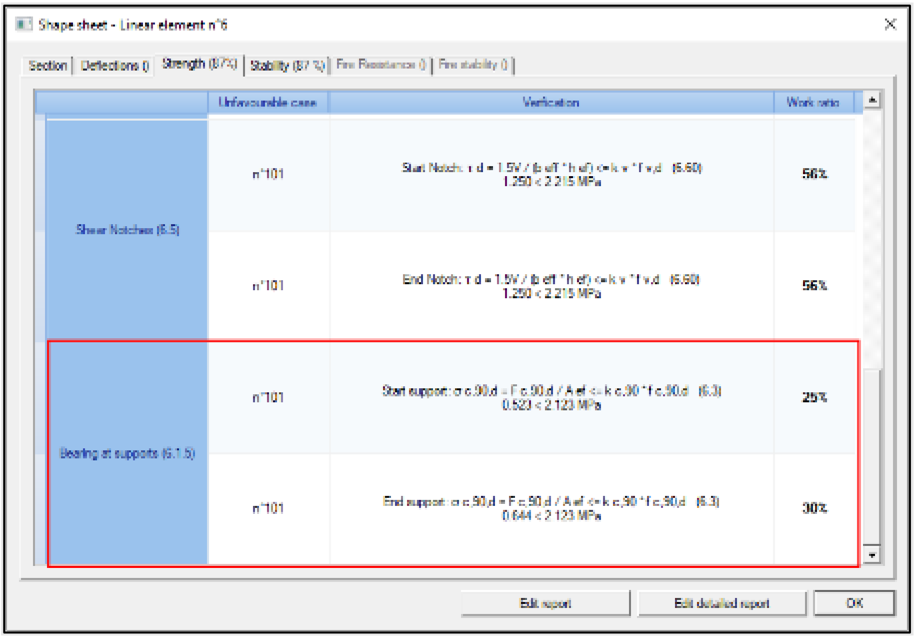
This effect is even bigger for glued laminated timber than for solid timber, resulting in a smaller $k_{c,90}$ for solid timber.

(4) For members on discrete supports, provided that $\ell_1 \geq 2h$, see Figure 6.2b, the value of $k_{c,90}$ should be taken as:

– $k_{c,90}$ = 1,5 for solid softwood timber

– $k_{c,90}$ = 1,75 for glued laminated softwood timber provided that $\ell \leq 400$ mm

After calculation, the results from this check are available in the shape sheet for the selected member.



Shape sheet

The detailed version of the shape sheet will also provide the various parameters involved in this verification ($F_{c,90,d}$, A_{ef}, $k_{c,90}$ and $f_{c90,,d}$).

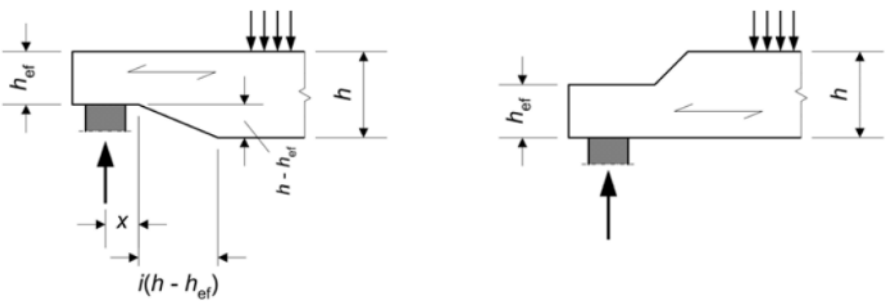
Bearing at supports (6.1.5)	Case no 101, Mesh No. 6.1 Start support: $F_{c,90,d}$ = 4.186 kN A_{ef} = 80.00 cm² $k_{c,90}$ = 1.50 $f_{c,90,d}$ = 1.415 MPa $\sigma_{c,90,d} = \frac{F_{c,90,d}}{A_{ef}} \leq k_{c,90} f_{c,90,d}$ (6.3) 0.523 < 2.123 MPa (25 %) Case no 101, Mesh No. 6.4 End support: $F_{c,90,d}$ = 4.186 kN A_{ef} = 65.00 cm² $k_{c,90}$ = 1.50 $f_{c,90,d}$ = 1.415 MPa $\sigma_{c,90,d} = \frac{F_{c,90,d}}{A_{ef}} \leq k_{c,90} f_{c,90,d}$ (6.3) 0.644 < 2.123 MPa (30 %)

Shape sheet (detailed)

5.3. Verification of beams with a notch at the support acc. EC5

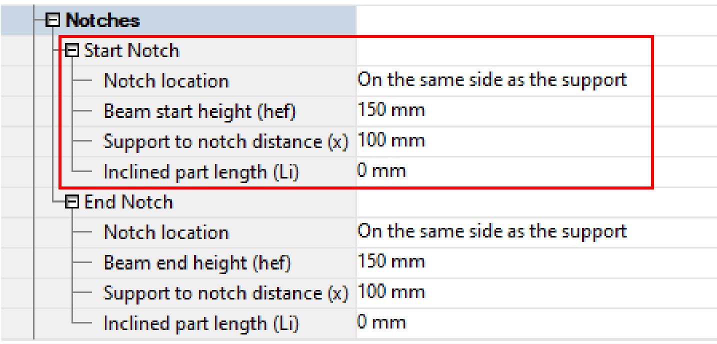
Possibility of performing additional verification for timber elements according to Eurocode - the check of the notch on an element ends.

Advance Design 2024 now manages the support verification of notched beams according to §6.5.2 from EN 1995-1-1.



End-notched beams

A new ‘Notches’ section has been added to the timber design properties.



Notches	
Start Notch	
Notch location	On the same side as the support
Beam start height (hef)	150 mm
Support to notch distance (x)	100 mm
Inclined part length (Li)	0 mm
End Notch	
Notch location	On the same side as the support
Beam end height (hef)	150 mm
Support to notch distance (x)	100 mm
Inclined part length (Li)	0 mm

Notch definition at the start of an element

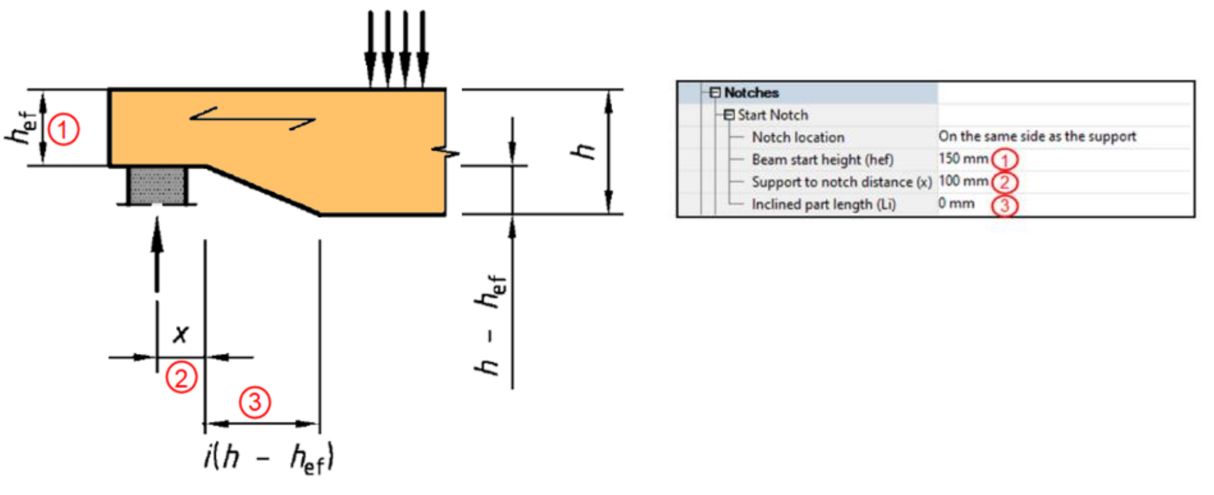
The ‘Notch location’ parameter allows the user to specify whether the notch is:

- On the opposite side of the support
- On the same side as the support

A notch located on the same side as the support (i.e., tension side) can lead to brittle failure due to stress concentration at the notched corner. Thus, the design shear strength of the member shall be reduced by a k_v factor. On the contrary, for a beam with the end notch at the compression side (i.e. opposite side to the support), the k_v factor is taken as 1,0.

This k_v reduction factor is determined based on:

- The effective depth of the beam (h_{ef}) at support location
- The distance from the line of action of the support reaction to the corner of the notch (x)
- The notch inclination (i), if any



The α and k_n parameters, also involved in the determination of k_v, are computed automatically during the timber design sequence:

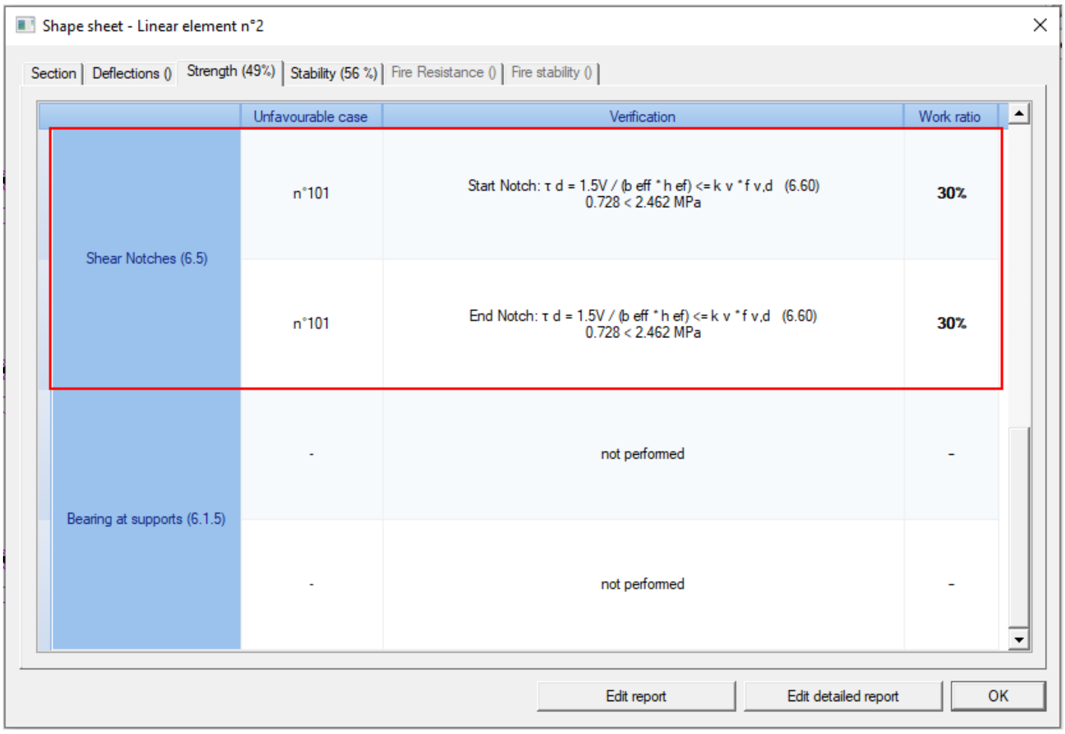
$$\alpha = \frac{h_{ef}}{h}$$

$$k_n = \begin{cases} 4,5 & \text{for LVL} \\ 5 & \text{for solid timber} \\ 6,5 & \text{for glued laminated timber} \end{cases}$$

The verification aims to ensure that the shear stress at the notch support (τ_d) does not exceed the design shear strength ($f_{v,d}$), reduced by the k_v factor.

$$\tau_d = \frac{1{,}5 V_d}{b_{ef} h_{ef}} \le k_v f_{v,d}$$

After calculation, the results from this check are available in the shape sheet for the selected member.



Shape sheet

The detailed version of the shape sheet also provides the various parameters involved in this verification (k_v, $f_{v,d}$...).

3) Cross sections strength	
Tension Compression	Case no -, Mesh No. -, 6.1: $\sigma_{t0,d} \le f_{t0,d}$: not done (-)
Shear	Case no 101, Mesh No. 2.1 6.13: $\tau_d \le f_{v,d}$: 0.485 < 2.462 MPa (20 %)
Oblique bending	Case no 101, Mesh No. 2.3 6.17: $\sigma_{t0,d} / f_{t0,d} + \sigma_{my,d} / f_{my,d} + k_m \sigma_{mz,d} / f_{mz,d} \le 1$: 0.48900 < 1 (49 %) Case no 101, Mesh No. 2.3 6.18: $\sigma_{t0,d} / f_{t0,d} + k_m \sigma_{my,d} / f_{my,d} + \sigma_{mz,d} / f_{mz,d} \le 1$: 0.34230 < 1 (34 %)
Torsion	Case no -, Mesh No. -, 6.14: $\tau_{tor,d} \le k_{shape} f_{v,d}$: 0.000 = 0.00 * 0.000 MPa: not done (-)
Shear with Torsion	Case no 101, Mesh No. 2.1 $(\tau_d / f_{v,d}) + (\tau_{tor,d} / (k_{shape} f_{v,d})) \le 1$: 0.19706 < 1 (20 %)
Shear Notches (6.5)	Case no 101, Mesh No. 2.1 Start Notch: k_v = 1.00 $f_{v,d}$ = 2.462 MPa $\tau_d = \frac{1.5V}{b_{ef} h_{ef}} \le k_v f_{v,d}$ (6.60) 0.728 < 2.462 MPa (30 %) Case no 101, Mesh No. 2.5 End Notch: k_v = 1.00 $f_{v,d}$ = 2.462 MPa $\tau_d = \frac{1.5V}{b_{ef} h_{ef}} \le k_v f_{v,d}$ (6.60) 0.728 < 2.462 MPa (30 %)

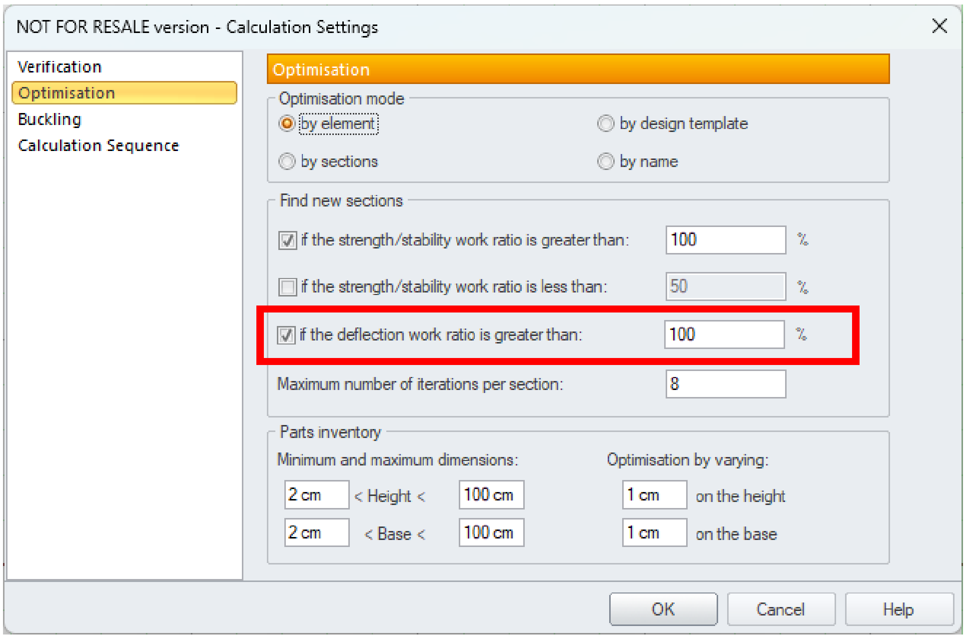
Shape sheet (detailed)

5.4. Optimization of timber elements according to the deflection (EC5)

Ability to automatically select the optimal timber section due to the deflection condition according to EN 1995-1-1.

Advance Design 2024 can now find the optimal dimensions for timber members based on their deflection results.

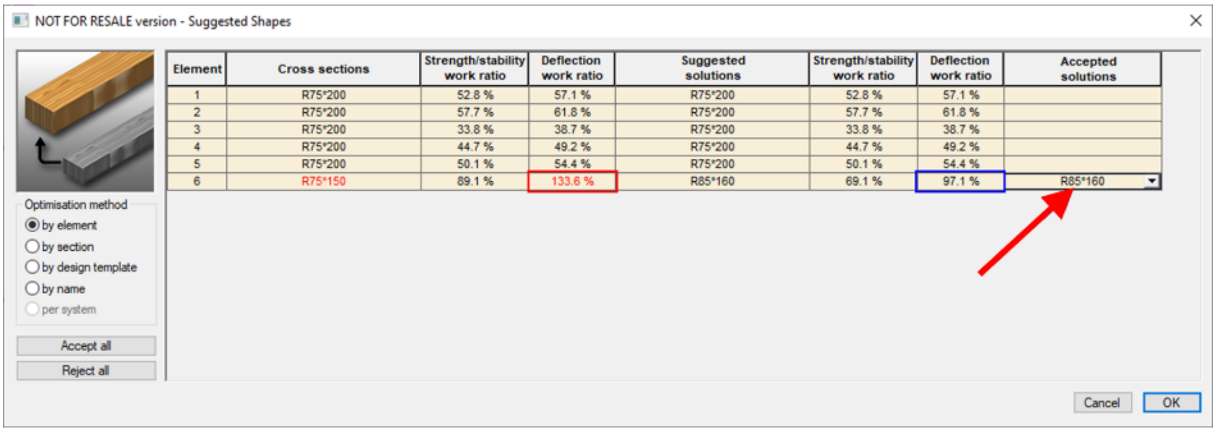
New dimensions are searched if the appropriate option is selected on the Timber Calculation setting window, and the current deflection ratio exceeds a limit, set as 100% as default.



Optimization parameters on the Timber Calculation Settings window

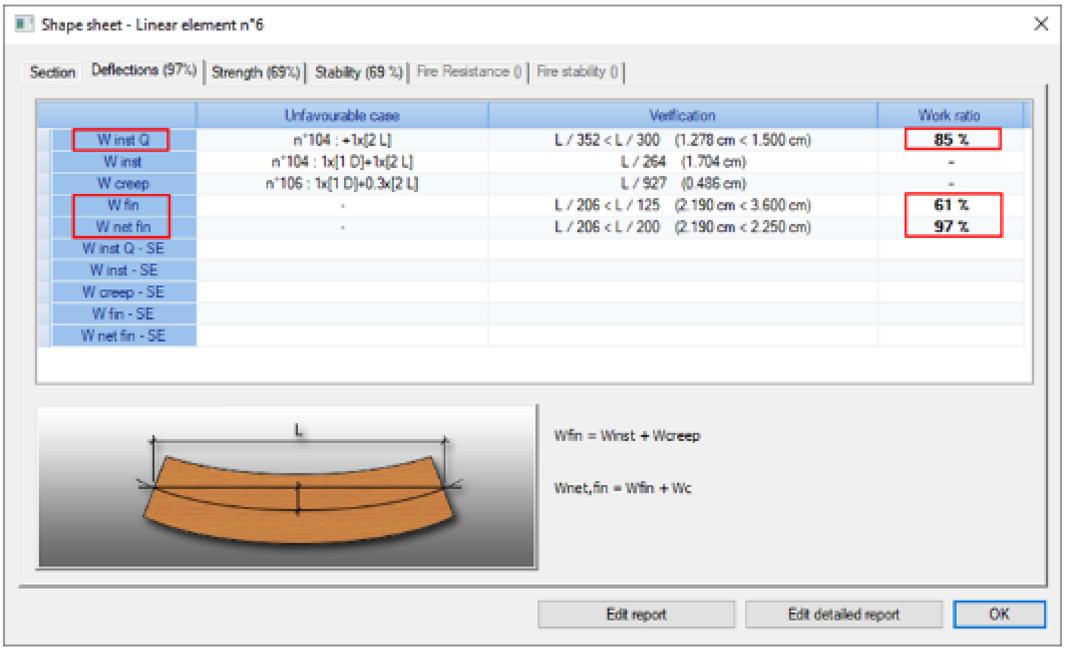
In this dialog, we can define the section size limits and the increments to apply on the base and height of the initial section.

The results of the optimization are given in the Suggested shapes dialog. The members for which the deflection is over the limit are displayed in red. A more appropriate, bigger section is then suggested.



Suggested shapes dialog

This feature ensures that none of the various deflection components exceeds the limit values defined in §7.2 from EN 1995-161 (or its national appendix).



Results of deflection verification for a timber element

6. Enhanced user experience and the comfort of program operation

A series of novelties and improvements related to user experience, resulting in increased efficiency and comfort of use.

6.1. User templates for Selection

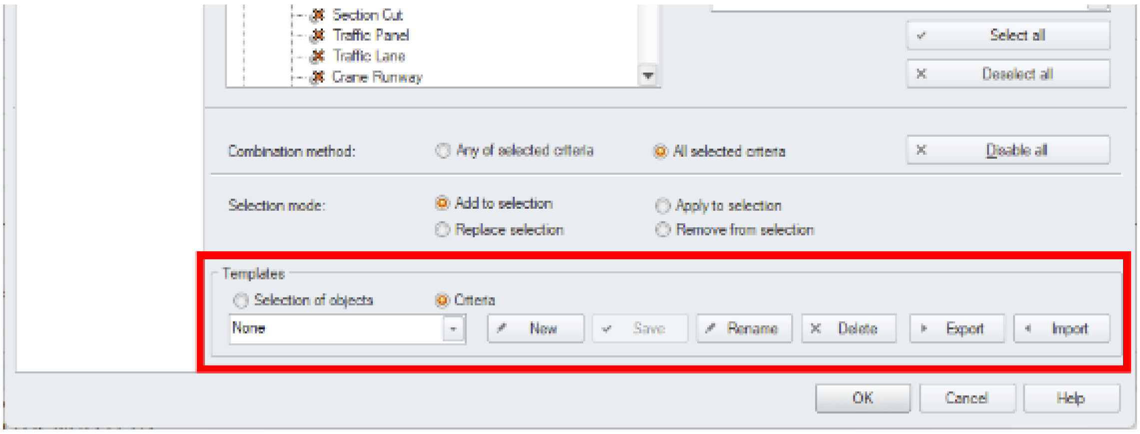
The ability to save and easily select templates with object selection.

The selection of objects is one of the more frequently performed operations. For this purpose, we have several tools available, including selection using the Project browser or using the window to define selection criteria. Starting with Advance Design 2024, we also can save and use previously saved object selections. This makes it easier to quickly select any set of objects at any time during work.

We can save a template with selections for two types:

- saving a selection of objects - that is, saving any selection (e.g., graphic) of any objects
- saving selection criteria - that is, saving of selection defined by criteria in the Element Selection window.

Templates can be added at any time when working with a structure. Creating a new template is done using the commands at the bottom of the Element Selection window. Using these commands, we can add, rename, and delete a template, and in the case of templates for criteria, we can also export and import templates to a file.



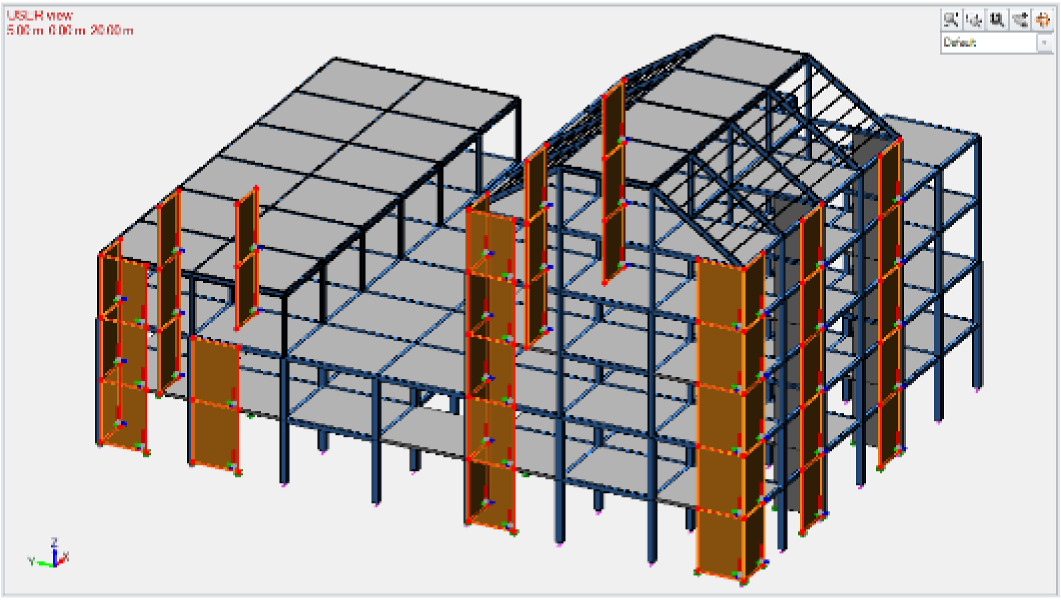
New options for selection templates on the Element Selection window

Templates for the selection of objects

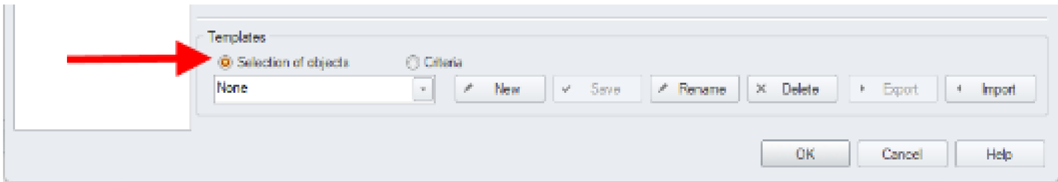
Object selection templates allow multiple quick selections of any object selection, regardless of how the selection was created and regardless of the type of object. For example, we can graphically select several different parts of a structure (walls, beams, supports, ...) and save this selection as a template. This type of template is based on saving the ID numbers of the elements.

The procedure for adding a template is as follows:

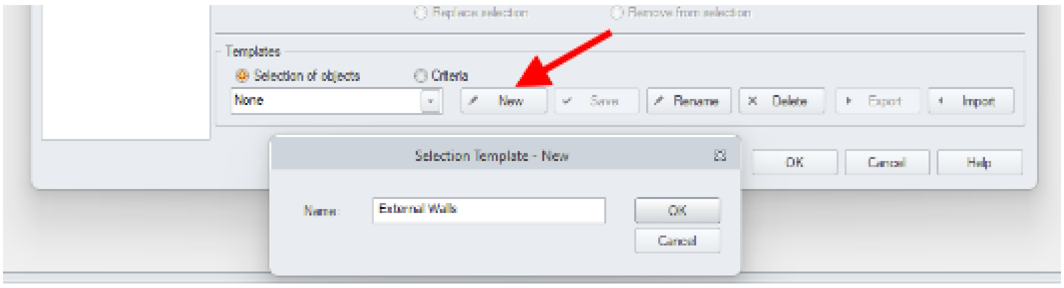
- To begin with, we create a selection. You can use any method (such as a graphical selection of elements) and it can contain various types of objects.



- Then open the Element Selection window (ALT+S) and set the template type to 'Selection of objects' at the bottom.



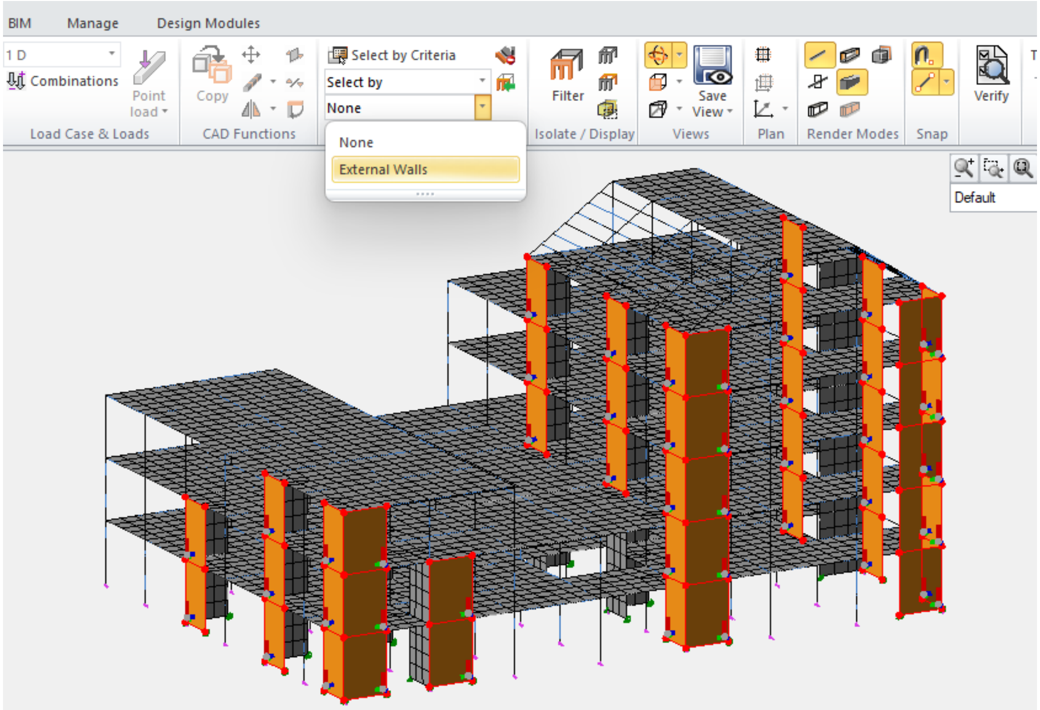
- Click on the New button and enter the name of the template



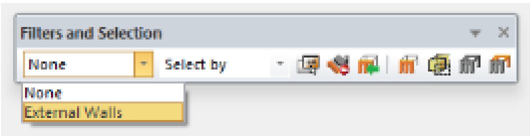
- The template is saved when the Save button is used or when the window is closed with the OK button.

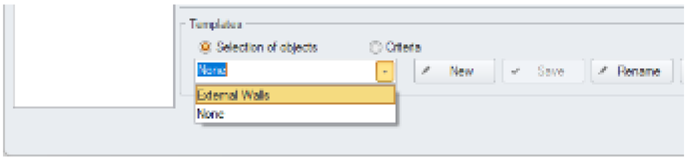
Selecting a selection from a template can be done in several ways.

The fastest way is to select a template from the list available directly on the ribbon. Once selected from the list, object selection is automatic. As presented in the image below, templates work in both descriptive and analytical models.



A list selection from a dedicated selection toolbar works on a similar basis, and you can also recall a selection using the Element selection window (ALT+S).





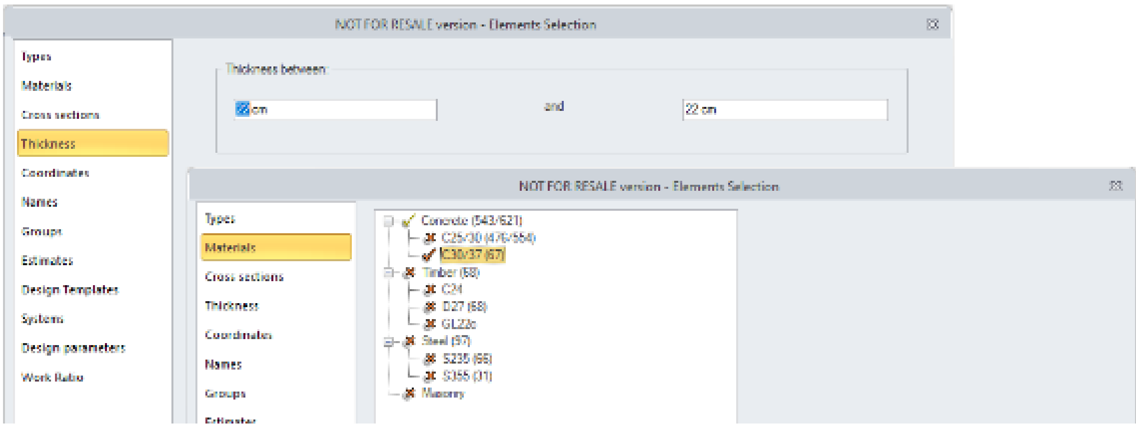
Since object selection templates are based on storing the ID numbers of selected elements, it should be noted that modifying the numbering of objects in the model may affect the selection effect of a previously saved template. Templates of this type are available only in the current project.

Templates for selection criteria

The selection template by saving criteria is used to save the current set of criteria set in the Element Selection window. This type of template is not based on ID numbers and thus is more versatile, although it requires a precise definition of criteria. Templates of this type can be used in the current project but can also be saved/exported to an external file and easily loaded and used in other projects.

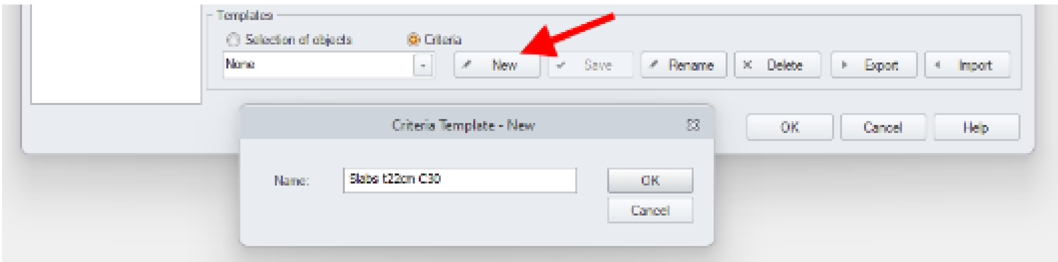
The procedure for adding a template is as follows:

- First, we open the Element Selection window (for example, using the ALT+S shortcut) and select criteria in the tabs.



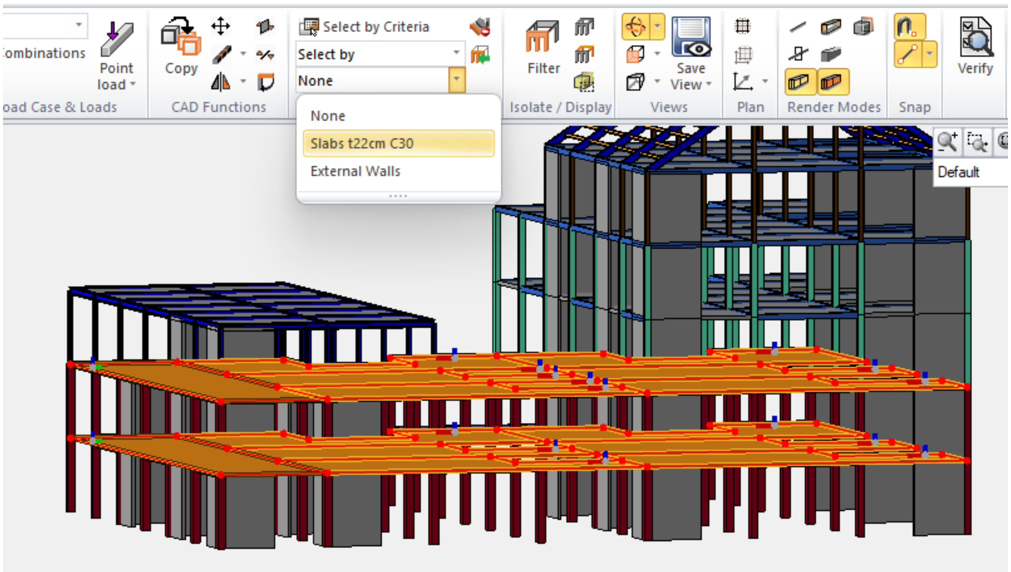
- We make sure that the template type is set to 'Criteria' at the bottom of the window (this is the default choice).

- Click on the New button and enter the name of the template.



- The template is saved when the Save button is used or when the window is closed with the OK button.

The application of the template is done in the same way as discussed earlier for object selection templates.

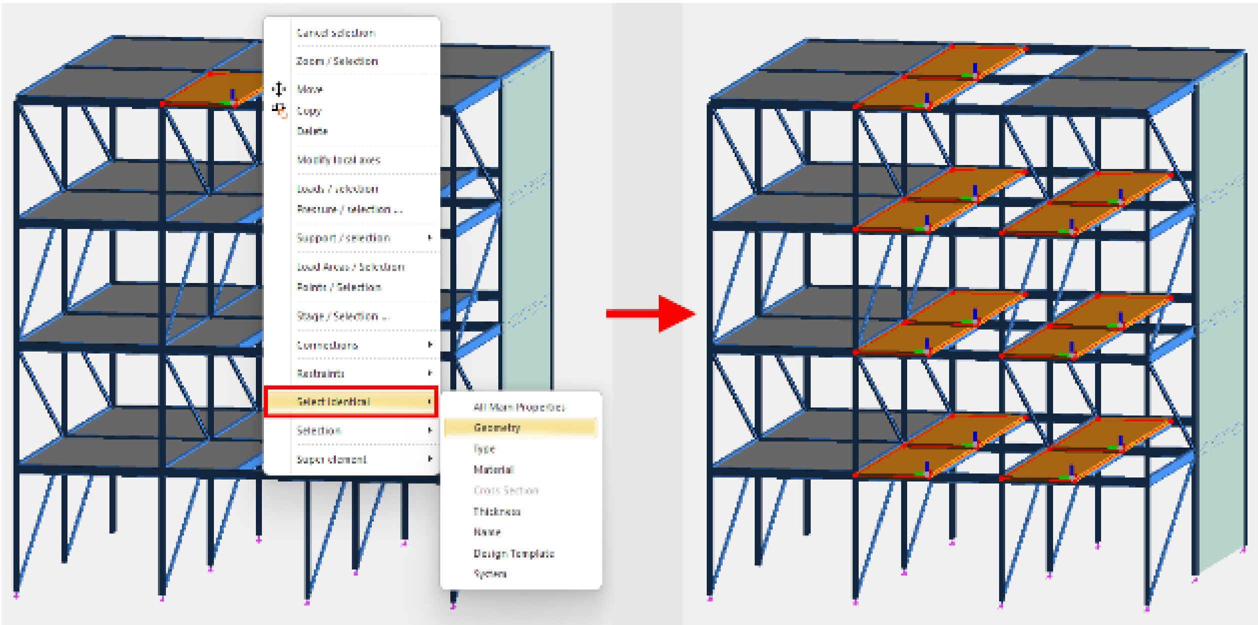


In the case of criteria selection templates, we additionally can save them to an external file. This can come in handy when we define several of our universal templates and want to use them in other projects. To do this, use the dedicated export and import buttons.

6.2. New options for selecting identical elements

Much faster and easier object selection.

To facilitate the graphic selection of elements, a new tool has been added for the quick selection of elements identical to the current selected one. To use the new function, select a single element (linear element / planar element/load area/support/load) and open the right-click context menu. The new 'Select identical' command contains several comparison criteria conveniently selectable from the menu, so you can quickly narrow down your search.



The following criteria are available:

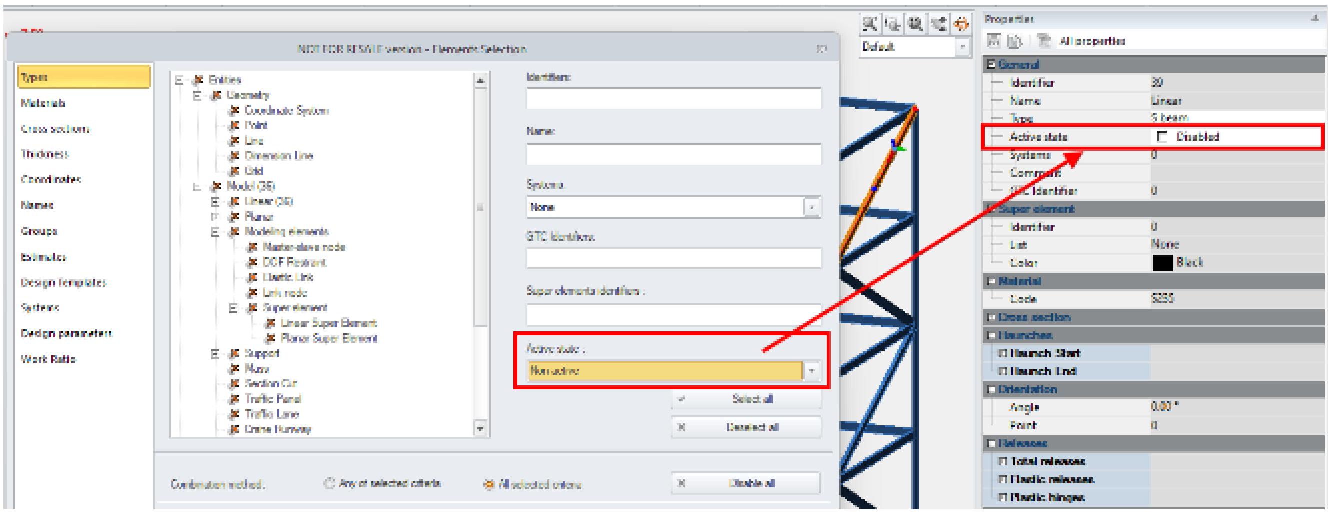
- **All main Properties** - selects elements with the same main properties. The list of main properties depends on the element type:
    - linear elements: geometry (length), material, and section
    - planar elements: geometry, thickness
    - load area: geometry
    - supports: category (Rigid Point, Elastic Point...) and the definition of restraints (values of TX, TY, ...).
    - loads: category (Point / Linear / Planar) and the definition (values of FX, FY, ...).
- **Geometry** - selects elements with the same geometric properties (for example the length for linear elements)
- **Type** - selects elements with the same Type property (like S beam or Tie for linear elements)
- **Material** - selects elements with the same material.
- **Cross Section** - selects elements with the same cross-section (linear elements)
- **Thickness** - selects elements with the same thickness (planar elements)
- **Name** - selects elements with the same Name property.
- **Design Template** - selects elements with the same general design template.
- **System** - selects elements from the same subsystem.

NOTE: *Depending on the selected element, some of the criteria may not be available. For example, the Thickness criterion is available for surface elements, but not for supports, loads, or linear elements.*

6.3. Possibility for filtering inactive elements

Faster search for inactive items.

To facilitate the search for elements that have been excluded from the generation of the calculation model (i.e., have the Active state property turned off), a new filter has been added to the Element Selection window. This filter allows you to quickly select elements whose state is active or inactive.



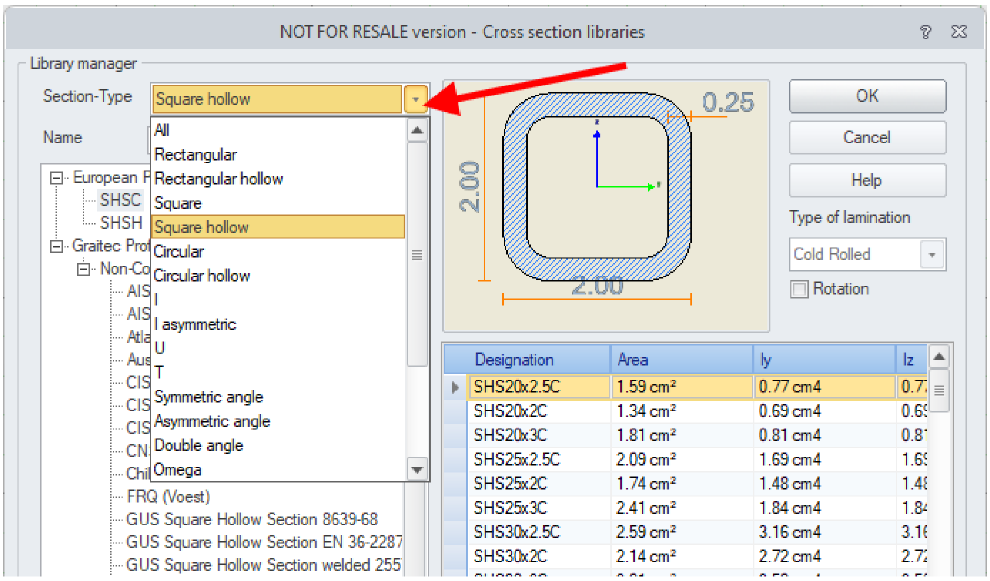
Faster search for inactive items

6.4. Filtering profiles from databases

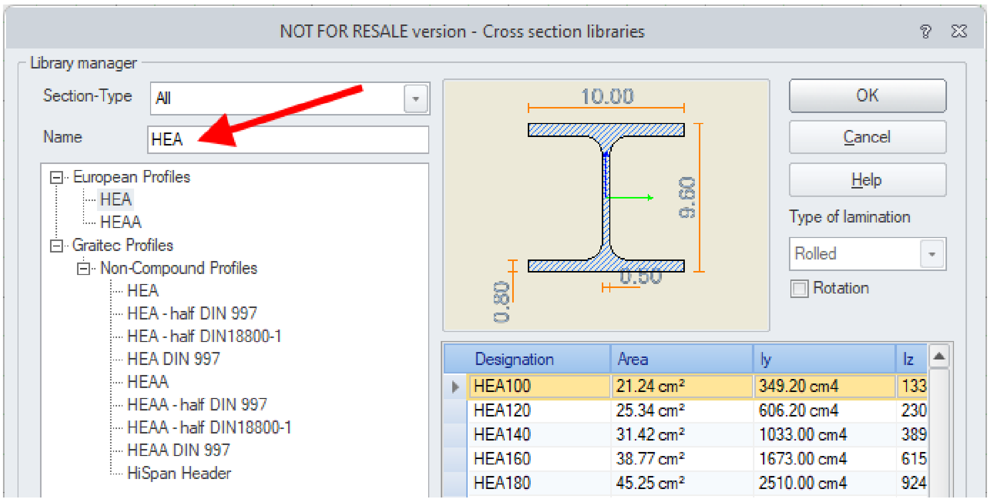
Faster profile search from databases.

To increase the speed and convenience of finding the cross sections you need, several improvements have been made to the window for selecting profiles from databases.

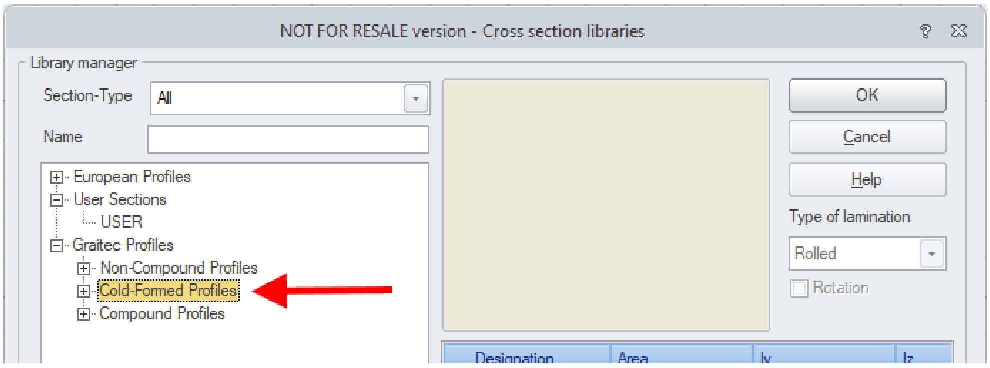
The first change is the addition of a filter by section type. The filter in the form of a drop-down list contains a set of available profile types, for example Rectangular, Rectangular hollow, U, C, T, Z, Zeta, etc. Selecting a type from the list automatically narrows down the list of the displayed profile libraries.



Another improvement is the addition of a text filter. The principle of operation is simple - you should enter any string of characters (it can be the beginning or any fragment of the library name) and press Enter. The list of available libraries will be automatically narrowed to those which contain the entered text in their names.



In addition, to make it easier to search for cold-formed steel profiles in the *Graitec Profiles* database, such profiles have been separated into a new group 'Cold-Formed Profiles'.



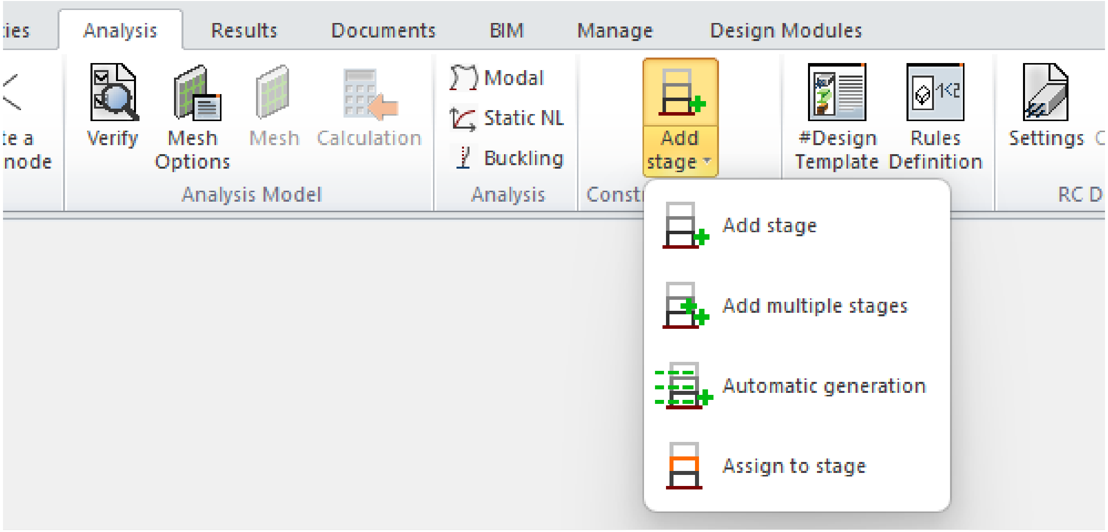
It should be noted that the cold-formed profiles in this group are profiles that contain a set of data required for verification calculations according to steel standards. However, in the *Graitec profiles* database, in the group 'Non-Compound Profiles' there are also other profiles which by their shape and name suggest that they are cold-formed profiles. However, these profiles cannot be verified according to the standard due to a lack of necessary data for verification calculations.

6.5. New commands on the ribbon for adding stages

Easier definition of construction stages.

To make it easier to define construction stages, four commands have been added to the Analysis ribbon:

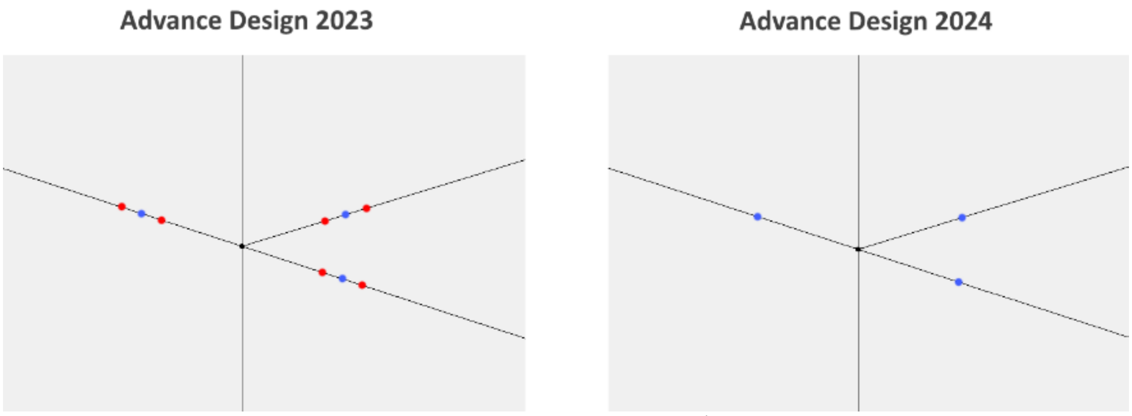
- **Add stage** – adds a single new construction stage.
- **Add multiple stages** – adds many new construction stages (based on the number entered)
- Automatic generation – automatically generates construction stages - one for each story.
- **Assign to stage** – calls a window to assign selected objects to stages.



6.6. Improved mesh generation for Pushover analysis

For the elements with plastic hinges, the finite element mesh is now regular, which significantly improves the mesh distribution in the adjacent surface elements.

The modeling of plastic hinges in line elements is enhanced in Advance Design 2024. Previously, to capture plastic rotations/translations of hinges, additional nodes were automatically generated on both sides of the plastic hinge. These nodes disturbed the regular meshing of structural elements, especially when linear elements were in contact with surface elements and increased the total number of Degrees of Freedom (DOF) in the structure. Better modeling of plastic hinges in Advance Design 2024 made it possible to capture plastic rotations/translations without the need for these additional nodes.

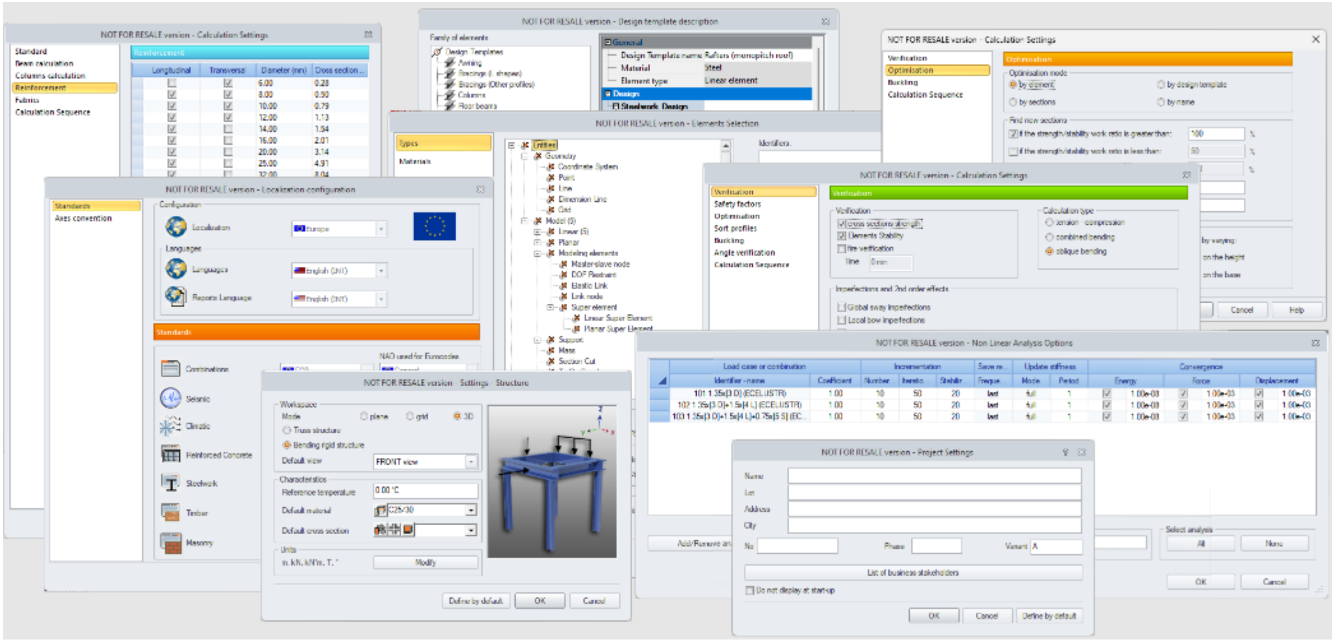


Example of plastic hinges modeling on beams (blue dots – plastic hinges, red dots – additional nodes)

6.7. The next stage of unification of dialog windows

More convenient operation in the program thanks to a clearer interface with a uniform window design.

In this version of Advance Design, another set of dialog windows has been updated, giving them a unified appearance. Although the layout and content of the windows have usually remained unchanged, the modifications concern the appearance and the components used. These changes have two main goals - to standardize the appearance of the content of windows to improve the user's perception during everyday work, as well as to modernize the program by switching to the use of newer technological components.



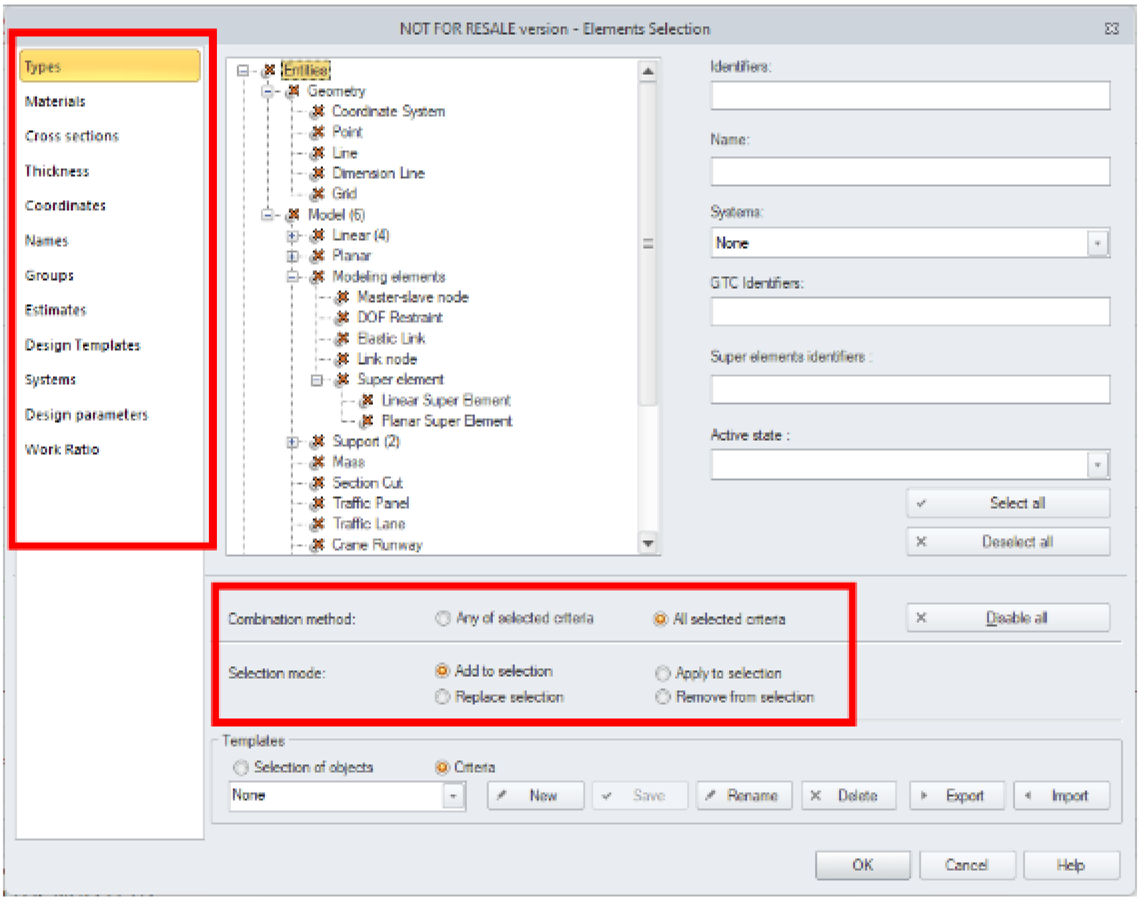
6.8. Improvements to the selection by criterion window

Improvements to the Element selection window for easier choosing selection methods.

The window for selection by criteria (opened from the ribbon or via the keyboard shortcut ALT+S) has been slightly expanded to expand the possibilities and make the work easier.

The first improvement is the change of the window content selection from tabs to a list. These speeds up the selection of criteria categories and eliminates the problem of scrolling through the tabs, which occurred especially with long names for some language versions.

The second change is new selection modes and a clearer definition of criteria combinations. For this purpose, several options have been added at the bottom of the window.



Improved window for selection by criteria

The first set of options is called the Combination method. There are two choices:

- **Any of the selected** criteria – it is a previous 'union' method. It selects all elements which correspond to at least one of the defined criteria. That is, for example, when you choose two criteria: Type=*Linear* elements and Material=*Concrete*, then with this method all linear elements (including non-concrete elements) and all concrete elements (including planar elements) will be selected.
- **All selected criteria** – it is a previous 'intersection' method. It selects only the elements which correspond to all defined criteria at the same time. That is, for example, when you choose two criteria: Type=*Linear* elements and Material=*Concrete*, then with this method only linear concrete elements will be selected.

The second set of options is called the Selection mode. These are used when the window for selection by criteria was called to create a new selection when any object selection already existed in the model.

There are four choices:

- **Add to selection** - adds new selection to the existing one. This is a default option.
- **Replace selection** - replaces the existing selection with a new one.
- **Remove from selection** - removes new selection from the current selection.
- **Apply to selection** - applies new criteria to the existing selection.

7. Other novelties and small improvements

Several various new features and small improvements introduced in the newest version of Advance Design.

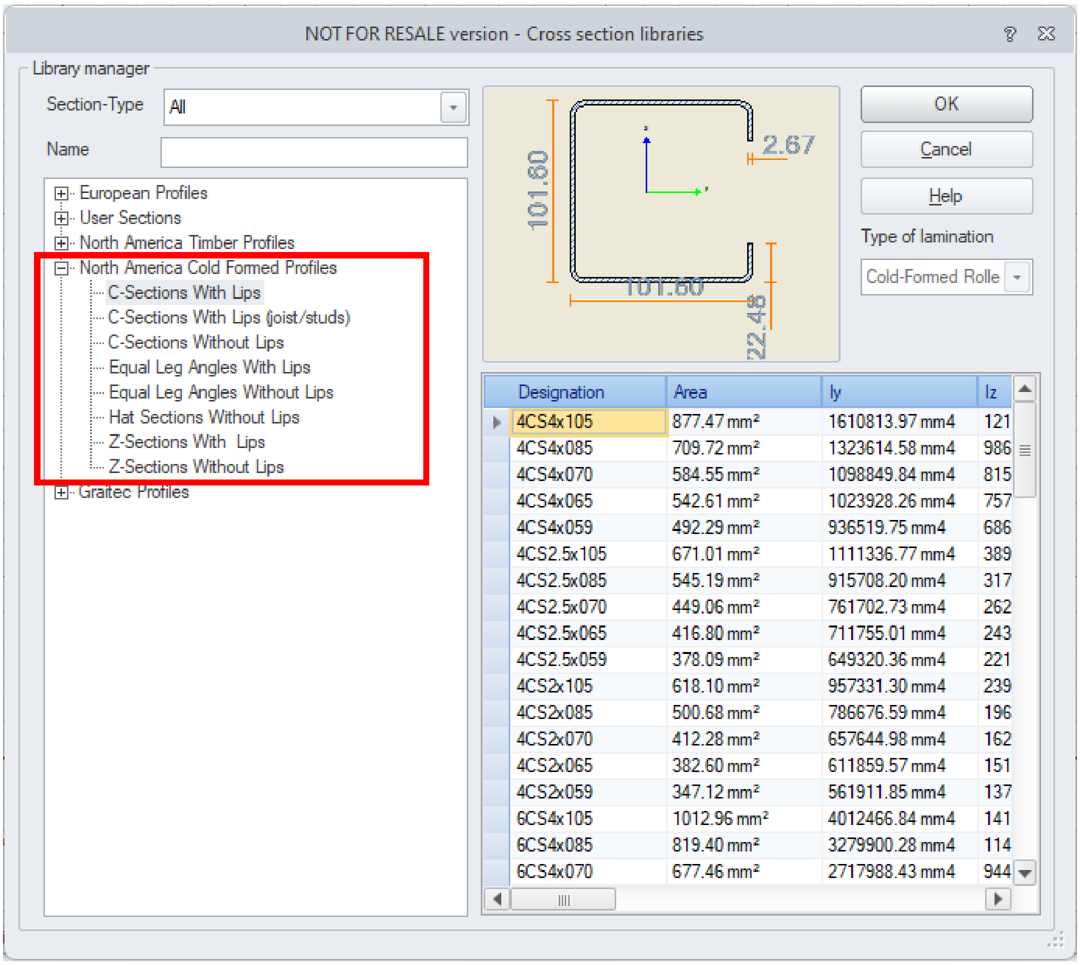
7.1. New database with cold-formed steel profiles for North America

Ability to model structures using cold-formed profiles typical of the North American market.

A new list has been added to the list of profile databases – **North America Cold-Formed Profiles**. It includes cold-formed steel profile data from tables I-1 to I-8 from the *AISI Cold-Formed Steel Design Manual (2017 edition)*.

There are 8 profile families available:

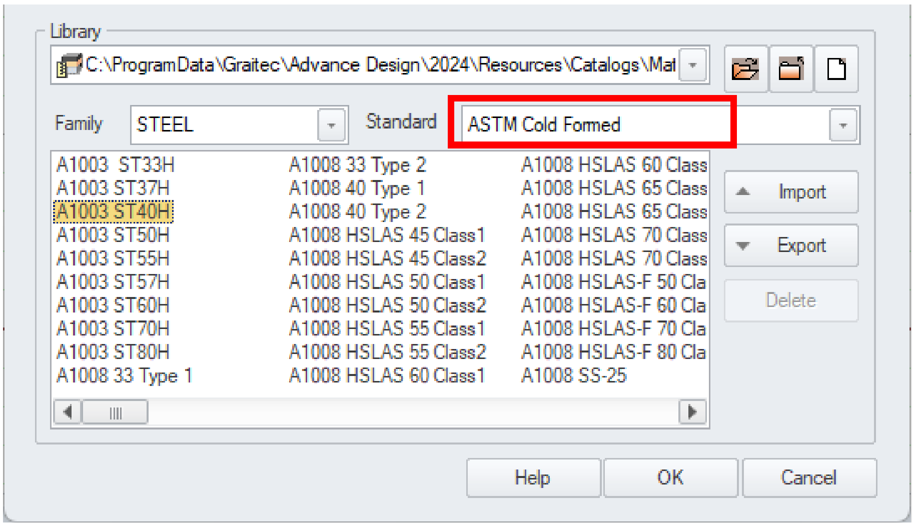
- C-Sections with Lips
- C-Sections with Lips (joist/studs)
- C-Sections without Lips
- Z-Sections with Lips
- Z-Sections without Lips
- Equal Leg Angles with Lips
- Equal Leg Angles without Lips
- Hat Sections without Lips



7.2. New steel material database for North American cold formed sections

Ability to model structures using steel materials for North American cold formed sections.

To the material library, a new standard has been added to the STEEL family list: *ASTM Cold Formed*. It contains new materials for use for North America cold-formed profiles. Each material has a specific application which is described in detail in Table 1.2 of the *AISI Cold-Formed Steel Design Manual*.

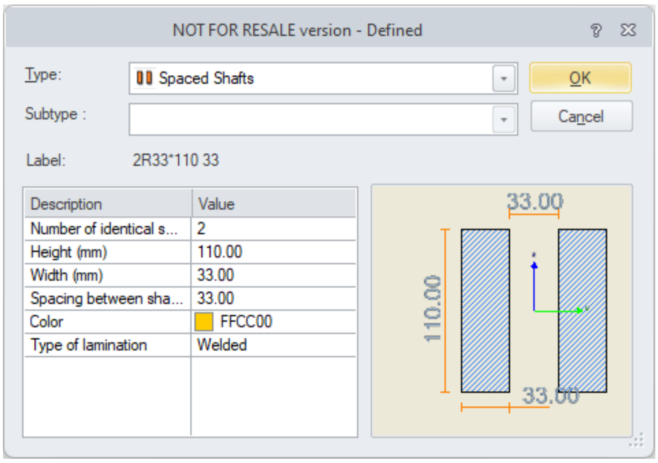


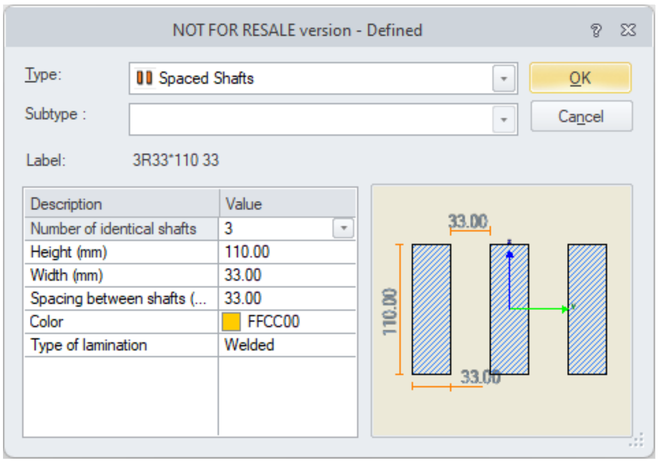
7.3. New parametric section – spaced shafts

Ability to easily define wooden compound profiles.

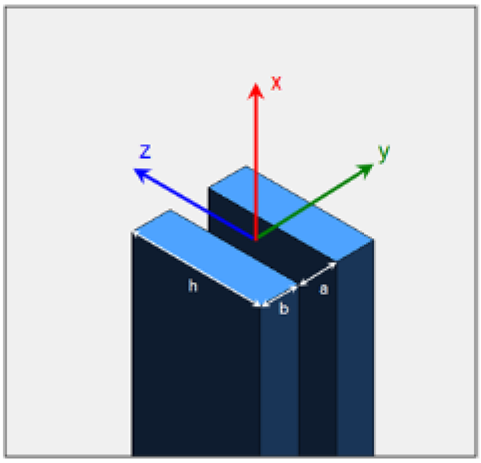
A new type - Spaced Shafts - has been added to the list of profile types that can be defined in a parametric way.

It allows the modeling of a profile consisting of 2, 3, or 4 identical rectangular sections, set in parallel at the same spacing. Profiles with two elements are most often used to model wooden roof elements, while 2, 3, and 4 element configurations are usually used to model wooden columns.

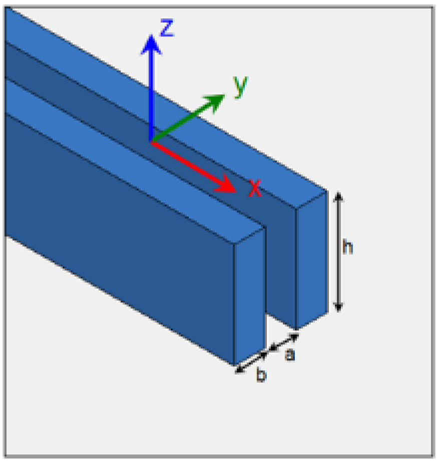




The axis convention used in Advance Design is with the local z axis parallel to shaft height (h).



Axes convention for spaced columns



Axes convention for spaced beams

The number of shafts and spacing between them have an impact on the section parameters of the member:

Area for n shafts:

$$A_{tot} = n \cdot A = n \cdot b \cdot h$$

In-plane inertia for n shafts:

$$I_{y,tot} = n \cdot I_y = n \cdot \frac{b \cdot h^3}{12}$$

Out-of-plane inertia

for **2 shafts:**

$$I_{z,tot} = \frac{h \cdot [(2 \cdot b + a)^3 - a^3]}{12}$$

for **3 shafts:**

$$I_{z,tot} = \frac{h \cdot [(3 \cdot b + 2 \cdot a)^3 - (b + 2 \cdot a)^3 + b^3]}{12}$$

for **4 shafts:**

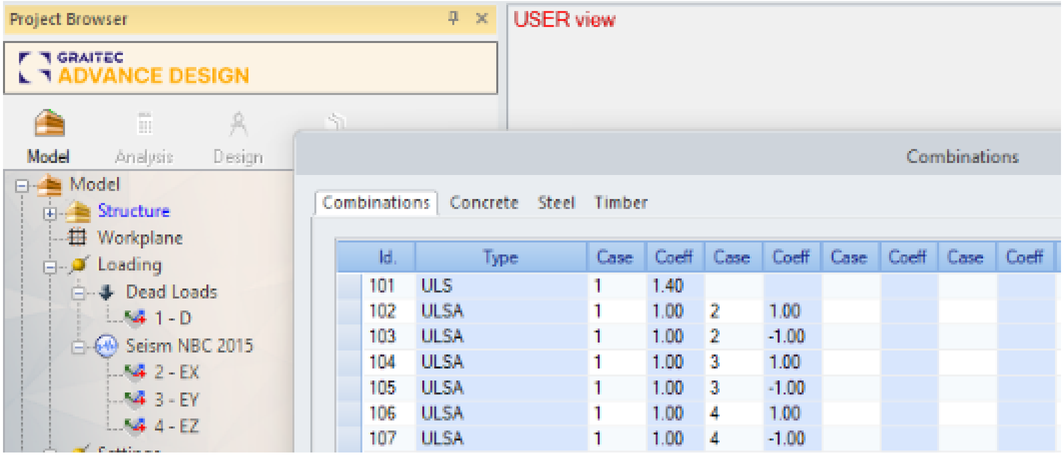
$$I_{z,tot} = \frac{h \cdot [(4 \cdot b + 3 \cdot a)^3 - (2 \cdot b + 3 \cdot a)^3 + (2 \cdot b + a)^3 - a^3]}{12}$$

7.4. Improved combination creation with seismic cases for Canada

Automatic consideration of the occurrence of opposite directions of seismic forces for Canada.

During the automatic generation of combinations with seismic cases according to the Canadian NBC standard, seismic combinations are now considering both directions for forces from seismic load cases (EX, EX, EY). Now seismic cases are considered in combination once with a positive sign and once with a negative sign.

This prevents the problem of force cancellation when we have concomitant loads on the structure, for example, soil pressure in one direction and seismic force in the opposite direction.



7.5. Defining Link at node on selection

Better control over modeling with a precise selection of the elements to be linked.

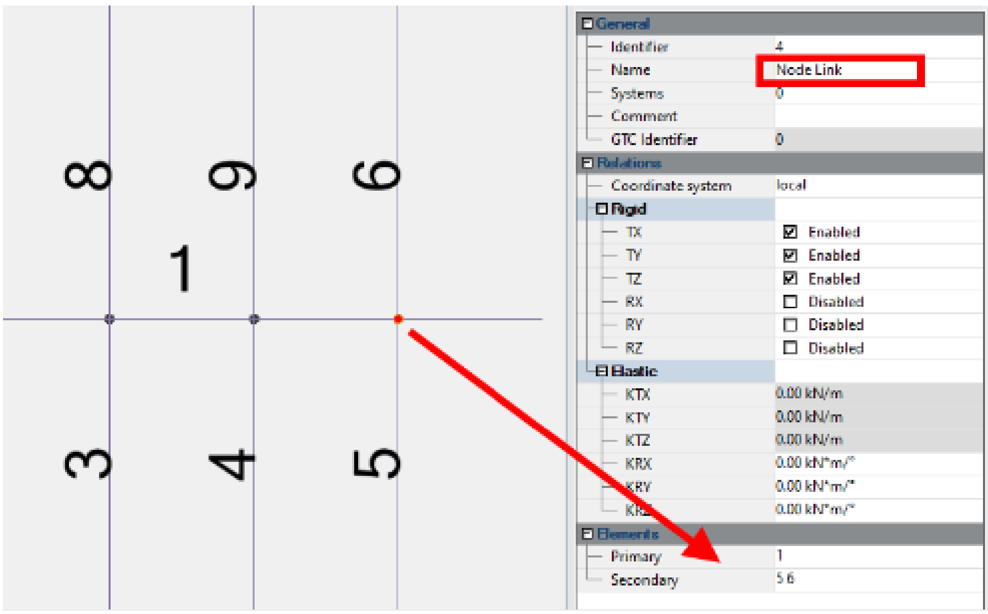
An additional method of quickly defining multiple links at node at the same time has been introduced. Now, after calling the command 'Link at Node' in the command line, you can choose to define it by selecting elements (by pressing x on the keyboard).

```
Command Line
Press p to input a point or press x to select elements >
```

You can then graphically indicate the main and secondary elements separately. You can include multiple main elements and multiple secondary ones.

```
Command Line
 > Select primary elements and press enter >  1
Command Line
 > Select secondary elements and press enter > 5 4 2 3 7 8 9 6
```

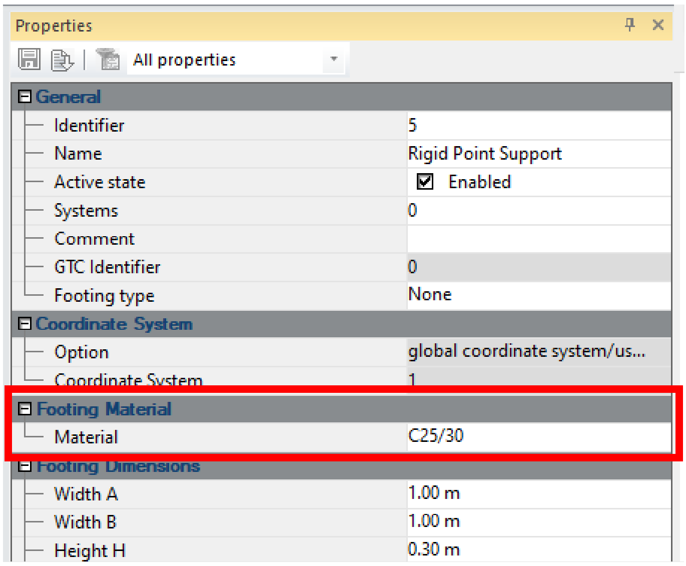
This allows for a precise and clear definition of links.



7.6. Possibility of material definition for supports

Ability to set the type of concrete for the foundation in the 3d model.

The option to assign a material (concrete type) to supports has been introduced. It is available in the list of properties of all types of punctual and linear supports.

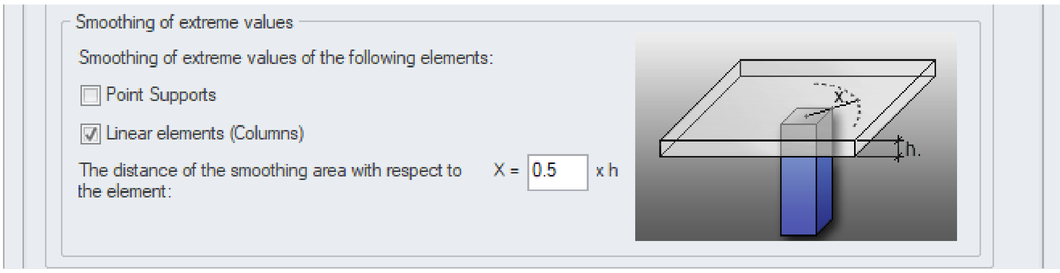


Assigning material to a foundation currently has two uses. First, together with information on foundation dimensions, it allows cost and CO2 emissions estimation. Secondly, it is used when exchanging information between the Advance Design model and the RC Footing design module.

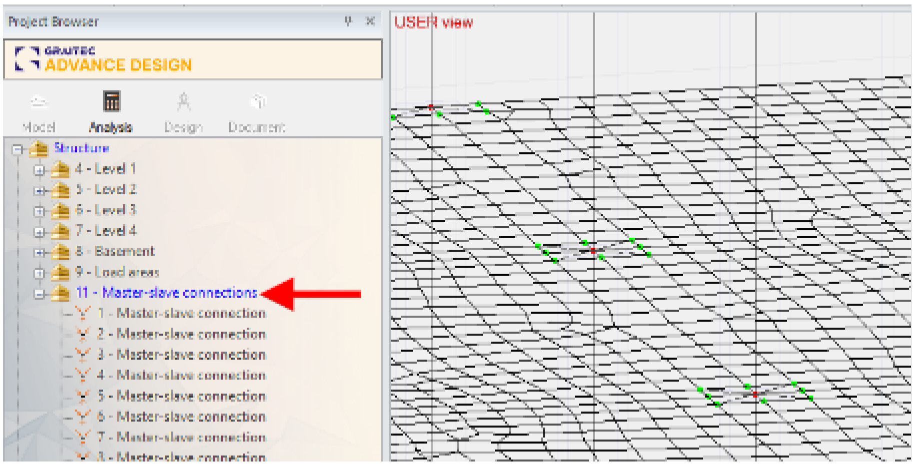
7.7. Dedicated system for master-slave links

Better organization of system content by grouping automatically generated master-slave connections.

One of the existing configurations of the program is the ability to automatically generate master-slave connections in the slab, to avoid the effect of point concentration of forces over a column or support.



Such connections are generated automatically when preparing the calculation model, and from this version are also automatically placed in a dedicated system in the Project browser window. This increases order and clarity of the project structure.

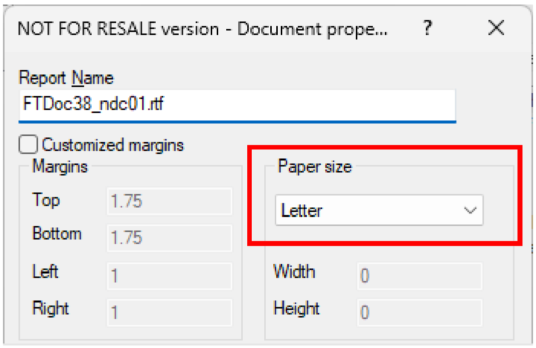


7.8. Improvements to the program defaults for North America

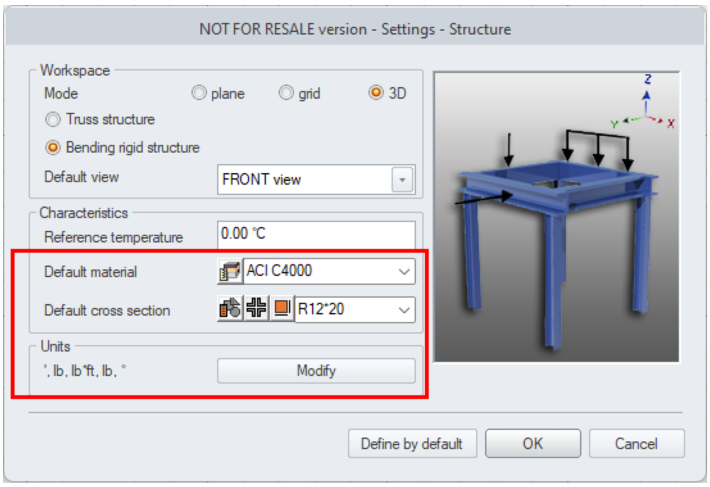
Better customization of some of the program defaults when selecting the localization of the US or Canada.

To make things easier for users who have chosen US or Canada as their localization in the program configuration, a few minor changes have been made to the default settings.

The first change concerns printing parameters. Now the default paper format for printing for the US and Canada is Letter format instead of A4.



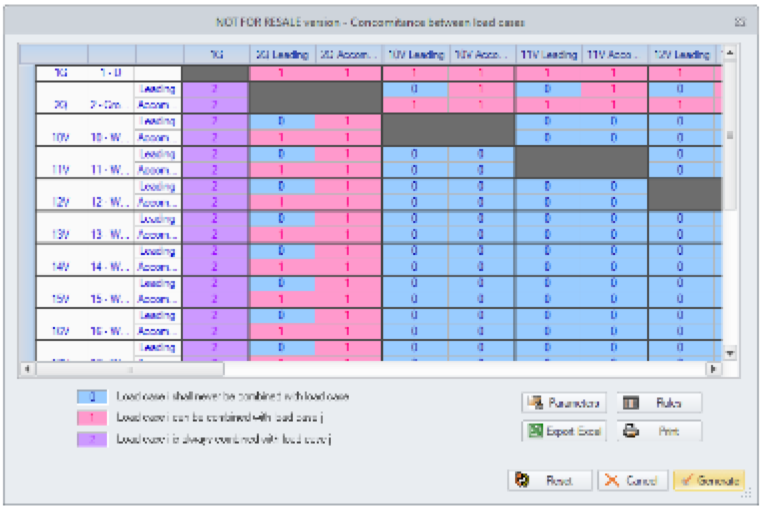
Another change concerns the default design settings. For both countries, the default material has been changed, and for the US, additionally, the default section and units of the project.



7.9. Shorter time of generation of load combination

Faster generation of load combinations.

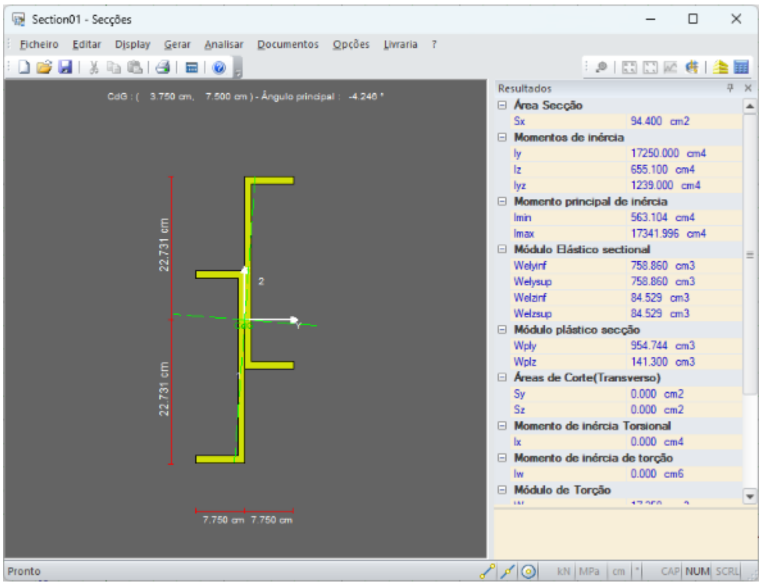
The algorithm for generating load combinations using the concomitance matrix has been improved so that for large tasks the combinations generation time has been reduced.



7.10. Improvements in localization for Spain and Portugal

Full interface translations in all modules are provided for Spain and Portugal.

The translations for Spanish and Portuguese languages have been improved and expanded on missing areas (like the Section editor).



Section Editor

8. Design Modules - Common improvements

New features and improvements common to design modules.

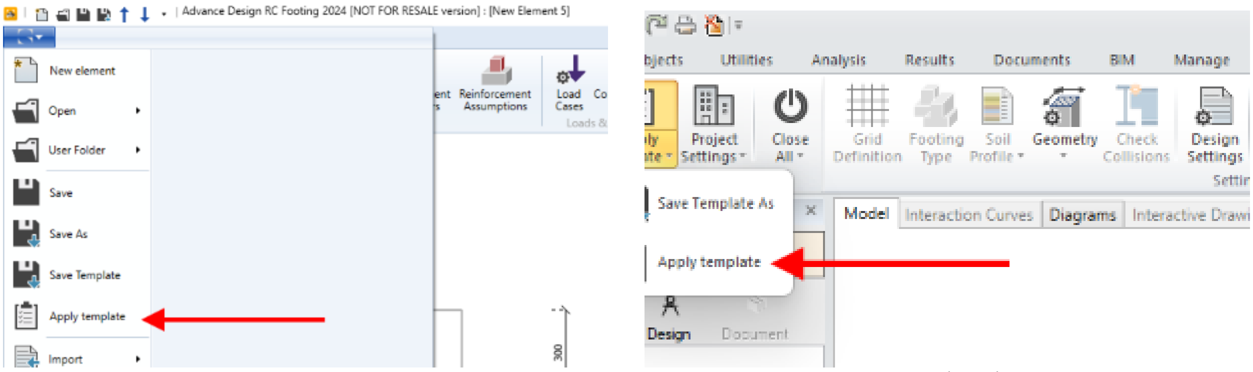
8.1. Allow applying project templates when the module is already opened

Ability to update settings from a template for a currently open project.

One of the ways to speed up and streamline your work while designing the next element is to use project templates with our predefined settings. Project templates can be selected when creating a new project for an element (for standalone versions of design modules) or before exporting/opening an element in a design module in the Advance Design environment.

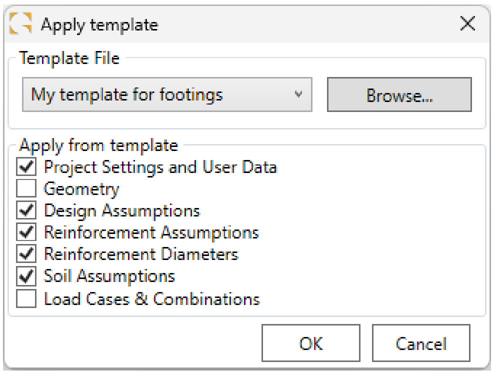
Starting with the newest version, it is possible to apply a project template to an already opened element. That is, at any time during the design, for example, after opening or importing an element from Advance Design, we can use a new command to apply a template.

When working in the standalone version of the modules, the 'Apply template' command is available from the menu. When working in the Advance Design environment, the new command is available on the ribbon.



Location of the new command in the standalone version (left) and on the ribbon in Advance Design (right)

Similarly, when starting a new project with a template, when assigning a template to an open project, we can also select the scope we want to overwrite from the template.



Selecting the scope of application of the template for RC Footing module

This functionality is available for RC Beam, RC Column, RC Wall, RC Footing, RC Slab, and Masonry Wall modules.

8.2. Display the steel ratio in the Info Panel

Quickly available information on steel ratio.

One of the basic qualitative results regarding the reinforcement of concrete elements is the reinforcement steel ratio. Until now this result has been presented only in the report and in the drawings. With the latest version of the program, this value is visible immediately after the calculation and also in the info panel window, making it possible to check quickly.

In all RC modules (Beam, Column, Foundation, Wall, and Slab), the global steel ratio value for the entire element is visible at the bottom of the Info Panel.

Verifications	Top x	Top value	Top WR	Bottom x	Bottom value
σ_crq Concrete	3500 mm	2,8 MPa	11,38%	0 mm	0,5 MPa
σ_crq Steel	0 mm	48,2 MPa	12,06%	3500 mm	157,5 MPa
σ_qp Concrete	3500 mm	2,8 MPa	11,38%	0 mm	0,5 MPa

Calculation results | Errors and warnings — Steel ratio = 51,7 kg/m³

In addition, for multi-span beams, the corresponding steel ratio value is presented in the header separately for each span.

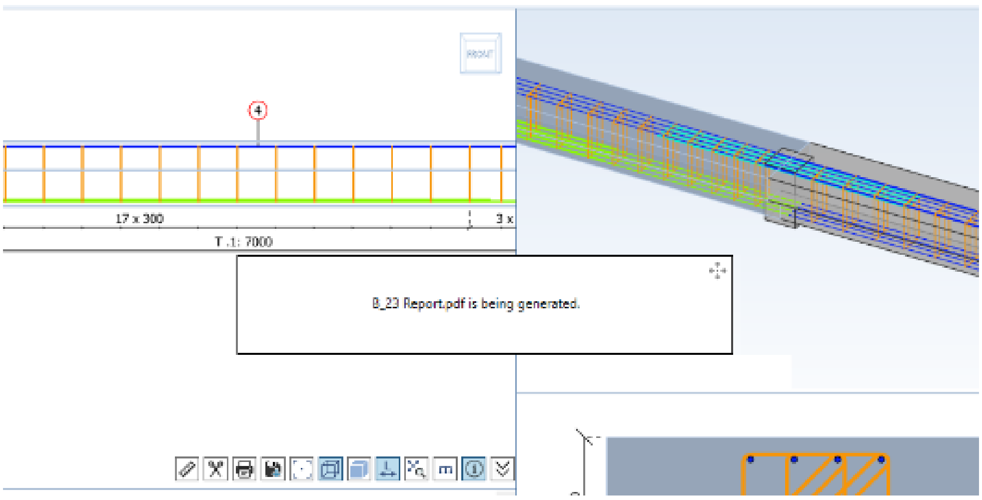
	T .1 (Steel Ratio = 108,4 kg/m³)										
Reinforcement	x	Design effort	Resist. effort	Comb.	Reinf. min.	Reinf. theo.	Reinf. real	Work Ratio	x	Desigr
Longitudinal top left	0 mm	-28 kN·m	-35,5 kN·m	104	178 mm²	178 mm²	180 mm²	78,94%	0 mm	-96,2
Longitudinal top right	7000 mm	-92,6 kN·m	-123 kN·m	104	178 mm²	508 mm²	653 mm²	75,26%	7000 mm	-17 k
Longitudinal bottom	3010 mm	112 kN·m	119,6 kN·m	104	178 mm²	596 mm²	628 mm²	93,66%	4200 mm	68,1

8.3. Small improvements

A set of smaller improvements affecting many design modules.

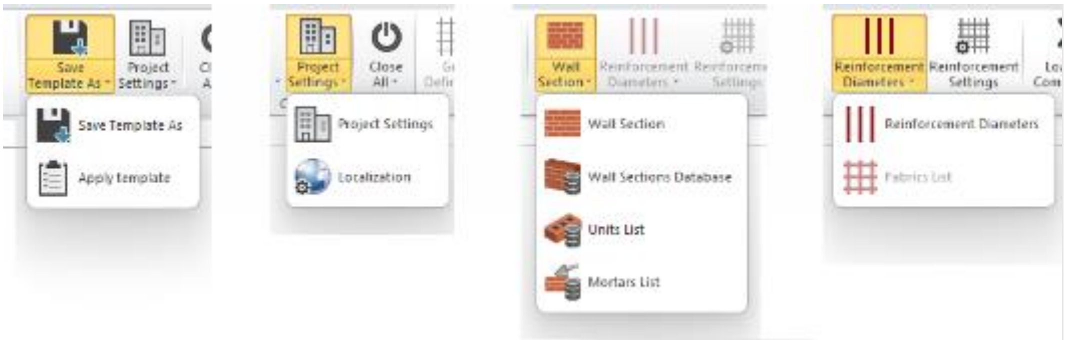
- **Better notification of report generation**

In order not to miss the information about the generation of the report, the small window indicating the generation of the report is now always visible on top. In addition, it can be easily moved to a new location.



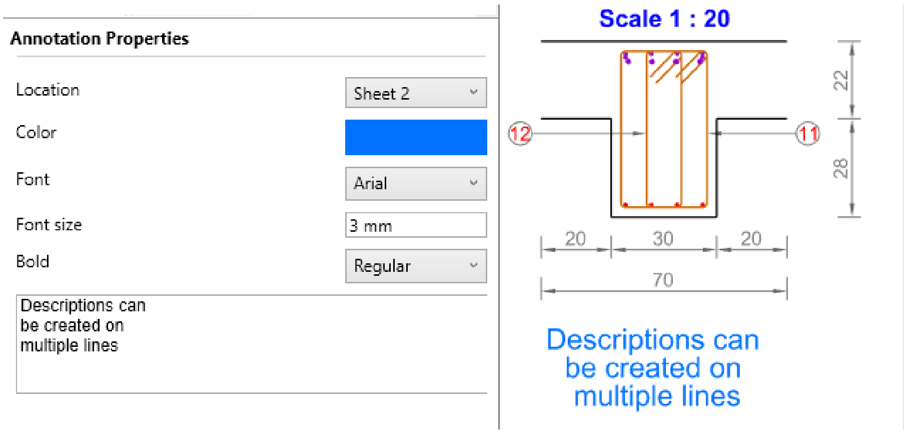
- **Grouping icons on a dedicated ribbon in Advance Design**

On the Design Modules ribbon in the Advance Design environment, some of the commands have been grouped to make it easier to find the right commands, especially when the ribbon is short for smaller screen resolutions.



- **Multiline descriptions on drawings**

When adding custom annotations on drawings, it is now possible to enter text with multiple lines (using the Enter key).



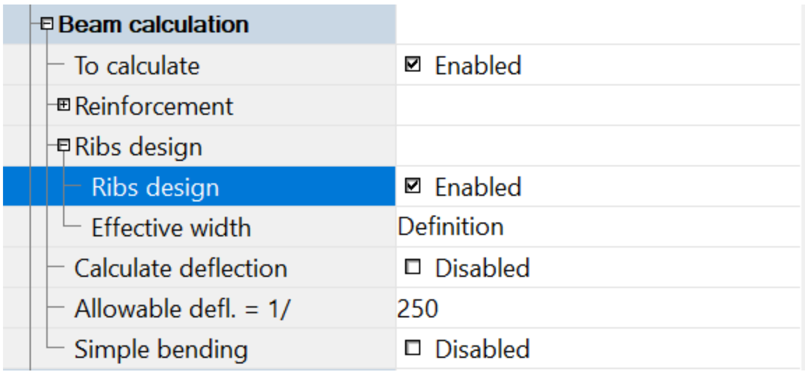
9. RC Beam

New features and improvements implemented in the latest version of the RC Beam module.

9.1. Import geometry and internal forces from Rib design

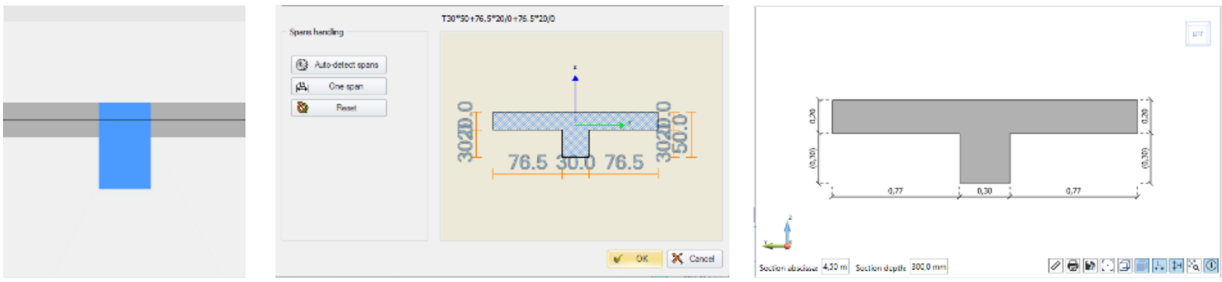
Consideration of effective section and design internal forces when importing beams with rib option active.

One of the options available for reinforced concrete beams supporting a concrete slab is the ability to consider them as ribs. Activation of this option causes that during the design analysis of a reinforced concrete element, the interaction of the beam with the slab is considered, and the design efforts are not just the forces occurring only in the rectangular beam part but are the combinations envelop of FEM (Finite Element Method) internal forces occurring on the entire monolithic T or L section.



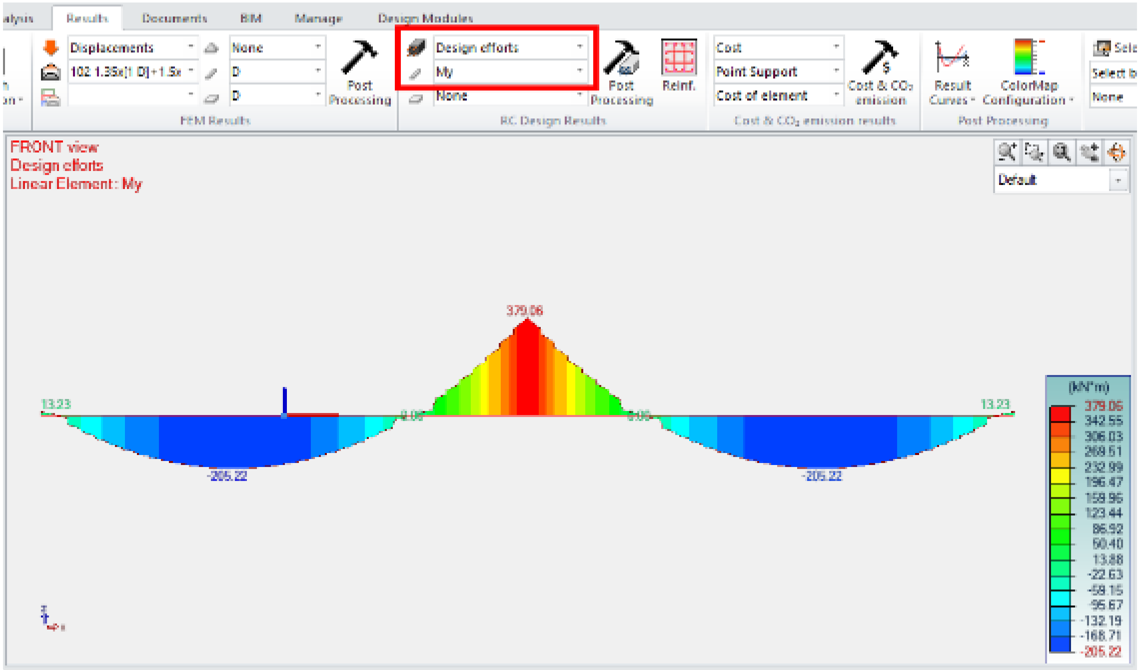
Ribs design option for reinforced concrete beam elements on Advance Design

With Advance Design 2024, when a rib beam is exported from Advance Design to RC Beam module, instead of a rectangular beam section, the equivalent monolithic T or L section is imported.

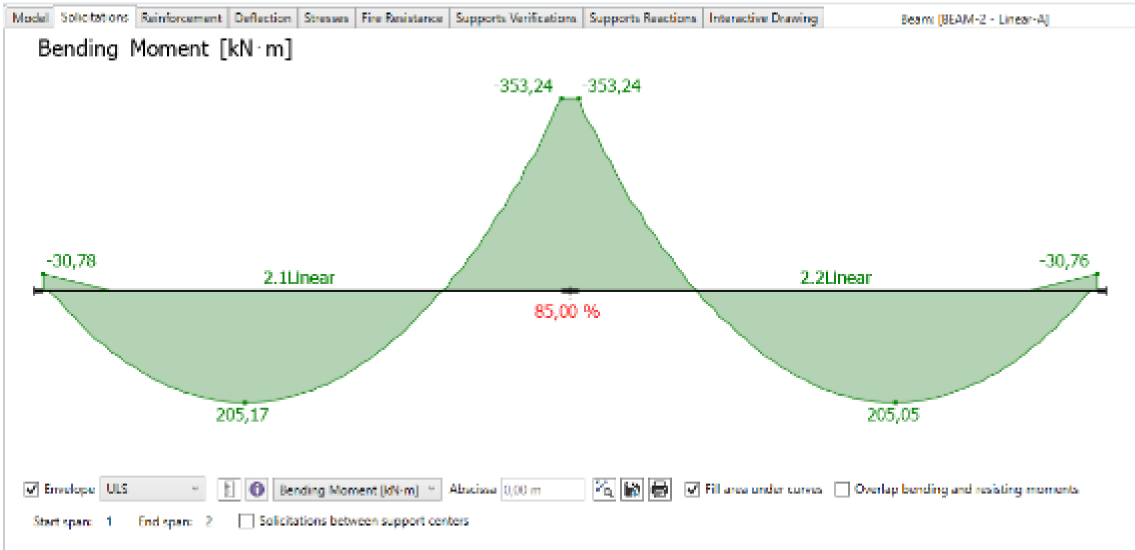


Initial rectangular section (left), effective section on Advance Design (middle), and imported section (right)

Also, instead of the FE internal forces from a rectangular beam, the equivalent integrated design internal forces are imported.



Design bending moment for the entire monolithic rib beam in Advance Design



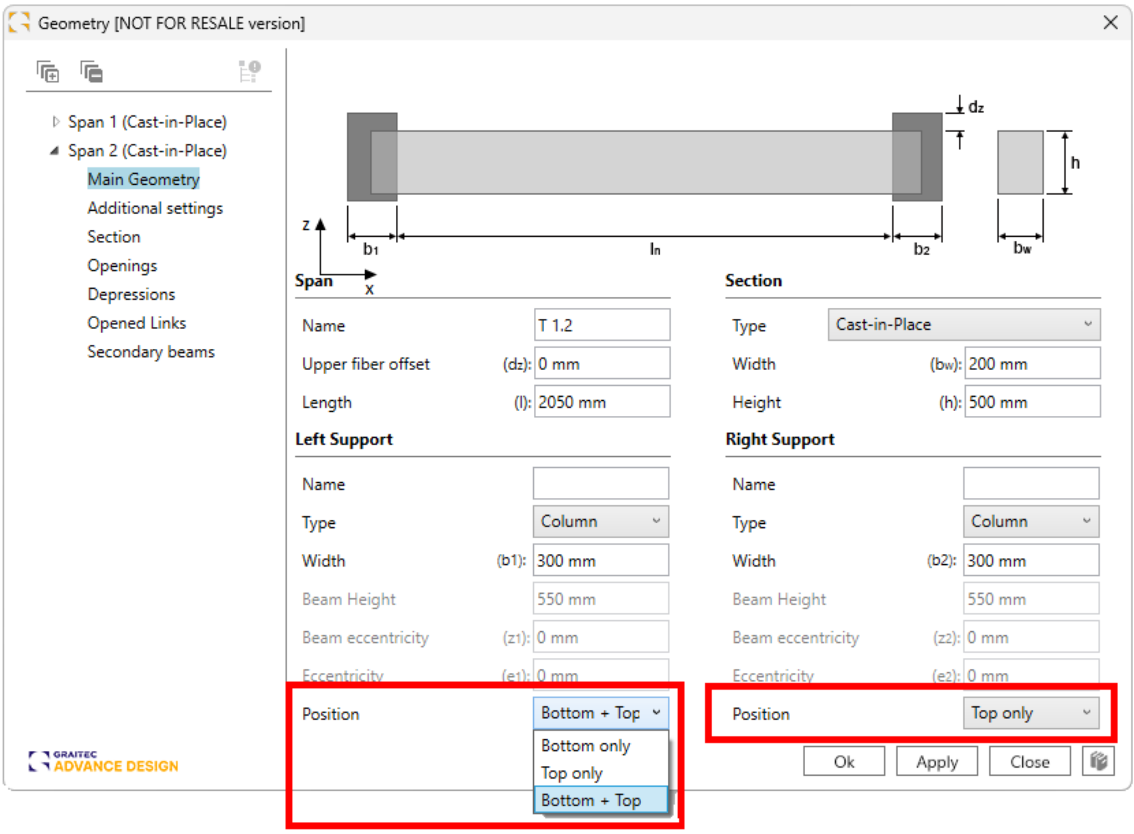
Design bending moment for rib beam imported to RC Beam module

9.2. Possibility to define the position of supports (columns/ walls)

Better quality of drawings due to showing the real support position.

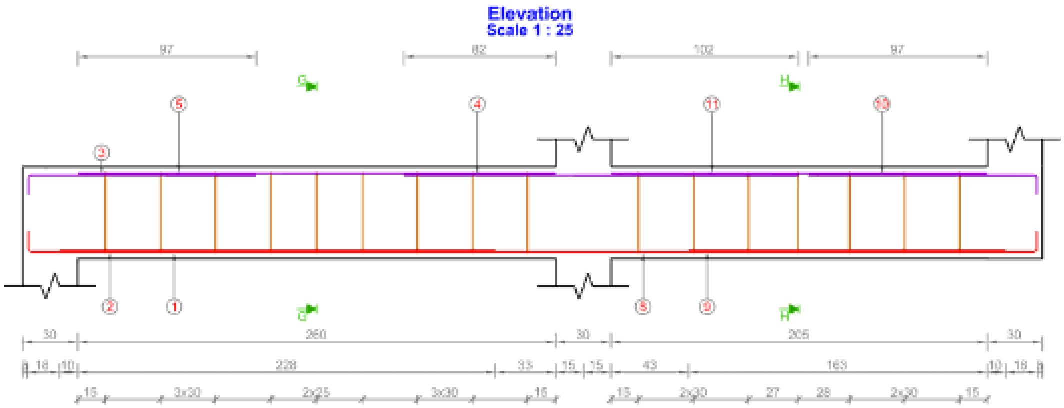
Supports (columns and walls) now distinguish whether they support a beam from below, above, or from both sides.

For this, an additional option is available on the 'Geometry' tab, where we can set the position independently for each support. This data is now also imported from the Advance Design model.



New options for the supports position on the Geometry dialog

The main purpose of this new possibility is to get correct drawings, where we can see the position of support (wall or column).

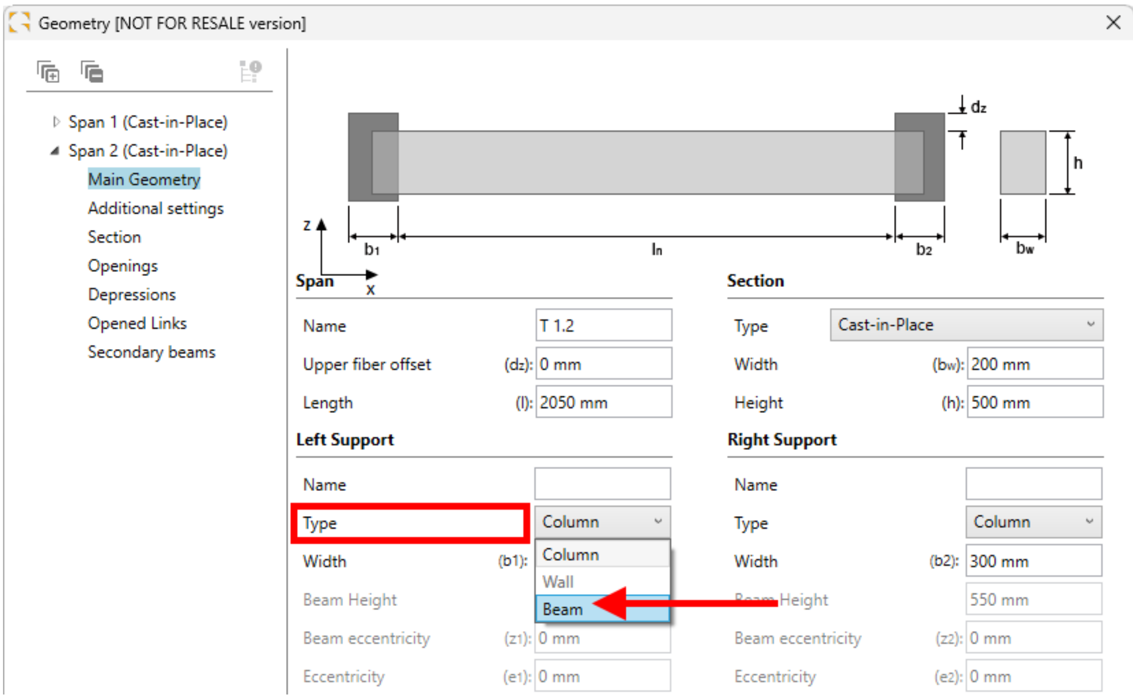


Example of a drawing with an elevation of a two-span beam with different support configurations

9.3. New support type "beam"

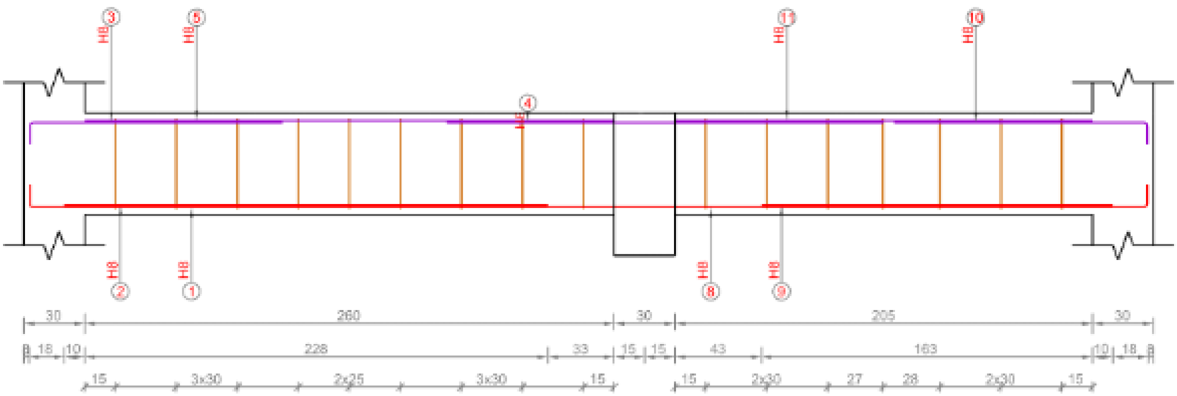
Possibility for generating correct drawings when the support is another beam.

With the latest version of the RC Beam module, it is possible to define as support another beam (perpendicular beam that acts as a support). For this, on the 'Geometry' dialog, the list of support types is expanded with a new entry - *Beam*. This data can be edited manually, but also, when importing data from Advance Design, beams supporting an element (beams of higher height) are recognized as supports of beam type.



New entry on the support type list

This enables the correct generation of drawings when the support is another beam.



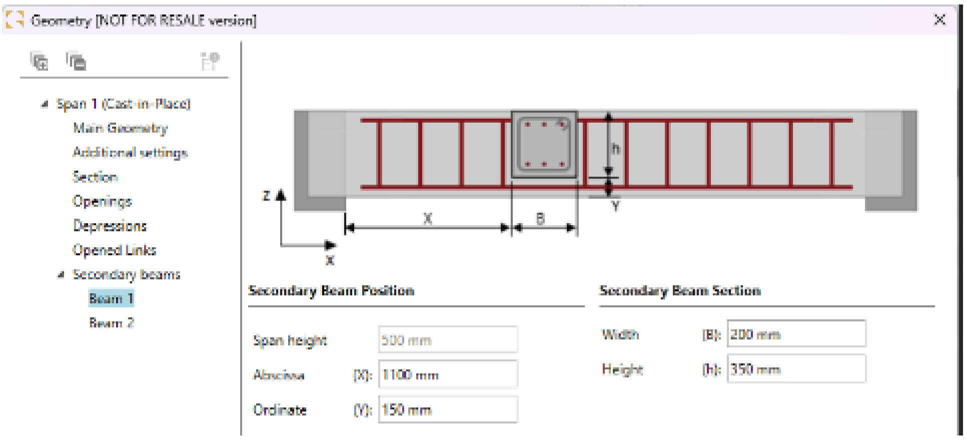
Drawing for a beam elevation with the center support modeled as a perpendicular beam

9.4. Ability to define secondary beams

Possibility for generating better drawings by considering the position of the secondary beams approaching perpendicularly.

In the latest version of the module, secondary perpendicular beams can be defined. Such beams have a lower height than the analyzed beam and are not treated as a support.

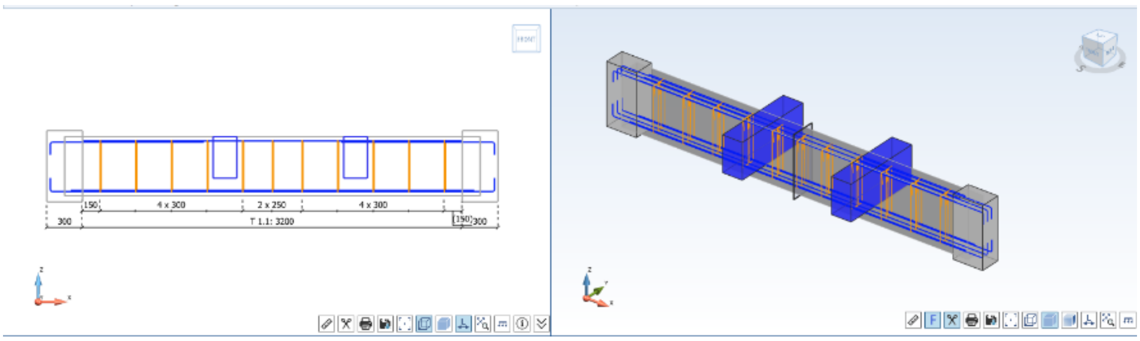
Secondary beams are added using a new tab in the Geometry window.



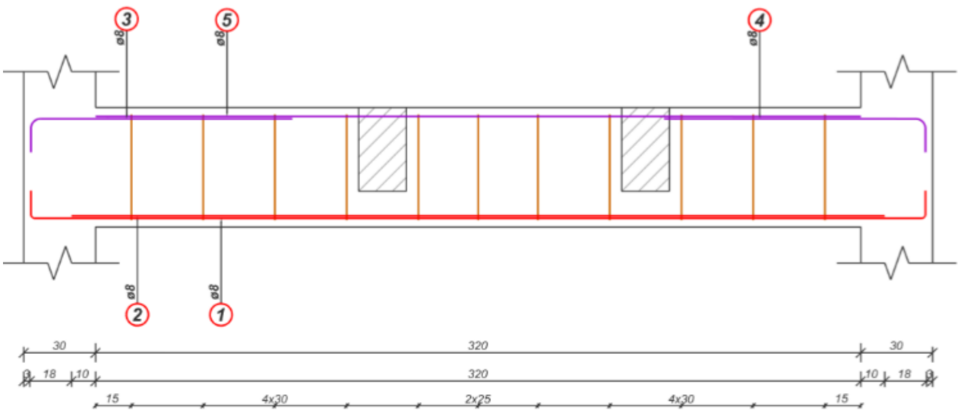
New tab for defining secondary beams

NOTE: *When importing data from the Advance Design model, secondary beams are automatically detected and added to the list of secondary beams.*

Secondary beams can be seen on the views and reinforcement drawings.



Secondary beams on views

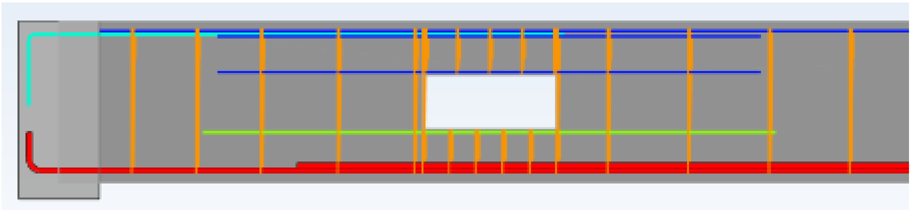


Secondary beams on a drawing

9.5. A new chapter in the report on openings reinforcement

Easier verification of required reinforcement for openings.

In the detailed report for beams, if there are openings in the beam, there is now a new chapter on the reinforcement of openings.



View of the beam with opening reinforcement

16 Openings reinforcement (required reinforcement)

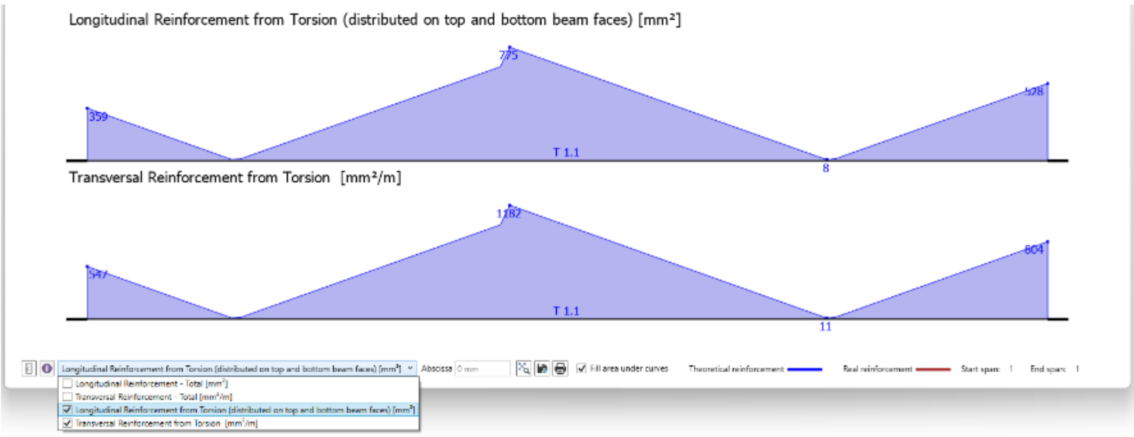
Openings Reinforcement								
Span	Opening	Lintel	A_t (mm²/m)	T_u (MPa)	A_{top} (mm²)	A_{bot} (mm²)	$A_{l,L}$ (mm²)	$A_{l,R}$ (mm²)
1	1	Top	792	1.1	53	208	186	167
		Bottom	743	1.2	156	0		

New table in report for opening reinforcement

9.6. New diagrams for reinforcement for torsion

Easier verification of reinforcement for torsion.

In the diagrams with information on the theoretical area of reinforcement now we can display two new diagrams – for the longitudinal reinforcement from torsion (distributed on top and bottom beam faces or distributed on all beam faces) and for the transversal reinforcement from torsion.

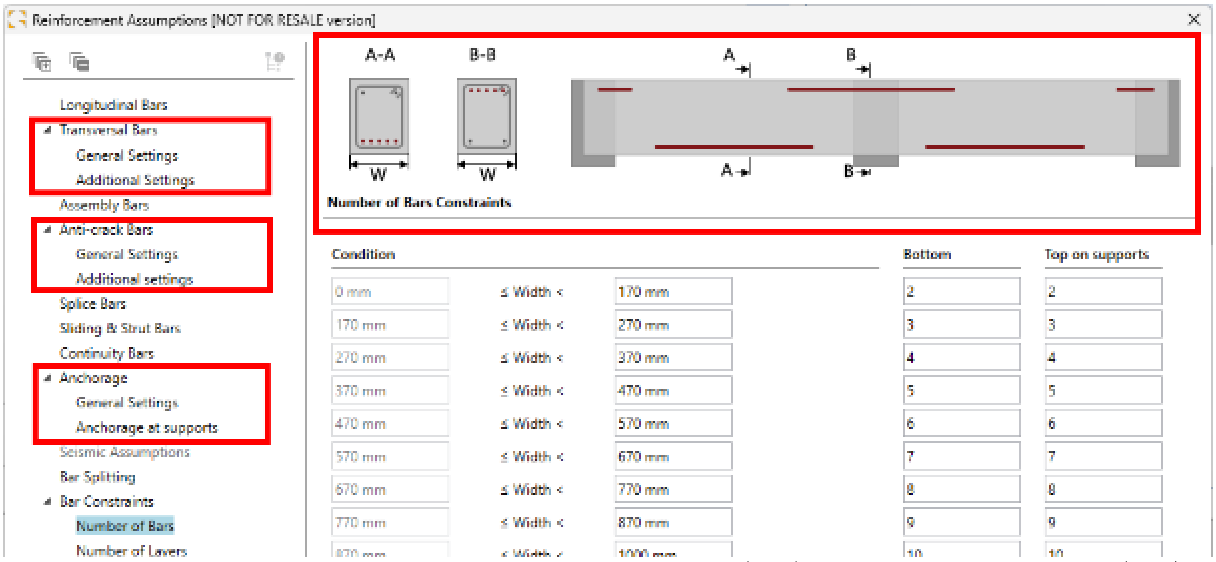


9.7. Small improvements

Improvements to Reinforcement assumptions dialog

For improving the user experience, two types of changes are applied to the Reinforcement assumption dialog – splitting of selected tabs and providing more pictures with explanations.

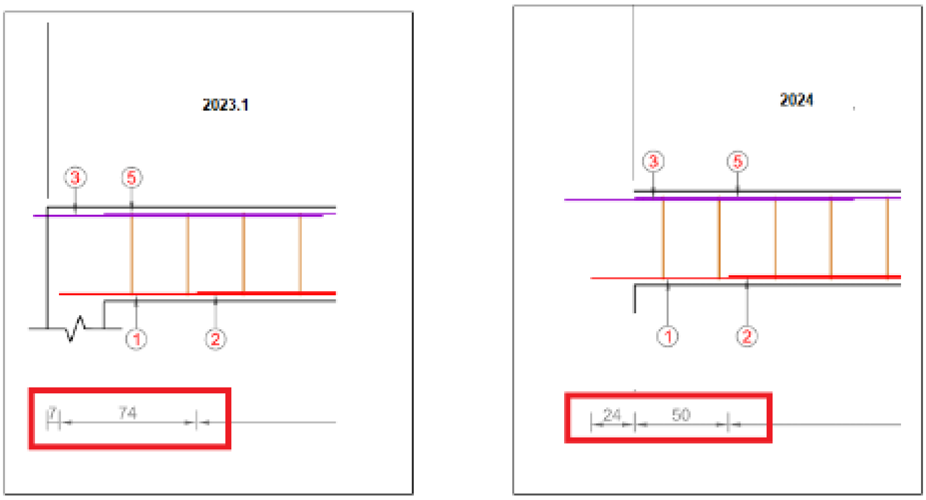
Splitting tabs was aimed at better grouping options to find them more quickly, as well as reducing the need to scroll through the contents of the window. These changes concern three tabs: Transversal bars, Anti-crack bars, and Anchorage. New explanatory pictures have appeared at the top of some tabs as well as in some selection lists.



The Reinforcement Assumption dialog with split tabs (left) and one of new picture (top)

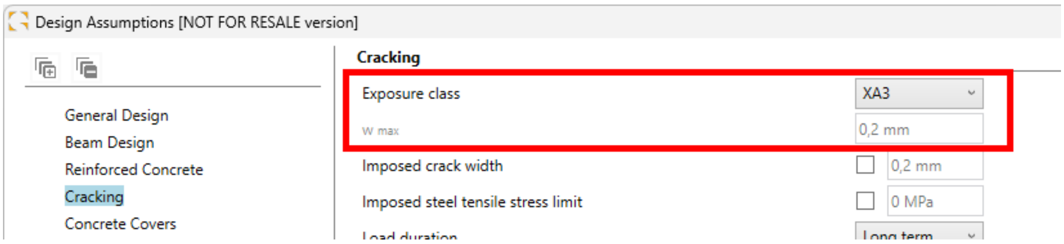
Drawing – improvement to side dimension lines for reinforcement

Dimension lines for bottom beam bars are now referenced from the inner edge of the support, making it easier to place bars accurately when the beam is supported by walls.



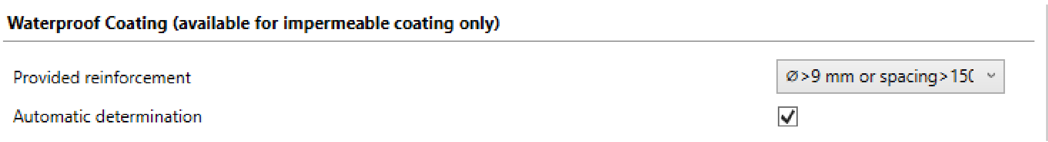
Modification of the default Wmax value for France

When the French national annex to Eurocode 2 is selected, then the max crack value (Wmax) for class XA3 is now set to 0.2 mm. The change was done according to the recommendation from the FD P 18-717 (published in August 2021).

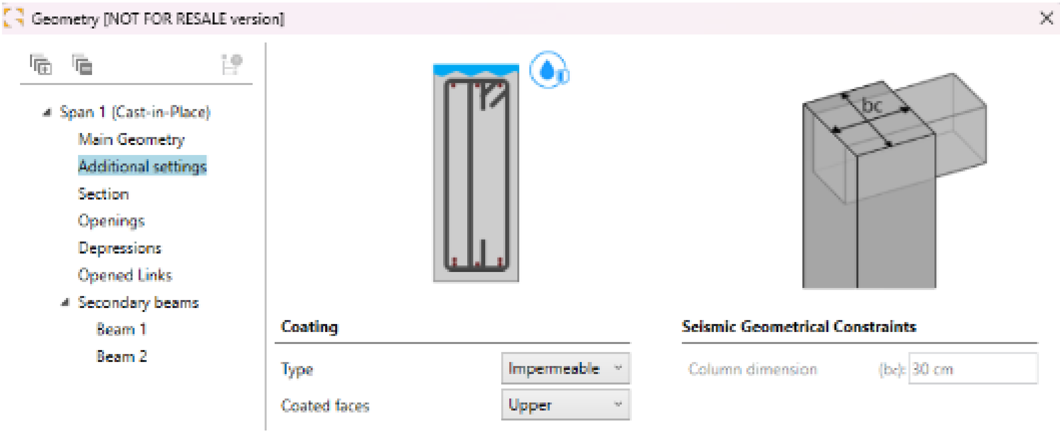


Adjustment of waterproofing data entry for France

To increase the comfort and clarity of the data, several minor modifications have been made related to the input of waterproofing data (available on the *Longitudinal Bars* tab of the **Reinforcement Assumptions** dialog and a new *Additional settings* tab on the **Geometry** dialog).



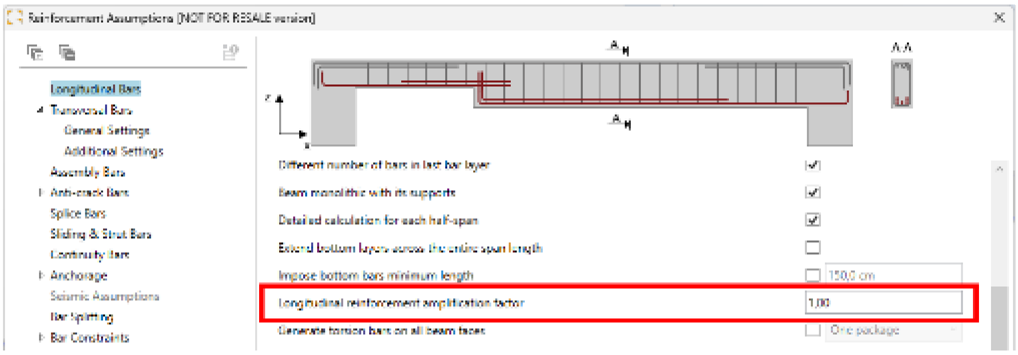
Settings on the Reinforcement Assumptions dialog



Settings on the Geometry dialog

The additional coefficient for modification of longitudinal reinforcement

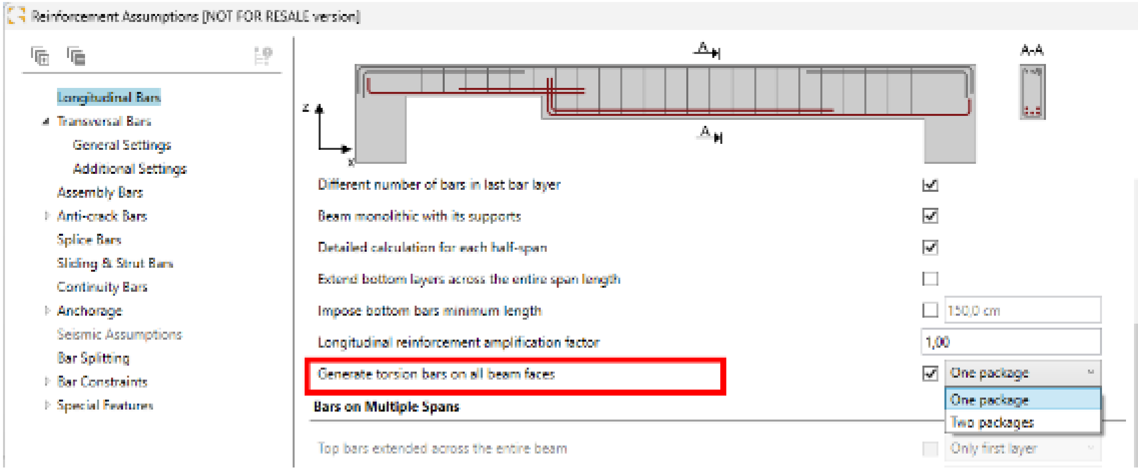
A new coefficient has been added to the beam longitudinal reinforcement settings to easily modify the required theoretical area of longitudinal reinforcement. The amplification factor is applied only to the theoretical reinforcement area resulting from the loads in the vertical plane (not for torsion). It applies to both bottom and top theoretical reinforcement.



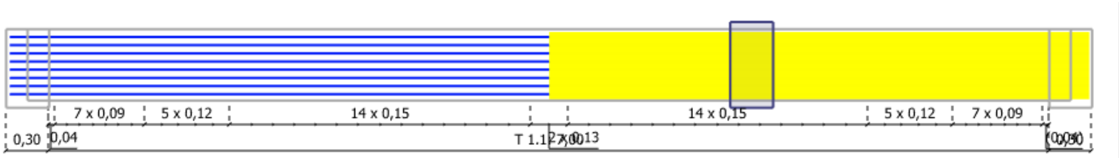
Possibility to choose the way of distribution of reinforcement for torsion

A new option is added to the Reinforcement Assumption window to decide if distribute the theoretical reinforcement area from torsion on the entire length of the beam or to generate two different packages (having different diameters / different numbers of bars) on each half of the beam. This might be helpful when the theoretical reinforcement area for torsion on one half is dramatically different (much smaller or bigger) than on the other half. Note that the torsion bars will be generated with two packages only when they are needed on the lateral sides of the beam due to the torsion. Note also that if bars are needed also for anti-crack reasons, full-length bars will be generated.

The new option is available for Eurocode and NTC standards.



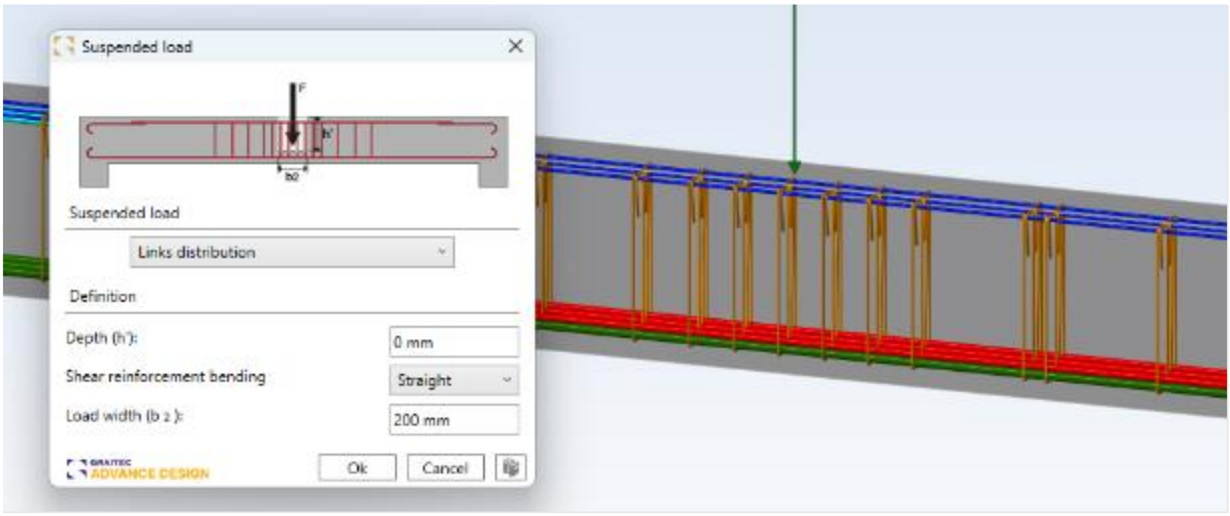
New option for torsion reinforcement separation



Torsion reinforcement in two groups

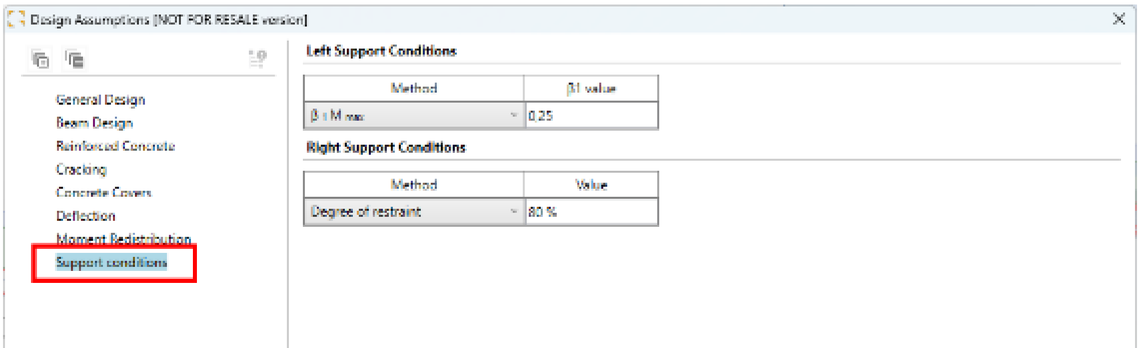
Improvement in defining suspended loads with link distribution

When a concentrated load is defined as suspended, but the Depth (h') value is not entered, the load is nevertheless applied as suspended if the Links distribution option is selected, assuming that it is above the middle fiber (and thus it is not necessary to reassemble the load).



Support conditions on the Design Assumptions dialog

To the Design Assumption dialog, a new Support Conditions tab has been added. It contains the options used for defining left and right-end support conditions, which were available on the ribbon of the standalone version of the module. This allows modifying such data when working in the Advance Design environment.



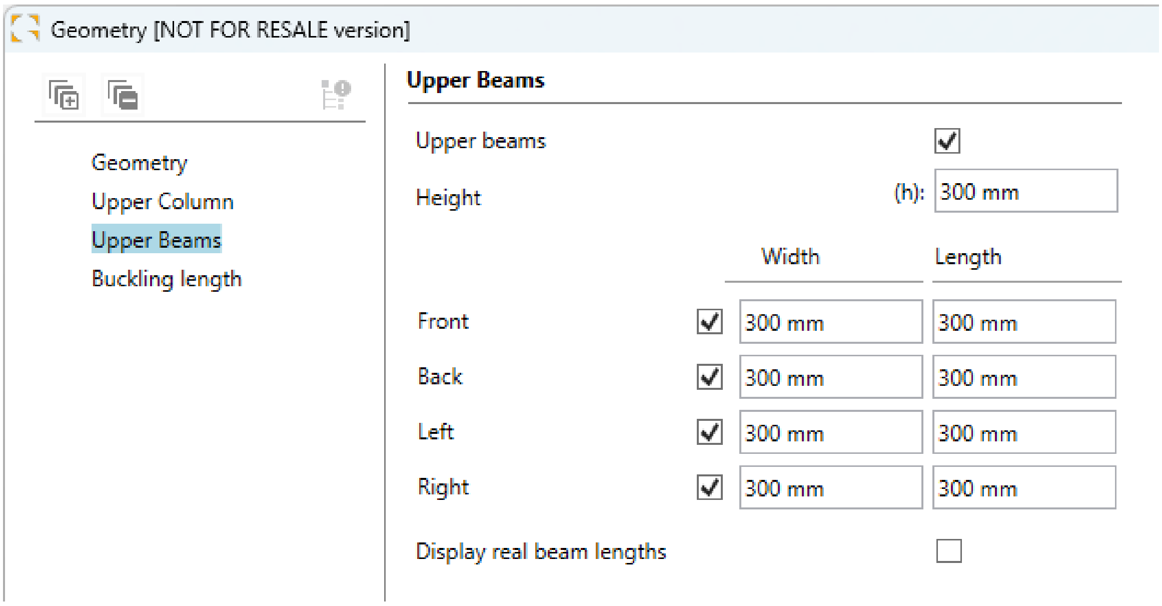
10. RC Column

New features and improvements implemented in the latest version of the RC Column module.

10.1. Different upper beams geometry

Possibility to individually specify the length and width of the top beams and to use that data for automatic determination of buckling lengths.

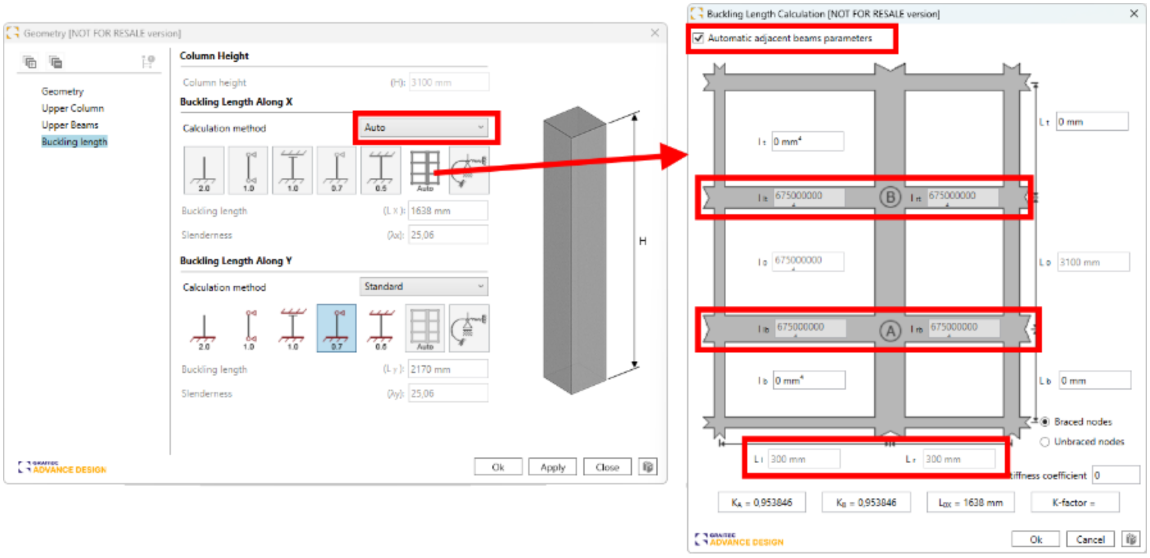
In the column geometry definition window, the scope of definitions for top beams has been expanded. For this, a new tab dedicated to these parameters has been added. Now, in addition to the possibility of defining the height and selecting the number of beams, the possibility of individual definitions of the width of beams and their length has been added.



Content of the new tab

Importantly, this information is now also imported from the Advance Design model along with the rest of the information for the column.

Beam geometry data from the new tab is now also automatically transferred to the window for the automatic determination of column buckling lengths if Auto is selected as the calculation method. In this case, moments of inertia and beam lengths are automatically filled in according to the geometric data. To be able to edit the values independently of the geometric data, you can disable the automatic synchronization option at the top of the window.

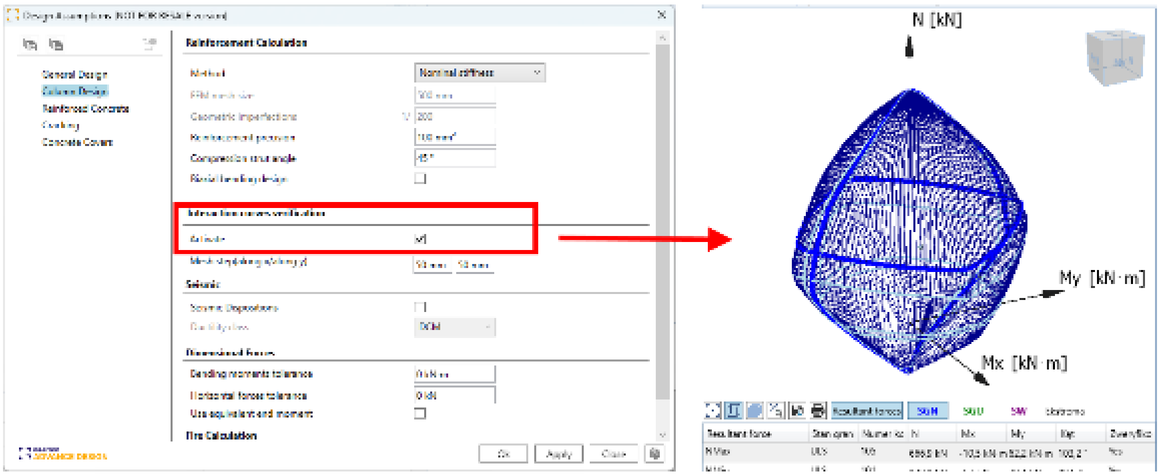


Data completion during automatic determination of buckling length

10.2. Possibility to disable the interaction curves checks

Ability to better customize verification.

On the Column Design tab of the Design Assumptions window, there is a new option to deactivate the verification of column resistance using interaction curves.



10.3. Minor improvements to ribbons

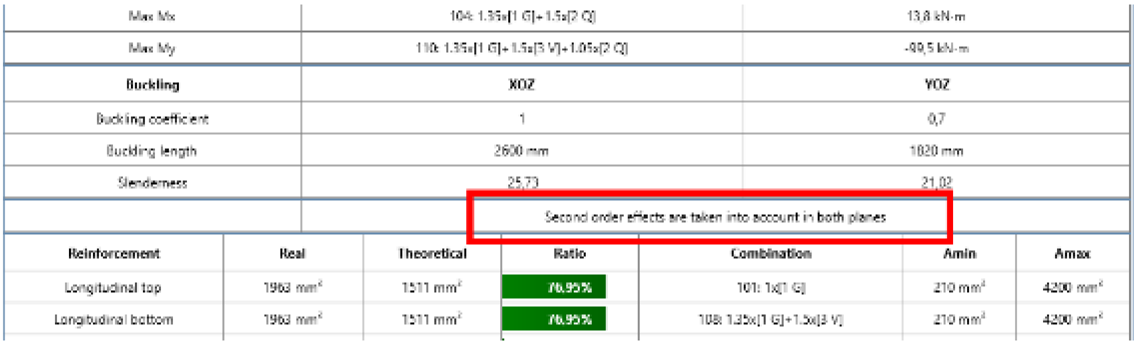
Separation of icons for defining geometry and specifying section type.

To accelerate the selection of options for geometry definition, two geometry-related icons are now separated on the ribbon of the standalone application - the first for selecting cross-section types, and the second for calling the window for defining other column geometry parameters.

10.4. Minor improvements in Info Panel

Faster information on the consideration of second-order effects.

On the Info Panel there is now displayed a piece of short information on whether the second-order effects are not included, or are included in both planes or in one plane (XOZ, YOZ).



Max Mx	104: 1.35x[1 G]+1.5x[2 Q]	13,8 kN·m
Max My	110: 1.35x[1 G]+1.5x[3 V]+1.05x[2 Q]	-99,5 kN·m
Buckling	**XOZ**	**YOZ**
Buckling coefficient	1	0,7
Buckling length	2600 mm	1820 mm
Slenderness	25,73	21,02
	Second order effects are taken into account in both planes	

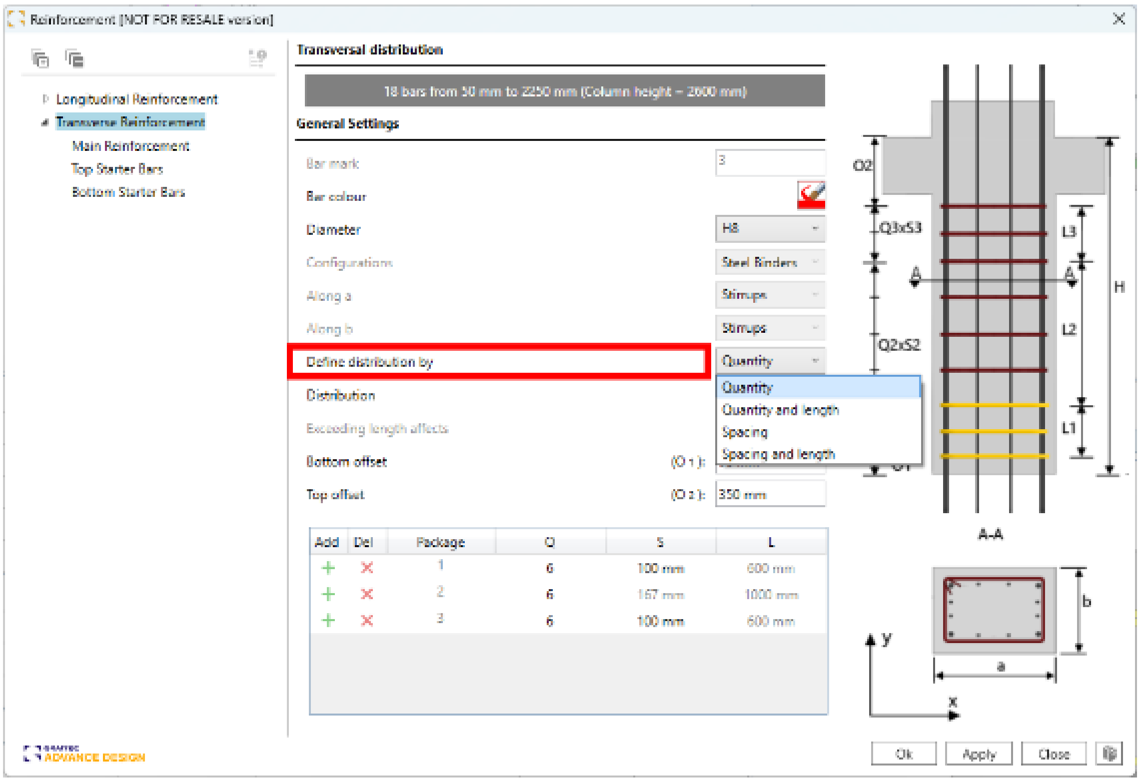
Reinforcement	Real	Theoretical	Ratio	Combination	Amin	Amax
Longitudinal top	1963 mm²	1511 mm²	76.95%	101: 1x[1 G]	210 mm²	4200 mm²
Longitudinal bottom	1963 mm²	1511 mm²	76.95%	108: 1.35x[1 G]+1.5x[3 V]	210 mm²	4200 mm²

10.5. Improvement to transverse reinforcement on Edit dialog

Quickly and precisely define and edit the distribution of transverse bars in the table with new definition methods.

In the latest version of RC Column module, the possibility of specifying the method of distribution definition has been added, which significantly increases the possibility of editing and custom defining the distribution of transverse bars.

For this purpose, in the Reinforcement editing window, on the Transversal Reinforcement tab, a drop-down list 'Define distribution by' has been added, which contains four definitions: Quantity, Quantity and Length, Spacing, Spacing, and length.



Selecting a given item changes the accessibility to editing the fields in the following reinforcement table and allows a comfortable definition of the distribution according to current needs. The method of operation is as follows:

- first - select how to place the bars by using the new list (for example – by Quantity)
- next – select the type of distribution / how should be grouped (for example – 3 Zones Distribution)
- next – define offsets and what offsets are affected by exceeding the length.
- and finally – define values on table / change the packages.

Although this may sound like a complicated procedure, in practice it is quite an easy and effective method that automatically determines all other parameters when editing selected values.

Also worth mentioning is another additional improvement in this window - now the distribution category parameter (3 Zones, 2 Zones, ….) is now automatically completed based on the resulting from calculation bar distribution.

11. RC Footing

New features and improvements implemented in the latest version of the RC Footing module.

11.1. Representation of the substitute footing

A visual representation of the substitute footing.

When we have defined a multilayer soil and activated the verification of bearing capacity using the substitute foundation method, then the location and dimensions of such a substitute foundation are now presented in the 2D elevation views automatically after calculations.

11.2. Additional information on concrete cracking in Info Panel and report

Greater control over calculations with additional concrete cracking information.

One of the available verifications according to Eurocode for the foundation is cracking analysis. If the crack width exceeds the limit value, an appropriate warning message appears.

With the latest version, the most important information about crack verification is also presented in the info Panel.

Verification type	Combination	Value	Limit	Work Ratio
Bearing resistance	111: 1x[1 G]+1x[2 Q]	314,80 kN	467,78 kN	67,3%
Seismic bearing capacity	118: 1x[1 G]+1x[101 COMB]+0.3x[2 Q]	0,59	1,00	59,02%
Sliding	117: 1x[1 G]-1x[101 COMB]	10,00 kN	71,64 kN	13,96%
Settlement	115: 1x[1 G]+0.3x[2 Q]	0,36 cm	5,00 cm	7,18%
Punching	118: 1x[1 G]+1x[101 COMB]+0.3x[2 Q]	0,33 MPa	0,96 MPa	34,43%
Cracking in x direction (bottom)	115: 1x[1 G]+0.3x[2 Q]	0,289 mm	0,300 mm	96,34%
Cracking in y direction (bottom)	115: 1x[1 G]+0.3x[2 Q]	0,241 mm	0,300 mm	80,5%
Reinforcement	**Real**	**Theoretical**		**Ratio**
Bottom Along X	7,07 cm²	6,35 cm²		89,77%
Bottom Along Y	7,85 cm²	7,09 cm²		90,21%

Info Panel

In addition, the report adds a table with details of the calculation of cracking in the footing.

12 Crack width check

Crack width verification							
Direction	Position	Combi	$S_{r,max}$ (mm)	$\varepsilon_{sm} - \varepsilon_c$ (‰)	W_k (mm)	W_{lim} (mm)	WR
Along X	Bottom	115	333	0.79	0.262	0.300	87.41 %
Along Y	Bottom	115	335	0.86	0.288	0.300	96.03 %

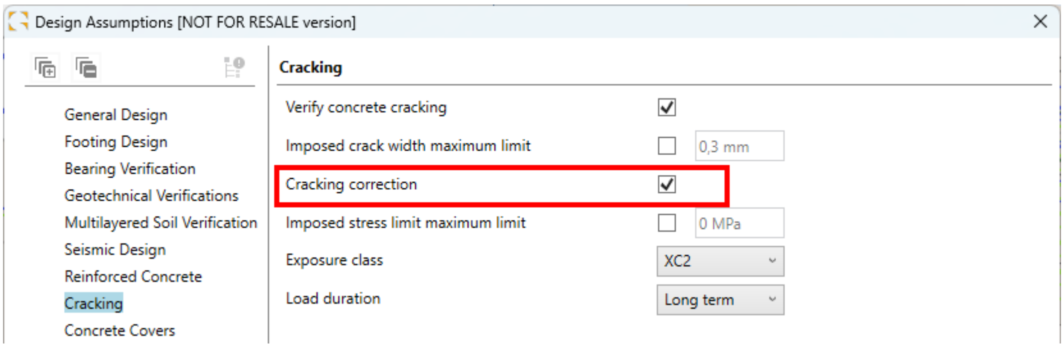
New section of the report

11.3. Auto correction of reinforcement when crack widths are too large

Keeping the cracking to the limit by automatically increasing the reinforcement.

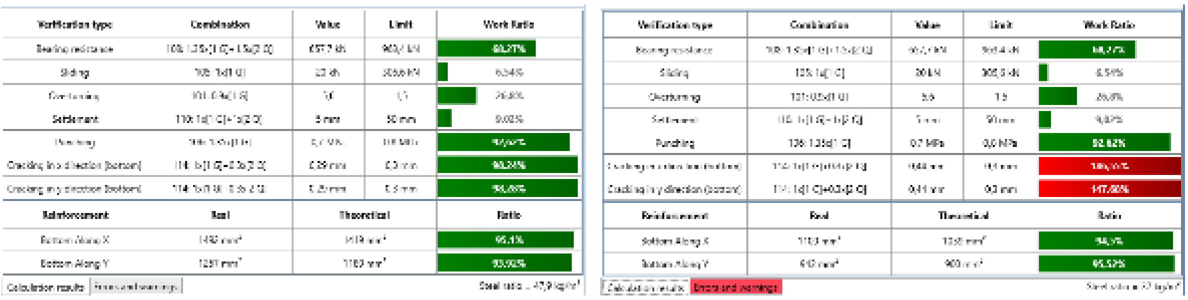
In the latest version of the module, in such cases, an automatic modification of the reinforcement in the foundation can be conducted so that the crack width does not exceed the limit value. This is useful in cases when it is more beneficial to keep a smaller footing thickness instead of more reinforcement.

The choice of whether the cracking should be automatically corrected is in the Design Assumptions window on the Cracking tab:



New option to correct cracking

As shown in the example below, enabling this option automatically reduces cracking at the expense of more reinforcement.

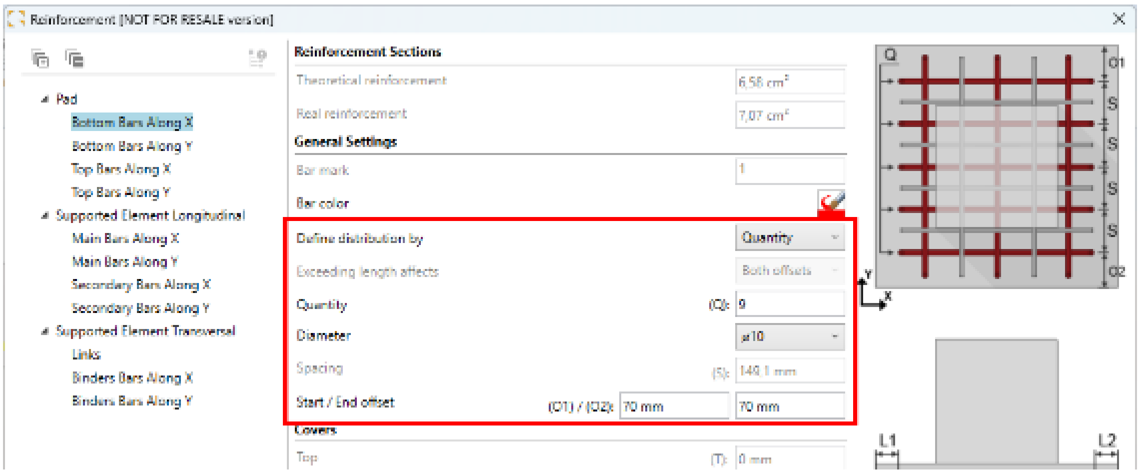


Cracking correction on (left) and off (right) for the same example

11.4. Improved editing of the distribution of bars

Easier definition and modification of bar distributions on a pad.

In the newest version of the program, the way of editing the distribution of top and bottom bars in the pad has been modified. The changes concern the Pad tabs on the Reinforcement edit window and are related to an option to select the method of distribution definition that has been added. We can now choose to define the bar distribution either by entering the number of bars or by entering the spacing.



11.5. Information in the report about the final reinforcement area

Additional information in the report to better describe the final reinforcement area

In the calculation report in the section on reinforcement results, new information about the final required reinforcement area was provided.

This is the final theoretical reinforcement area, based on which the provided reinforcement is selected. This area may be larger than the original theoretical reinforcement area determined from the formulas because it considers possible additional adjustments resulting from the need to meet additional conditions/verifications.

This can be an iterative increase of the reinforcement area to meet the limit of cracking or stresses in the reinforcement bars, but also a recalculation of the theoretical reinforcement resulting from a change in concrete cover after the application of other than the originally assumed diameters of the real reinforcement.

Theoretical reinforcement	$A_{theo} = \frac{2N_{Ed} \cdot \delta}{l_1 \cdot f_{yd}} = \frac{2 \times 689.3\ kN \times 113\ mm}{400\ mm \times 434.8\ MPa} = 832\ mm^2$	
Minimum reinforcement	$A_{min} = 1.3\ ‰ \cdot L_2 \cdot d = 1.3\ ‰ \times 1200\ mm \times 210\ mm$ $A_{min} = 336\ mm^2$	
Required reinforcement	$A_{reqd} = \max\{A_{theo}; A_{min}\} = \max\{832\ mm^2; 336\ mm^2\} = 832\ mm^2$	
Final required reinforcement (after correction for other verification to be satisfied - stresses, cracking etc.)	$A_{reqd,f} = 952\ mm^2$	
Provided reinforcement	$A_{prov} = 1021\ mm^2$	13 × ø10 (Spacing = 88 mm)

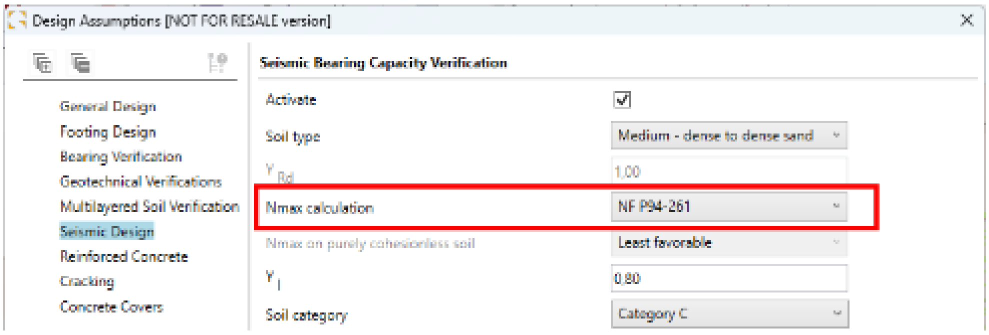
11.6. Improvements to seismic bearing capacity calculations for France

Possibility to select another method for Nmax calculation for France, and more details on related checks on reports.

An additional method for determining the ultimate seismic bearing capacity of the footing under vertical axial load is available for French project settings. Until now, the Nmax value was always determined following EN 1998-5. Now we can determine this value following NF P94-261, using the formula:

$$N_{max} = \frac{q_{net} \cdot A}{\gamma_{R,v} \cdot \gamma_{R,d,v}}$$

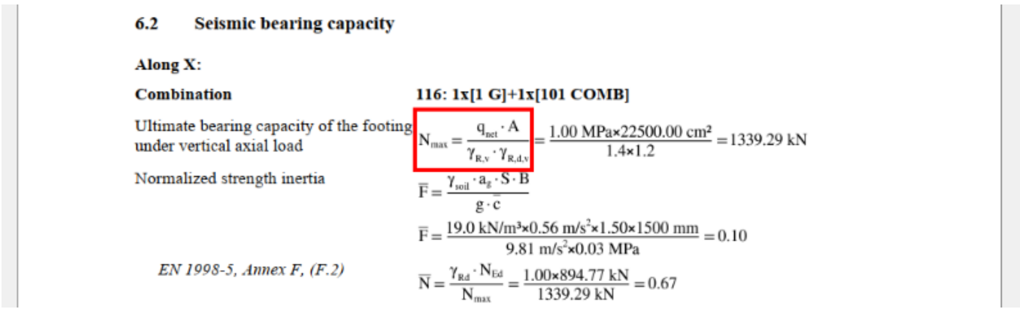
The choice of method is available as a list on the Seismic Design tab of the Design Assumptions window. Note that it is available for editing only for French settings.



Choice of method for determining Nmax

It is worth mentioning that this method requires user-defined bearing pressure/resistance.

If seismic bearing capacity verification is performed, details of calculations using the selected method are available in the relevant section of the report.



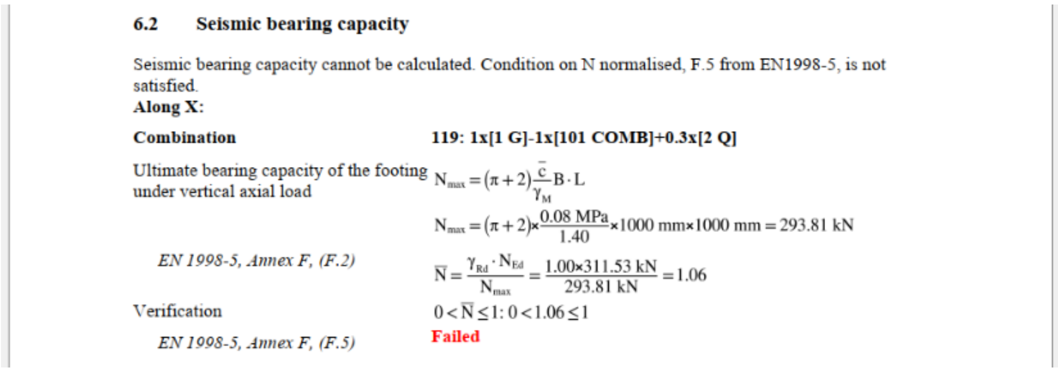
6.2 Seismic bearing capacity

Along X:

Combination	**116: 1x[1 G]+1x[101 COMB]**
Ultimate bearing capacity of the footing under vertical axial load	$N_{max} = \frac{q_{net} \cdot A}{\gamma_{R,v} \cdot \gamma_{R,d,v}} = \frac{1.00\ MPa \times 22500.00\ cm^2}{1.4 \times 1.2} = 1339.29\ kN$
Normalized strength inertia	$\bar{F} = \frac{\gamma_{soil} \cdot a_g \cdot S \cdot B}{g \cdot \bar{c}}$
	$\bar{F} = \frac{19.0\ kN/m^3 \times 0.56\ m/s^2 \times 1.50 \times 1500\ mm}{9.81\ m/s^2 \times 0.03\ MPa} = 0.10$
EN 1998-5, Annex F, (F.2)	$\bar{N} = \frac{\gamma_{Rd} \cdot N_{Ed}}{N_{max}} = \frac{1.00 \times 894.77\ kN}{1339.29\ kN} = 0.67$

Excerpt from the report with details on used method

The latest version of the program has also expanded the report's content when seismic bearing capacity cannot be calculated. Details of the calculation of the associated condition from the French National Annex are now provided.



6.2 Seismic bearing capacity

Seismic bearing capacity cannot be calculated. Condition on N normalised, F.5 from EN1998-5, is not satisfied.

Along X:

Combination	**119: 1x[1 G]-1x[101 COMB]+0.3x[2 Q]**
Ultimate bearing capacity of the footing under vertical axial load	$N_{max} = (\pi + 2)\frac{\bar{c}}{\gamma_M} B \cdot L$
	$N_{max} = (\pi + 2) \times \frac{0.08\ MPa}{1.40} \times 1000\ mm \times 1000\ mm = 293.81\ kN$
EN 1998-5, Annex F, (F.2)	$\bar{N} = \frac{\gamma_{Rd} \cdot N_{Ed}}{N_{max}} = \frac{1.00 \times 311.53\ kN}{293.81\ kN} = 1.06$
Verification	$0 < \bar{N} \leq 1 : 0 < 1.06 \leq 1$
EN 1998-5, Annex F, (F.5)	**Failed**

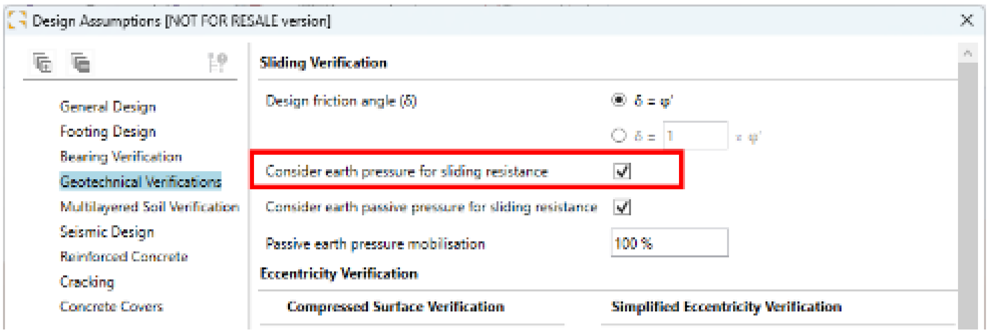
Excerpt from the report with details on not satisfied condition on normalized N

11.7. A new method of determining soil pressure

Determining active and passive earth pressure according to Eurocode 7 provisions.

In the latest version of the module, a new algorithm for determining active and passive earth pressure has been introduced. Earth pressures are considered during sliding verification and are now calculated following annex C of EN 1997-1. The changes apply to calculations according to the Eurocode, excluding the German National Annex (using a different method).

NOTE: *A consideration of earth pressure for sliding verification is not enabled by default and can be enabled in the Design Assumptions window.*



Option to enable consideration of earth pressure for sliding

Detailed data and results of earth pressure calculations are presented in the report.

8.1 Sliding verifications at ULS

Combi	Condition	Layer	H_d (kN)	$R_{h,d}$ (kN)	V_d (kN)	WR
106	Drained	None	307.6	451.6	743.8	68.1 %

Drained conditions

Combination: **106: 1.35x[1 G]**

Horizontal load $H_d = \sqrt{H_x^2 + H_y^2} = \sqrt{(300\ kN)^2 + (67.8\ kN)^2} = 307.6\ kN$

Vertical load (initial) $V_{d,0} = 760.8\ kN$

Vertical load (including the effect of earth pressures)

$V_d = Max\{V_{d,0} + \gamma_G \cdot P_{av,x} - P_{pv,x} + \gamma_G \cdot P_{av,y} - P_{pv,y}; 0\}$

$V_d = Max\left\{\begin{matrix}760.8\ kN + 1.35 \times 0.1\ kN - 8.6\ kN + \\ +1.35 \times 0.1\ kN - 8.6\ kN; 0\end{matrix}\right\}$

$V_d = 743.8\ kN$

Sliding resistance (initial) $R_{d,0} = \frac{V_d \cdot \tan(\delta_k)}{\gamma_{R,h} \cdot \gamma_\varphi} = \frac{743.8\ kN \times \tan(30\ °)}{1 \times 1} = 429.4\ kN$

EN 1997-1, (6.5.3)

The resistance will be modified by lateral active and passive earth pressure on footing:

Active earth pressure coefficient $K_a = 0.34$

C.1 (2) from EN 1997-1, Annex C

Passive earth pressure coefficient $K_p = 3.63$

C.1 (2) from EN 1997-1, Annex C

Horizontal component of the active earth pressure on X $P_{ah,x} = 0.2\ kN$

Vertical component of the active earth pressure on X $P_{av,x} = 0.1\ kN$

Horizontal component of the passive resistance on X $P_{ph,x} = 18.5\ kN$

Vertical component of the passive resistance on X $P_{pv,x} = 8.6\ kN$

Horizontal component of the active earth pressure on Y $P_{ah,y} = 0.2\ kN$

Vertical component of the active earth pressure on Y $P_{av,y} = 0.1\ kN$

Horizontal component of the passive resistance on Y $P_{ph,y} = 18.5\ kN$

Vertical component of the passive resistance on Y $P_{pv,y} = 8.6\ kN$

Applied load component on X with load from earth trust $H_x = |H_{dx}| + \gamma_G \cdot P_{ah,x} =$

$H_x = |299.7\ kN| + 1.35 \times 0.2\ kN = 300\ kN$

Applied load component on Y with load from earth trust $H_y = |H_{dy}| + \gamma_G \cdot P_{ah,y} =$

$H_y = |67.5\ kN| + 1.35 \times 0.2\ kN = 67.8\ kN$

Resultant from passive earth pressure $R = \sqrt{R_{p,d,x}^2 + R_{p,d,y}^2} = \sqrt{(18.5\ kN)^2 + (18.5\ kN)^2} = 26.2\ kN$

Resistance load from passive earth pressure along X $R_{p,d,x} = \frac{P_{ph,x}}{\gamma_{Rd}} = \frac{18.5\ kN}{1} = 18.5\ kN$

Resistance load from passive earth pressure along Y $R_{p,d,y} = \frac{P_{ph,y}}{\gamma_{Rd}} = \frac{18.5\ kN}{1} = 18.5\ kN$

Passive resistance projected along the axis of resultant between Hdx and Hdy $R_{p,d} = R \cdot \cos(\alpha - \beta)$

$R_{p,d} = 26.2\ kN \times \cos(12.7\ ° - 45\ °) = 22.1\ kN$

Sliding resistance (final) $R_d = R_{d,0} + R_{p,d} = 429.4\ kN + 22.1\ kN = 451.6\ kN$

Sliding verification $H_d \le R_d$: $307.6\ kN \le 451.6\ kN$

Work Ratio **68.1 % (Passed)**

12. RC Wall

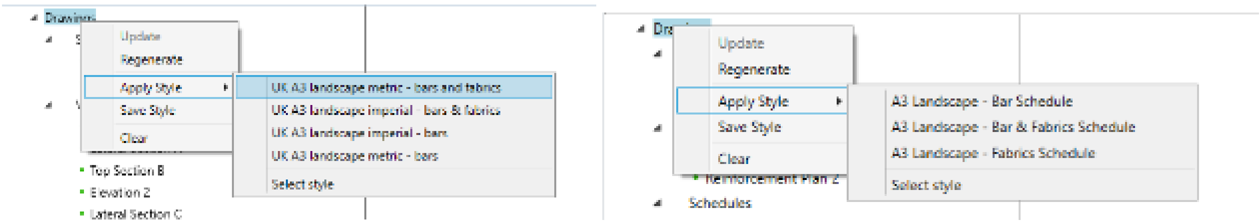
New features and improvements implemented in the latest version of the RC Wall module.

12.1. Generation of drawing with schedules for bars and fabrics

Automatic simultaneous generation of schedules for bars and fabrics.

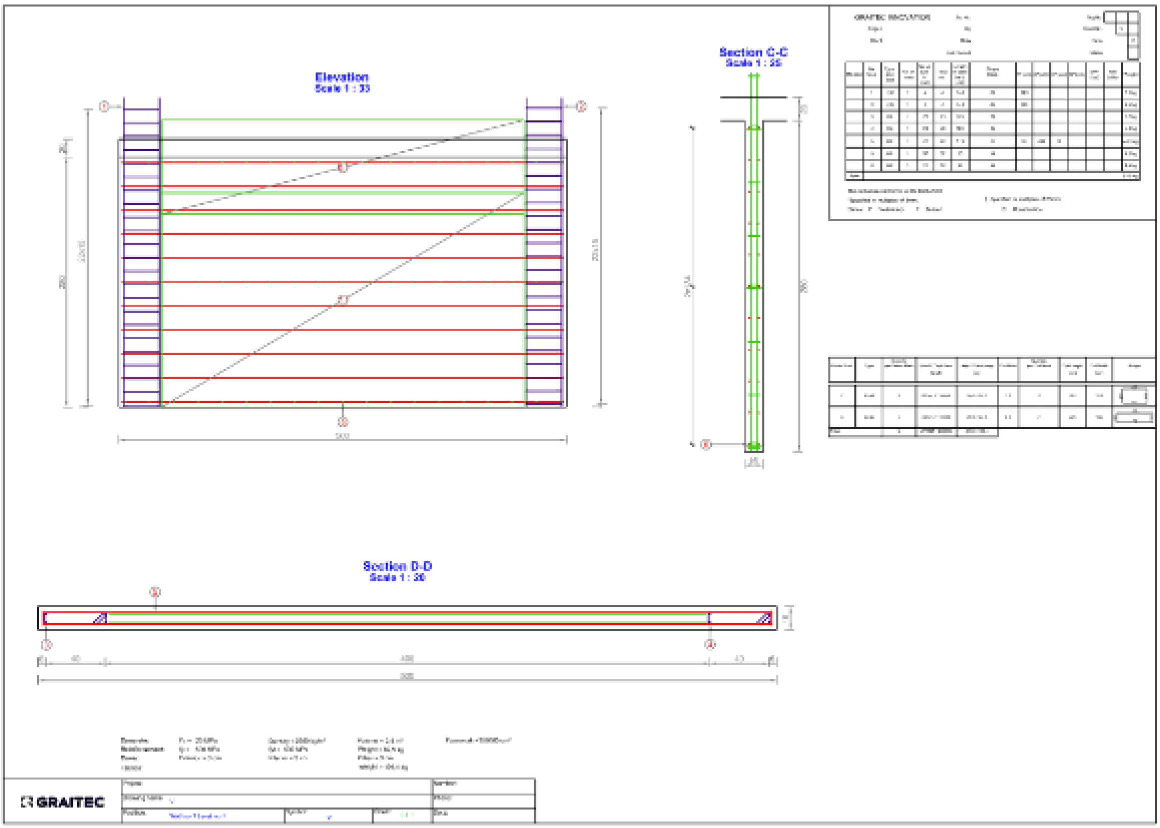
Previously, when generating drawings for walls, a bar schedule was generated, while if there were also reinforcement meshes, their schedule had to be manually generated and added to the drawing.

In the latest version, this process has been simplified, and depending on your needs, you can use either a drawing style for members only or a new template - for members and meshes. The same mechanism is also available for RC Slab module.



Selecting a drawing style on RC Wall (left) and RC Slab (right)

A new drawing style was added to allow the automatic simultaneous generation of schedules for bars and fabrics.

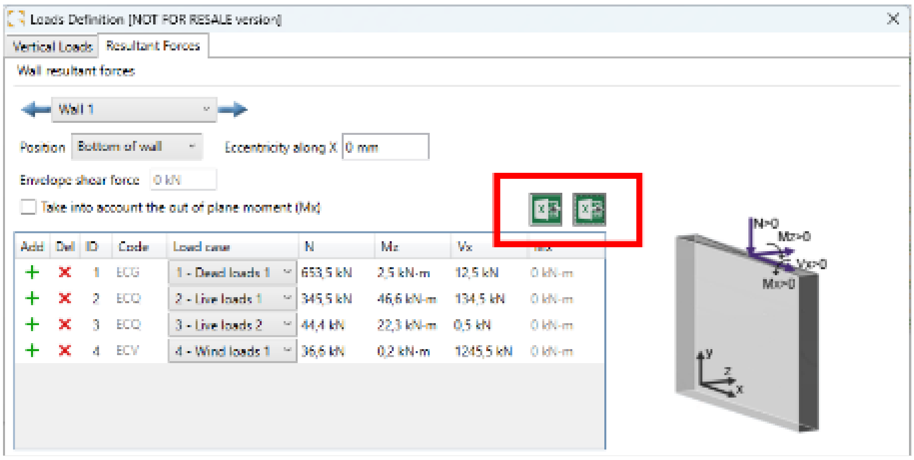


Automatically generated drawing for wall with schedules for bars and meshes

12.2. Import and export forces using Excel file

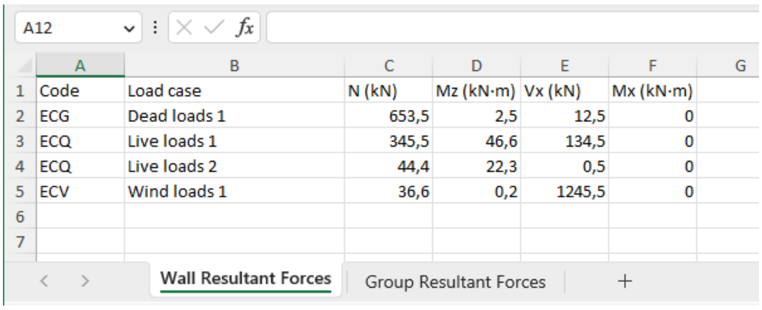
Easy entry of loads from external data using an Excel sheet.

For Shear walls, it is now possible to import and export resultant forces using an Excel sheet. For this purpose, two new icons are available in the window of load definitions - one for export and the other for import from an Excel spreadsheet.



Import and export to Excel commands

With the ability to import resultant forces per wall or wall group, you can easily use the module to analyze walls when load data is available from external sources, as well as quickly edit load data.



Loads exported to an Excel spreadsheet

13. Masonry Wall

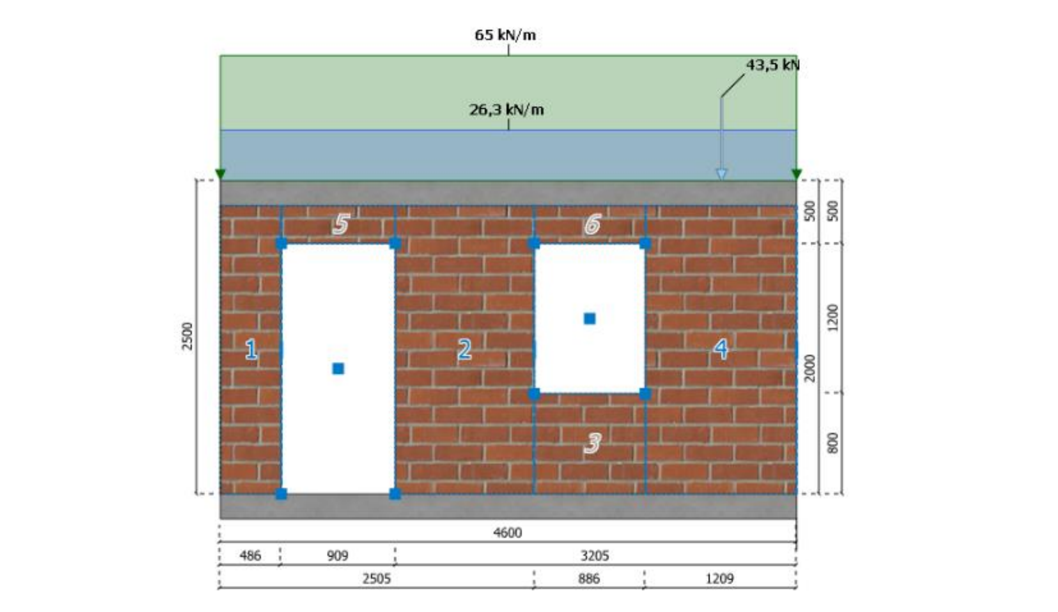
New features and improvements implemented in the latest version of the Masonry Wall module.

13.1. Loads representation in viewports

Easy verification of defined loads.

To make it easier to view defined loads as well as imported internal forces, it is now possible to display loads and internal forces graphically

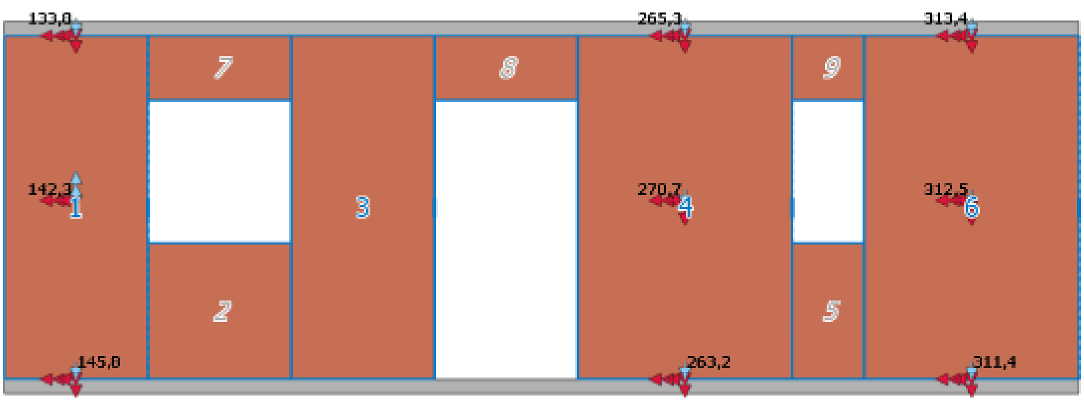
We can now see loads in the 2d and 3d views.



View of external loads on the wall

Depending on the needs, we can easily enable or disable the display of loads directly from the window of a particular view. From the window, with graphic settings, we can also control their display range - for example, for which load case.

In the case where the wall was imported from Advance Design, we can also graphically display the calculated internal forces at the bottom middle, and top of the panels for each load case.

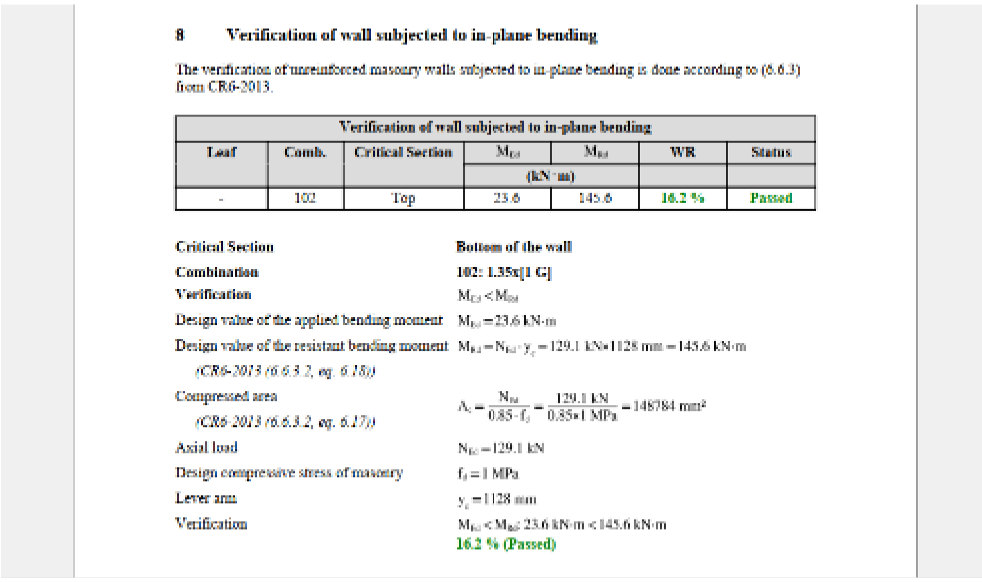


Resultant forces determined for vertical design sections

13.2. Expanded report for RC6 calculations

Richer project documentation for Romanian clients.

A new section has been added in the reports for verification of a wall subjected to in-plane bending according to the Romanian CR6 standard.



8 Verification of wall subjected to in-plane bending

The verification of unreinforced masonry walls subjected to in-plane bending is done according to (6.6.3) from CR6-2013.

Verification of wall subjected to in-plane bending						
Leaf	**Comb.**	**Critical Section**	M_{Ed}	M_{Rd}	**WR**	**Status**
			(kN·m)			
-	102	Top	23.6	145.6	16.2 %	Passed

Critical Section	**Bottom of the wall**
Combination	**102: 1.35x[1 G]**
Verification	$M_{Ed} < M_{Rd}$
Design value of the applied bending moment	$M_{Ed} = 23.6$ kN·m
Design value of the resistant bending moment *(CR6-2013 (6.6.3.2, eq. 6.18))*	$M_{Rd} = N_{Ed} \cdot y_c = 129.1 \text{ kN} \cdot 1128 \text{ mm} = 145.6$ kN·m
Compressed area *(CR6-2013 (6.6.3.2, eq. 6.17))*	$A_c = \frac{N_{Ed}}{0.85 \cdot f_d} = \frac{129.1 \text{ kN}}{0.85 \cdot 1 \text{ MPa}} = 148784 \text{ mm}^2$
Axial load	$N_{Ed} = 129.1$ kN
Design compressive stress of masonry	$f_d = 1$ MPa
Lever arm	$y_c = 1128$ mm
Verification	$M_{Ed} < M_{Rd}$: 23.6 kN·m < 145.6 kN·m **16.2 % (Passed)**

13.3. Possibility to choose the section for the stress diagram

Better control over the display of results.

When displaying the results in the form of stress diagrams, it is now possible to select the position of the section (bottom/ middle/ top). For the selected section, the height of the section level is also presented.

Section position selection options

Stress diagrams for the section at the bottom of the wall

13.4. Saving the masonry databases data per model

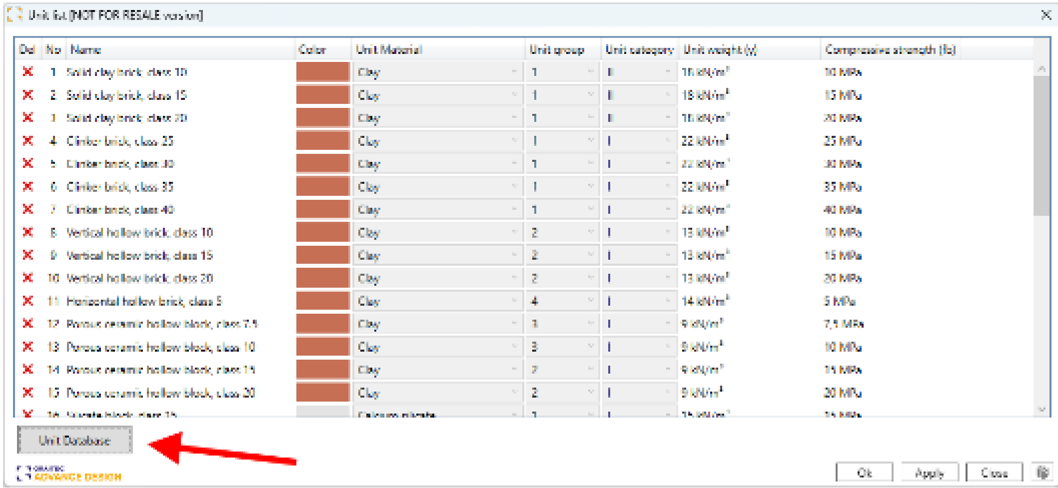
A new way of selecting data from databases to make it easier to manage their contents.

To facilitate the management of data from databases used by the Masonry Walls module, databases have been separated from the data of the current project.

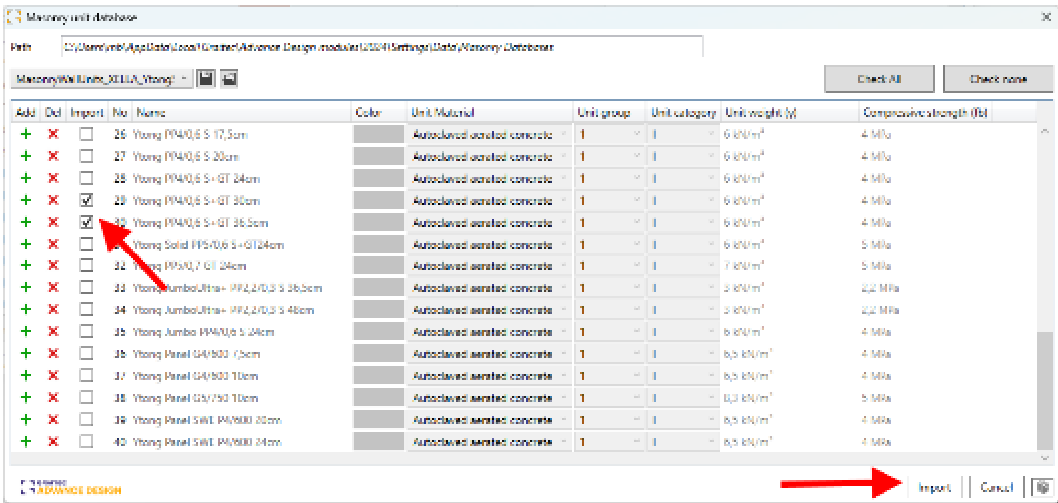
The changes apply to databases for mortar, masonry units, and wall sections.

Thanks to these changes, the change of data, e.g., of a given section in the masonry wall section database will not affect other projects that use the same section.

In the case of Masonry Units and Mortars, the data is managed in the same way as, for example, the management of reinforcement fabrics or soils in the other modules. For example, if you click on the Unit List icon, you will see a window with a list of masonry units loaded/available in the project. This list can of course be defined individually and depends on the project template. To delete from the list unwanted entry, you can click on the icon on the first column. To add new entries to the list, you need to use the Unit database button.



In this way, another window is available, in which you can see a list of all available Masonry units in the databases. Using checkboxes, we select the items we want to add to the current project and then we can use the Import button.

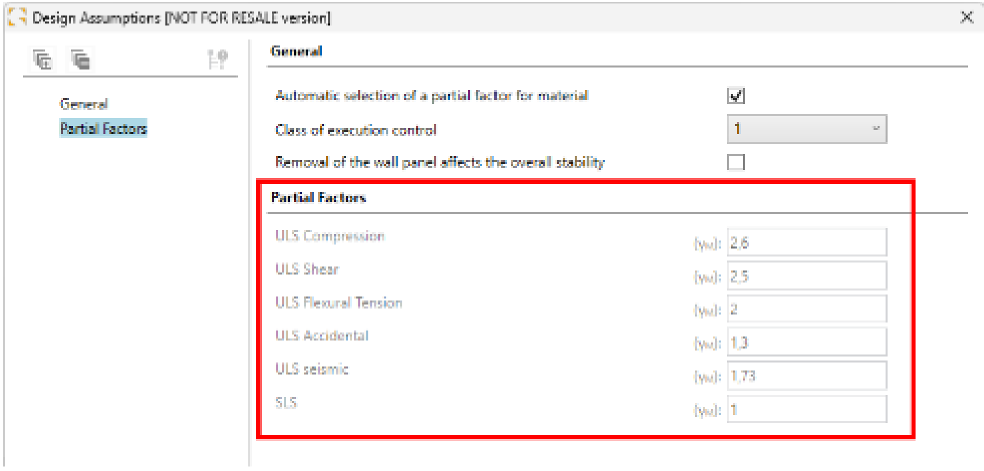


As it was available in the previous version, in this window we can, of course, select a database from the disk, edit the contents of the database itself, including extending it with our own items.

13.5. Presentation of partial factors

Quick verification of the applied partial factors.

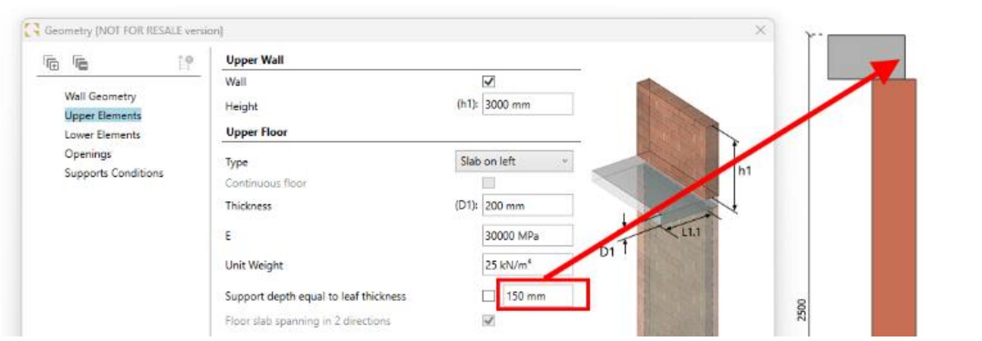
To make it possible to verify the determined partial factors for the current masonry section, they are now available in the Design Assumptions window. Note that the values of the partial factors are determined automatically, as most national annexes depend on the parameters and structure of the wall cross-section. Therefore, they may be different for different walls.



13.6. Small improvement: Support depth representation in viewports

Displaying the depth of the slab support on viewers to easily check the data.

The depth of the slab support is now visible to viewers, so you can more easily make sure that the data you entered is correct.



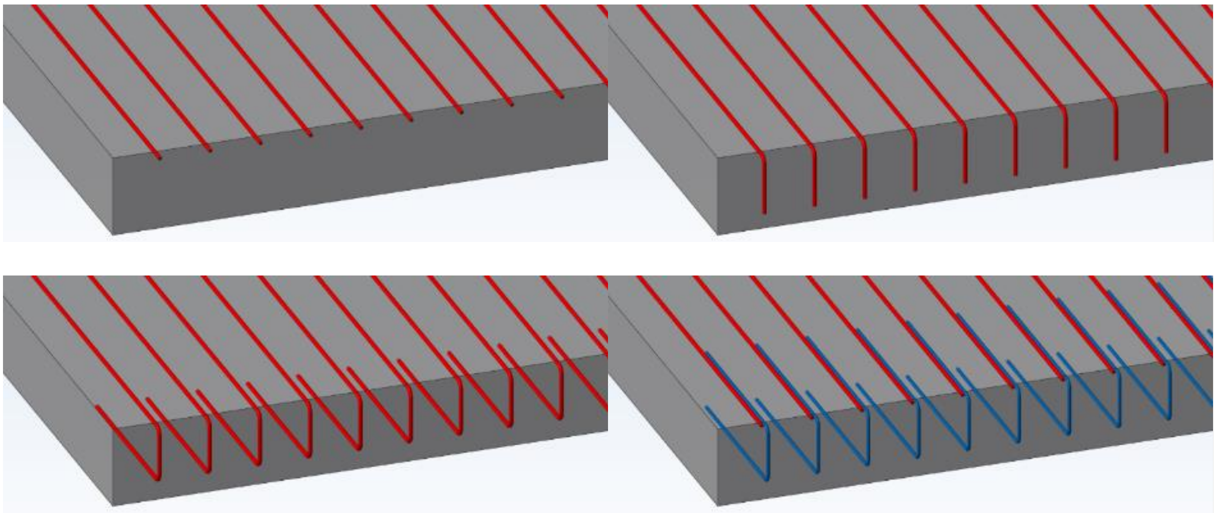
14. RC Slab

New features and improvements implemented in the latest version of the RC Slab module.

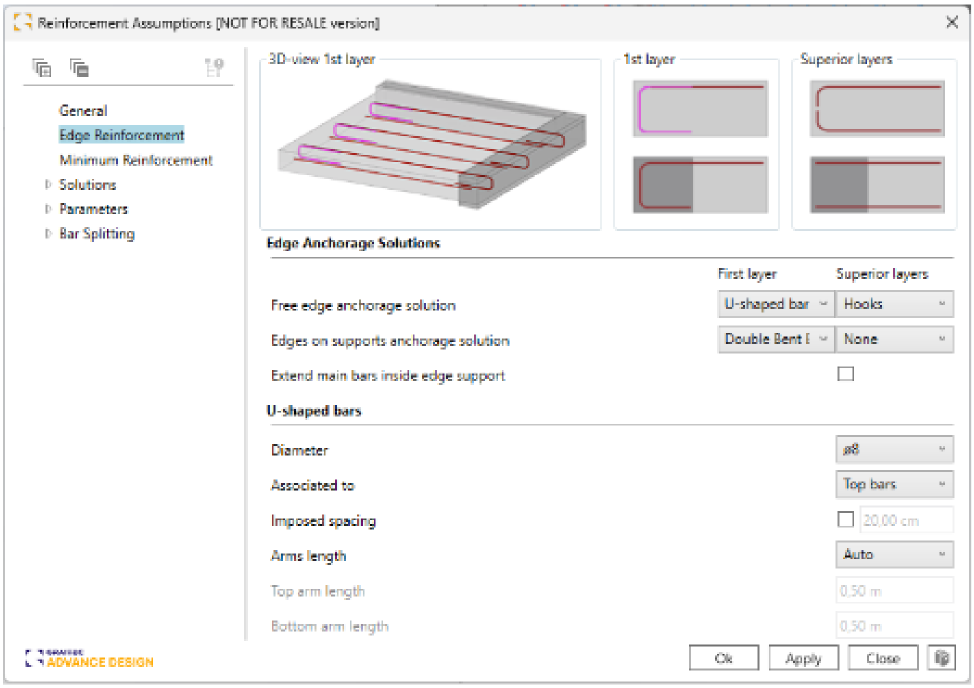
14.1. New bar shapes

Possibility to generate u-shaped and double-bend bars.

The latest version of the program has expanded the list of types of rebar shapes that can be used in concrete slabs. All these shapes refer to the method of bar end, i.e. bar anchoring solution. Now there are 4 anchoring methods available: straight bars, hooks, double-bent bars, and using u-shaped bars.

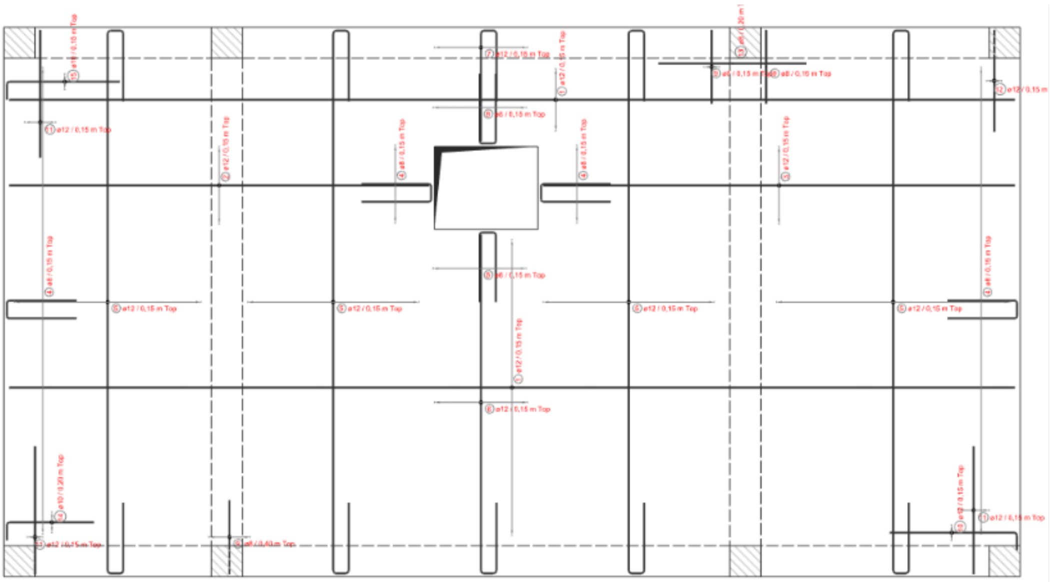


The choice of anchorage type is available in the settings (Reinforcement Assumptions window), on the new Edge Reinforcement tab. We can define different settings for the first and second layers of reinforcement, independently for free edges and edges on supports.



In addition, this window allows us to set parameters for u-shaped bars, including diameter, and assignment to bottom or bottom reinforcement, and we can also impose spacing or arm length.

During the generation of reinforcement drawings, bar anchorage is presented as projected, so you can easily recognize its shape.

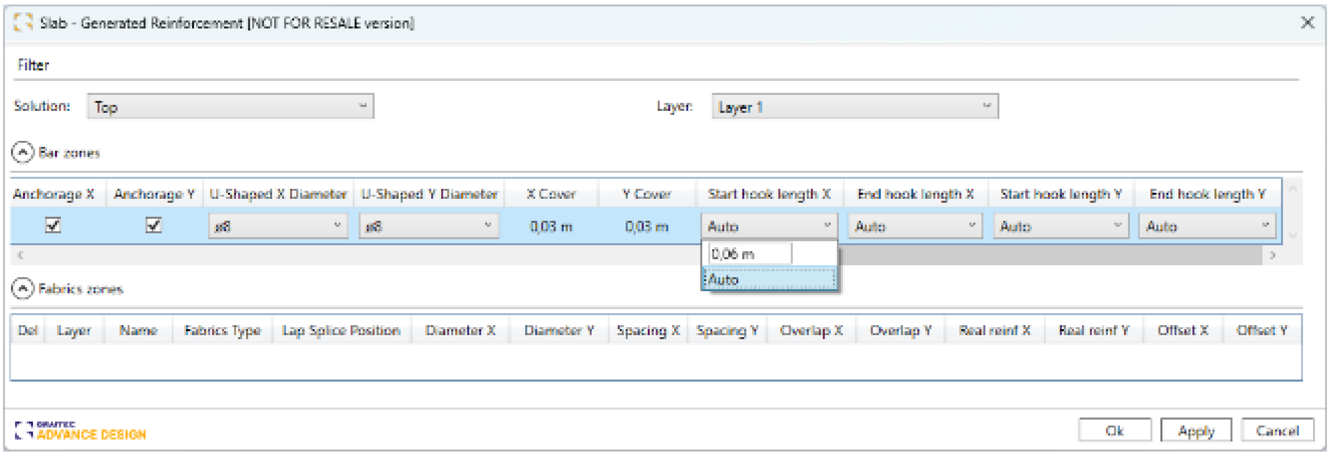


Automatically generated reinforcement plan for top reinforcement

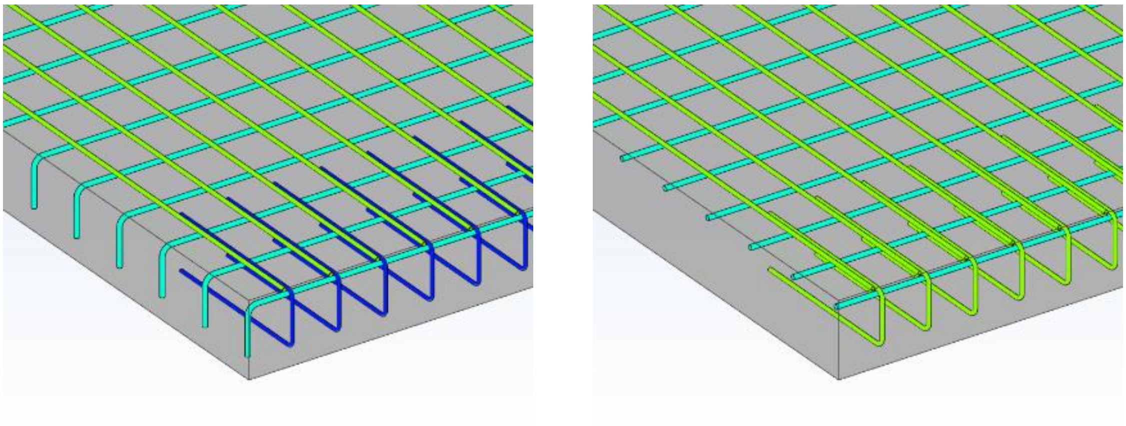
14.2. Possibility to manage hooks for reinforcement zones

Possibility for editing hooks separately for each reinforcement zone.

In the latest version of the program, the window for editing generated reinforcement has been expanded with a set of new columns for modifying anchorage parameters. Separately for each reinforcement zone you can decide whether the anchorage is to be generated, you can set the diameter (for anchorage by U-shape bar), set the cover value, and also modify the start or end hook length.



Part of the Generated Reinforcement dialog with anchorage parameters

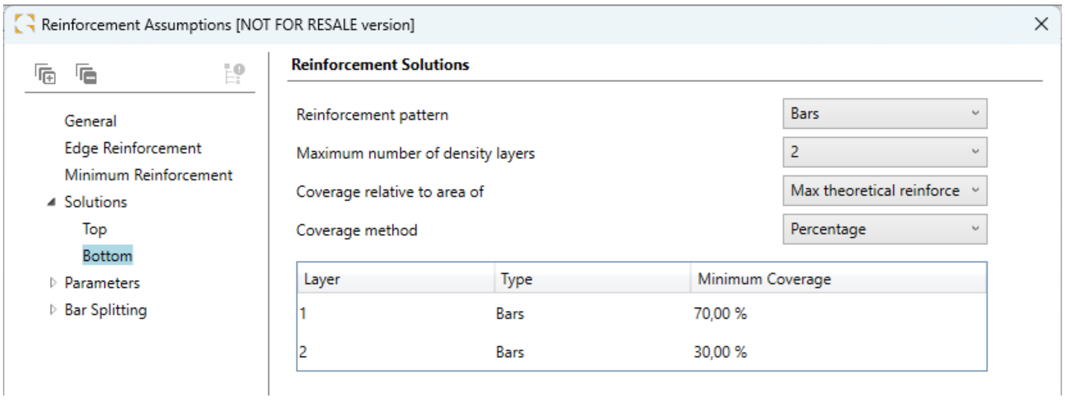


Editing example - removing the hooks in one direction and changing the diameter of the U-shaped bars in the other

14.3. New options for defining reinforcement solution

Possibility to define reinforcement coverage with reinforcement areas.

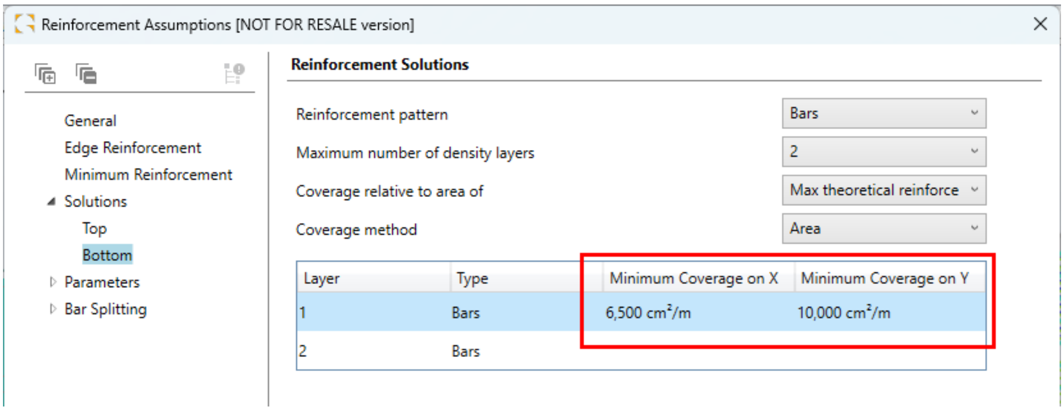
For the automatic definition of reinforcement zones, we can use the setting (on the Reinforcement Assumption window) so that when defining reinforcement layers, they cover a fixed percentage of the theoretical reinforcement area. That is, for example, for the bottom reinforcement, the first zone (covering the entire surface) should cover 70% of the required reinforcement, and the remaining 30% should be covered by the second layer.



Minimum coverage defined as the percentage of maximum theoretical reinforcement area

In the latest version, we can specify the coverage method not only by percentages but also by absolute values of the reinforcement area. This facilitates the precise selection of coverage.

To be able to select the method, in the *Reinforcement Assumption* window, under the *Solution* tab, you must set the **Coverage** method to **Area**. Then in the table below, we can specify the minimum coverage area separately for X and Y directions.



Minimum coverage defined as the value of reinforcement area

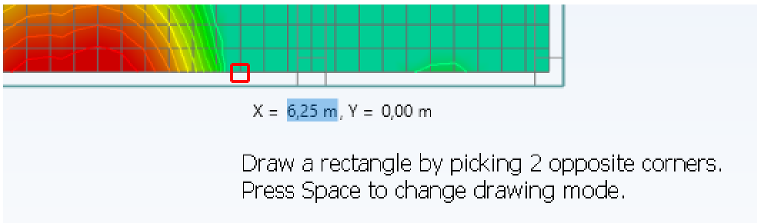
14.4. New methods for defining reinforcement zone graphically

Faster modeling of reinforcement zones with additional graphic definition methods.

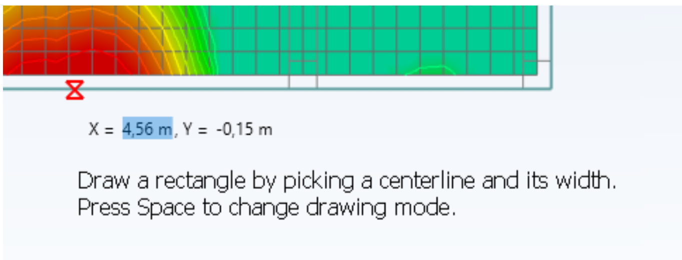
When defining reinforcement in concrete slabs based on the calculated theoretical reinforcement, we can use either automatic determination of reinforcement zones, or we can define such zones by ourselves, as well as we can mix both methods (e.g., one zone on the entire surface of the slab is defined automatically and zones of additional reinforcement are defined manually).

Until now, the graphical definition of a zone consisted of entering the zone area via a closed polyline. In the latest version, two additional methods of drawing rectangular reinforcement zones are available - by 2 points and by the centerline.

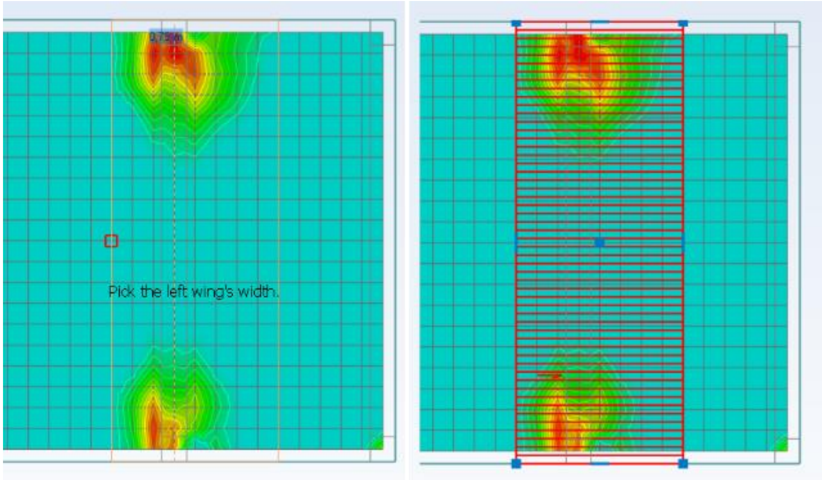
The new method of defining rectangles by 2 points is quite simple. We just need to pick 2 opposite points of the rectangle's diagonal. This model is now a default one.



The second new method is a definition of a rectangle by a centerline and side offsets. We need to pick the start and end point for the line, and next we pick a point for defining the right and left edges of the rectangle. This mode is especially useful for defining rectangular zones over a line of supports.



Note that the drawing mode can be changed by pressing the Space key when adding a zone.



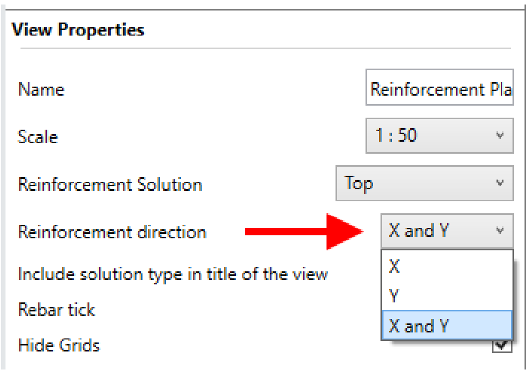
Zone definition with centerline and side offsets (left), and defined reinforcement zone (right)

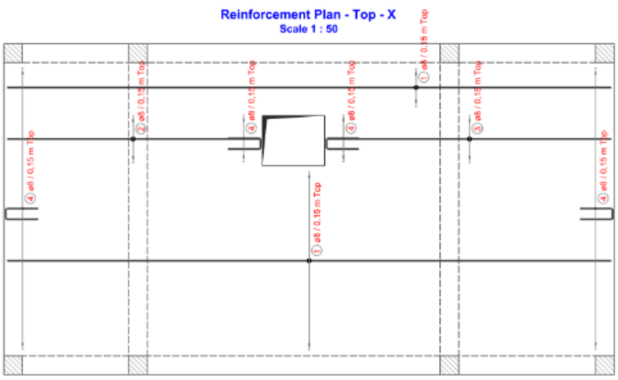
14.5. Set of improvements to drawings

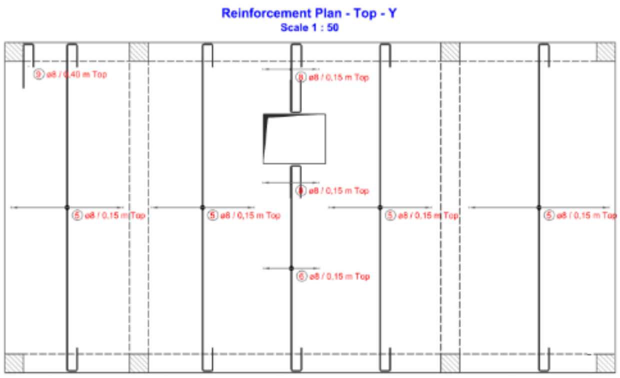
Increased attractiveness and greater customization of automatic reinforcement drawings for RC slabs.

Separate drawings for reinforcement directions

With this version, we can generate drawings either simultaneously or separately for each reinforcement direction. For this purpose, in the properties of the plans, we can choose whether a given view presents rebars in the X or Y or in both directions.



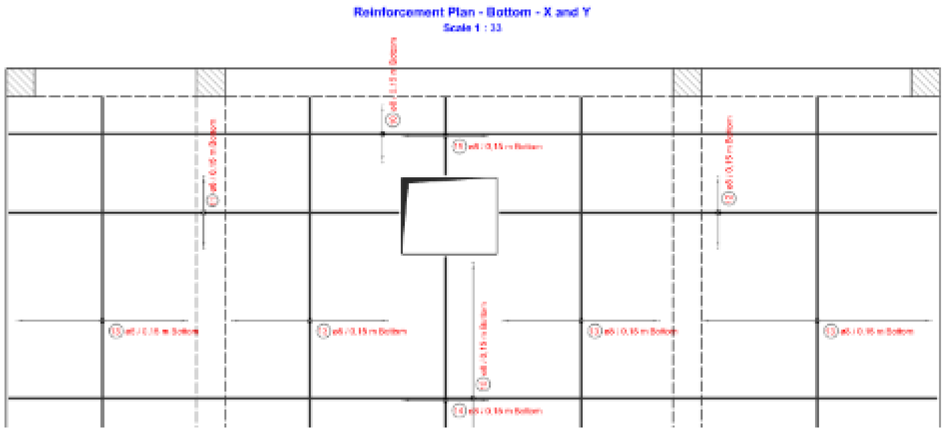




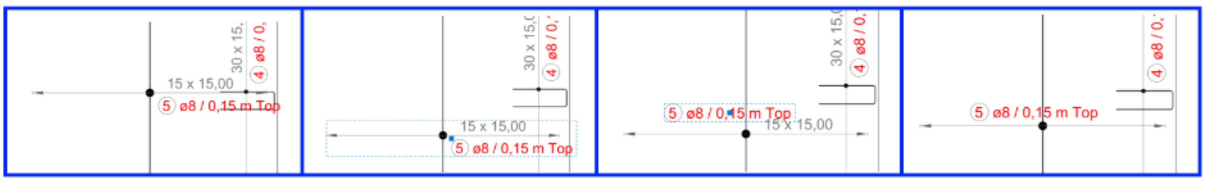
Improvements to bar annotations

To increase the functionality of the drawings and improve their reception, several improvements have been made to the generation and editing of reinforcement descriptions.

First, the algorithm for the automatic placement of descriptions has been improved to increase their readability and avoid bar collisions.



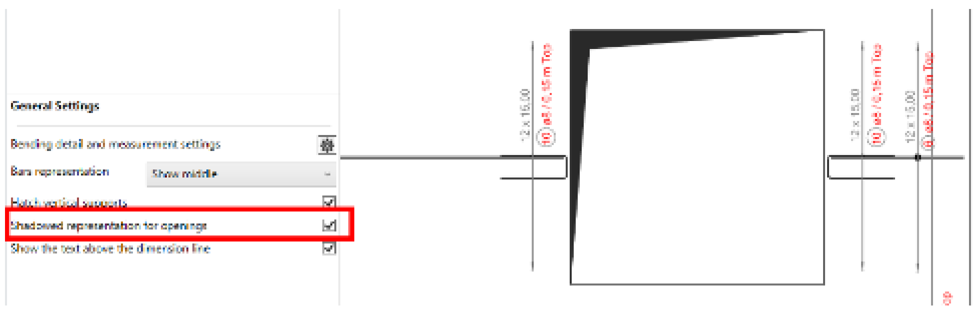
Secondly, in situations where you want to modify the positions of the distribution descriptions, you can move the entire description, as well as the bar mark and the dimension line independently. You can also use the new option to disable descriptions above the dimension line.



Initial position / moved the whole description / moved bar mark / without dimension line's description

Better representation of openings

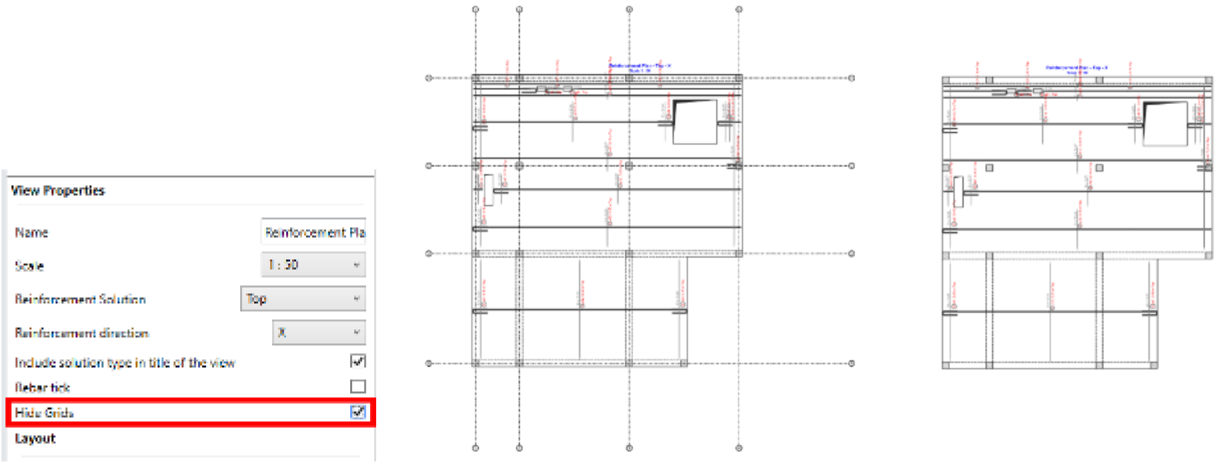
Another improvement of RC slab drawings is the ability to display a special graphic representation for openings (as shown in the image). In case you do not want such an additional symbol for openings, you can disable it with a new option in the drawing settings.



Option for changing the display style for openings

New option to hide grids in drawings

To increase the functionality of the parameterization of slab drawings, an option has been added to decide whether the structure axes are to be presented in the drawings.



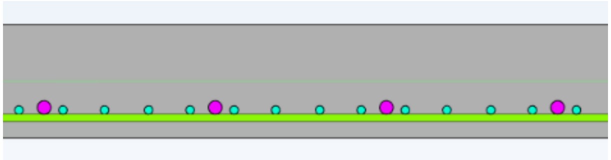
Option to exclude axes and application example

14.6. Set of smaller improvements

Increased functionality and comfort with a set of small improvements.

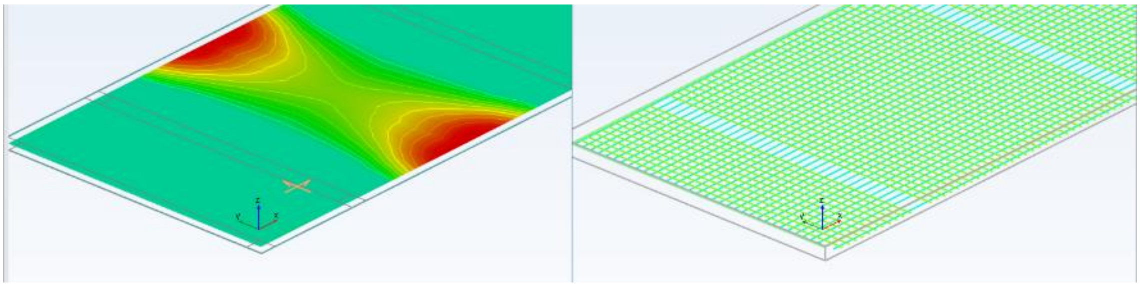
Allow the second layer to have a bigger diameter than the first one

When defining bars in each direction in multiple layers, it is now possible to use any configuration of bar diameters in different layers, including adding bars with larger diameters in the next layer than in the first.



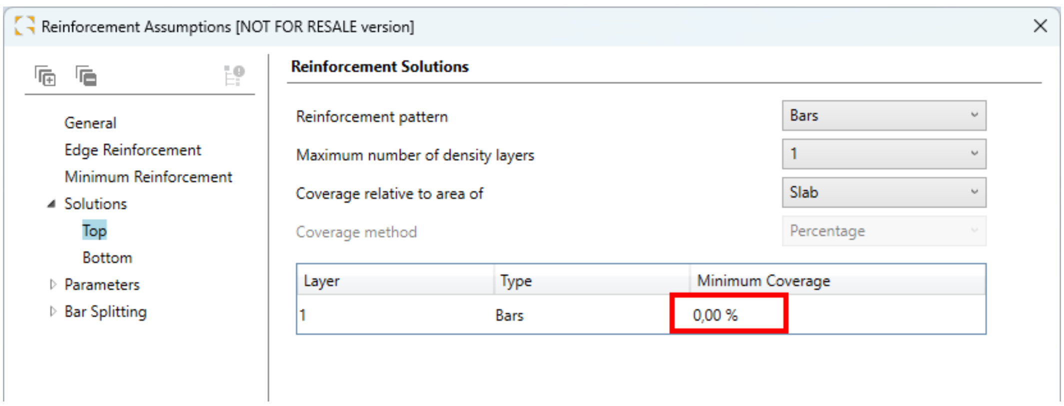
Displaying local axes for each slab

Since the reinforcement for slabs refers to their local axis system, to facilitate orientation during zone definition and reinforcement analysis, the views now show a graphical symbol for the local axis system of the slab.



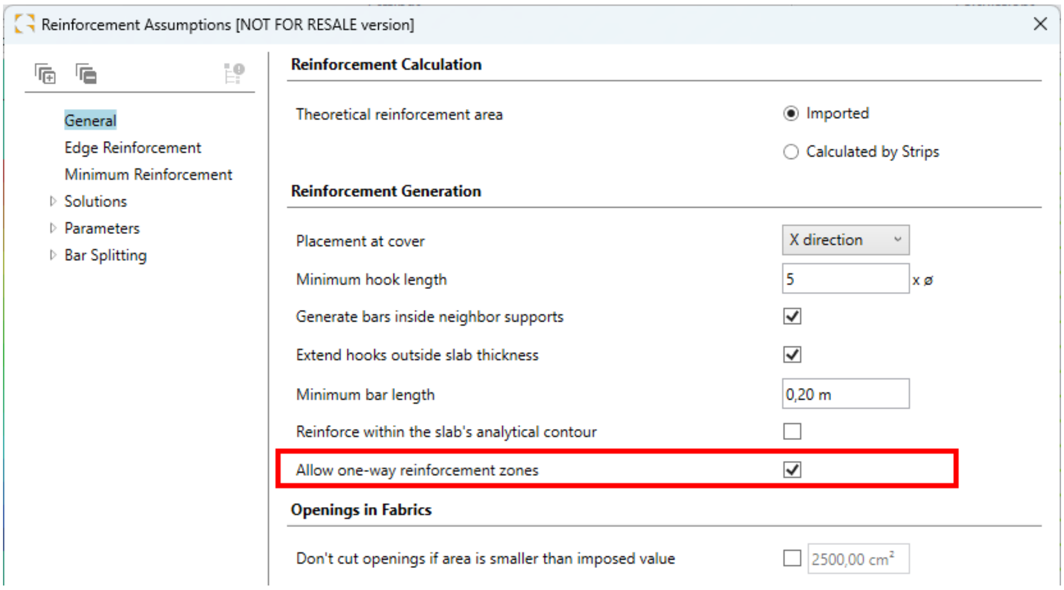
Possibility for disabling reinforcement layer

In case you want to generate reinforcement automatically, but not in all layers, you can now enter minimum coverage as zero for a layer when setting parameters. An example application is the quick automatic generation of the bottom reinforcement, while the top reinforcement is then defined manually only in selected areas (e.g., over supports).

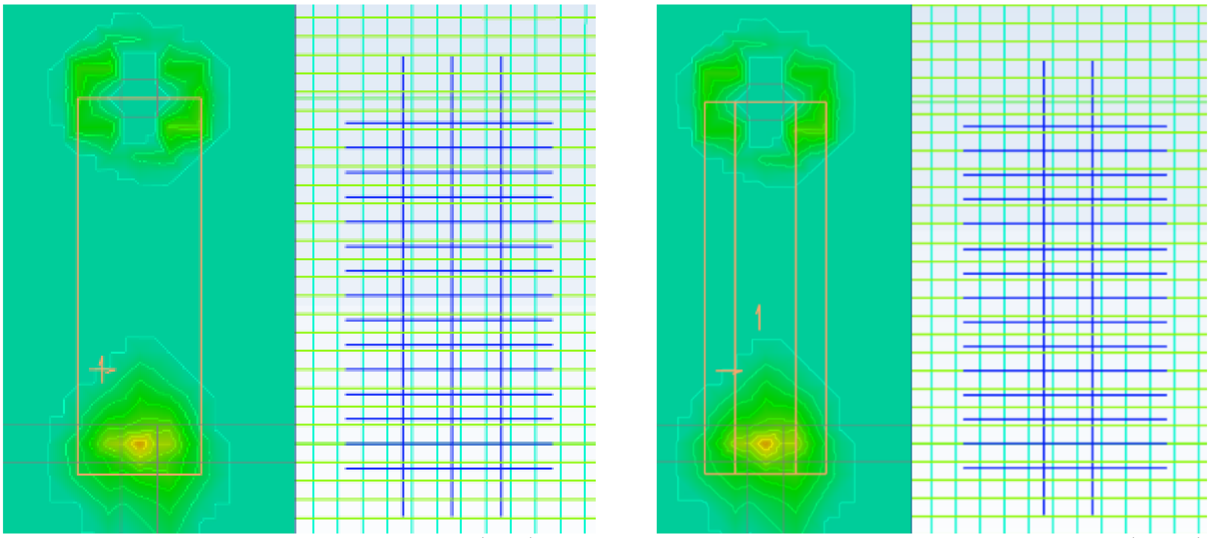


New option to generate one-way reinforcement zones

A new option 'Allow one-way reinforcement zones' is available in the Reinforcement Assumptions window. This option is used during automatic determination of reinforcement zones and causes some of the zones to be generated not as two-way, but as one or two one-way zones. Although not all zones can be modified in this way, in many cases the result is a smaller number of bars.



Location of a new option



Zone with two-way reinforcement (left) and two zones with one-way reinforcement (right)

15. Steel Connections

New features and improvements implemented in the latest version of the Steel Connection module.

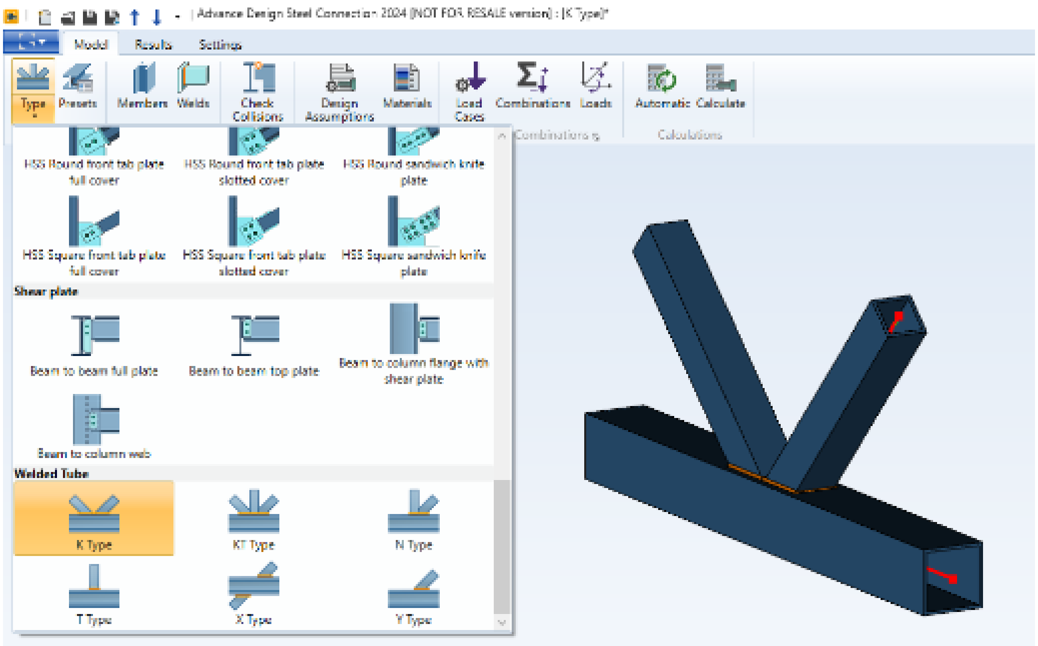
15.1. Welded truss tube connections for square/rectangular tubes

New welded truss connection type with multiple diagonals configurations of square and rectangular tubes.

We are excited to introduce a new functionality in our Advance Design Steel Connection module - the analysis of welded connections for rectangular/square tube sections, providing engineers with more flexibility in their design process. With this new connection category, users can confidently analyze and optimize connections using rectangular/square tubes, further expanding the capabilities of our software.

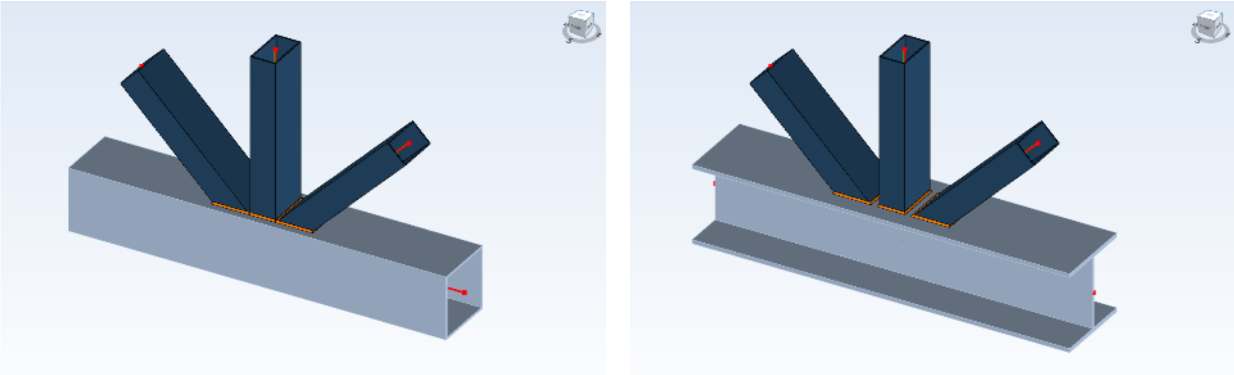
The calculations are based on the provisions of Eurocode 3 (EN 1993-1-8).

In the current version 2024, the users can verify welded truss connections for steel profiles for the following configurations: K Type, KT Type, N Type, T Type, X Type and Y Type. When creating a new connection in the standalone environment, the connection type can be selected from a list of available predefined types.



Selection of a connection type

The Welded Tubular Truss connection allows a chord (main beam) and brace elements (diagonals) to join. The main beam can be square, rectangular tubular or I steel cross-section, while the diagonals can be only square or rectangular tubular steel cross sections.



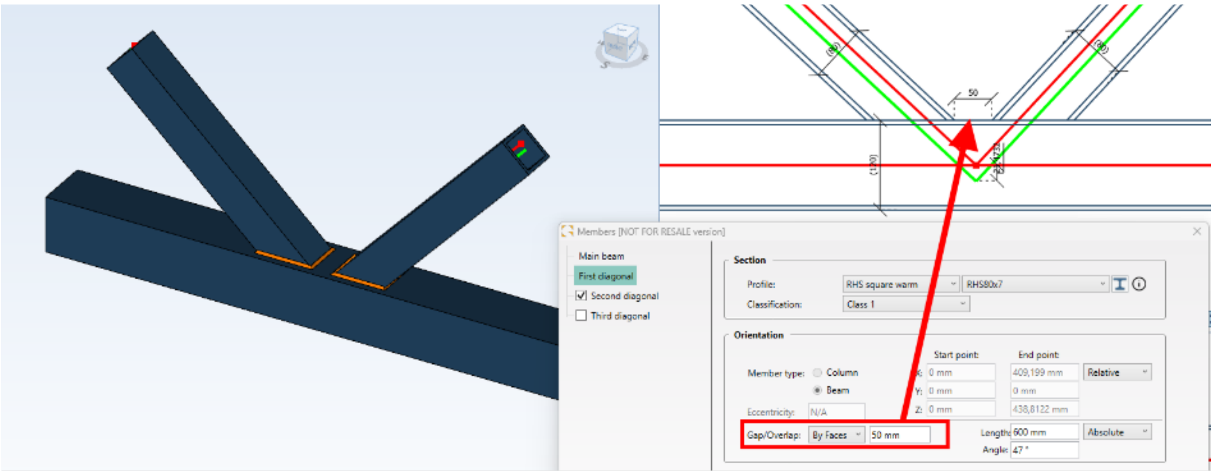
A chord modeled by a square hollow section (left) and by an I section (right)

New connections can be modeled directly on the Steel Connection module but can be also imported from the Advance Design model. In the second case, along with data on geometry, profiles, and materials, internal forces are also transferred.

NOTE: *As Eurocode mentions, the angle between the chord and diagonals cannot be less than 30 degrees. This Eurocode restriction is imposed in Advance Design and the standalone module - if the angle is smaller than 30 degrees, the connection is not generated in Advance Design, while in the standalone module, the angle is recalculated.*

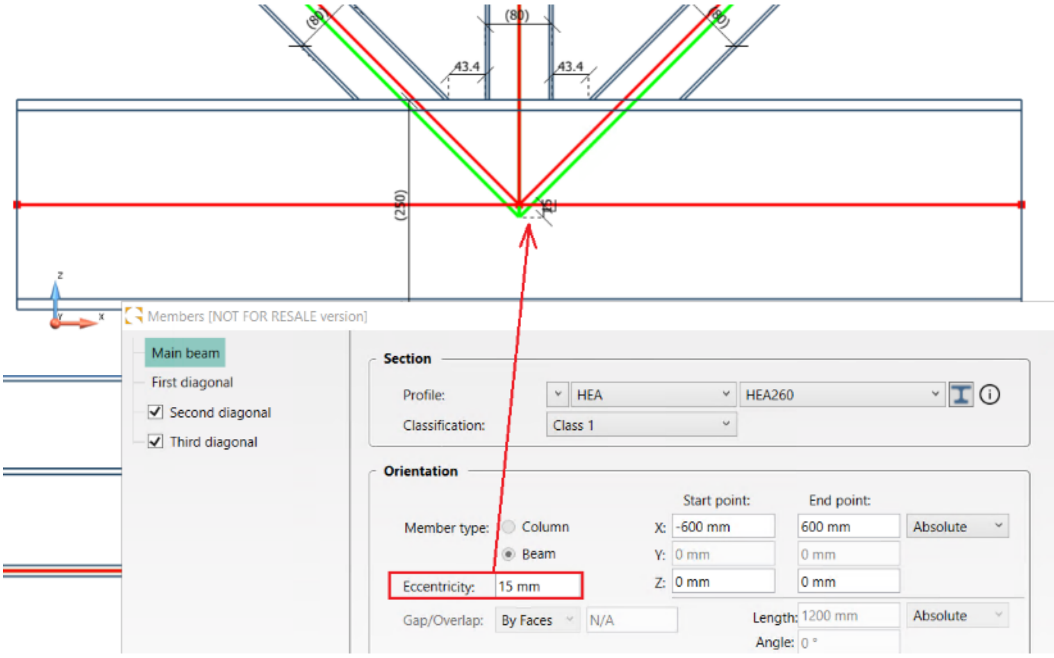
Whether the connection is defined fully in the module or imported from a FEM model, the details related to the position of the diagonals can be changed from the module.

For example, the gap/overlap offset between the axes/edges of the members, which may be different from that resulting from typical axial modeling of elements in a finite element model, can be changed from diagonals member definition.



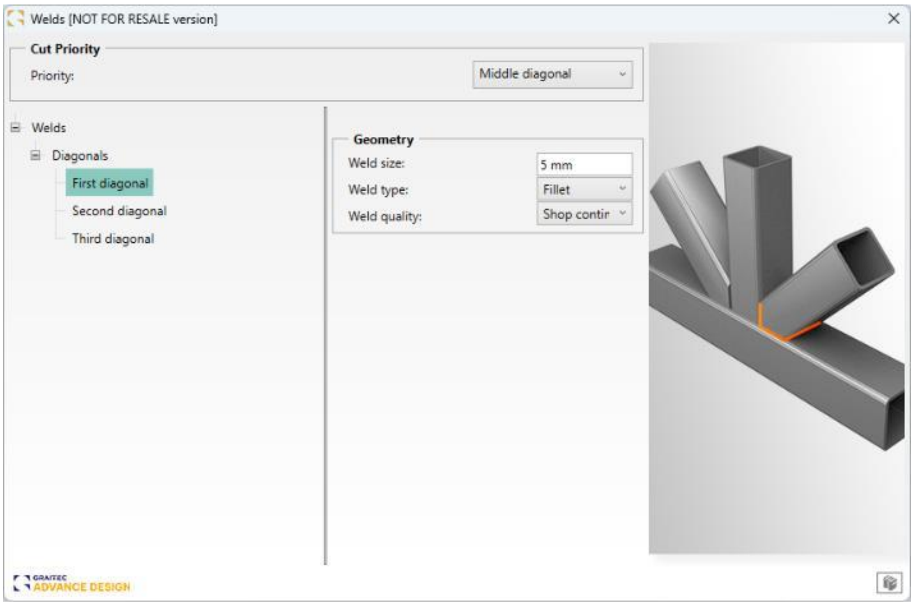
Modification of the gap between diagonals

Also, the eccentricity of the main beam can be controlled from the member's dialog.



The eccentricity of the main beam

The elements in this category of connections are joined by welds, so the user can easily set their parameters.

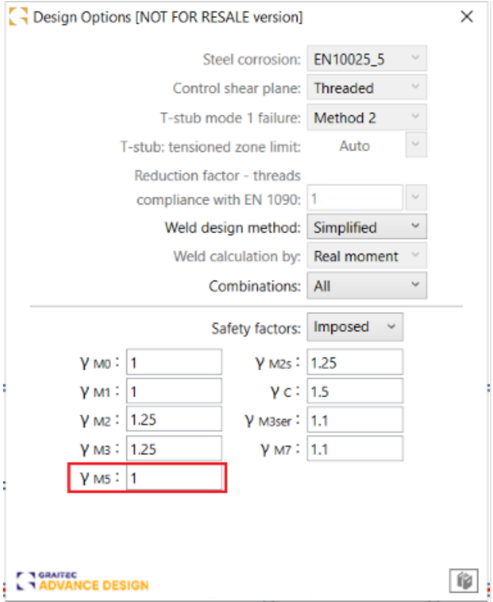


Weld parameters

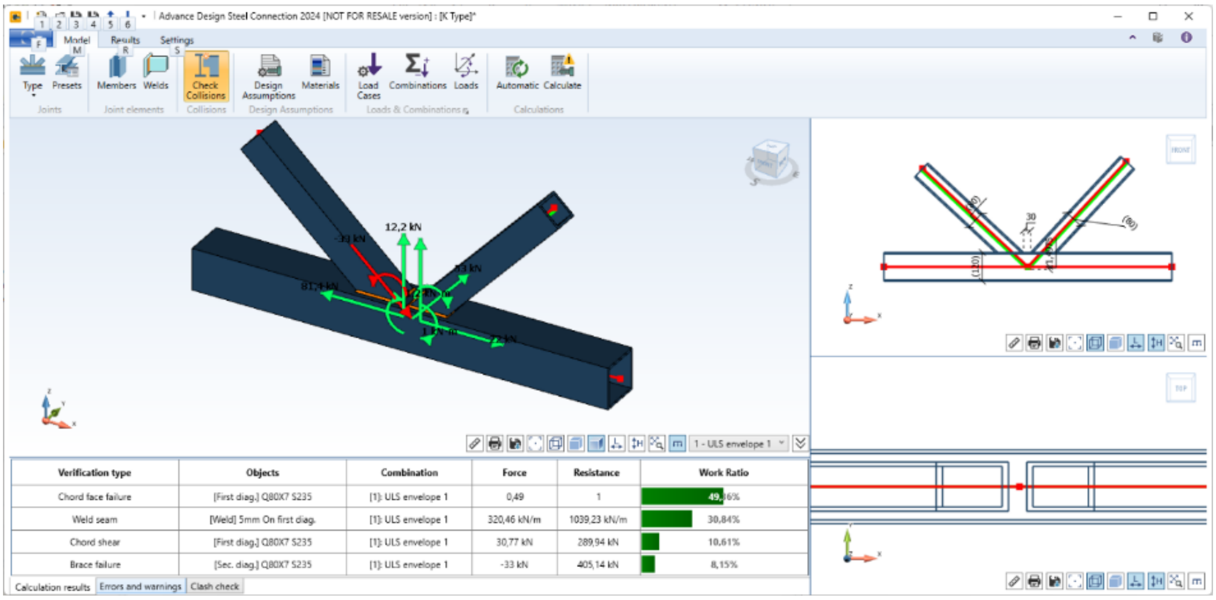
In the Welds dialog, the user can define which diagonal is cut or not by setting the "Cut Priority" option.

Cut Priority

For the Design Options a new parameter has been added: the partial safety factor for resistance of joints in hollow section lattice girder γ_{M5}, with the default value of 1. The value that can be changed if needed.



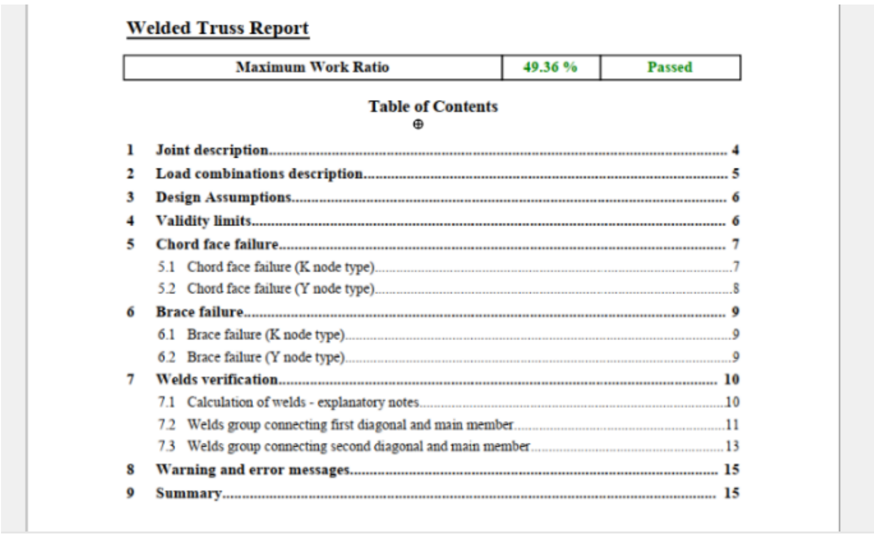
Once the loads are entered (or imported from a model in Advance Design), calculations can be performed. Basic information about verification results is available in the Info Panel.



Quickly accessible basic results in the Info Panel table

The analyses include all required verifications according to EN 1993-1-8, including weld verifications considering the geometry of the joint and the effective width of the weld.

More details of the calculations can be found in the report. It contains a detailed description of the data, as well as a set of results, for most of which detailed descriptions and formulas are available.

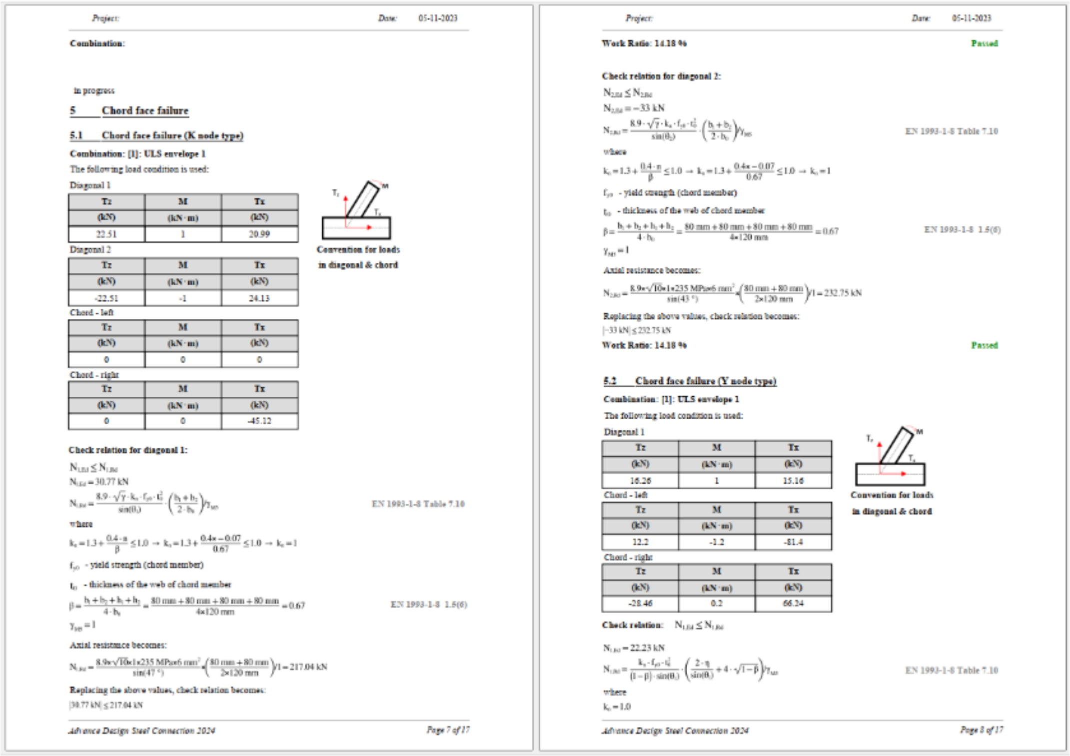


Welded Truss Report

Maximum Work Ratio	49.36 %	Passed

Table of Contents

Table of contents for a selected connection report



Project: Date: 05-11-2023

Combination:

in progress

5 Chord face failure

5.1 Chord face failure (K node type)

Combination: [1]: ULS envelope 1

The following load condition is used:

Diagonal 1

Tz	M	Tx
(kN)	(kN·m)	(kN)
22.51	1	20.99

Diagonal 2

Tz	M	Tx
(kN)	(kN·m)	(kN)
-22.51	-1	24.13

Chord - left

Tz	M	Tx
(kN)	(kN·m)	(kN)
0	0	0

Chord - right

Tz	M	Tx
(kN)	(kN·m)	(kN)
0	0	-45.12

Convention for loads in diagonal & chord

Check relation for diagonal 1:

$N_{1,Ed} \leq N_{1,Rd}$

$N_{1,Ed} = 30.77$ kN

$N_{1,Rd} = \frac{8.9 \cdot \sqrt{\gamma} \cdot k_n \cdot f_{y0} \cdot t_0^2}{\sin(\theta_1)} \cdot \left(\frac{b_1 + b_2}{2 \cdot b_0}\right) / \gamma_{M5}$ EN 1993-1-8 Table 7.10

where

$k_n = 1.3 + \frac{0.4 \cdot n}{\beta} \leq 1.0 \rightarrow k_n = 1.3 + \frac{0.4 \times -0.07}{0.67} \leq 1.0 \rightarrow k_n = 1$

f_{y0} - yield strength (chord member)

t_0 - thickness of the web of chord member

$\beta = \frac{b_1 + b_2 + h_1 + h_2}{4 \cdot b_0} = \frac{80\ mm + 80\ mm + 80\ mm + 80\ mm}{4 \times 120\ mm} = 0.67$ EN 1993-1-8 1.5(6)

$\gamma_{M5} = 1$

Axial resistance becomes:

$N_{1,Rd} = \frac{8.9 \times \sqrt{10} \times 1 \times 235\ MPa \times 6\ mm^2}{\sin(47°)} \times \left(\frac{80\ mm + 80\ mm}{2 \times 120\ mm}\right) / 1 = 217.04$ kN

Replacing the above values, check relation becomes:

$|30.77\ kN| \leq 217.04$ kN

Advance Design Steel Connection 2024 Page 7 of 17

Project: Date: 05-11-2023

Work Ratio: 14.18 % Passed

Check relation for diagonal 2:

$N_{2,Ed} \leq N_{2,Rd}$

$N_{2,Ed} = -33$ kN

$N_{2,Rd} = \frac{8.9 \cdot \sqrt{\gamma} \cdot k_n \cdot f_{y0} \cdot t_0^2}{\sin(\theta_2)} \cdot \left(\frac{b_1 + b_2}{2 \cdot b_0}\right) / \gamma_{M5}$ EN 1993-1-8 Table 7.10

where

$k_n = 1.3 + \frac{0.4 \cdot n}{\beta} \leq 1.0 \rightarrow k_n = 1.3 + \frac{0.4 \times -0.07}{0.67} \leq 1.0 \rightarrow k_n = 1$

f_{y0} - yield strength (chord member)

t_0 - thickness of the web of chord member

$\beta = \frac{b_1 + b_2 + h_1 + h_2}{4 \cdot b_0} = \frac{80\ mm + 80\ mm + 80\ mm + 80\ mm}{4 \times 120\ mm} = 0.67$ EN 1993-1-8 1.5(6)

$\gamma_{M5} = 1$

Axial resistance becomes:

$N_{2,Rd} = \frac{8.9 \times \sqrt{10} \times 1 \times 235\ MPa \times 6\ mm^2}{\sin(43°)} \times \left(\frac{80\ mm + 80\ mm}{2 \times 120\ mm}\right) / 1 = 232.75$ kN

Replacing the above values, check relation becomes:

$|-33\ kN| \leq 232.75$ kN

Work Ratio: 14.18 % Passed

5.2 Chord face failure (Y node type)

Combination: [1]: ULS envelope 1

The following load condition is used:

Diagonal 1

Tz	M	Tx
(kN)	(kN·m)	(kN)
16.26	1	15.16

Chord - left

Tz	M	Tx
(kN)	(kN·m)	(kN)
12.2	-1.2	-81.4

Chord - right

Tz	M	Tx
(kN)	(kN·m)	(kN)
-28.46	0.2	66.24

Convention for loads in diagonal & chord

Check relation: $N_{1,Ed} \leq N_{1,Rd}$

$N_{1,Ed} = 22.23$ kN

$N_{1,Rd} = \frac{k_n \cdot f_{y0} \cdot t_0^2}{(1-\beta) \cdot \sin(\theta_1)} \cdot \left(\frac{2 \cdot \eta}{\sin(\theta_1)} + 4 \cdot \sqrt{1-\beta}\right) / \gamma_{M5}$ EN 1993-1-8 Table 7.10

where

$k_n = 1.0$

Advance Design Steel Connection 2024 Page 8 of 17

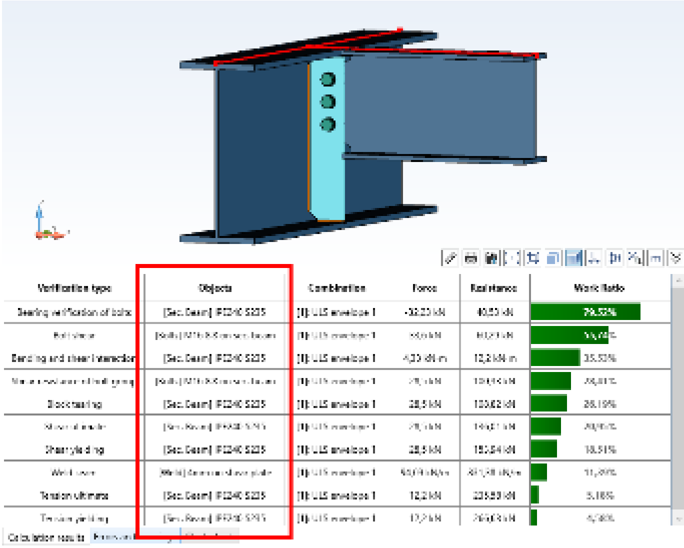
Selected pages from the results report

NOTE: *This is the first version of the module with welded truss connections, and it is planned to further develop the range of available options, configurations, and results in future versions of the module. One of the options not available for these connections in the current version is drawings.*

15.2. Improvement on the Info Panel for Shear plate joint

Easier connection verification with critical component information.

In the case of Shear plate connections (beam end connections in which the connecting element is a plate or rib working in shear), a new column is added to the Info panel for the indication of the element in which the verification is made. Thanks to this, it is much easier to verify which component is decisive for a given verification, without having to check the report.



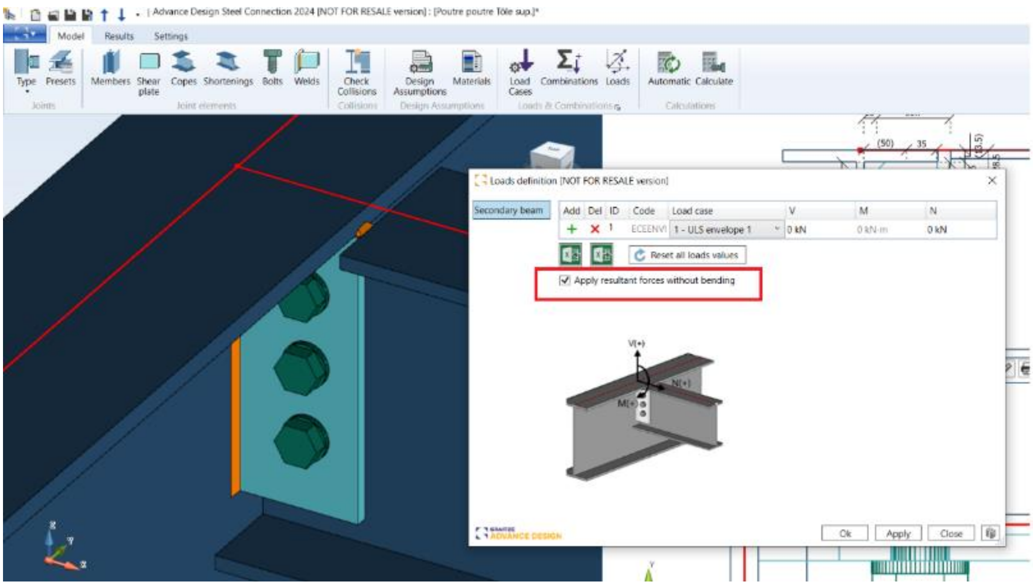
New column in Info Panel for Shear connections

15.3. New option to ignore bending moment in the calculation for hinged beams

More control over the type of forces used for verifications.

For connections where the beams are hinged on the column's flanges or to beams, such as Shear Plate or Clip Angle, the position of the V force influences the calculation. This situation is often encountered especially when the forces are imported from an Advance Design model.

Generally, in practice, the moment is considered zero at the face of the column/beam.
To control this behavior in the Steel Connection module, a new option has been added in the Loads definition dialog:



New option for hinged connections

If this option is checked, the value of the bending moment from the Loads definition tab will not be considered in the joint calculation.