GRAITEC MODELING THE FUTURE

What's New
in Advance Design 2025

Table of contents

1. Welcome to Advance Design 2025

GRAITEC is pleased to present the latest version of the leading structural analysis software – **Advance Design 2025**..

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 2025 is no exception, proving they are still top of their game in terms of providing top-level Construction, AEC, and Building Design software solutions worldwide.

This version 2025 of Advance Design is enhanced with a lot of users-centric new functionalities with high end benefits, and is articulated around few main subjects:

Computing capabilities
- Possibility for easy editing stiffnesses of planar elements using factors.
- New method of load distribution from Load areas.

Modelling capabilities
- Quick modelling of typical structures for solar panels
- Possibility of importing from Excel linear and point objects
- Visualization of foundations on model

Enhancing steel structure design capabilities
- Parametric modelling of a number of new cold-formed sections, including double C and double Sigma
- Verifications of new types of cold-formed sections according to EC3 and AISC codes.
- Possibility for defining reinforce plates on welded tube truss connections

Enhancing timber structure design capabilities acc. Eurocode
- Verification of single tapered timber beams
- Significant reduction of the calculation time of verification of timber elements.

Enhancing concrete structure design capabilities
- Increase the speed of reinforcement calculations and comfort working with RC planar elements
- New possibilities with automatic strip generation on RC slabs
- Verifications of confined and reinforced masonry walls

Enhanced user experience and the comfort of program operation
- Configuration of keyboard shortcuts
- Easy displaying envelope (min/max) values for FEM result tables
- New mechanism for creating reports using native DOCX format

Version 2025 of Advance Design also comes with a vast number of improvements and adjustments following the feedback received from thousands of users worldwide.

2. Quick list

This is a condensed / short list of new features for Advance Design 2025.

Computing capabilities

- **Surface element stiffness modifiers**
  - Possibility to get geometry orthotropic behavior of planar elements by editing the stiffness using factors.
- **Updated Q4 finite element**
  - Changed the definition of the surface 4-node finite element to a newer, more advanced one that improves results related to shear locking phenomena.
- **New method of load distribution from Load areas**
  - New method of load distribution from Load area to linear elements using FEM approach. It allows the distribution of loads from any type and shape of load.

Modelling

- **Generator for photovoltaic panel support structures**
  - Possibility for quick and easy modelling support structures used for supporting photovoltaic panels. It covers geometries used on solar farms, flat roofs, and parking places.
- **Defining planar objects by using 2 points**
  - Define rectangular surface elements, such as planar elements or load areas, using a 2-point indication.
- **Bidirectional conversion of Load area to Planar element**
  - Possibility of bidirectional conversion of Load area to Planar element. Helpful in many cases when, for example, you want to modify a model imported from another software.
- **Possibility to define in a table or import from Excel linear elements**
  - Ability to create new objects (linear or punctual) using Data grid tables and to import elements from an Excel spreadsheet.
- **Visualization of foundations on model**
  - Possibility for displaying in 3D model a visualization of foundations. This allows for the visualization of assumed, or calculated by the RC Footing module, geometrical parametric of foundations.
- **Snow generation considering snow guards on roof (Eurocode)**
  - Possibility to take snow guards into account during automatic snow load generation acc. Eurocode 1. Snow guards are elements preventing the sliding of snow located along the roof slope.

Design of Steel structures

- **Additional sections for cold-formed design**
  - Possibility of parametric modelling and performing code verifications (acc. EC3 and AISC) of a number of new cold-formed sections, including double C and Sigma.

- **Displaying Steel design results for selected element/mesh**
  - Easy selection of element for already opened shape sheet and quick check of results for selected mesh node.

- **Shape sheep results on a selected mesh**
  - Detailed steel design results on a specific portion of a linear element

Design of Timber structures

- **Timber optimization by system (Eurocode)**
  - Possibility for timber elements to run the optimization per system.

- **New entries in graphical verifications for deflection (Eurocode)**
  - Possibility for selecting for graphical postprocessing new results for deflection from the timber design results.

- **Single tapered beam (Eurocode)**
  - Verification of single tapered timber beams according to the rules of Eurocode 5.

- **Deflection for brittle finishes criterion (France)**
  - Thorough design of timber floors and ceilings at the SLS according to the French National Annex to EN 1995-1-1.

- **Reduction of calculation time (Eurocode)**
  - Significant reduction of the calculation time of verification of timber elements.

Design of Concrete structures

- **Possibility for editing Young modulus for reinforcing steel**
  - The possibility for editing the Young modulus (Es) value for steel used for reinforcement. This allows for simulating the reinforcement made of non-standard materials when analysing the theoretical reinforcement of elements.

- **Considering the local system of support for foundations**
  - Consideration of the local support layout (set according to the supported element) when transferring foundation dimensions and reaction forces to RC Footing module.

- **Reduction of reinforcement calculation time for surface elements**
  - Accelerate calculation time for calculating reinforcement for surface elements by optimizing the program code.

Results

- **Additional data properties on Result tables**
  - A set of new fields with properties to be selected when creating custom tables with results.

- **Display of extreme values on Result tables**
  - New modes of displaying values in tables with results with displaying extreme results (envelope - min/max) . This allows for easy looking for only an extreme value of a force and its localization.

- **New commands on postprocessing ribbon**
  - The ability to easily access from the Ribbon some frequently used postprocessing options.

- **New mechanism for creating reports**

    - A new report creation mechanism that allows direct generation of content in Microsoft Word (docx) format.

Enhanced user experience

- **Infill for planar loads presentation**
    - A new option to the Planar loads to fill surface by color. Helpful both when working and creating documentation, especially to present loads in a top view.
- **Quick display of object identification numbers and load values**
    - Quick display of object identification numbers and load values using the right click menu. This makes it faster and easier to manage the display of components.
- **Easier and faster creation of element selection templates**
    - Easier and faster creation of element selection templates by saving selection templates from the right click menu.
- **Defining linear releases for selected edges for multiple planar elements**
    - The possibility to set linear releases on one (or more) selected edges for multiple elements having the same geometry.
- **Ability to sort the elements from a system**
    - New options to easily sort the elements in a system using different criteria.
- **Configuration of keyboard shortcuts**
    - Possibility to configure custom keyboard shortcuts in Advance Design environment. You can check the current mapping of keyboard shortcuts and add your own.
- **Easier seismic data entry for Spain.**
    - Quick coordinate search for localities in Spain when entering seismic load data.
- **Improvements to editing material properties**
    - Automatically creating a new user material if a manual change of parameters has been made for an existing material.
- **The next stage of unification of dialog windows**
    - More convenient operation in the program thanks to a clearer interface with a uniform window design.
- **Online help in local languages**
    - Easy access to local language help content for Advance Design and design modules.

RC Design Modules

- **Export reinforcement schedules to excel files**
    - Easily transfer detailed reinforcement information from bar schedules directly to the Excel sheet.
- **New parameter for displaying the number of bars per distribution**
    - The ability to describe bars on drawings of RC elements by the total number of bars of a given bar mark or the number of bars occurring in a given distribution.
- **RC Beam - Set of improvements to facilitate the daily work**
    - Set of small improvements to all RC modules designed to work more efficiently with the module..
- **RC Beam - Drawings with cross sections on supports**

- Possibility for creating sections not only along the clear span, but also on supports.

- **RC Beam - Displaying bending details of top bars over a beam**
  - The possibility for generating on drawings bending details for top reinforcement of a beam above the beam elevation.

- **RC Beam – Low carbon concrete (Eurocode)**
  - Ability to include low carbon concrete in calculations allowing to reduce carbon footprint by using environmentally friendly materials.

- **RC Beam – Weakening Hook Factor for precast beams**
  - A possibility for imposing the value of the weakening hook factor in case of precast beams.

- **RC Beam – Expansion of torsion reinforcement report chapter**
  - The update of the torsional reinforcement chapter on the report with a link spacing verification.

- **RC Footing – Improvements to preliminary sizing of continues footing**
  - The ability to perform preliminary sizing for continuous foundations, considering its limitations with respect to the width of the pad in each direction, as well as with the ability to specify a specific eccentricity.

- **RC Column - Improvements related to the fire verification (Eurocode)**
  - Set of improvements related to the fire verifications of RC Columns, including displaying on the Info panel a set of additional results coming from the fire verification, as well as providing additional warning messages.

- **RC Column – Possibility for imposing the moment ratio for slenderness limit**
  - The ability to impose the moment ratio value used in slenderness limit calculations according to Eurocode.

- **RC Slab – Performance improvements**
  - Increase the comfort of work by significantly increasing the speed of the module, including flat work during editing and generation of reinforcement for models with a large number of finite elements.

- **RC Slab – Enhancement of the automatic strip generation mechanism**
  - Several updates addressing strip generation for RC slabs, implementing automatic strip definition in accordance with the standards laid out in the American ACI code.

Masonry Wall design module

- **Confined masonry**
  - Possibility for performing verifications of masonry walls bound with reinforced concrete columns, and verifications of masonry walls that includes reinforcement.

Steel Connection design module

- **Welded tube truss connection – Reinforcement plates**
  - Possibility for defining for the welded tubular truss connections additional plates needed to reinforce the contact area between the chord and the brace members. These plates can be two types: horizontal and lateral.

- **Welded tube truss connection - Drawings**

  - Drawing generation capabilities for welded tube truss connections.

- **Welded tube truss connection - Punching shear verification**
  - Implementing a punching shear failure verification for welded tube sections.

- **Welded tube truss connection – Set of small improvements**
  - A set of smaller enhancements to the Welded tube truss connection to increase functionality.

- **Shear plate connection - Improvement to bolts positioning**
  - Adjusted the existing behavior for the bolts positioning in the case when the secondary beam is sloped.

- **Splice connection - Check for U splice profile on Info panel**
  - The check for U splice profile is now available on the Info panel.

- **Base Plate connection - Update default properties of the hooked anchors**
  - Changes related to properties of hooks to get correct mandrel diameter for hooks.

3. New computing capabilities

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

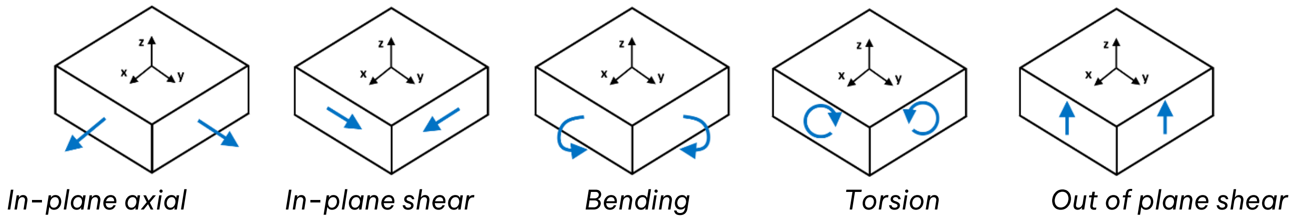
3.1. Surface element stiffness modifiers

Possibility to get geometry orthotropic behavior of planar elements by editing the stiffness using factors.

The new surface element stiffness modifiers in Advance Design 2025 gives the user an ultimate control on every stiffness aspect in shells, plates, and membranes. Advance Design is now capable of modeling special types of surface elements requiring different stiffnesses for each direction in bending, axial and shear behaviors.

Introduction

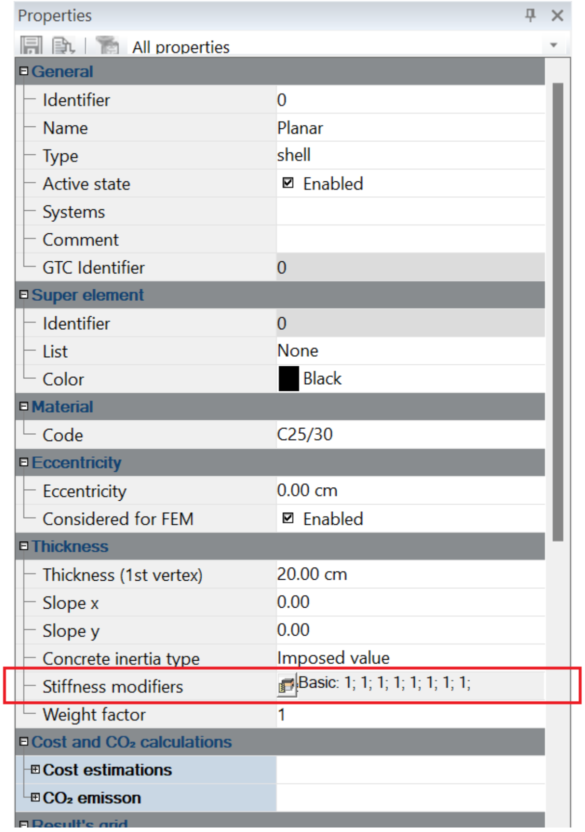
By having the capacity to modify separately each stiffness component of surface elements (shells, plates, and membranes), engineers will gain a higher control on the structural behavior of surface finite elements. Thanks to the new surface element stiffness modifiers in Advance Design 2025, designers of can now control independently the in-plane axial stiffness in each direction, the in-plane shear stiffness, the bending stiffness in each direction, the torsional stiffness and the out of plane shear stiffness in each direction.



Internal forces in surface element.

Stiffness modifiers

In Advance Design 2025, the stiffness modifiers of a surface element are accessible from its properties list.



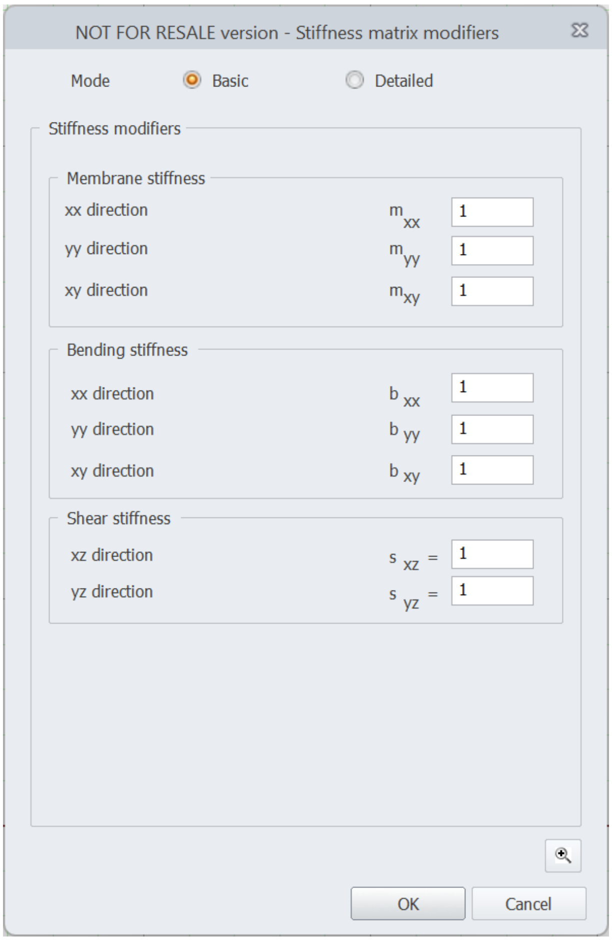
Stiffness modifiers in surface element properties list.

For concrete elements with auto calculation of cracked section, Advance Design will automatically calculate the bending stiffness modifiers and the user can modify all remaining stiffnesses. For every other case, all the stiffnesses can be modified by the user.

Two input modes are available for modification factors, the *Basic* and the *Detailed*. The choice of method is up to the user and is made after using the icon in the *Stiffness modifiers* field. Regardless of the selected method, it is also possible to change the coefficient for modifying the automatically determined element self-weight. This is done with the Weight factor option below.

- **Basic stiffness modifiers**

As the name indicates, basic stiffness modifiers are the most basic and simplest way to input the modification factors. This is the default mode in Advance Design and is intended for all users.



Basic stiffness modifiers.

In the Basic mode, modification factors are placed in three groups: membrane stiffness, bending stiffness and shear stiffness.

For membrane stiffness:

- $\mathbf{m_{xx}}$ - modification factor for in-plane axial stiffness along local x axis
- $\mathbf{m_{yy}}$ - modification factor for in-plane axial stiffness along local y axis
- $\mathbf{m_{xy}}$ - modification factor for in-plane shear stiffness

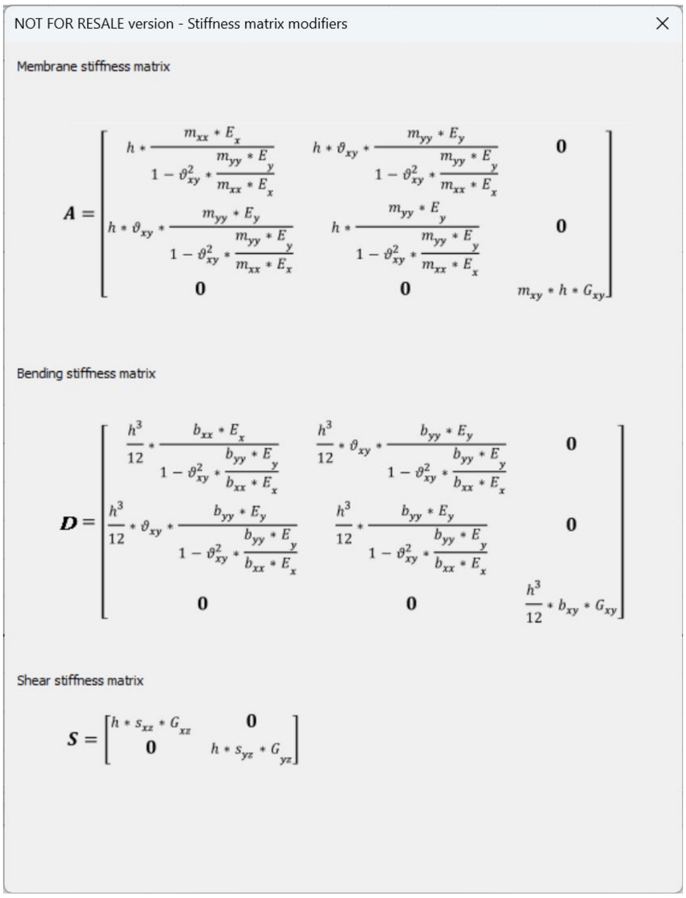
For bending stiffness:

- $\mathbf{b_{xx}}$ - modification factor for bending stiffness along local x axis (bending moment around local y axis which generates normal stresses along local x axis)
- $\mathbf{b_{yy}}$ - modification factor for bending stiffness along local y axis (bending moment around local x axis which generates normal stresses along local y axis)
- $\mathbf{b_{xy}}$ - modification factor for torsion stiffness

For shear stiffness:

- $\mathbf{s_{xz}}$ - modification factor for out of plane shear stiffness in local x axis direction
- $\mathbf{s_{yz}}$ - modification factor for out of plane shear stiffness in local y axis direction

These modification factors multiply locally the corresponding Young or Shear modulus in the stiffness matrix. For more details on how these factors work, pressing the magnifier icon will show how these factors affect the stiffness matrix calculation of the surface element.



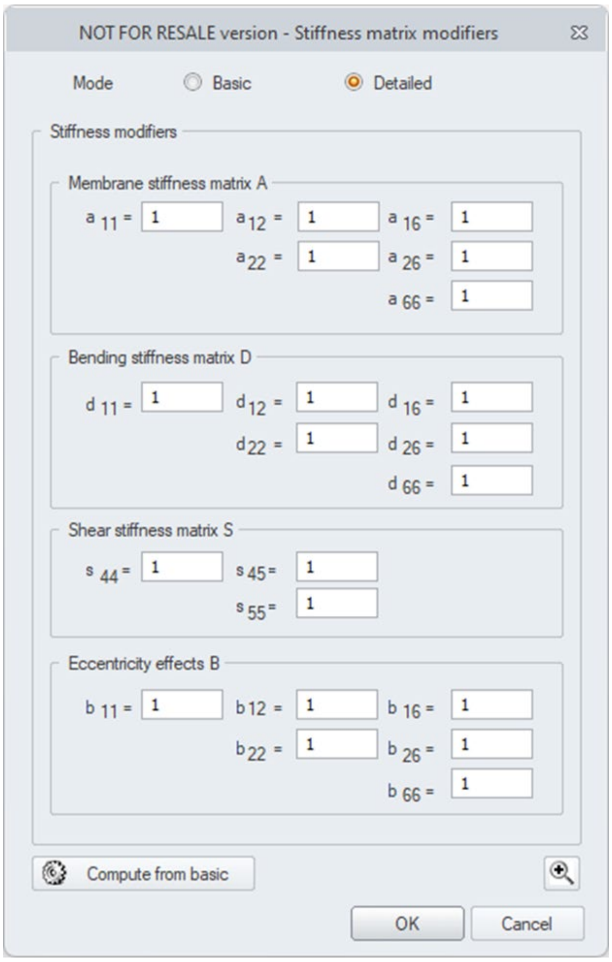
Stiffness matrix formulation with basic modifiers.

With h the thickness of the element, E_x and E_y the Young modulus on local x and y axis respectively, ν_{xy} the xy Poisson ratio, G_{xy} the in-plane shear modulus, G_{xz} and G_{yz} the out of plane shear modulus on local x and y axis respectively.

For isotropic material $E_x = E_y = E$ and $G_{xy} = G_{xz} = G_{yz} = G$

- **Detailed stiffness modifiers**

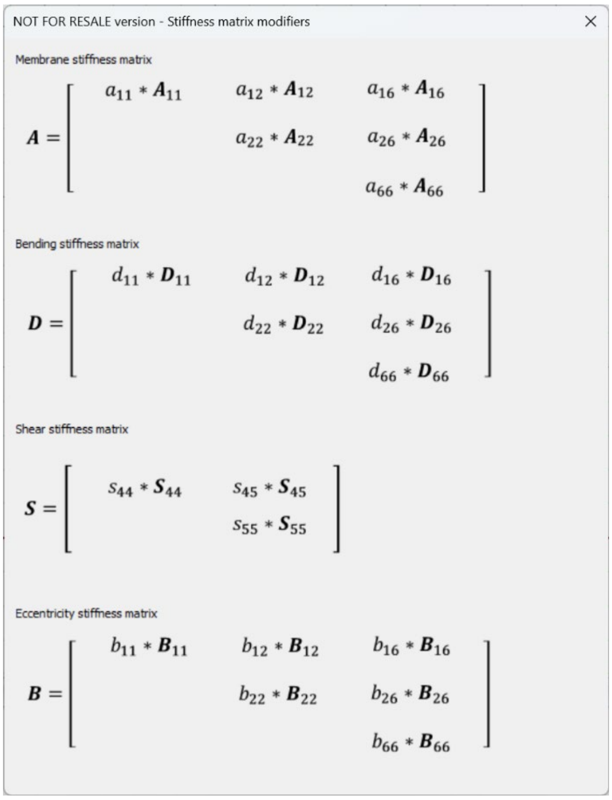
Detailed stiffness modifiers are advanced modification factors that give the user a full control on every component of the stiffness matrix. This mode is intended for users with a good knowledge in finite element theory.



Detailed stiffness modifiers

It is possible to convert the modification factors of basic mode into their detailed equivalent by clicking the '*Compute from basic*' button.

Detailed modification factors are placed in four groups: membrane stiffness matrix A, bending stiffness matrix D, shear stiffness matrix S and eccentricity effects B. In addition to controlling all components on the diagonal of stiffness matrix, the detailed mode gives the user the possibility to control the interaction stiffnesses between different degrees of freedom (off diagonal component) and the eccentricity effects (for eccentric elements). Each modification factor multiplies a specific corresponding component in the stiffness matrix. For more details on how these factors work, pressing the magnifier icon will show how these factors multiply the initial components of the stiffness matrix.

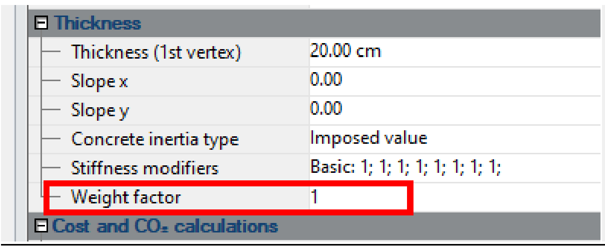


Stiffness matrix formulation with detailed modifiers.

The lower-case parameters are the advanced modification factors and the upper-case parameters are the initial components of the stiffness matrix.

Weight factor

In parallel with the introduction of stiffness modifiers, the possibility of easy modification of the self-weight of a given surface element using a coefficient was also introduced. Thanks to this coefficient, it is quite easy to obtain the self-weight of a surface element for cases in which it is necessary, without the need to create fictitious materials.

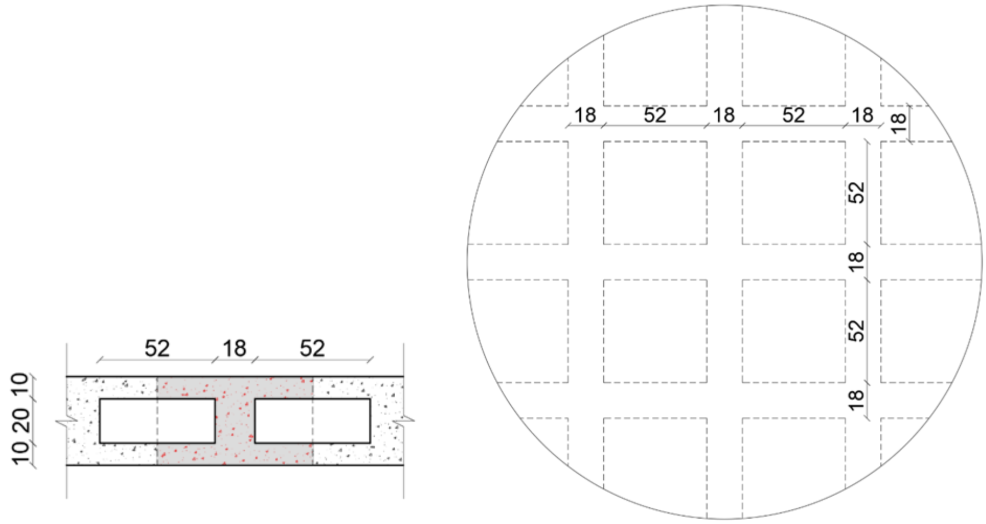


Application examples

Having a full control on every stiffness aspect of the surface element, gives the designer a wide range of practical applications. In the following we will highlight just two common applications, the first regarding modeling of voided reinforced concrete slabs and the second concerning modeling secondary reinforced concrete walls.

- **Example 1 - Voided reinforced concrete slab**

Voided slabs are reinforced concrete slabs with air pockets inside of them. These voids are created by incorporating rectangular plastic cuboids or spheres around the middle line of the slab thickness. In fact, concrete in the center of the slab section is not remarkably effective in bending resistance. So, removing it will save material and make the slab lighter without losing too much bending stiffnesses. By reducing the concrete quantity and using recycled plastic voided volumes, the voided slab is a great solution for a sustainable construction process.



Cross-section (left) and plan view (right) of the analyzed example of the voided slab. The units are cm.

To correctly model a voided slab, the presence of voids should be considered when calculating its geometric properties, stiffnesses and self-weight. Let us consider the voided slab as on the above picture and calculate its properties with the presence of voids and without it (full section).

	Full section	Voided section	Voided/full ratio
Area	2800 cm^2	1760 cm^2	**0,629**
Volume	196000 cm^3	141920 cm^3	**0,724**
Moment of inertia	373333,33 cm^4	338666,67 cm^4	**0,907**

This voided slab can be modeled as solid 40 cm thick while using the following modifiers:

- All stiffnesses proportional to the section area (membrane stiffnesses and shear stiffnesses) should be multiplied by the Area ratio

$$Area\ ratio\ = \frac{Voided\ section\ area}{Full\ section\ area} = 0.629$$

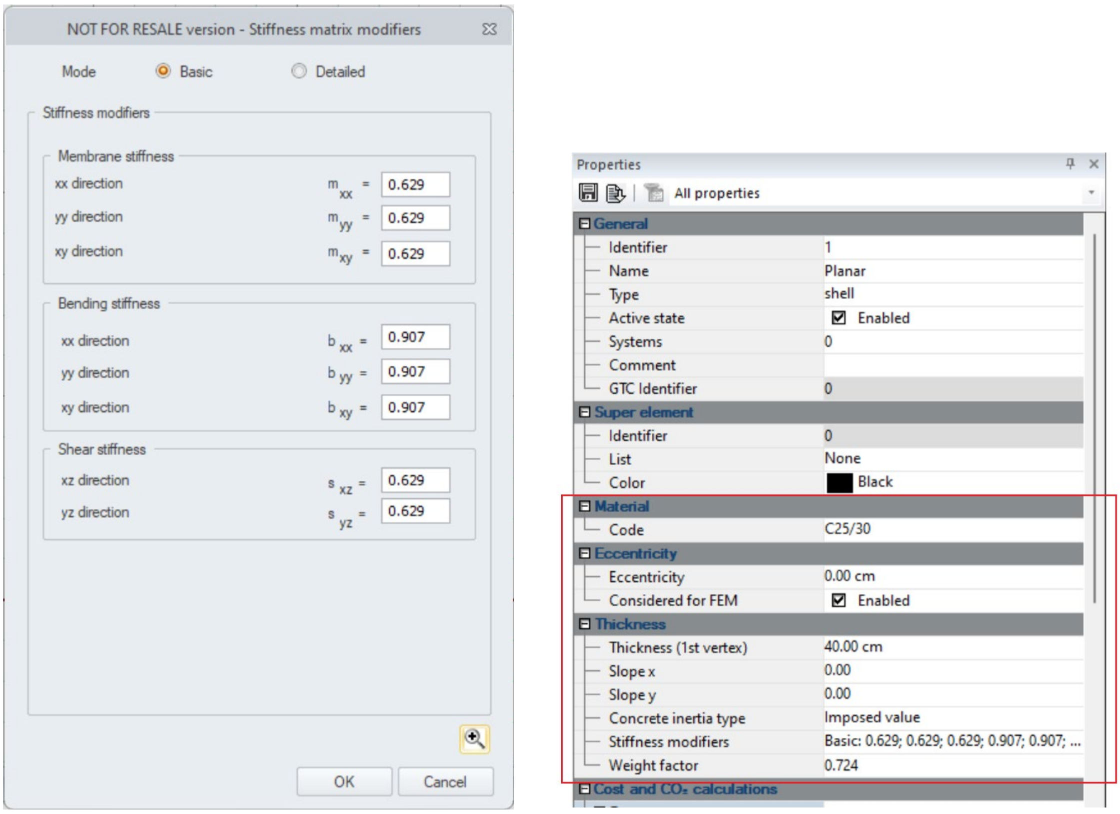
- All stiffnesses proportional to the section moment of inertia (bending stiffnesses) should be multiplied by the moment of inertia ratio

$$Moment\ of\ inertia\ ratio\ = \frac{Voided\ section\ moment\ of\ inertia}{Full\ section\ moment\ of\ inertia} = 0.907$$

- The slab self-weight should be multiplied by the volume ratio

$$Volume\ ratio\ = \frac{Voided\ section\ volume}{Full\ section\ volume} = 0.724$$

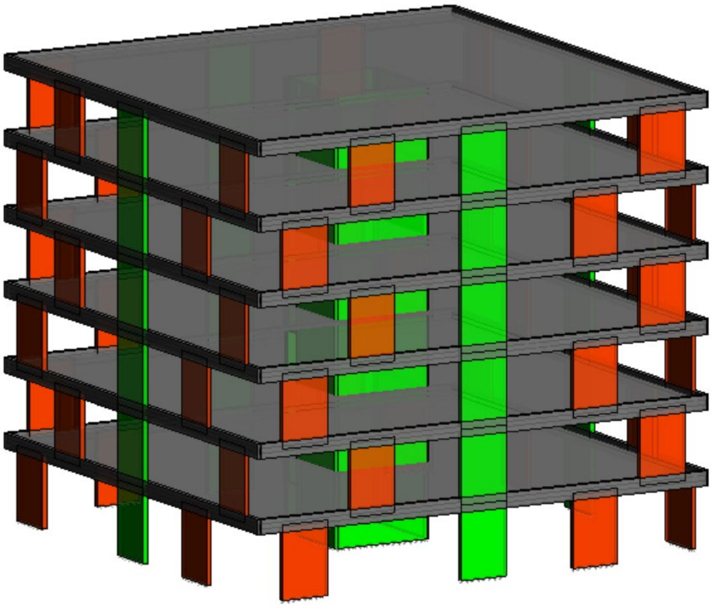
By applying the above stiffness modifiers and weight factor, Advance Design is now capable of accurately modeling the voided slab.



Basic stiffness modifiers and the properties list for the voided slab example.

- **Example 2 - Secondary reinforced concrete walls**

In building design, it is sometimes common that architecture and facade constraints result in irregular structural lateral resisting system. For example, the building as on a picture below, has discontinuous concrete walls on all its sides (elements in red) while also having a continuous central core and side walls (elements in green). In such cases, seismic codes advise to consider these discontinuous walls as secondary members that do not participate in the lateral resisting system of the structure. So, for this structure, the green walls resist gravity and lateral loads while the red walls carry only gravity loads (do not resist to lateral forces). Modeling the accurate behavior of these red walls is now possible with surface element stiffness modifiers.



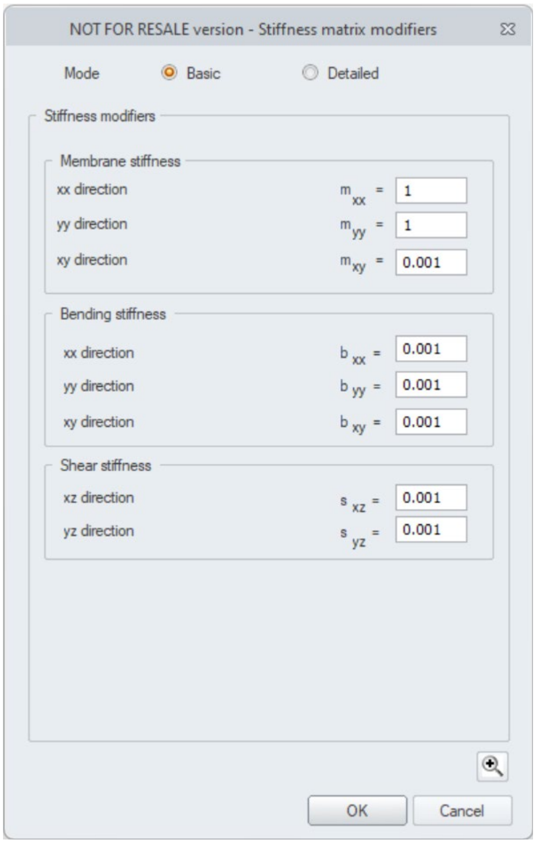
Building with discontinuous concrete walls.

To stop the red walls from carrying lateral forces along their strong axis, we should make their in-plane shear stiffness negligeable. This is possible by setting the basic membrane stiffness modifier $m_{xy} = 0.001$ (absolute zero cannot be placed because it will make the shell element unstable).

To stop the red walls from carrying lateral forces along their weak axis two options are possible. Option 1: make pin side releases at the walls edges. Option 2: make the walls bending stiffnesses and out of plane shear stiffness negligeable. This is possible by setting the basic bending and shear stiffness modifiers to 0.001 (absolute zero cannot be placed because it will make the shell element unstable).

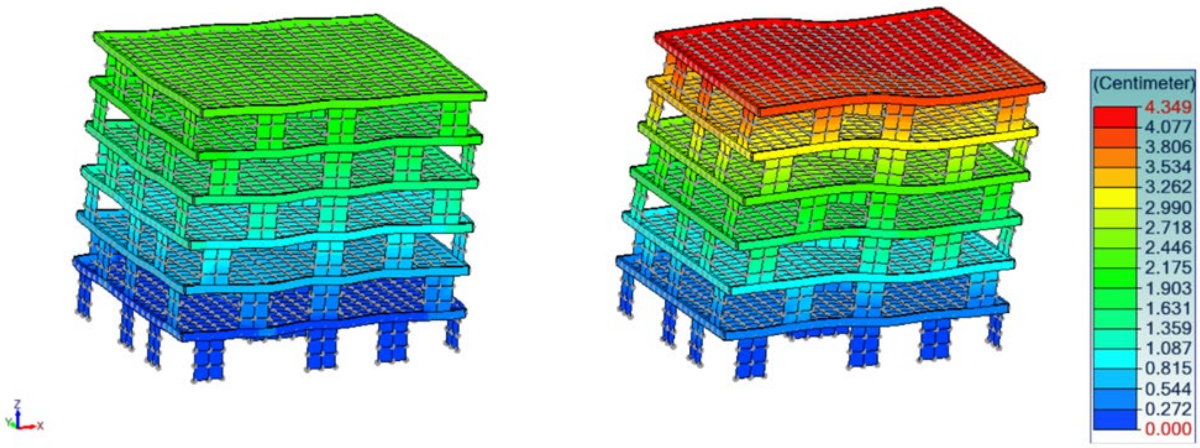
$$b_{xx} = b_{yy} = b_{xy} = s_{xz} = s_{yz} = 0.001$$

Since we are putting in focus the use of stiffness modifiers, we will go with option 2.



Red walls stiffness modifiers.

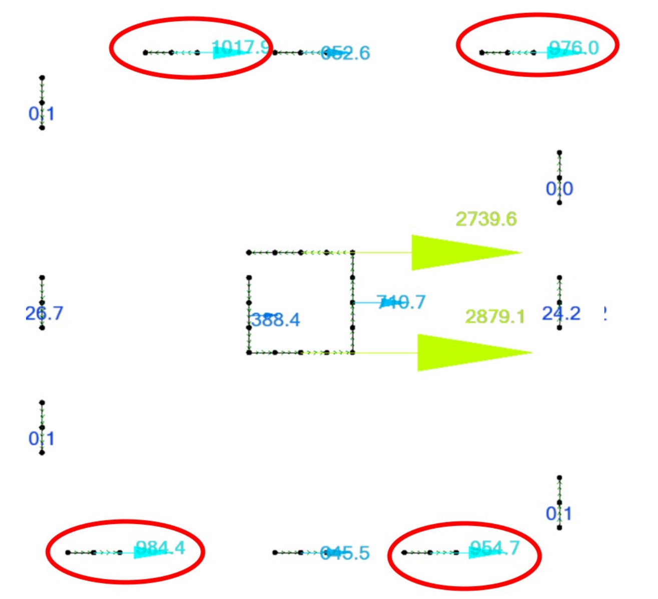
To display how Advance Design is now capable of accurately modeling the secondary concrete walls we will do a comparison between the structural results of a model with no stiffness modifiers and a model with the modifiers set for the red walls. Both models are subjected to the same uniform lateral forces applied at all stories. Figure below shows the displacement of structures under the lateral loads



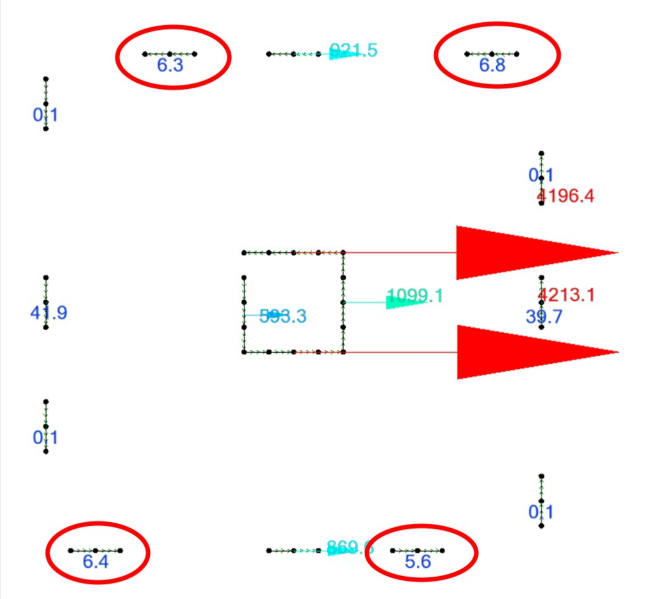
Displacement comparison for structures subjected lateral loads.
Model without stiffness modifiers (left) and with stiffness modifiers (right).

It is clear that preventing the discontinuous red walls from resisting to horizontal forces will reduce the lateral stiffness of the building. Thus, the structure will have an increased lateral displacement. Figures below shows the base shear transmitted to supports when the structures are subjected to the lateral loads.

Model without stiffness modifiers	Model with stiffness modifiers



*Base shear on the structure with red walls *<u>*resisting*</u>* to lateral loads.*



*Base shear on the structure with red walls *<u>*not resisting*</u>* to lateral loads.*

By comparing above figures we can clearly see how the load path of lateral forces has changed when the red walls did not resist to lateral forces. In this case, the lateral load was carried solely by the central core wall and continuous side walls (green wall elements). Thus, we notice a remarkable increase of base shear on green walls supports in the right figure.

3.2. Updated Q4 finite element

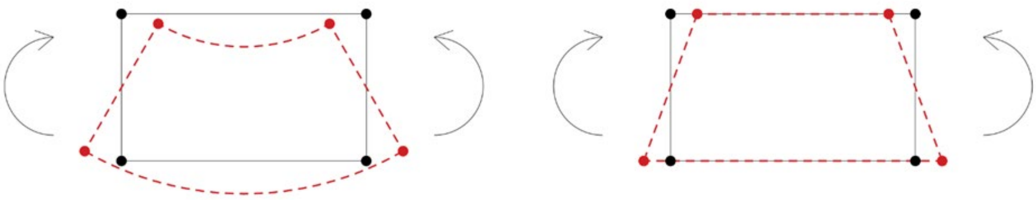
Changed the definition of the surface 4-node finite element to a newer, more advanced one that improves results related to shear locking phenomena.

Advance Design 2025 introduces an enhancement to the in-plane shear formulation of the basic type of surface finite element that improves the accuracy of some results especially for poor mesh models, while maintaining the same excellent calculation performance.

Introduction

Q4 mesh elements are the most commonly used type of finite elements for modeling shells, plates, and membranes. They consist of a quadrilateral surface element with a node on each corner (in total 4 nodes). Q4 elements are popular due to their simple calculation. In fact, they use linear bidimensional shape functions which makes them quite easy to analyze.

However, this simplification comes at a price. Linear shape functions cannot correctly capture the curvature of the Q4 element edges when subjected to in-plane bending. Instead of deforming with a curvature on the edge, Q4 elements will deform into a trapezoidal shape when subjected to in-plane bending.



Comparison between real deformation (left)and Q4 element deformation (right).

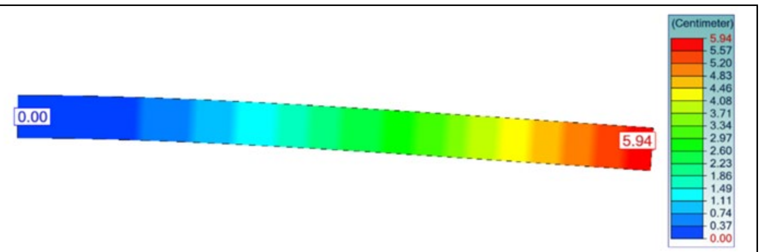
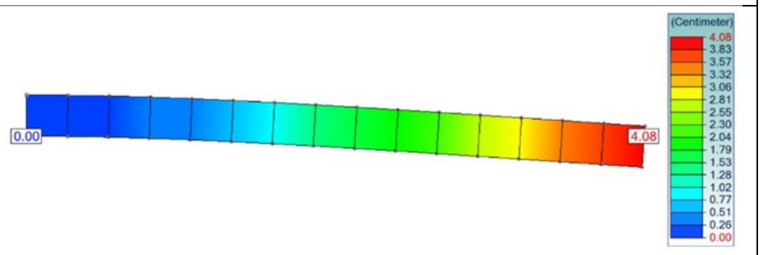
This inaccuracy in deformation will result in an overestimation of the in-plane shear carried by the finite element while underestimating the in-plane bending. In other words, the element will appear stiffer in bending and will displace less. This phenomenon is known as in-plane shear locking.
The inaccuracy due to in-plane shear locking is negligeable for shells and membranes with enough mesh density (a logical mesh size with no less than 3 mesh elements per direction is enough). So, shells and membranes with coarse meshing are the most susceptible to in-plane shear locking problems.
By adopting a new formulation for the in-plane shear calculation based on [H.Choi & P.Lee 2024], Advance Design 2025 is now capable of better handling this in-plane shear locking problem even for structures with coarse meshing. This improvement has no impact on calculation cost, so, Advance Design 2025 will give more accurate results while maintaining its top performance.

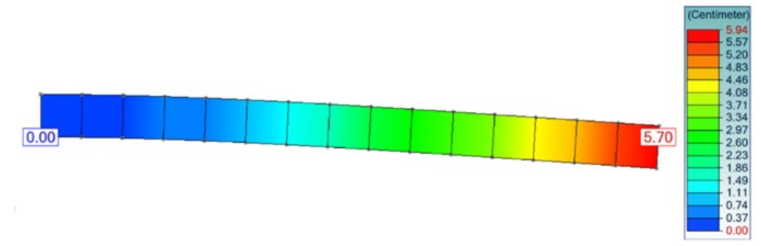
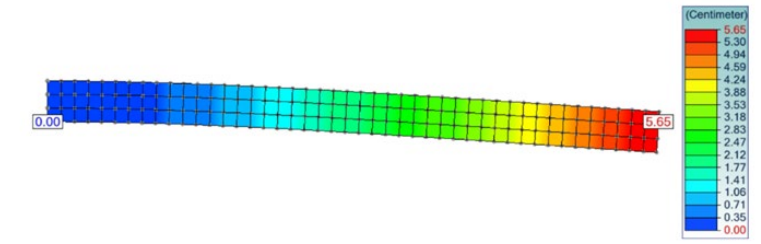
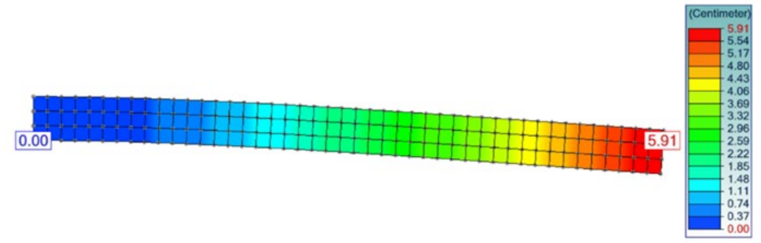
Comparison between AD 2025 and previous versions

To highlight the improvement of Advance Design 2025 in managing the in-plane shear locking, we will consider a cantilever deep beam element with a span of 15 m and a rectangular cross-section of 30 cm width and 100 cm depth. This deep beam will be subjected to its self-weight. In a first approach, to get benchmark reference results, the structure will be analyzed as a beam line element. Then, the deep beam will be modeled by shell elements with a Q4 meshing in a previous AD version (2024.1.2) and in the new AD 2025 version. For models with shell elements, we will start with a mesh size of 1 x 1 m. This is considered as a coarse mesh size to the studied structure since we will have one single mesh element along the entire depth of the structure. This initial coarse meshing should bring out the in-plane shear locking problems. After that, we will use a good mesh size 0.33 x 0.33 m which should make the structure less sensitive to shear locking even in previous AD versions.

- **Deflection comparison**

A deflection comparison is conducted between the beam model, the shell model (with coarse and good meshing) in previous and new versions of Advance Design. The results are displayed in a table below.

Beam model	5.94 cm	
Shell model (coarse mesh) Advance Design 2024.1	4.08 cm	

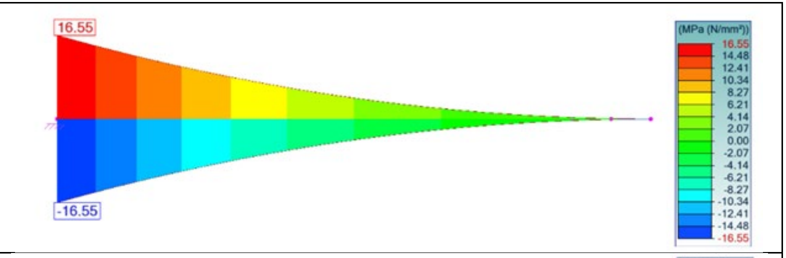
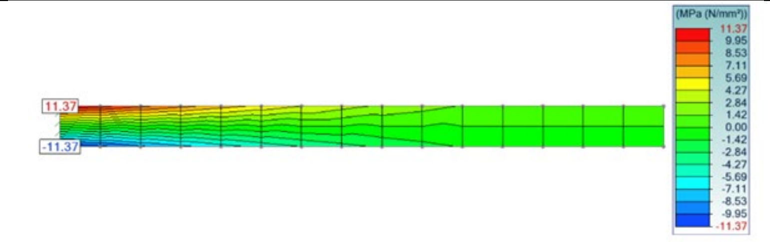
Shell model (coarse mesh) Advance Design 2025	5.70 cm	
Shell model (good mesh) Advance Design 2024.1	5.65 cm	
Shell model (good mesh) Advance Design 2025	5.9 cm	

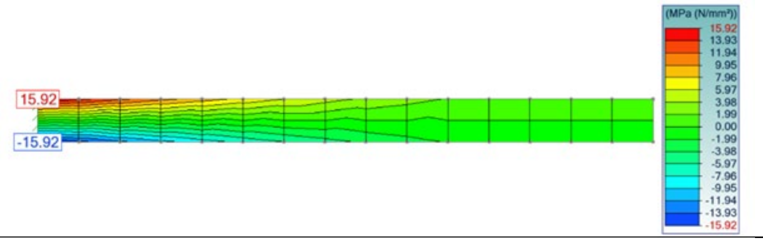
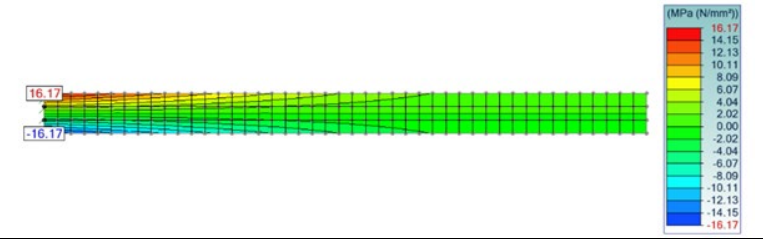
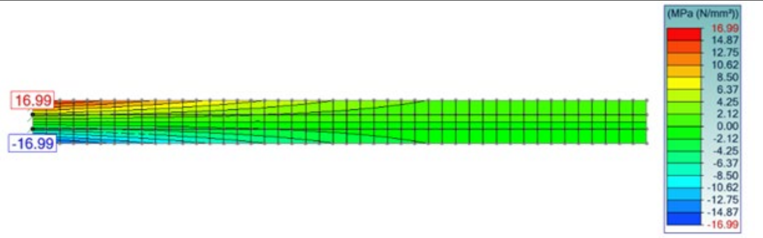
It is clear that for a coarse meshing, previous versions of Advance Design encountered in-plane shear locking problems and the structure was stiffer than it should be which resulted in an inaccurate lower deflection. When a better mesh was adopted we got results closer to the benchmark (beam model) values.

Advance Design 2025 is a lot better in handling in-plane shear problems even with a coarse mesh. For both coarse and good meshing, AD 2025 gives accurate results that are close to the benchmark beam model.

- **Axial stress comparison**

Now, a bending axial stress comparison is conducted between the different models.

Beam model	16.55 MPa	
Shell model (coarse mesh) Advance Design 2024.1	11.37 MPa	

Shell model (coarse mesh) Advance Design 2025	15.92 MPa	
Shell model (good mesh) Advance Design 2024.1	16.17 MPa	
Shell model (good mesh) Advance Design 2025	16.99 MPa	

The results and conclusions are similar to the previous example. For a coarse mesh, previous versions of Advance Design encountered in-plane shear locking problems and the axial stress in the structure was underestimated. When a better mesh was adopted we got results closer to the benchmark (beam model) values.

Advance Design 2025 is a lot better in handling in-plane shear problems even with a coarse mesh. For both coarse and good meshing, AD 2025 gives accurate axial stress results that are close to the benchmark beam model.

It should be noted here again that for Q4 mesh elements, in-plane shear locking causes inaccuracy problems only in shells and membranes having a coarse meshing.

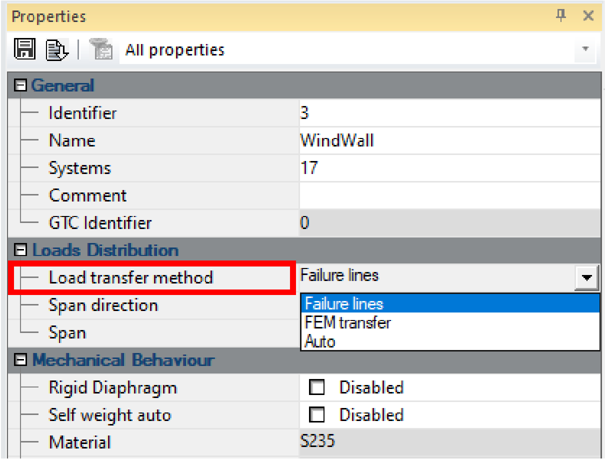
3.3. New method of load distribution from Load areas

New method of load distribution from Load area to linear elements using FEM approach. It allows the distribution of loads from any type and shape of load.

Advance Design uses Load Areas to transfer loads from surfaces such as roofing and walling to the supporting linear element. Previously, the load distribution from load area to linear elements was conducted solely according to failure lines theory. Despite its practicality, this approach is mostly accurate for quadrilateral load area with uniform load distribution all over it. In such cases, we get the classic analytical envelope load distribution. However, when dealing with a more complex load system and more complex load area geometry, the analytical method is not able to represent the actual distribution of forces on the bars, as the inaccuracy of the failure line theory increases.

To improve load distribution results in cases where the load was irregular or defined only on a part of the load area, a new option has been introduced in the 2024.1.2 update to convert the defined load to equivalent uniform load, which for typical regular systems has improved the distribution of forces on members. (For more information, see *What's New in Advance Design 2024.1.2*).

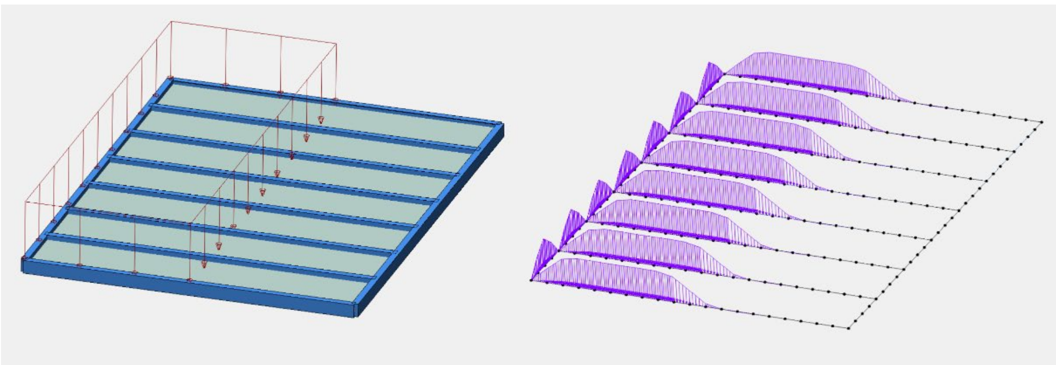
Nevertheless, in the latest version of Advance Design 2025, to cover all loading scenarios with good precision, an additional new finite element load distribution algorithm has been introduced. The new algorithm can be activated individually for Load area, and for this purpose in its parameters appeared a field with a choice of three options: Failure lines, FEM transfer and Auto.



For **Failure lines**, as the name indicates, it is a load transfer based on failure lines theory and is most suitable for quadrilateral load areas with uniform load distribution. **FEM transfer** is for FEM load distribution and is suitable for all other loading scenarios. **Auto** lets the software choose the proper option (failure lines or FEM transfer) based on the applied loadings and geometry. Auto option is the default .

The new FEM transfer option is based on a background FEM analysis in which the following steps are conducted for each load area:

- The load area and its surrounding support elements (beams and columns) are isolated.
- The load area is considered as a thick shell made of RIGID material.
- The supporting beams and columns are considered as fixed line and point supports respectively.
- Loadings are applied on this model and a general meshing is conducted.
- This background model is analyzed and the resulting forces on its supports are converted to line loads and applied on the corresponding structural elements of the global structure.

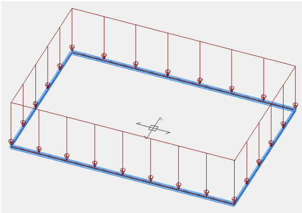


For all transfer methods, we can also indicate the span direction (x, y, xy, Other), however, for the selection of Other, the method in which the distribution of the area of influence on each edge can be modified, is available only for the Failure lines method.

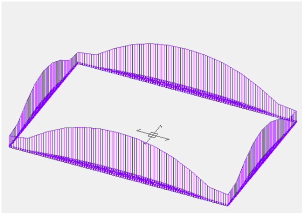
Note that the shape of the load distribution using the FEM transfer method is affected by the density of the FEM mesh, defined in the global mesh settings. In addition, it should be considered that the use of the new method may slightly increase the time of model generation and the calculations themselves.

Example

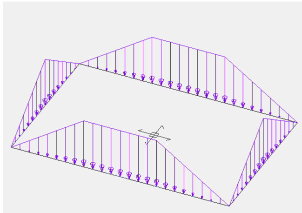
Let us see examples of the effects of load transfer to linear elements for a quite simple geometry. For Failure modes, the results are given with the option of converting surface loads to uniform loads turned off and with the conversion option turned on (the option available from the version 2024.1.2).



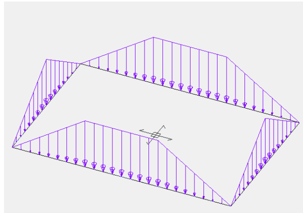
Defined load



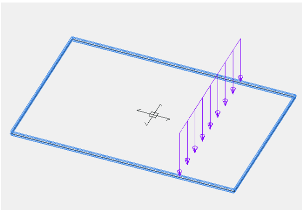
FEM transfer



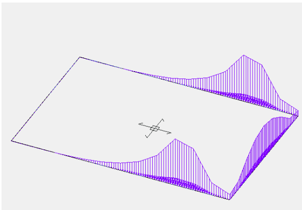
Failure lines (conversion OFF)



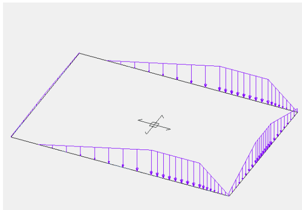
Failure lines (conversion ON)



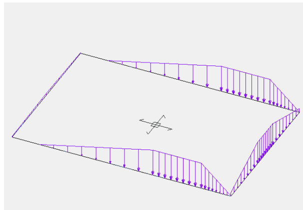
Defined load



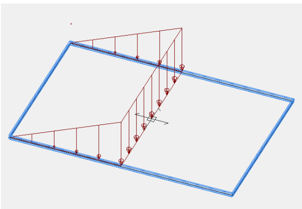
FEM transfer



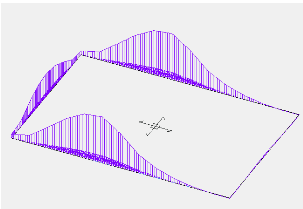
Failure lines (conversion OFF)



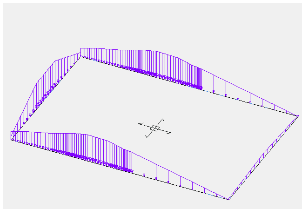
Failure lines (conversion ON)



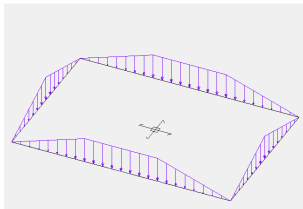
Defined load



FEM transfer



Failure lines (conversion OFF)



Failure lines (conversion ON)

4. Modelling

A series of new features and improvements related to the preparation of the calculation model.

4.1. Generator for photovoltaic panel support structures

Possibility for quick and easy modelling support structures used for supporting photovoltaic panels. It covers geometries used on solar farms, flat roofs, and parking places.

As everyone becomes more aware of the need to obtain energy from renewable sources, the presence of photovoltaic installations is also increasing. Looking around, it is clear that photovoltaic panels can be seen more often, whether on the roofs of buildings, home gardens, parking spaces or, finally, in dedicated solar farms. In all cases, PV panels are supported on a structure made mostly of steel profiles, and the geometry of the support structures is in the vast majority of cases similar. At the same time, for reasons of the need to adapt to the conditions of a given location and the need to obtain an economical solution, these structures are usually individually designed.

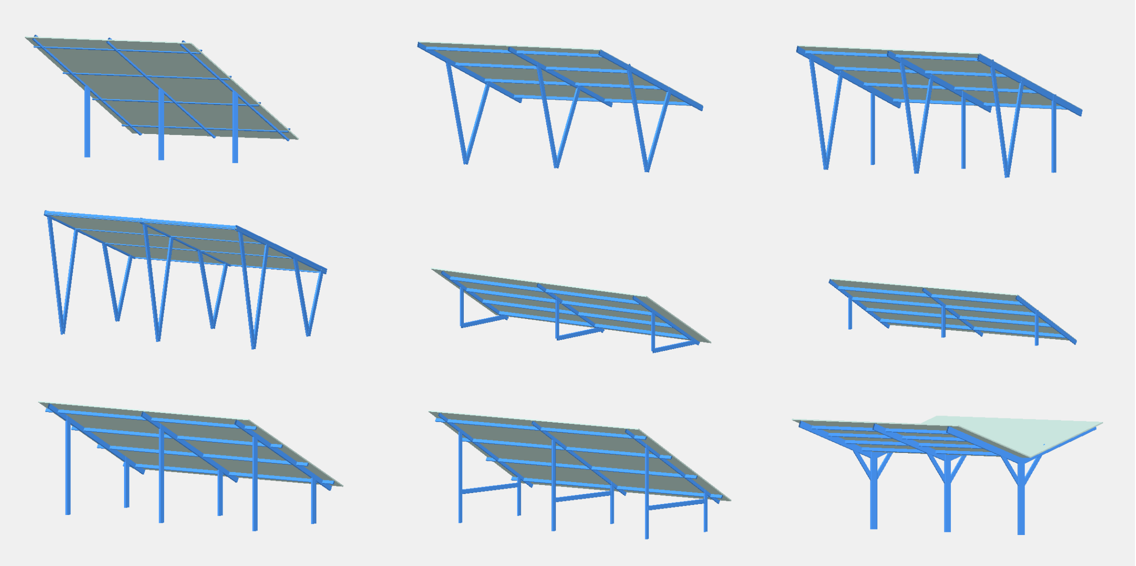
Unsurprisingly, more and more Advance Design users are designing these types of structures, taking advantage of its capabilities, including easy modeling, fast load definition, as well as the ability to optimize steel structures. To further save valuable user time during the most labor-intensive part, which is modeling the structure, the latest version of Advance Design software introduces a new geometry generator tailored for photovoltaic (PV) panel support structures. The introduction of the new generator marks a significant advancement in Advance Design, empowering engineers to design PV panel support structures efficiently.

Key features of a new generator

- **Versatile geometry modeling:** The generator allows users to model nine different types of geometries, covering the most common solutions used in PV panel support constructions. These geometries are adaptable, enabling the definition of various geometric subtypes to suit diverse project requirements.

  Available geometry types:

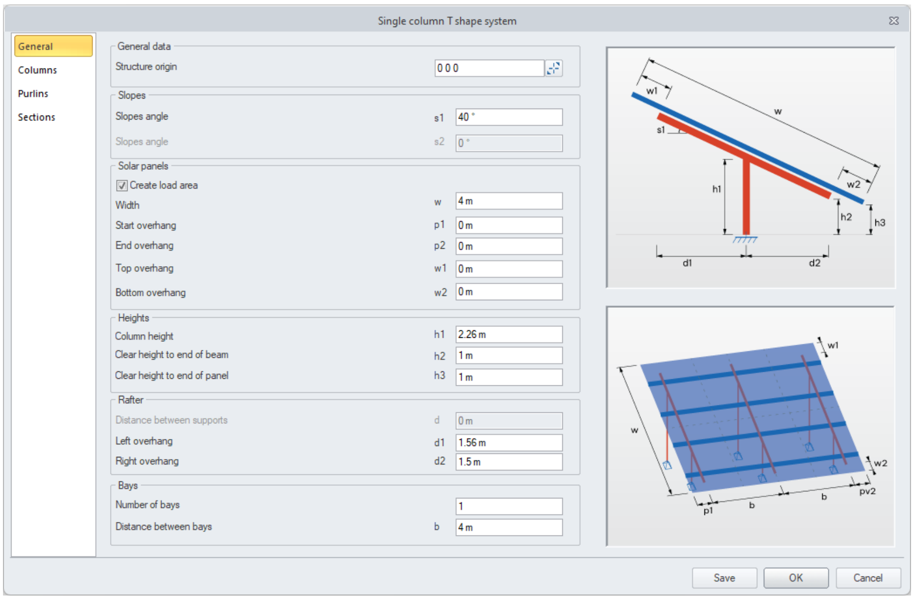
  - Sigle column T shape system
  - Two columns V system
  - Three column VI system
  - Four column VV system
  - Supported beam system with horizontal bracing
  - Supported beam system
  - Two column system
  - Two column system with horizontal bracing
  - Single column Y shape system



- **Wide editing capabilities:** Most geometric parameters can be modified, allowing for modeling flexibility. For example, columns can be moved and inclined as desired, and can be reinforced by additional bracing bars. In addition, you can define any number of spans, any number of purlins, as well as extensions or shortenings of purlins and beams.

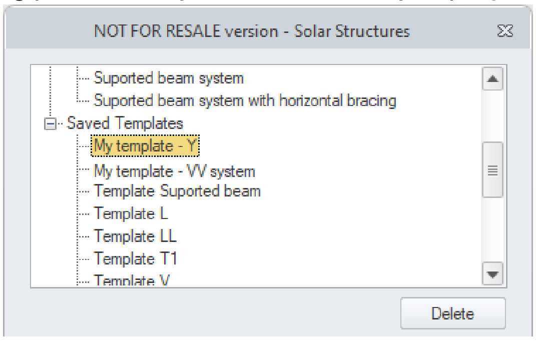
- **Wide range of applications**: The generated geometries are applicable across a spectrum of projects, including large-scale solar farms, residential rooftop installations, and carport structures. This versatility ensures that engineers can utilize the tool across different contexts, promoting widespread adoption of solar energy solutions.

- **Efficient and intuitive interface**: Designed for ease of use, the module offers a user-friendly interface that simplifies the modeling process. Engineers can quickly navigate through the options and customize the geometry according to specific project needs, saving valuable time and resources.



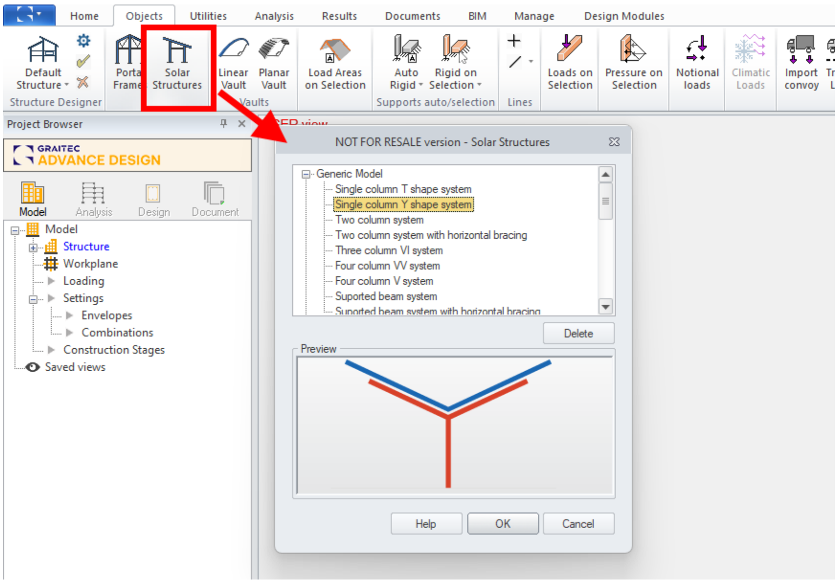
- **Comprehensive output generation**: In addition to generating geometric models, the generator produces load areas, supports, as well as material data and design template, making the design process much faster. Supports can be defined as rigid or pinned, placed at the ends of columns, and in the case of vertical columns, it is possible to define the length of the ground embedment and the definition of an elastic linear support.

- **Customization and reusability:** Users have the flexibility to save their input data as custom templates for future use. Any number of customized versions of the entered data can be saved, building a custom library of typical solutions. This feature streamlines the design process for similar structures, enhancing productivity and consistency in project delivery.

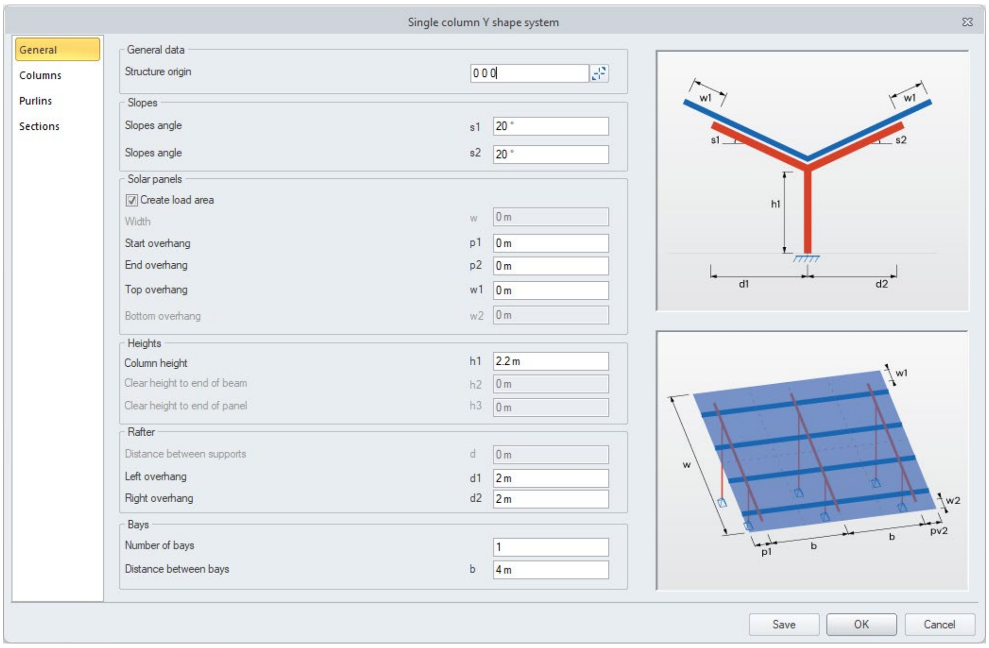


How to generate a new structure

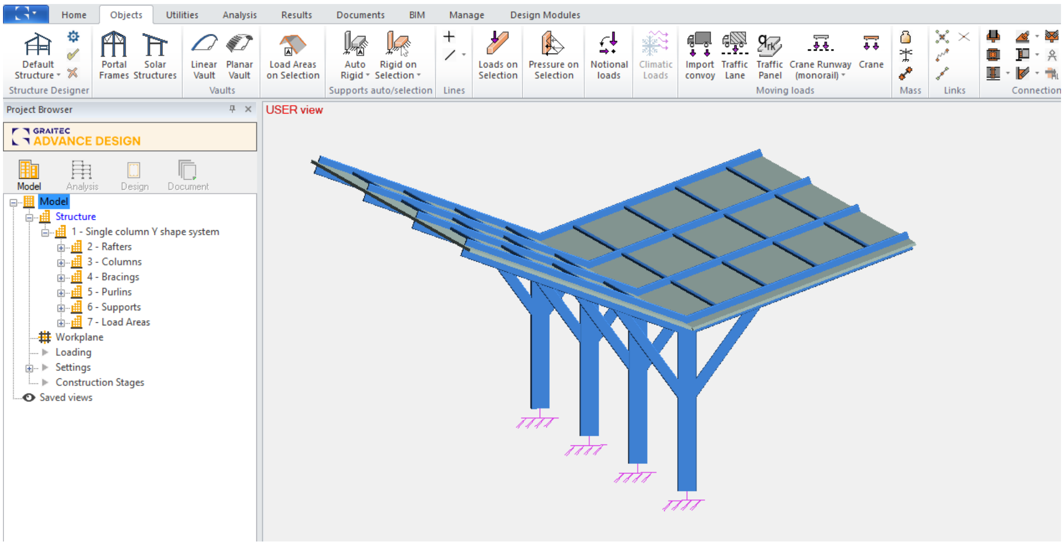
1. On the **Object** ribbon select the **Solar Structures** command



2. Select the desired type from the list in the manager window
3. In the dialog window, adjust the dimensions and parameters according to your needs.



4. [Optional] To save the entered data as a template for the next application, press **Save** and enter a name for the new template. Saved templates can be seen in the manager window.
5. Press **OK** – the new model is automatically created. Note that the generated elements are automatically grouped into the appropriate systems, which makes further work much easier.



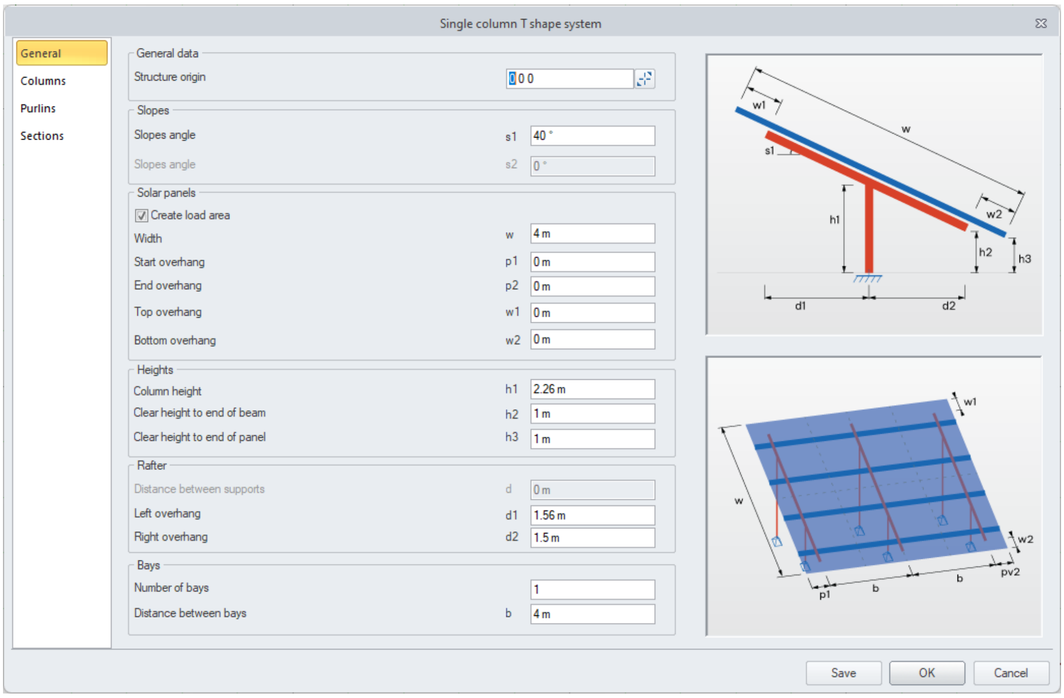
Additional information about the available parameters

The generator contains four tabs with parameters for model definition: General, Columns, Purlins and Sections.

- **General**

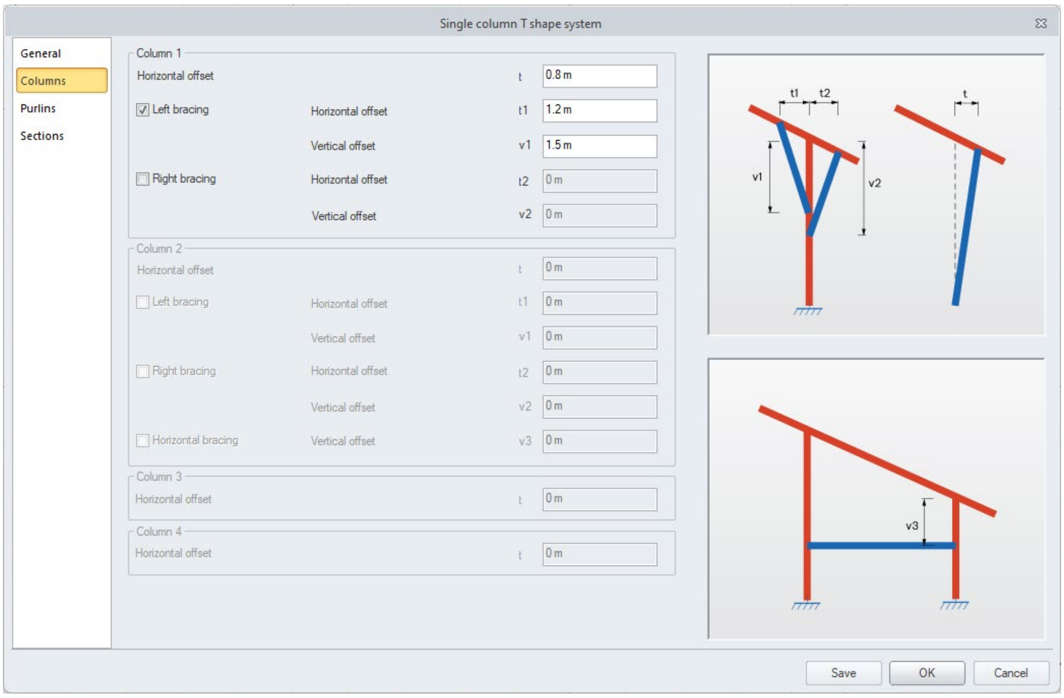
The following sections are available in this window:
- **General data** - allows indicating or entering the insertion point of the structure
- **Slopes** - allows you to enter the angle of the slope
- **Solar panels** - allows you to enter the dimensions of the area covered by PV panels, as well as to decide whether the corresponding load area should be created in the model.
- **Heights** - allows you to enter the distance between the ground level and the selected element of the structure.
- **Rafter** - allows you to enter widths defining the size of the rafter.
- **Bays** - allows you to define the number and spacing between frames.

Note that some of the entered dimensions are related to each other and modification of one of them may cause recalculation of the others. The primary parameters are slope angle (s) and solar panel dimensions (w). Introducing a dimension that prevents the generation of geometry automatically restores the previous value with an appropriate warning.



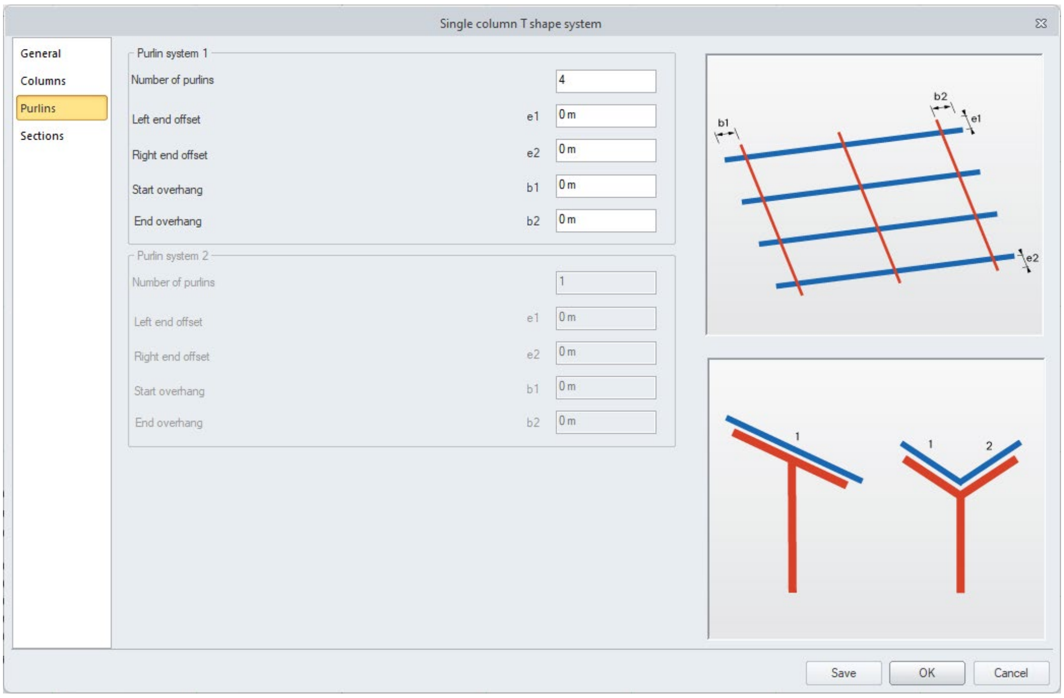
- **General**

This tab is used to define inclination of columns as well as optional bracings. Depending on the geometry type, data for one or more columns is available. Note that some of the horizontal offsets may take negative values, while the vertical and horizontal offsets for bracings are measured from the current position of the top end of the column.



- **Purlins**

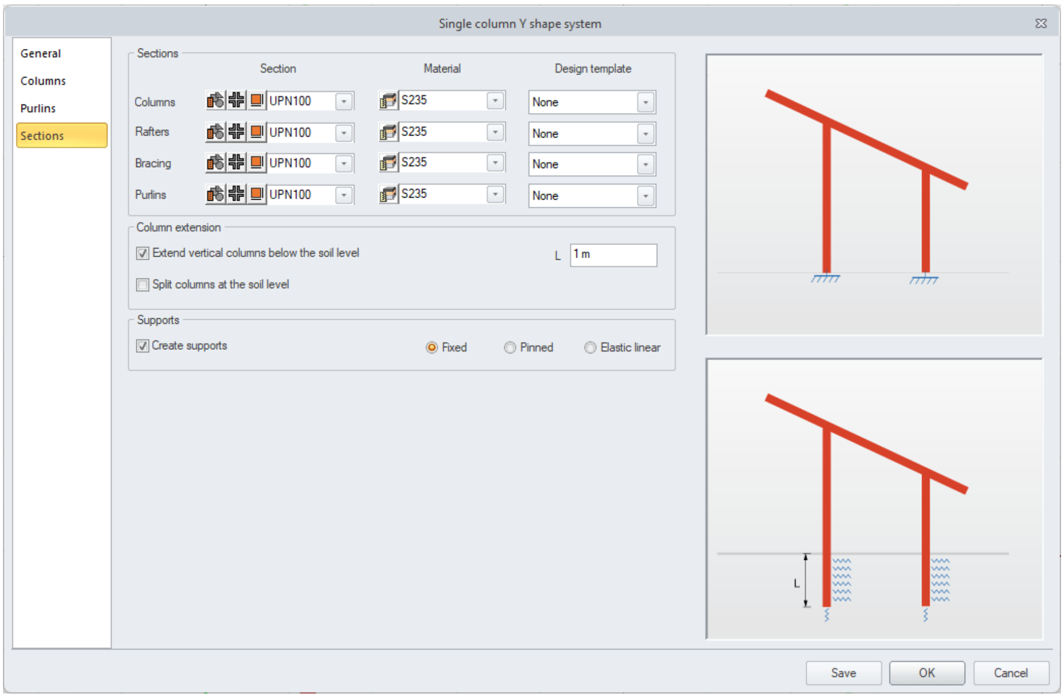
This tab is used to define the distribution of purlins and their positioning relative to beams. A twin purlins system is only available for single column Y shape system.



- **Sections**

The following groups of options are available in this window:

- **Sections** - allows selecting section, material, and design template independently for four categories of elements: columns, beams, bracing and purlins. For greater convenience when reviewing the generator's operation, default sections and materials have been adopted, but you can select other data. It should also be noted that after selecting your own data, you can save all data as your own template.
- **Columns extension** – allows you to select whether the columns are directly fixed at ground level or whether they are embedded by a certain value. In the case of embedded columns, you can choose to automatically divide them at ground level, which can facilitate the subsequent definition of various verification parameters for steel elements.
- **Supports** – allows you to decide whether to generate a support of basic type at the lower end of the column. In the case of columns embedded in the ground, it is possible to define a flexible linear support at this length; note that its stiffness must be modified accordingly after the model is generated.

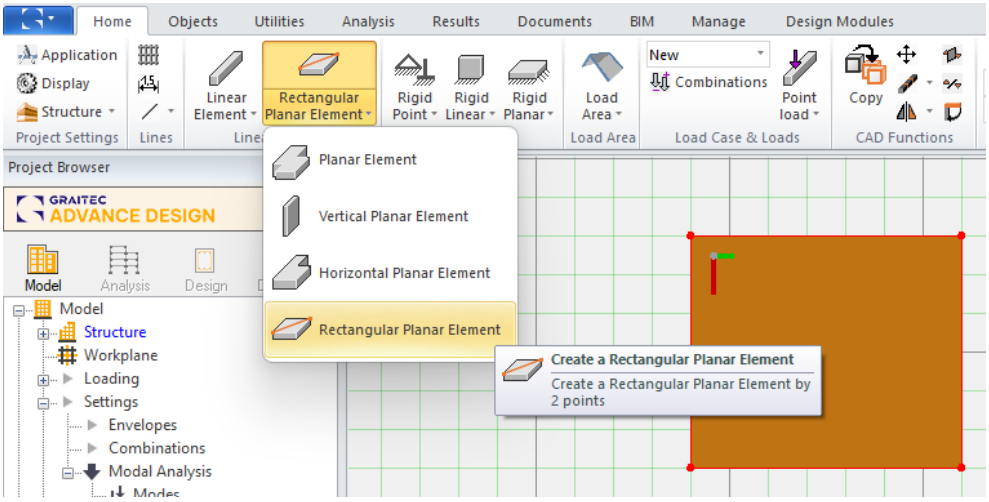


4.2. Defining planar objects by using 2 points

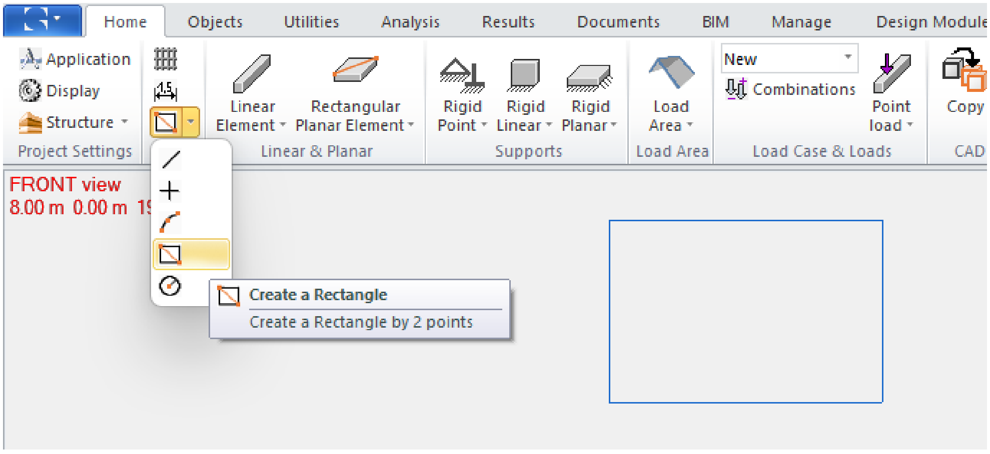
Defining rectangular surface elements, such as planar elements or load areas, using a 2-point indication.

Starting with the latest version of the application, a faster method of defining surface elements is available, by indicating 2 points on the diagonal. This functionality is quite basic but significantly increases the comfort and speed of modeling typical rectangular surface elements.

The new input mode is available for surface elements and load areas.



The same method can be used for defining rectangular-shaped closed polylines – useful for easy defining opening. For this purpose, a new *Crate a rectangle* command has been added next to the grouped line/point/arc/circle creation commands.

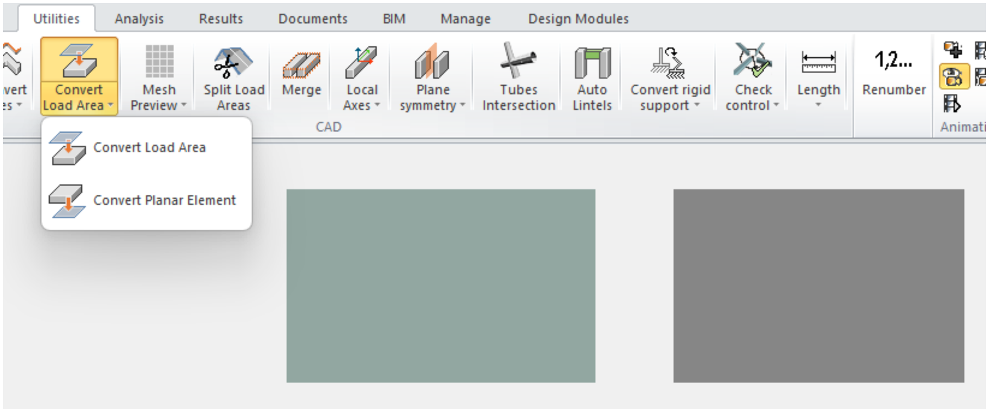


4.3. Bidirectional conversion of Load area to Planar element

Possibility of bidirectional conversion of Load area to Planar element. Helpful especially for improving models imported from another software.

Two new commands are available on the Utilities ribbon, in the CAD command group:

- **Convert Load Area** - converts a Load Area into a Planar element
- **Conver Planer Element** - converts a Planar element into a Load Area



Both commands work on the selection of one or more elements of one type - Planar elements or Load areas.

These commands can be used at any time during the modeling of the structure and are particularly useful when you have mistakenly defined a surface element of a different type than intended, or, for example, to better adapt a model imported from another software.

The conversion is based only on the geometry of the elements. This means that a given element is removed, and a new element of a new type, but with the same geometry, is created at the same place. If the surface element had holes inside the contour, they are removed during conversion. All parameters of the new elements are set as default, except for the assignment to System, which remains the same as in the original element.

4.4. Possibility to define in a table or import from Excel linear and punctual elements

Ability to create new objects (linear or punctual) using Data grid tables and to import elements from an Excel spreadsheet.

Data grid tables available in Advance Design have so far allowed viewing and editing of existing objects.
Starting with the latest version 2025, these tables can also be used to remove and create new linear and point elements. Creation of new elements is available for objects of type Point, Linear element, Point and Linear supports, Point and Linear loads.

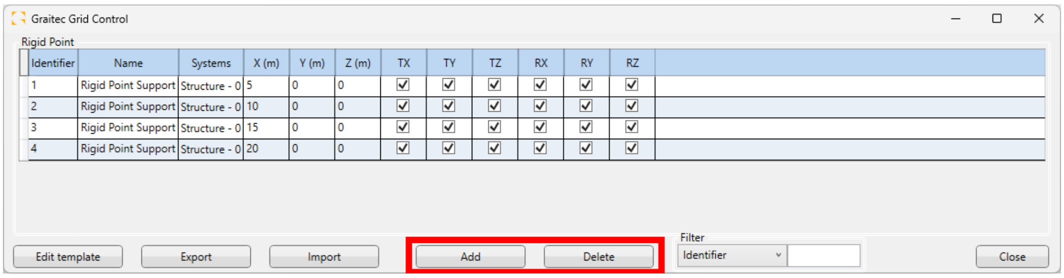
Creating new objects is possible using two scenarios - by adding new rows in a table or by importing table contents from an Excel spreadsheet.

Adding elements directly in the table

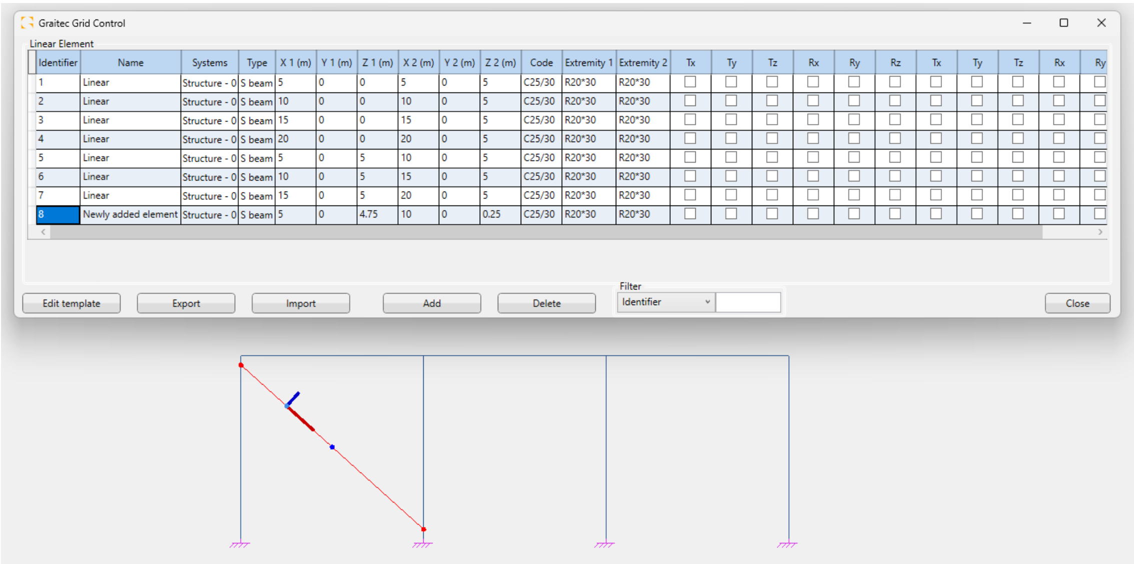
Entering new rows in the table, and thus new elements in the model is useful when we want to enter elements using manually entered data, like coordinates.

New **Add** and **Remove** buttons are used to add new items or remove existing ones directly in the table:

- **Remove** button is active for any type of objects and removes selected line.
- **Add** button is active only on objects that can be defined by one or two coordinates (punctual and linear objects) and adds a new line with a default data. Based on this a new object is automatically created on the model.

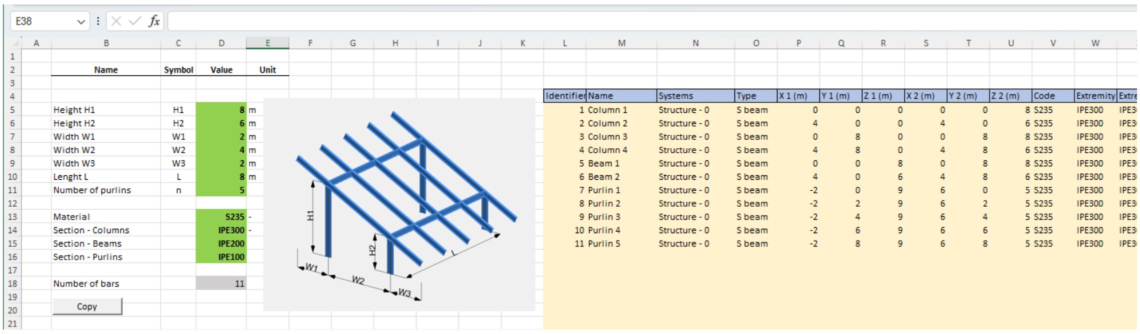


When creating new elements, they have a default setting and their starting point is set to coordinate 0,0,0. Of course, by editing the coordinates of new objects we can precisely define their location.



Importing elements from an Excel sheet

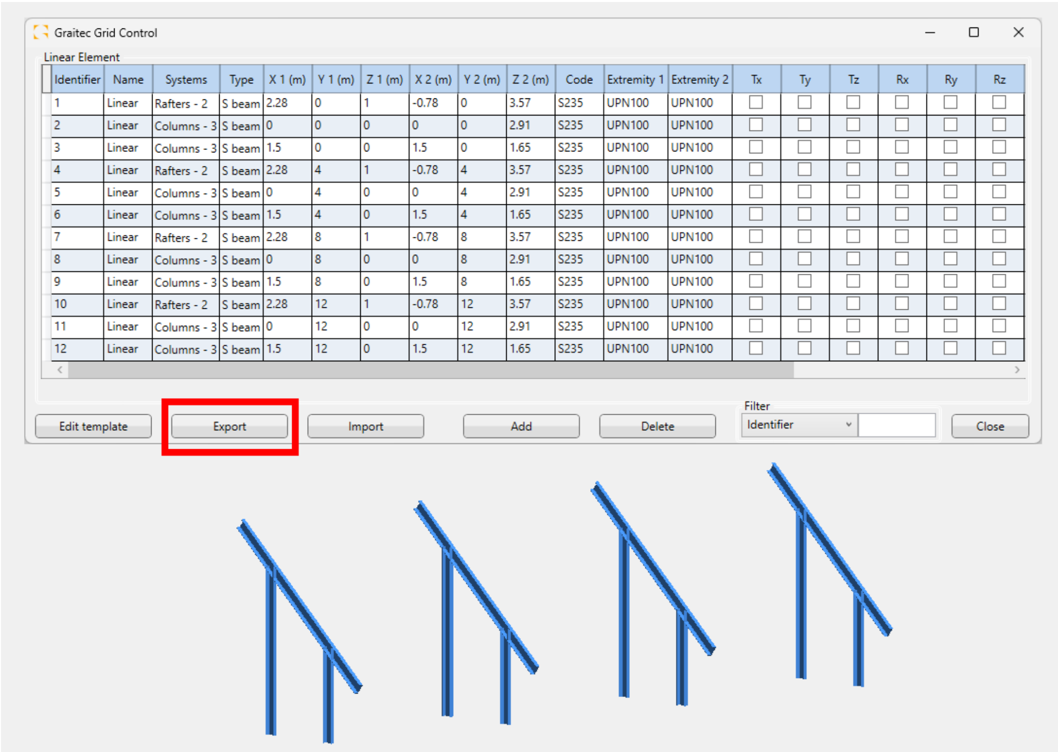
Import objects from Excel allows you to quickly create structures whose geometry is defined in the spreadsheet. Importing objects from an Excel sheet opens up many new model generation scenarios. One of them is the ability to create custom geometry generators in the Excel environment using formulas or macros.



Example of a simple geometry generator in a spreadsheet

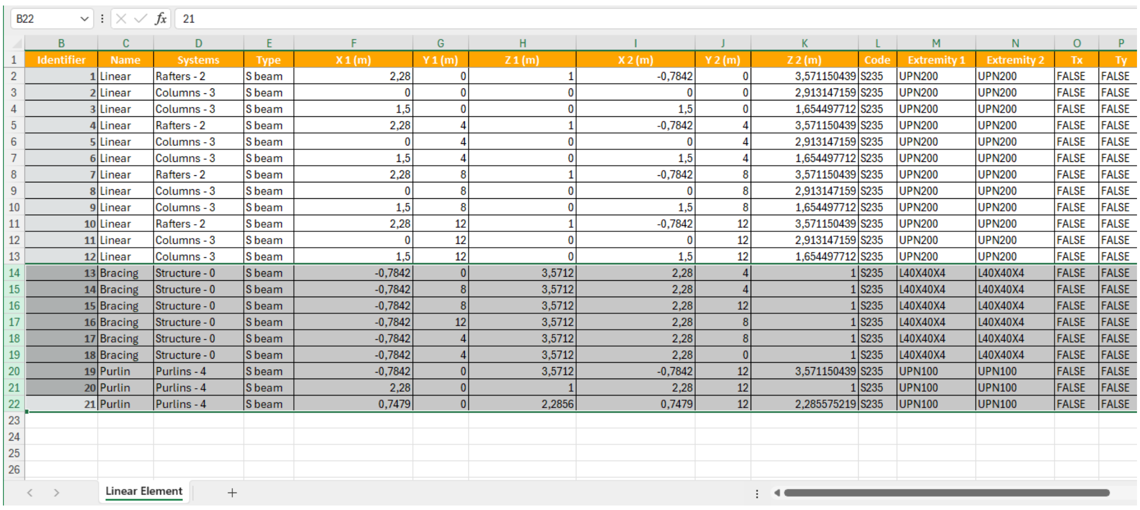
Regardless of whether you want to define a new geometry from scratch or edit an existing structure, the process is similar and consists of 3 steps:

- export data to an Excel spreadsheet



Exporting linear elements from an existing structure

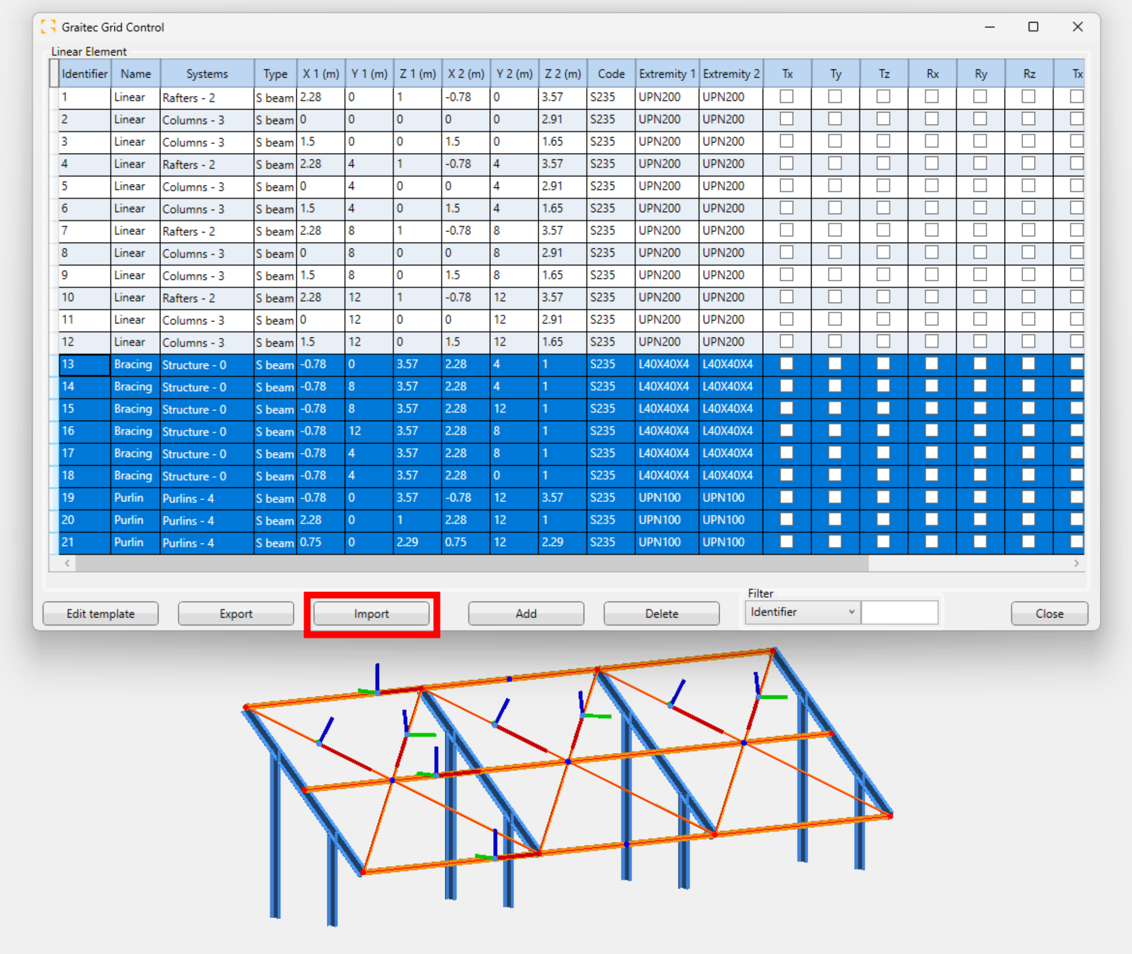
- add elements in the spreadsheet



Identifier	Name	Systems	Type	X 1 (m)	Y 1 (m)	Z 1 (m)	X 2 (m)	Y 2 (m)	Z 2 (m)	Code	Extremity 1	Extremity 2	Tx	Ty
1	Linear	Rafters - 2	S beam	2,28	0	1	-0,7842	0	3,571150439	S235	UPN200	UPN200	FALSE	FALSE
2	Linear	Columns - 3	S beam	0	0	0	0	0	2,913147159	S235	UPN200	UPN200	FALSE	FALSE
3	Linear	Columns - 3	S beam	1,5	0	0	1,5	0	1,654497712	S235	UPN200	UPN200	FALSE	FALSE
4	Linear	Rafters - 2	S beam	2,28	4	1	-0,7842	4	3,571150439	S235	UPN200	UPN200	FALSE	FALSE
5	Linear	Columns - 3	S beam	0	4	0	0	4	2,913147159	S235	UPN200	UPN200	FALSE	FALSE
6	Linear	Columns - 3	S beam	1,5	4	0	1,5	4	1,654497712	S235	UPN200	UPN200	FALSE	FALSE
7	Linear	Rafters - 2	S beam	2,28	8	1	-0,7842	8	3,571150439	S235	UPN200	UPN200	FALSE	FALSE
8	Linear	Columns - 3	S beam	0	8	0	0	8	2,913147159	S235	UPN200	UPN200	FALSE	FALSE
9	Linear	Columns - 3	S beam	1,5	8	0	1,5	8	1,654497712	S235	UPN200	UPN200	FALSE	FALSE
10	Linear	Rafters - 2	S beam	2,28	12	1	-0,7842	12	3,571150439	S235	UPN200	UPN200	FALSE	FALSE
11	Linear	Columns - 3	S beam	0	12	0	0	12	2,913147159	S235	UPN200	UPN200	FALSE	FALSE
12	Linear	Columns - 3	S beam	1,5	12	0	1,5	12	1,654497712	S235	UPN200	UPN200	FALSE	FALSE
13	Bracing	Structure - 0	S beam	-0,7842	0	3,5712	2,28	4	1	S235	L40X40X4	L40X40X4	FALSE	FALSE
14	Bracing	Structure - 0	S beam	-0,7842	8	3,5712	2,28	4	1	S235	L40X40X4	L40X40X4	FALSE	FALSE
15	Bracing	Structure - 0	S beam	-0,7842	8	3,5712	2,28	12	1	S235	L40X40X4	L40X40X4	FALSE	FALSE
16	Bracing	Structure - 0	S beam	-0,7842	12	3,5712	2,28	8	1	S235	L40X40X4	L40X40X4	FALSE	FALSE
17	Bracing	Structure - 0	S beam	-0,7842	4	3,5712	2,28	8	1	S235	L40X40X4	L40X40X4	FALSE	FALSE
18	Bracing	Structure - 0	S beam	-0,7842	4	3,5712	2,28	0	1	S235	L40X40X4	L40X40X4	FALSE	FALSE
19	Purlin	Purlins - 4	S beam	-0,7842	0	3,5712	-0,7842	12	3,571150439	S235	UPN100	UPN100	FALSE	FALSE
20	Purlin	Purlins - 4	S beam	2,28	0	1	2,28	12	1	S235	UPN100	UPN100	FALSE	FALSE
21	Purlin	Purlins - 4	S beam	0,7479	0	2,2856	0,7479	12	2,285575219	S235	UPN100	UPN100	FALSE	FALSE

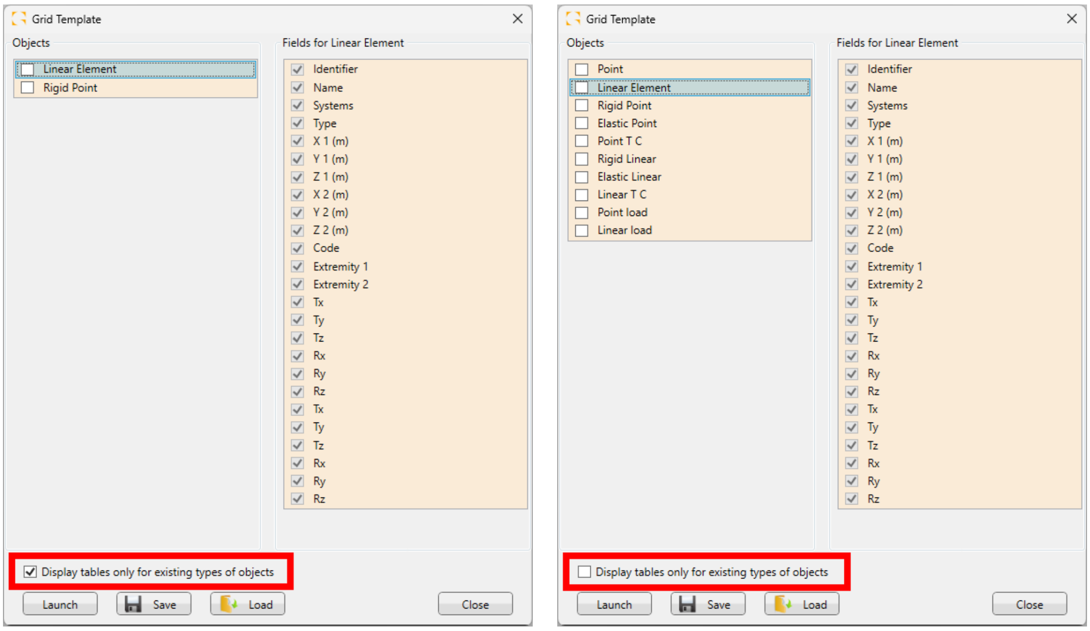
Expanding the list of members with new ones in the Excel sheet

- import into Advance Design.



Importing a spreadsheet from Excel during which new elements are added

During data export, objects visible in Data grid tables are transferred, which are then grouped according to object type on separate tabs of the Excel sheet. It is also possible to export empty tables in case objects of a given type do not yet exist in the current model, which allows you to add new elements of a given type in the Excel sheet. To decide whether to display and export all tables or only for existing object types, use the option visible from the bottom of the Data grid window.



When editing the data in the worksheet, it is important to remember that the element in the category is recognized by its ID number. That is, if we change the data of an element without changing the ID, it will be updated in the model. If we add an item with a new ID number, it will be created in the model. If we remove an item with a given ID number in the table, it will be removed from the model. And if there are multiple elements with the same ID number in the table, then the import will stop, and the details of the problem will be described in the log file.

When importing data from an Excel sheet, validation is conducted, and any errors are recorded in a text log file. Here are some rules for editing and importing data:

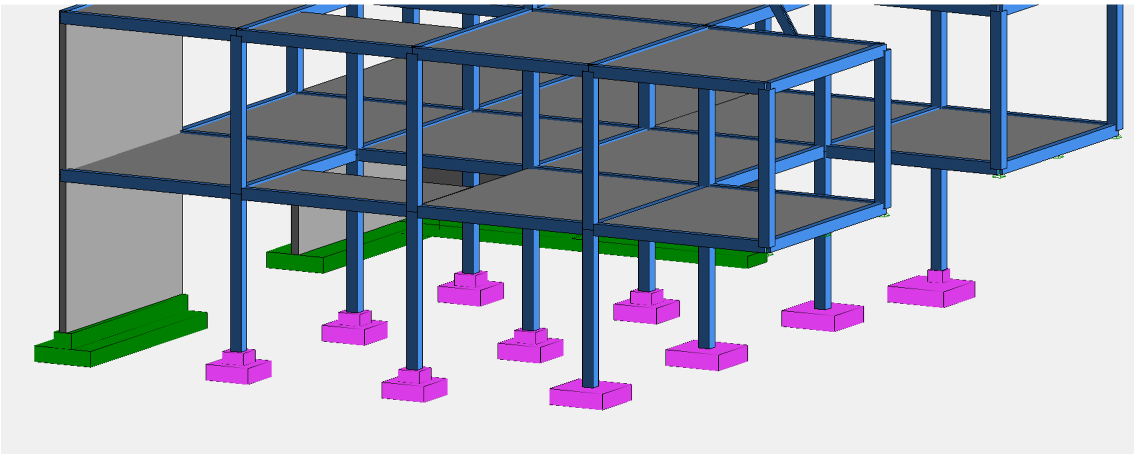
- Deleting an element from a table removes it from the model
- Adding an element with a new ID number adds it to the model
- Changing the data of an element without changing the ID causes it to be updated in the model.
- When there are multiple elements with the same ID number in a table, the import will stop.
- When the entered material or section is unknown, then for existing elements the update will not be made, while for new elements the default material/section will be assigned.
- When the data in the cells is inappropriate (e.g., letters instead of numbers) then the given row is skipped.
- When objects of a given type exist in the model, but the corresponding Excel table is empty when importing, these objects are removed from the model.
- When there are objects of a given type in the model, but during import the Excel file does not contain a table referring to this type of object, then the objects remain in the model unchanged.

4.5. Visualization of foundations on model

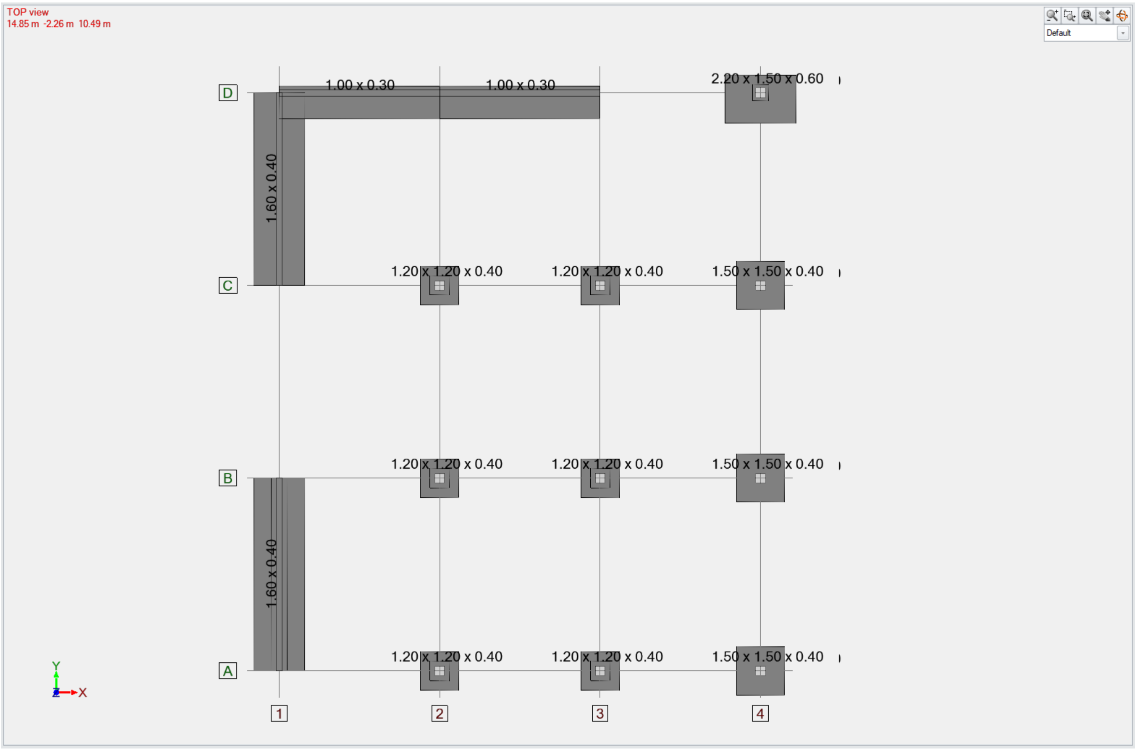
Possibility for displaying in 3D model a visualization of foundations. This allows for the visualization of assumed, or calculated by the RC Footing module, geometrical parametric of foundation.

When creating a model for FEM calculations, we use supports, such as linear or point supports, with specific mechanical properties (e.g., rigid or pinned) to model the foundations. In Advance Design, it was previously possible to specify the basic dimensions of the foundation in the support properties, and then either include the volume of the foundation in the estimation of costs and CO2 emissions, as well as pass these dimensions to the RC Footing module.

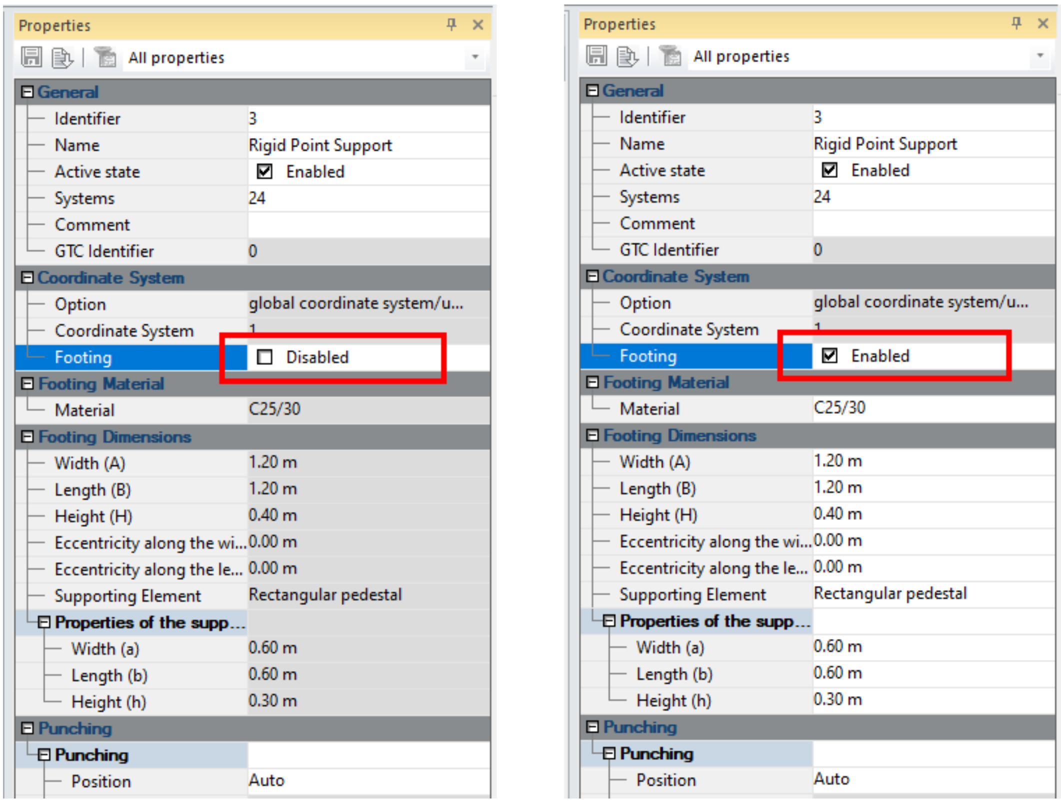
In the latest version of Advance Design 2025, the possibilities of parameterization of the foundations have been extended and the possibility of foundation visualization has been introduced.



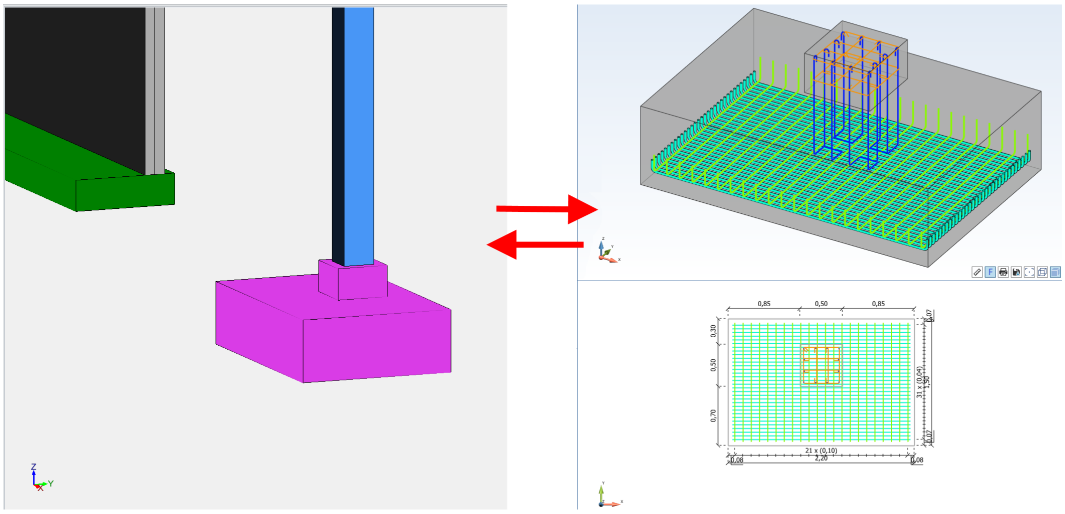
This facilitates the cooperation with RC Footing module, makes it easier to verify the dimensions of foundations, allows to create better visualization and documentation.



To distinguish foundation supports from supports of other types, a new option 'Footing' has been added to the support properties. When it is not active, this support performs all mechanical functions, but is not treated as a foundation. When the new option is enabled, all other foundation-related properties become active, including material, foundation dimensions, design template and parameters for cost and CO2 estimation.

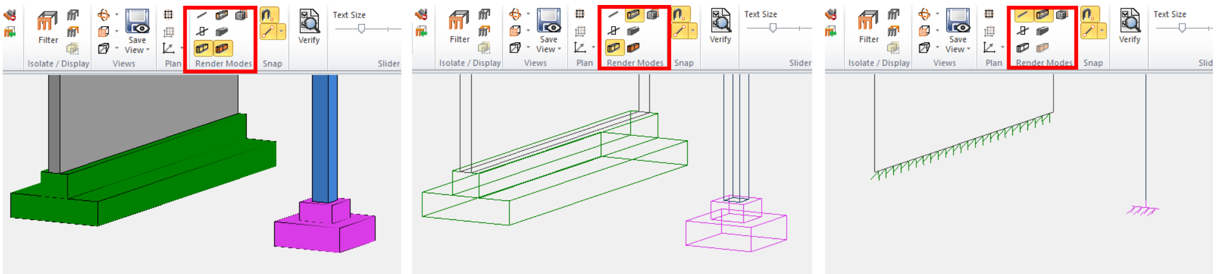


To describe the foundation geometry more completely, information about the eccentricity as well as the type and dimensions of the supporting element (pedestal type) are added to the basic foundation dimensions. These parameters are not only used during visualization but are also bidirectionally exchanged with RC Footing. Therefore, if the initial foundation dimensions are optimized in RC Footing, the final foundation geometry can be displayed and described thanks to data synchronization in the 3D model.



Note here that only foundation type supports can be exported to RC Footing module.

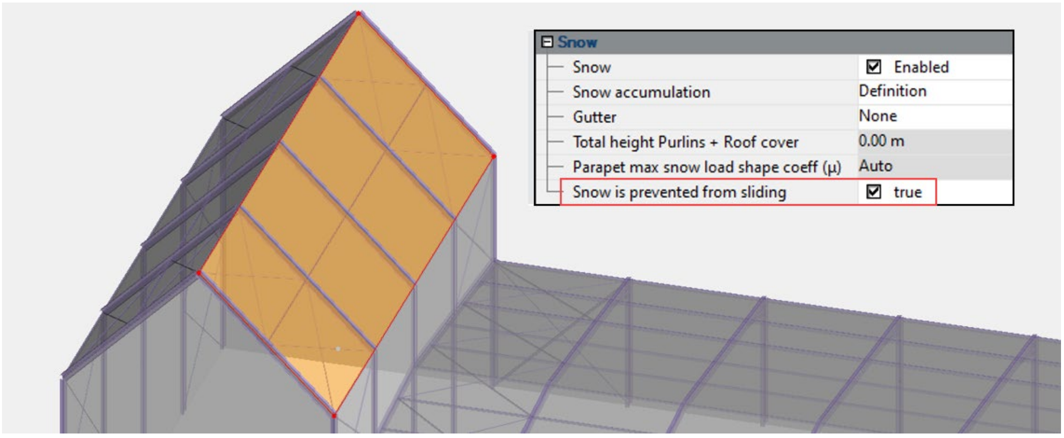
The graphical representation of foundations in the model, like the profiles of other elements, depends on the rendering settings.



4.6. Snow generation considering snow guards on the roof

Ability to define the presence of a snow fence on the roof (Eurocode).

It is now possible to define the presence of a device preventing the snow from drifting off the roof, such as a snow fence. For this, in the properties of the load area object, a new option has been added: *Snow is prevented from sliding*. The new parameter is available when the Eurocode is set as the current standard for climatic loads.



The impact of this option will be noticeable on high-pitched roofs ($\alpha > 30°$).
Indeed, for such roof geometries, Table 5.2 from EN 1993-1-3 usually allows for a linear decrease of the shape coefficient, accounting for the tendency of the snow to drift off the roof.

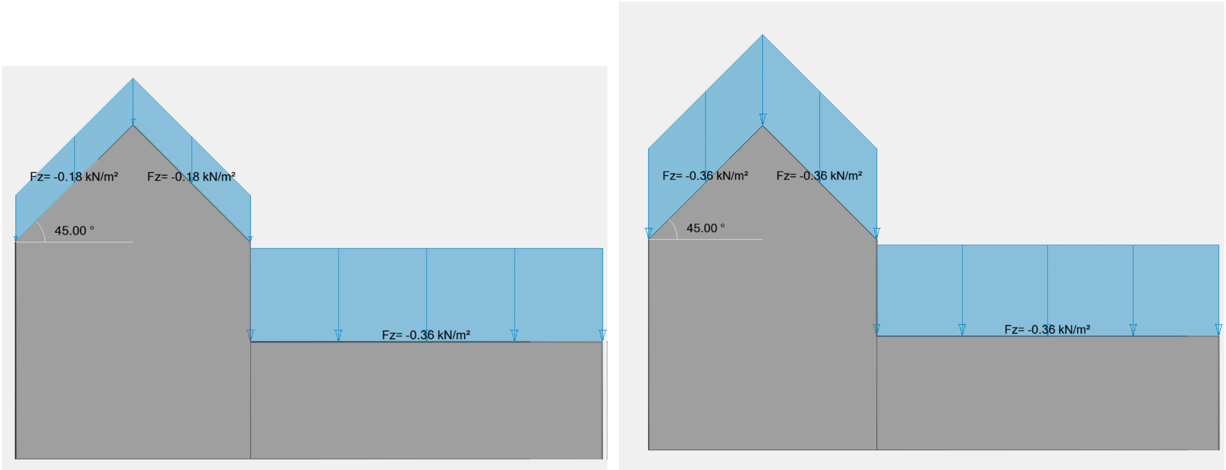
Angle of pitch of roof α	$0° \leq \alpha \leq 30°$	$30° < \alpha < 60°$	$\alpha \geq 60°$
$\mu_1(\alpha)$	$\mu_1(0°) \geq 0.8$	$\mu_1(0°) \geq 0.8$	$\mu_1(0°) \geq 0.8$
$\mu_2(\alpha)$	0.8	$0.8\frac{(60° - \alpha)}{30°}$	0

Yet, by activating the presence of a snow fence on the load area, such a reduction is no longer permitted.
Therefore, a μ = **0,8** value should be considered, as stated in §5.3.2 (2) (monopitch roof) and §5.3.3 (2) (pitched roofs) from EN 1993-1-3.

> (2) The values given in Table 5.2 apply when snow is not prevented from sliding off the roof. Where snow fences or other obstructions exist or where the lower edge of the roof is terminated with a parapet, then the snow load shape coefficient should not be reduced below 0,8.

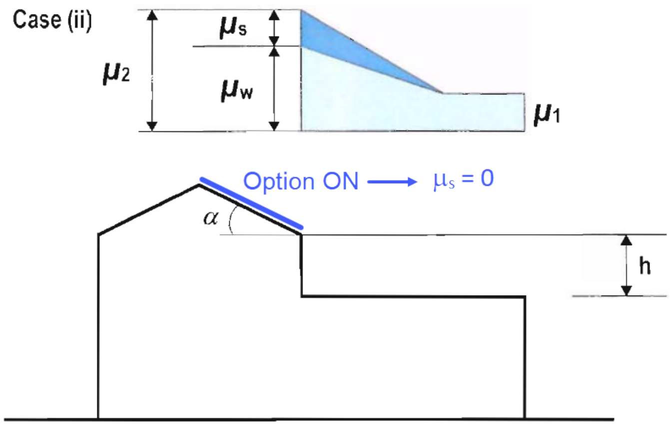
In Advance Design, in this case, the value of 0.8 is taken for all supported slope angles (from 0 to 80 degrees).

On the example below, with the option OFF, the taller construction, with its high-pitched roof, gets a smaller snow force (0,18 kN/m²) due to the reduction of its shape coefficient. By activating the new **Snow is prevented from drifting** option in load area properties, this reduction no longer occurs. The snow force on the taller construction is now 0,36 kN/m², same as on the adjacent flat roof.



Snow is prevented from drifting = OFF (left) and ON (right)

As for the drifted load arrangement, be aware that activating this new option on a taller construction will automatically nullify the contribution of μ_s (shape coefficient due to sliding of snow from the upper roof), as snow can no longer slide from the upper roof.



5. Enhance steel structure design capabilities

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

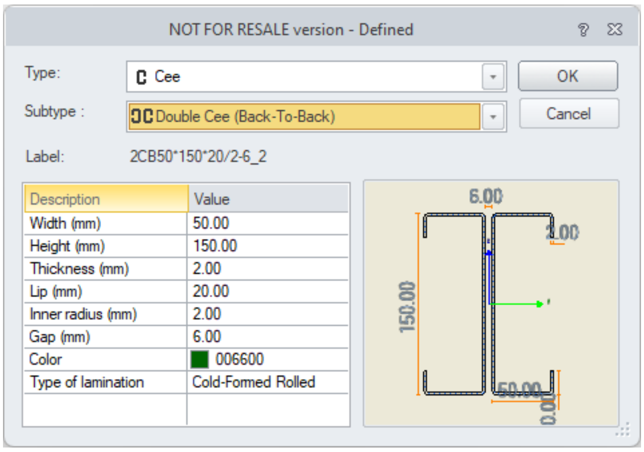
5.1. Additional sections for cold-formed design

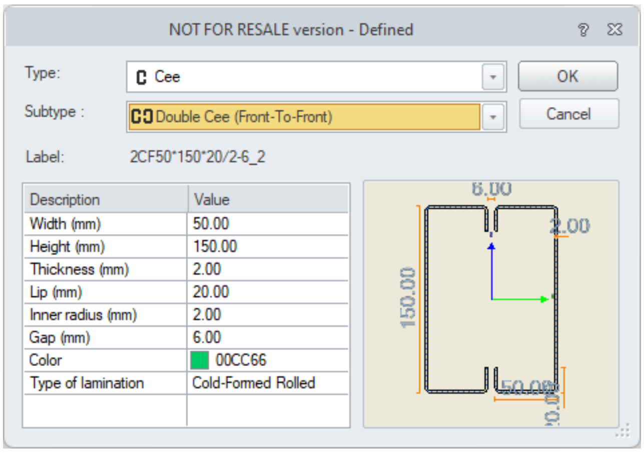
Possibility to model and design a range of new cold-formed sections.

With Advance Design 2025, the capabilities for analyzing cold-formed sections have been expanded, with the introduction of the ability to parametrically define a number of new section shapes, as well as perform verification for new section types according to EN 1993-1-3, as well as ACSI and CSA standards.

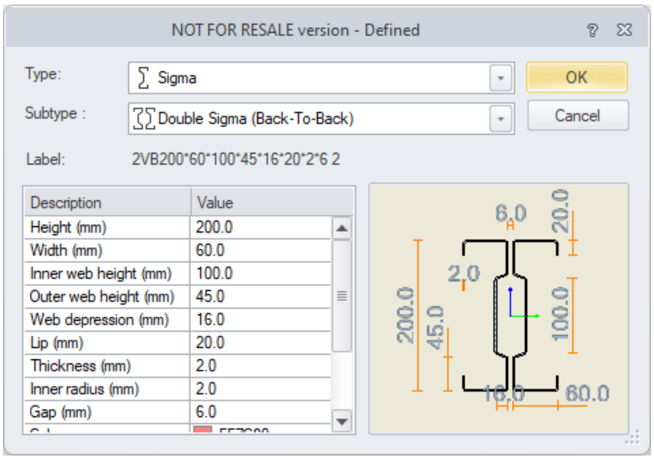
List of new profile types:

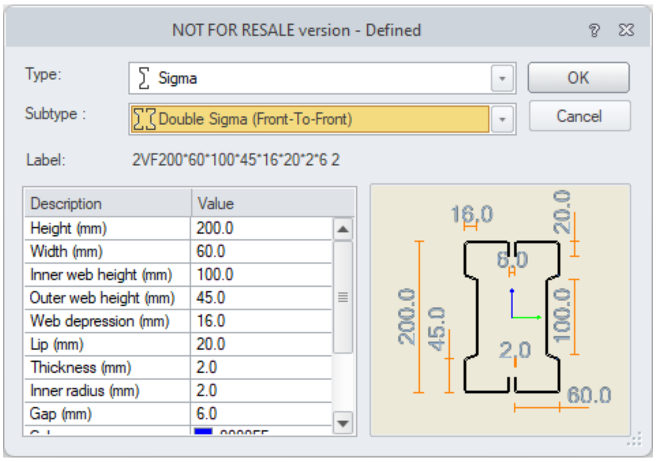
- **Double lipped channels (back-to-back or front-to-front)**



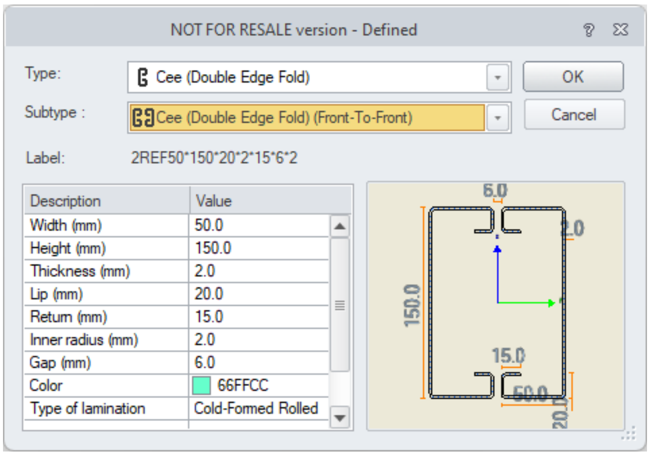


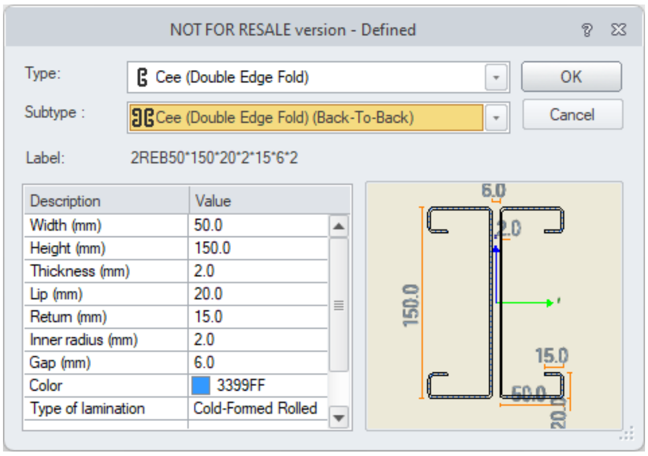
- **Double Sigma (back-to-back or front-to-front)**



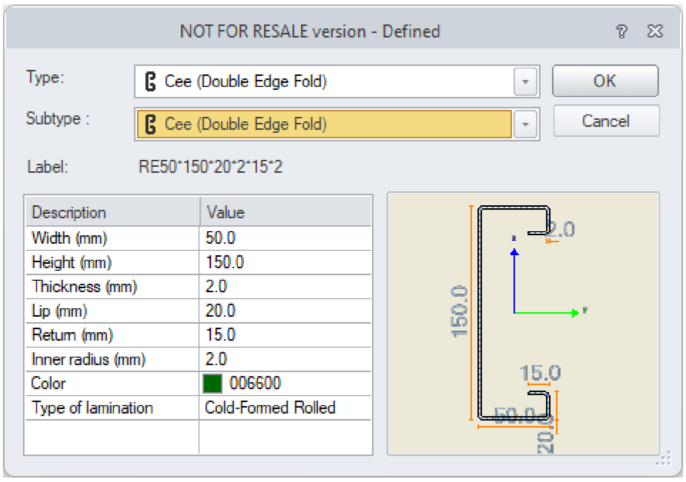


- **Double channels with double edge fold (back-to-back or front-to-front)**

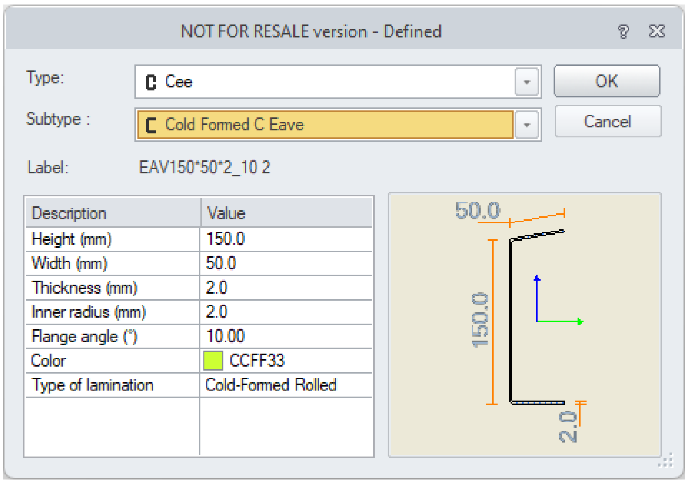




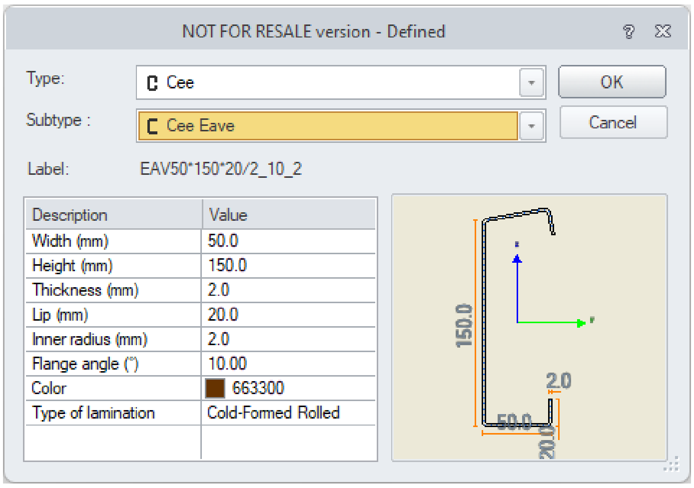
- **Channels with double edge fold**



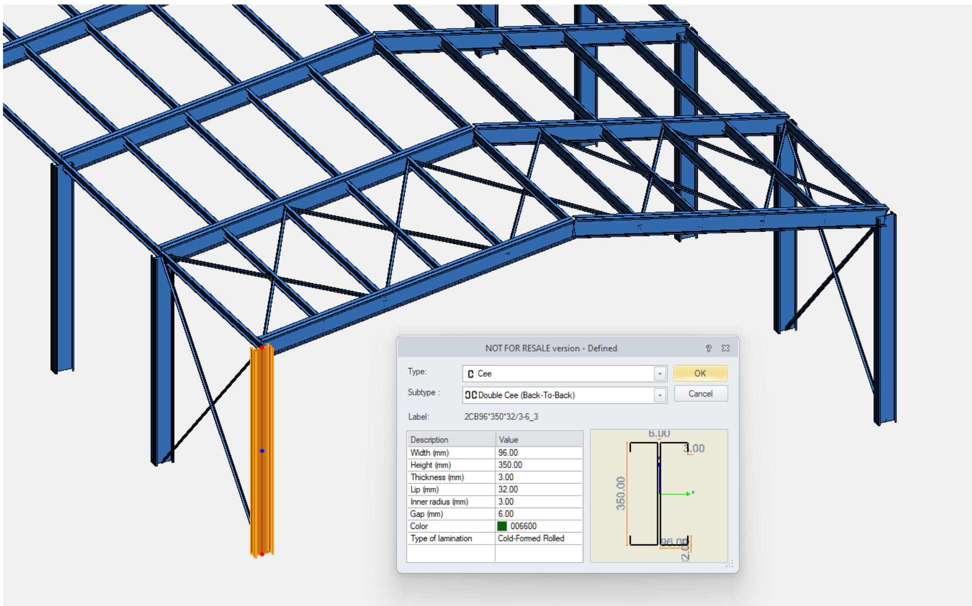
- **Channels with an inclined eave**



- **Lipped channels with an inclined eave**



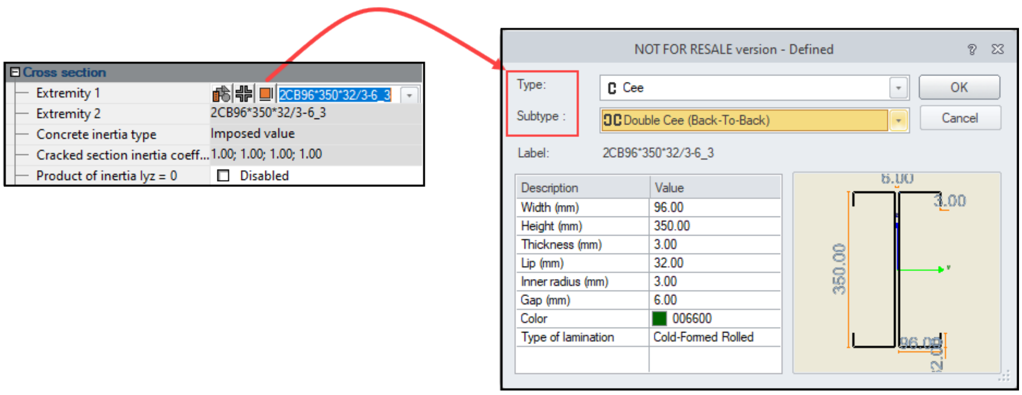
The new built-up sections introduced with Advance Design 2025 are commonly used as primary structural elements (columns, rafters...) and therefore, they were highly requested by our customers.



Example - Compound 2C section - definition and verification according to Eurocode

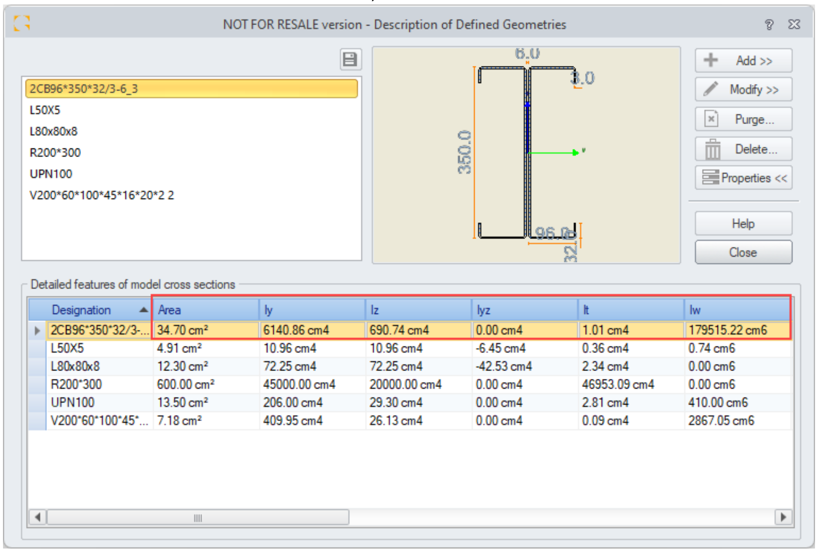
- **Defining new sections**

These new sections can be defined on a model by defining a Parametric section and choosing the desired Type and Subtype:



- **Gross cross-section properties**

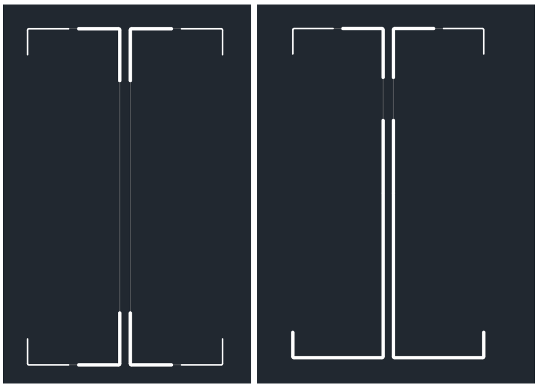
Once a built-up section is introduced in a model, its characteristics are automatically computed.



Designation	Area	Iy	Iz	Iyz	It	Iw
2CB96*350*32/3-...	34.70 cm²	6140.86 cm4	690.74 cm4	0.00 cm4	1.01 cm4	179515.22 cm6
L50X5	4.91 cm²	10.96 cm4	10.96 cm4	-6.45 cm4	0.36 cm4	0.74 cm6
L80x80x8	12.30 cm²	72.25 cm4	72.25 cm4	-42.53 cm4	2.34 cm4	0.00 cm6
R200*300	600.00 cm²	45000.00 cm4	20000.00 cm4	0.00 cm4	46953.09 cm4	0.00 cm6
UPN100	13.50 cm²	206.00 cm4	29.30 cm4	0.00 cm4	2.81 cm4	410.00 cm6
V200*60*100*45*...	7.18 cm²	409.95 cm4	26.13 cm4	0.00 cm4	0.09 cm4	2867.05 cm6

- **Effective properties**

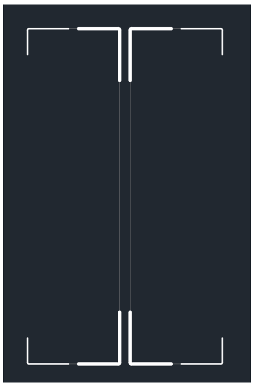
The effective cross-section is determined in accordance with EN 1993-1-5 and EN 1993-1-3.
On the built-up section below, one can see the effects of local plate buckling (resulting in ineffective widths in webs, flanges, and lips) and distortional buckling (resulting in a reduced thickness for the end stiffener):



Pure compression *Pure bending*

- **Resistance of cross-sections**

Because these built-up sections remain symmetric in pure compression, no shift of the centroid needs to be considered:

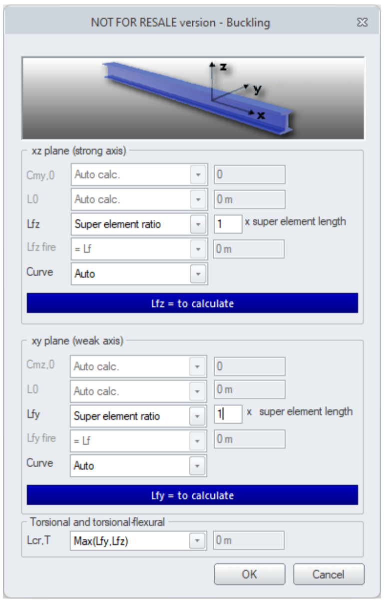


Therefore, the corresponding $\Delta_{My,Ed}$ and $\Delta_{Mz,Ed}$ moments will usually be null.

Combined compression and bending (6.1.9)	Case no 102 : 1.35x[1 D]+1.5x[2 S], Mesh No. 13.4 4/4 Cross section : Class 4 $\left\lvert\frac{N_{Ed}}{N_{c,Rd}}\right\rvert + \left\lvert\frac{M_{y\,Ed} + \Delta M_{y\,Ed}}{M_{cy,Rd,com}}\right\rvert + \left\lvert\frac{M_{z\,Ed} + \Delta M_{z\,Ed}}{M_{cz,Rd,com}}\right\rvert < 1$ (6.25) : $\left\lvert\frac{-45.33\ kN}{689.66\ kN}\right\rvert +$ $\left\lvert\frac{-69.86\ kN{*}m + 0.00\ kN{*}m}{108.64\ kN{*}m}\right\rvert + \left\lvert\frac{0.00\ kN{*}m + 0.00\ kN{*}m}{21.20\ kN{*}m}\right\rvert < 1$ (71 %)

- **Buckling resistance**

The effects of flexural buckling (in both planes), torsional buckling and torsional-flexural buckling will be considered by computing the corresponding critical forces ($N_{cr,y}$, $N_{cr,z}$, $N_{cr,T}$ and $N_{cr,TF}$)



Once the limiting critical force has been identified, the relative slenderness and the buckling coefficient is computed:

$$\bar{\lambda} = \sqrt{\frac{A_{eff} \cdot f_y}{\min\left(N_{cr,y}, N_{cr,z}, N_{cr,T}, N_{cr,TF}\right)}}$$

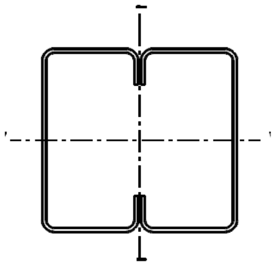
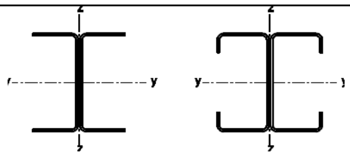
$$\phi = 0,5 \cdot \left[1 + \alpha \cdot \left(\bar{\lambda} - 0,2\right) + \bar{\lambda}^2\right]$$

$$\chi = \frac{1}{\phi + \sqrt{\phi^2 - \bar{\lambda}^2}}$$

Buckling (6.2.2)	Case no 102 : 1.35x[1 D]+1.5x[2 S], Mesh No. 13.1 0/4 Cross section : Class 4 $N_{Ed} < N_{b,Rd}$: 46.78 kN < 465.76 kN L_{fy} = 4.00 m, $\lambda_{bar\ y}$ = 0.29, α_y = 0.21 L_{fz} = 4.00 m, $\lambda_{bar\ z}$ = 0.88, α_z = 0.34 $L_{cr,TF}$ = 4.00 m, $N_{cr,TF}$ = inf kN, $L_{cr,T}$ = 4.00 m , $N_{cr,T}$ = 1222.63 kN $\chi = \min(\chi_y, \chi_z, \chi_T, \chi_{TF})$ = min(0.98, 0.68, 0.75, 1.00) $N_{Ed} < \frac{\chi * A_{eff} * f_{yb}}{\gamma_{M1}}$ (6.48 EN1993-1-1) $46.78\ kN < \frac{0.68 * 19.70\ cm^2 * 350.00\ MPa}{1.00}$ (10 %)

The α value used to determine the ϕ parameter will depend on the buckling curves defined in Table 6.3 from EN 1993-1-3.

Table 6.3: Appropriate buckling curve for various types of cross-section

Type of cross-section		Buckling about axis	Buckling curve
	if f_{yb} is used	Any	b
	if f_{ya} is used *)	Any	c
		y - y	a
		z - z	b

As for lateral-torsional bucking, the critical moment (M_{cr}) may be determined as per Annex I from EN 1999-1-1.

$$M_{cr} = \frac{C_1 \cdot \pi^2 \cdot E \cdot I_z}{k_z^2 \cdot L^2} \cdot \left(\sqrt{\left(\frac{k_z}{k_w}\right)^2 \cdot \frac{I_w}{I_z} + \frac{k_z^2 \cdot L^2 \cdot G \cdot I_t}{\pi^2 \cdot E \cdot I_z} + \left(C_2 \cdot z_g - C_3 \cdot z_j\right)^2} - \left(C_2 \cdot z_g - C_3 \cdot z_j\right) \right)$$

Then, for χ_{LT} determination, Advance Design will use curve b (α_{LT} = 0,34) as advised by EN 1993-1-3:

> **6.2.4 Lateral-torsional buckling of members subject to bending**
>
> (1) The design buckling resistance moment of a member that is susceptible to lateral-torsional buckling should be determined according to EN 1993-1-1, section 6.3.2.2 using the lateral buckling curve b.
>
> (2) This method should not be used for the sections that have a significant angle between the principal axes of the effective cross-section, compared to those of the gross cross-section.

Lateral-torsional Buckling (6.2.4)	Case no 102 : 1.35x[1 D]+1.5x[2 S], Mesh No. 13.4 4/4 Cross section : Class 4 $M_{y,Ed} < M_{by,Rd}$: 69.86 kN*m < 89.10 kN*m $\lvert M_{y,Ed} + \Delta M_{Ed}\rvert < \frac{\chi_{LT} * W_{eff,y} * f_{yb}}{\gamma_{M1}}$ $\lvert -69.86\ kN*m + 0.00\ kN*m\rvert < \frac{0.82 * 310.39\ cm^3 * 350.00\ MPa}{1.00}$ k_z = 1.00, k_w = 1.00, k_{wt} = 5.33, ψ_f = 0.00, C1 = 1.85, C2 = 0.00, C3 = 1.00, Lds = 4.00 m, Ldi = 4.00 m, Lcr = 4.00 m, zg = 0.00 cm, zj = 0.00 cm, Mcr = 271.07 kN*m, $\lambda_{bar\ LT}$ = 0.63, χ_{LT} = 0.82 Mcr was computed analytically. (78 %)

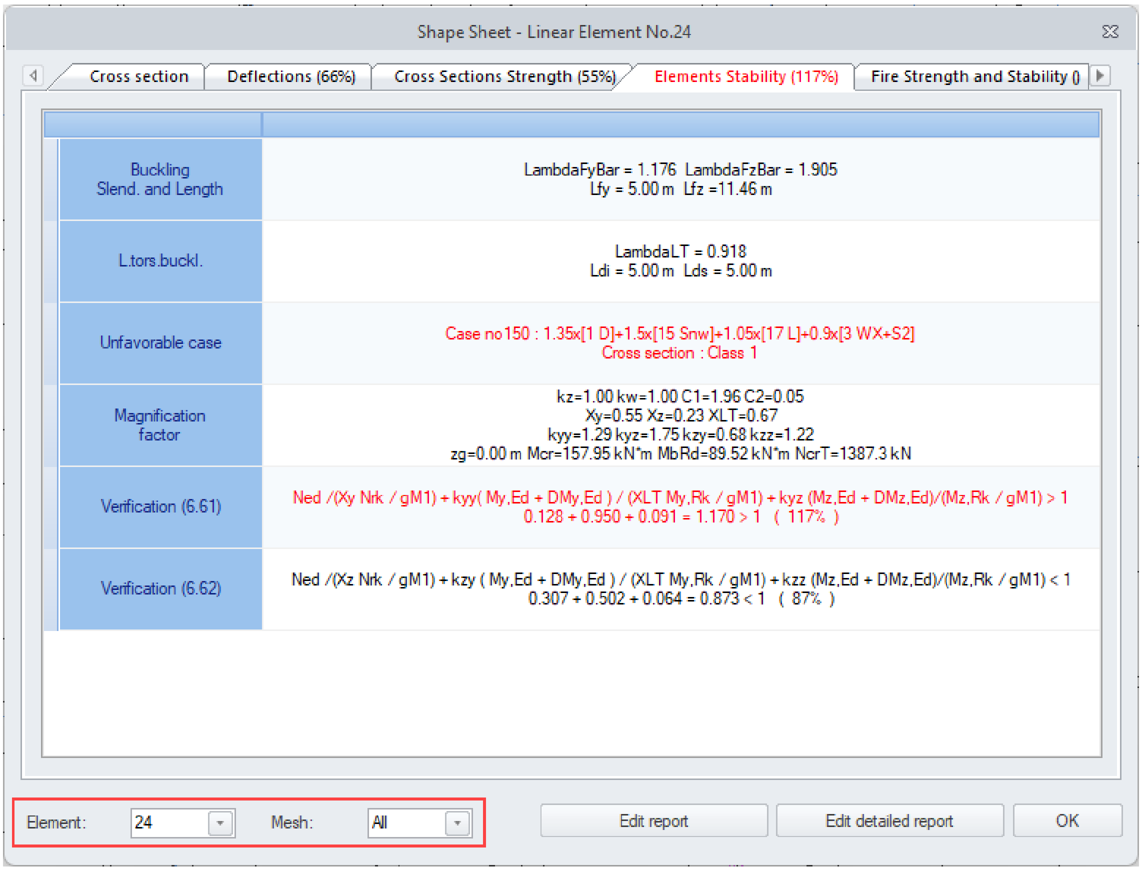
Buckling resistance is then checked in accordance with equation (6.36) from EN 1993-1-3:

Bending and Axial Compression (6.2.5)	Case no 102 : 1.35x[1 D]+1.5x[2 S], Mesh No. 13.4 4/4 Cross section : Class 4 $\left\lvert\frac{N_{Ed}}{N_{b,Rd}}\right\rvert^{0.8} + \left\lvert\frac{M_{y,Ed}}{M_{yb,Rd}}\right\rvert^{0.8} + \left\lvert\frac{M_{z,Ed}}{M_{zc,Rd}}\right\rvert^{0.8} < 1$ $\left\lvert\frac{-45.33\ kN}{465.76\ kN}\right\rvert^{0.8} + \left\lvert\frac{-69.86\ kN*m}{89.10\ kN*m}\right\rvert^{0.8} + \left\lvert\frac{0.00\ kN*m}{21.20\ kN*m}\right\rvert^{0.8} < 1$ Eccentricity favorable effects are ignored. (98 %)

5.2. Shape sheep results on a selected mesh

Detailed steel design results on a specific portion of a linear element

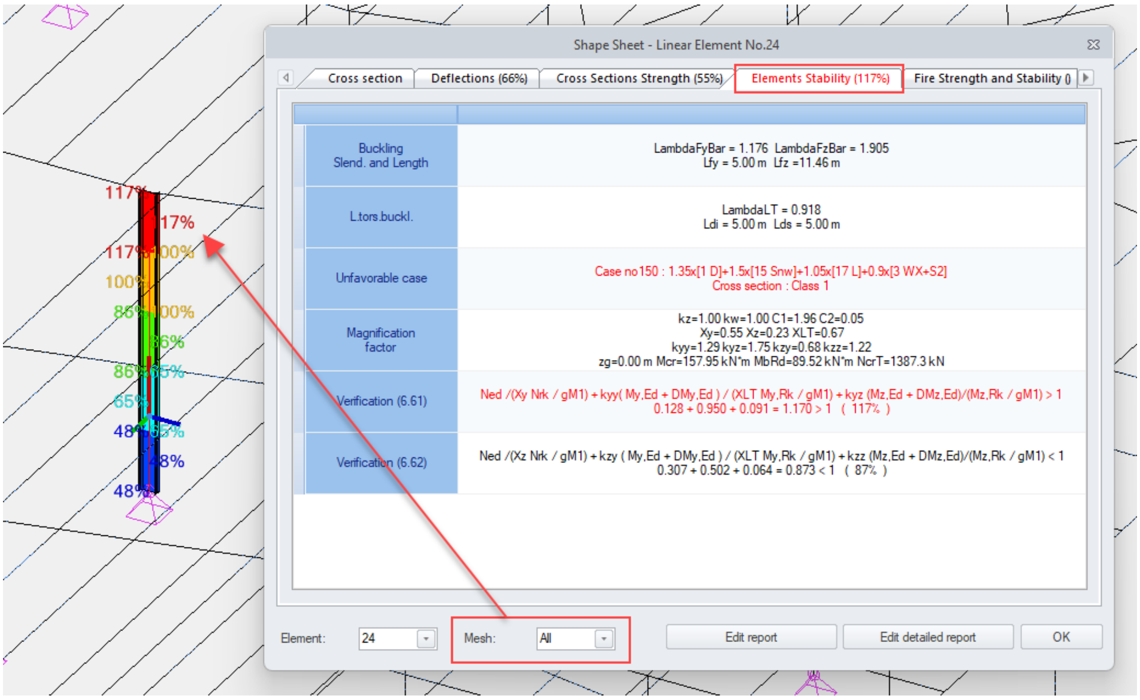
The shape sheet for steel members now offers two new controls to easily change the element under consideration, as well as the location along the member.



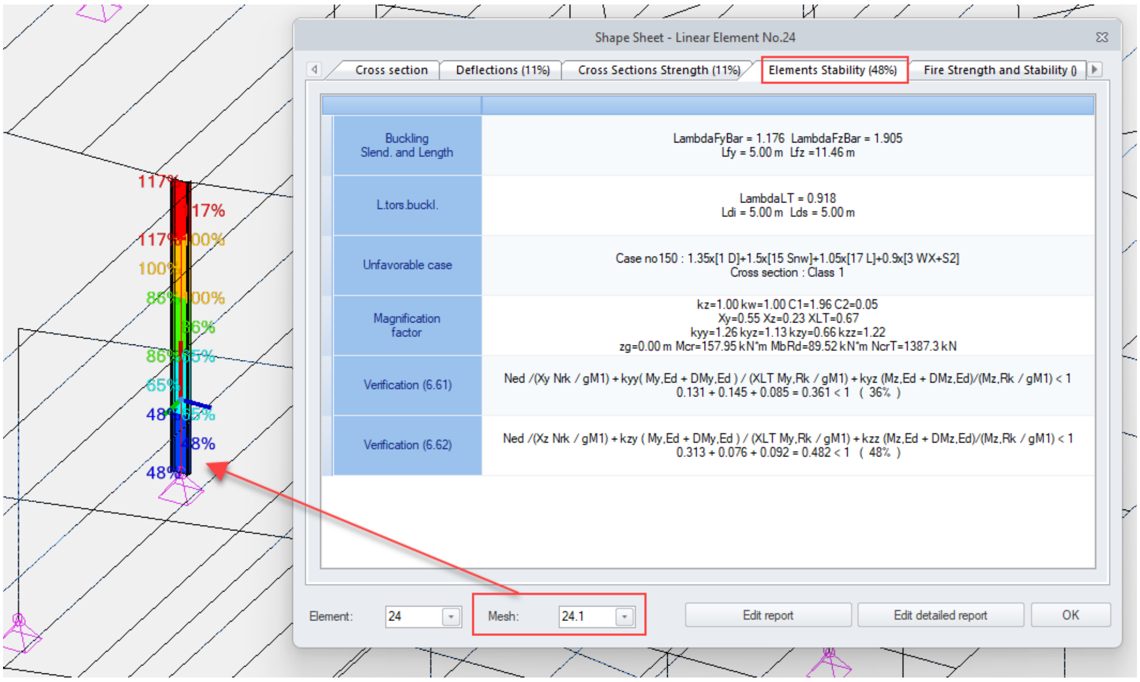
The **Element** field displays the numbers of linear steel elements present in the selection when the Shape Sheet window is opened. If the selection includes one element, only one number is available in the secret list. If there were more elements in the selection, selecting a particular one from this list allows you to display the window contents for that selected element.

The **Mesh** field displays a list of finite elements for the current element (selected in the Element field). Selecting a particular finite element allows you to view results for the selected element location. In addition, there is an *All* item (default selection), which allows you to display critical results for the entire element. This is the same effect as when opening a window in earlier versions of Advance Design.

On the example below, with Mesh = All, the shape sheet returns the steel design results for the critical location (i.e., top of the column).



Then, by selecting a different linear mesh from the drop-down list, the shape sheet will instantly update and show the results at this new location (i.e., bottom of the column).



This setting is also considered for the detailed shape sheet.
Indeed, if a specific linear mesh has been selected by the user (for example Mesh 24.1), all the verifications will be returned at this location:

3) Cross sections strength	
Tension Compression (6.2.4)	Case no 150 : 1.35x[1 D]+1.5x[15 Snw]+1.05x[17 L]+0.9x[3 WX+S2], Mesh No. 24.1 0/4 Cross section : Class 2 Fx < Nc,Rd : 90.8 < 1263.6 kN (7 %)
Shear on Y direction (6.2.6)	Case no 148 : 1.35x[1 D]+1.5x[18 SndrftX+]+1.05x[17 L]+0.9x[2 WX+S], Mesh No. 24.1 0/4 Cross section : Class 2 $\frac{h_w}{t_w} < 72\frac{\varepsilon}{\eta}$ (6.22) : 33.27 < 55.46 Fy,Ed < Vy,pl,Rd : 2.7 < 459.9 kN (1 %)
Shear on Z direction (6.2.6)	Case no 162 : 1.35x[1 D]+1.5x[15 Snw]+1.05x[17 L]+0.9x[7 WX-S2], Mesh No. 24.1 0/4 Cross section : Class 2 $\frac{h_w}{t_w} < 72\frac{\varepsilon}{\eta}$ (6.22) : 33.27 < 55.46 Fz,Ed < Vz,pl,Rd : 15.1 < 351.5 kN (4 %)
Bending on Y-Y (6.2.5)	Case no 156 : 1.35x[1 D]+1.5x[15 Snw]+1.05x[17 L]+0.9x[5 WX+S4], Mesh No. 24.1 4/4 Cross section : Class 1 My,Ed < My,c,Rd : 14.84 < 133.10 kN*m (11 %)
Bending on Z-Z (6.2.5)	Case no 148 : 1.35x[1 D]+1.5x[18 SndrftX+]+1.05x[17 L]+0.9x[2 WX+S], Mesh No. 24.1 4/4 Cross section : Class 2 Mz,Ed < Mz,c,Rd : 2.14 < 26.66 kN*m (8 %)
Bending on Y-Y and axial force (6.2.9)	Case no 187 : 1x[1 D]+1x[16 Snwa]+0.3x[17 L], Mesh No. 24.1 4/4 Cross section : Class 1 $N_{Ed} < 0.25 \cdot N_{pl,Rd}$ (6.33) : 68.2 kN < 315.9 kN $N_{Ed} < \frac{0.5 \cdot h_w\, t_w\, f_y}{\gamma_{M0}}$ (6.34) : 68.2 kN < 245.0 kN Clause 6.33 & 6.34 fulfilled. Check not done. 0.00000 < 1 (0 %)
Bending on Z-Z and axial force (6.2.9)	Case no 187 : 1x[1 D]+1x[16 Snwa]+0.3x[17 L], Mesh No. 24.1 4/4 Cross section : Class 1

6. Enhancing the analysis of timber structures

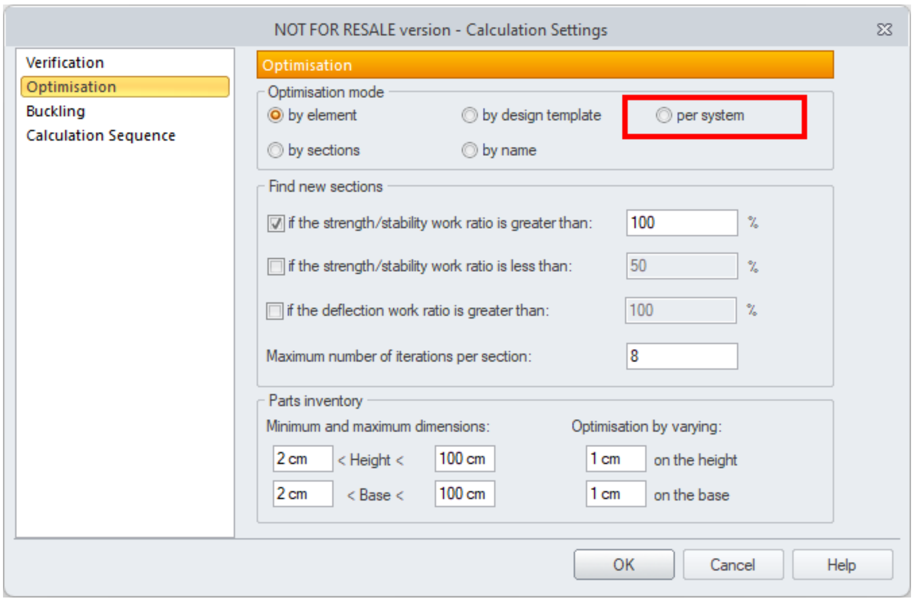
A series of novelties and improvements related to the verification and optimization of timber element structures.

6.1. Timber optimization by system (Eurocode)

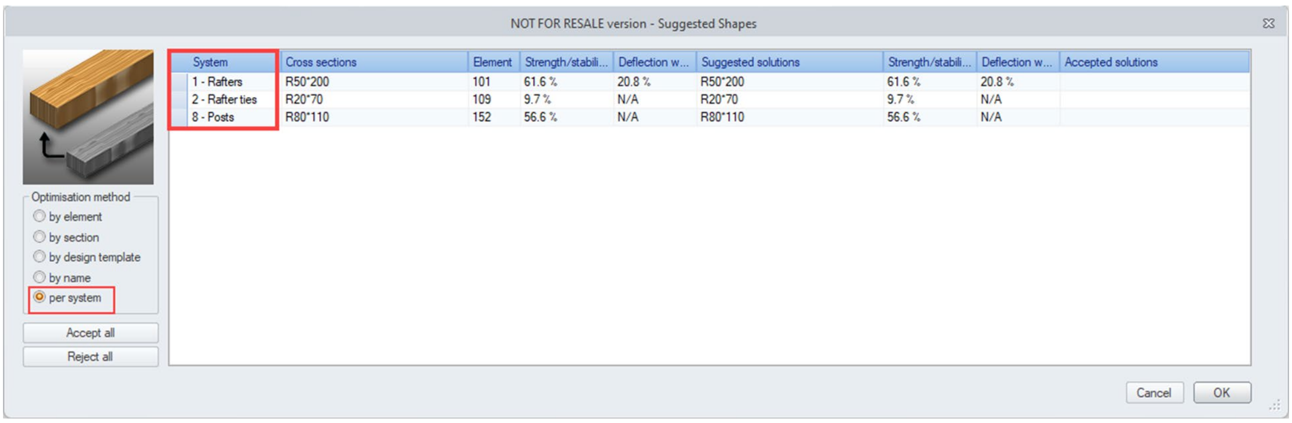
Possibility for timber elements to run the optimization per system.

Once the timber calculation according to Eurocode is completed, the timber design expert performs an optimization of the elements shape, according to the settings made in the timber design settings dialog box. The timber expert compares the work ratio of the timber elements with the specified criterion, and proposes other cross sections, which would correspond to the defined conditions.

The simplest method of proposing better profiles is to analyze each section independently. However, it is much more practical to group profiles according to different criteria. Until now, grouping of timber profiles for optimization could be done by section, by name and by design template. From Advance Design 2025 it is also possible to automatically group timber profiles by system, so the suggested shapes can be applied to all the elements within the system.



The optimization table on timber elements can now be organized by system for an easier control of the utilization of the structure.

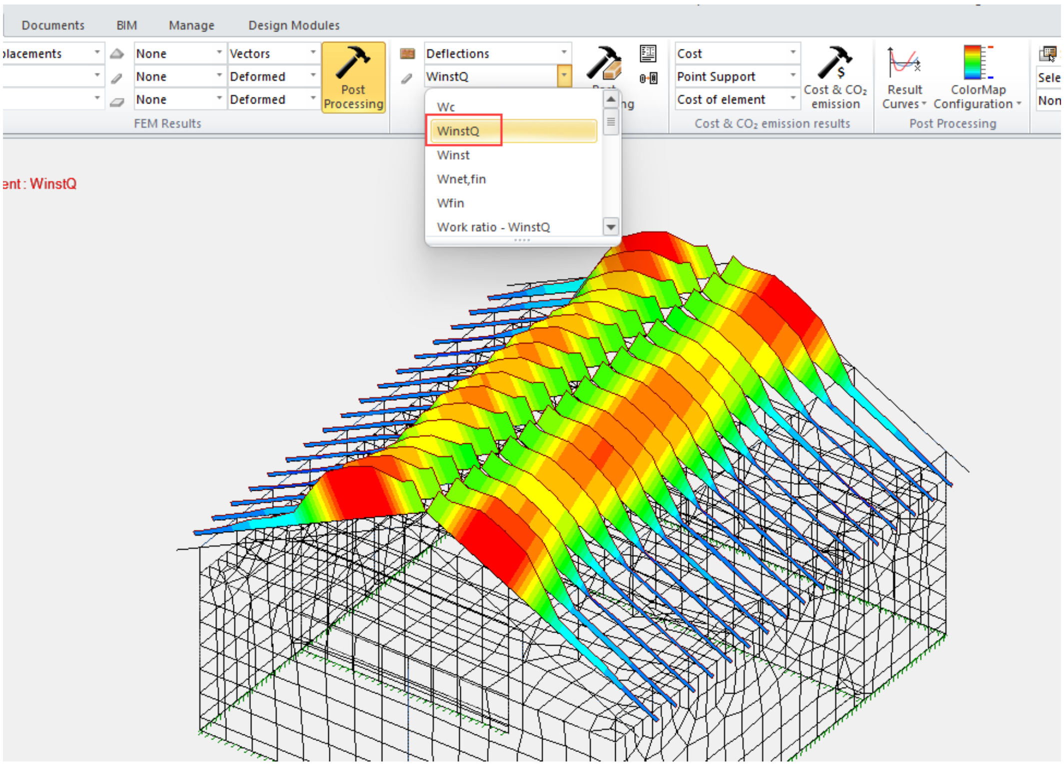


6.2. New entries in graphical verifications for deflection (Eurocode)

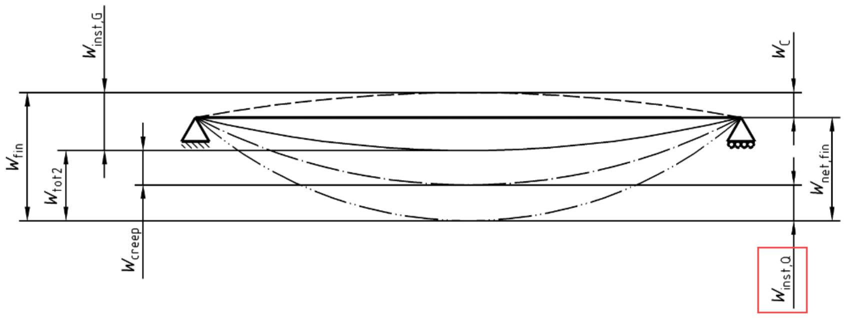
Possibility for selecting for graphical postprocessing new results for deflection from the timber design results.

The list with results available for graphical display for deflection for timber elements, in the case of Eurocode analysis results, has been expanded with new entries, making it faster and easier to verify and document a given result.

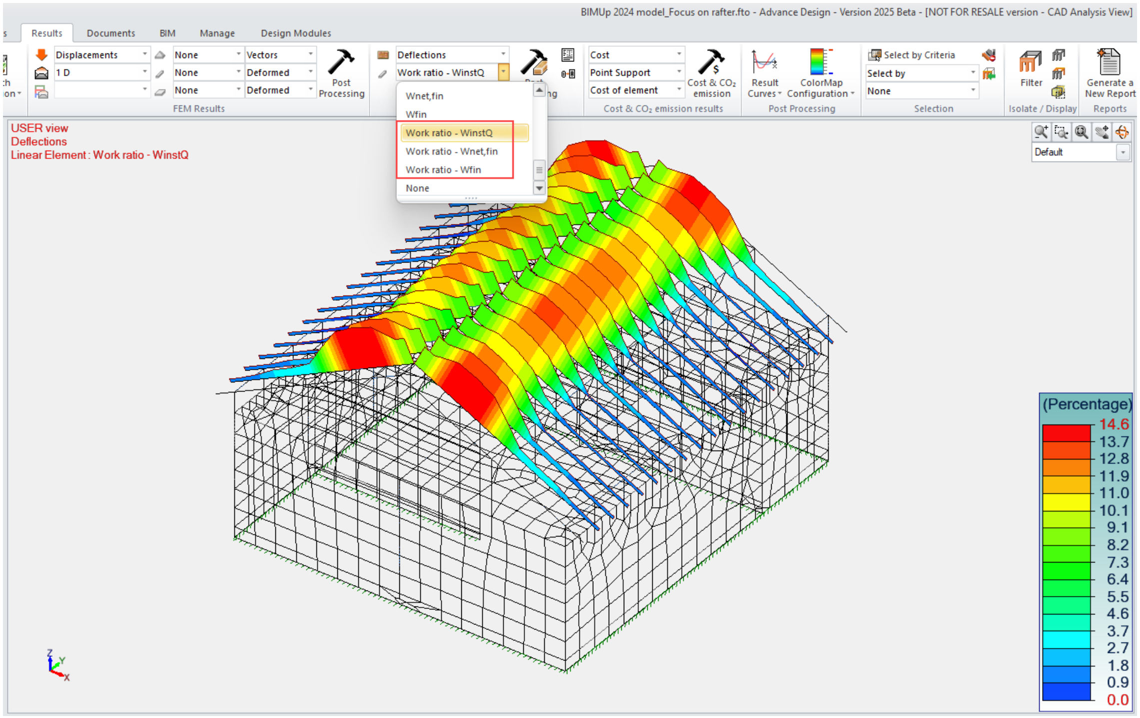
The first new element available in the drop-down list for visualization is the $w_{inst,Q}$ deflection.



This entry corresponds to the instantaneous deflection under variable loads.



In previous versions, this $w_{inst,Q}$ deflection was checked but the results were only available in the shape sheet.
The next new entries in the same list are the Work ratio values (in %) for the successive types of deflection.



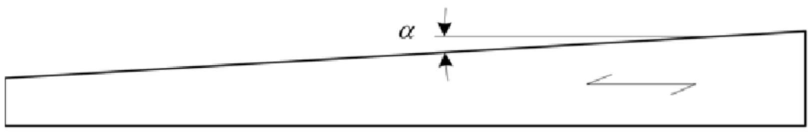
6.3. Single tapered beam (Eurocode)

Ability to design single tapered beams as per EN 1995-1-1

Advance Design 2025 is now able to design single tapered beams defined in §6.4.2 from EN 1995-1-1.

- **Definition**

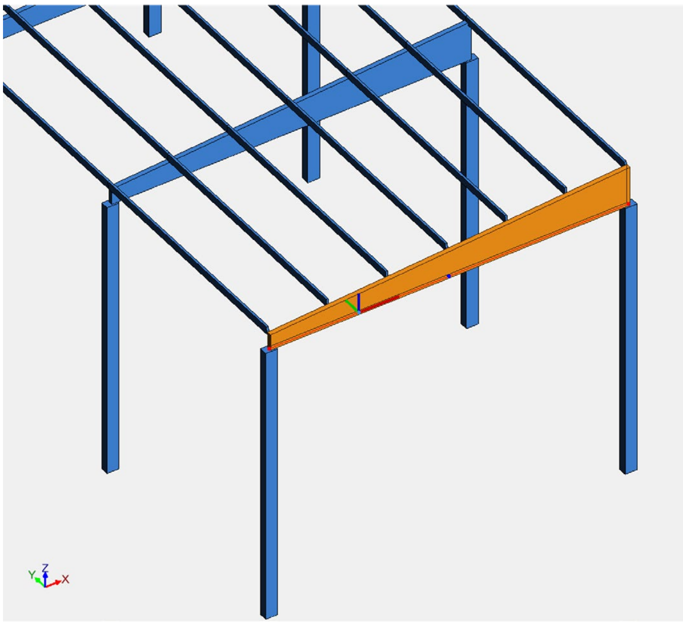
Tapered beams are rectangular in section with a linear slope from one end to the other.

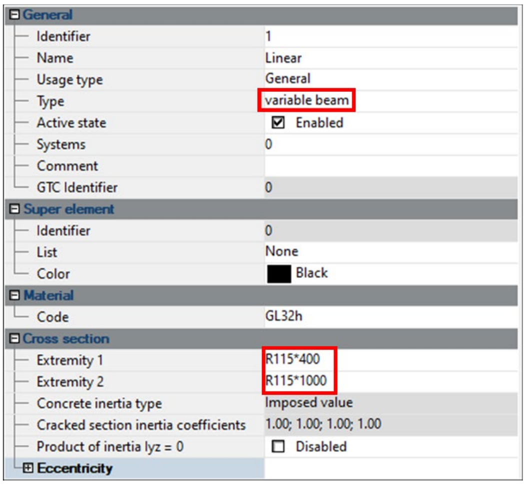


The angle of slope (α) is usually under 10° although no limit is defined in the Eurocode 5. Such tapered beams, typically made out of glued laminated timber, are common practice in roof construction.

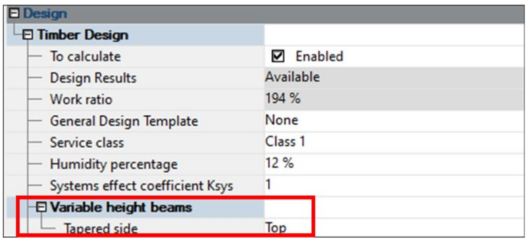
- **Modeling**

Tapred beams may be introduced in a model by defining a variable beam, with a different height at each end.
Then, in the timber design assumptions, the users will define the tapered side either on the top or bottom face.





General	
Identifier	1
Name	Linear
Usage type	General
Type	variable beam
Active state	☑ Enabled
Systems	0
Comment	
GTC Identifier	0
Super element	
Identifier	0
List	None
Color	Black
Material	
Code	GL32h
Cross section	
Extremity 1	R115*400
Extremity 2	R115*1000
Concrete inertia type	Imposed value
Cracked section inertia coefficients	1.00; 1.00; 1.00; 1.00
Product of inertia Iyz = 0	☐ Disabled
Eccentricity	



Design	
Timber Design	
To calculate	☑ Enabled
Design Results	Available
Work ratio	194 %
General Design Template	None
Service class	Class 1
Humidity percentage	12 %
Systems effect coefficient Ksys	1
Variable height beams	
Tapered side	Top

- **Design**

When checking the bending stress on the tapered edge, the bending strength should be reduced by a $k_{m,\alpha}$ factor. So at the outermost fiber of the tapered edge, the stress should satisfy the following expression:

$$\sigma_{m,\alpha,d} \leq k_{m,\alpha} f_{m,d} \qquad (6.38)$$

This factor is computed according to equation 6.39(for tensile stresses parallel to the tapered edge) or 6.40 (for compressive stresses parallel to the tapered edge):

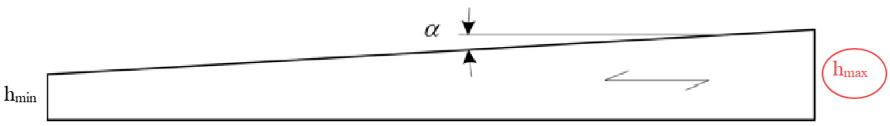
$$k_{m,\alpha} = \frac{1}{\sqrt{1+\left(\frac{f_{m,d}}{0,75 f_{v,d}}\tan\alpha\right)^2+\left(\frac{f_{m,d}}{f_{t,90,d}}\tan^2\alpha\right)^2}} \qquad (6.39)$$

$$k_{m,\alpha} = \frac{1}{\sqrt{1+\left(\frac{f_{m,d}}{1,5 f_{v,d}}\tan\alpha\right)^2+\left(\frac{f_{m,d}}{f_{c,90,d}}\tan^2\alpha\right)^2}} \qquad (6.40)$$

Where the tapered edge is under compression, the effects of lateral instability is accounted for, resulting in $k_{m,\alpha}$ and $k_{m,\alpha}$ both acting along to reduce the bending strength.

$$\sigma_{m,y,d} \leq k_{crit} f_{m,y,d} \longrightarrow \sigma_{m,y,d} \leq k_{crit} k_{m,\alpha} f_{m,y,d}$$

As a simplification, a conservative value of k_{crit} is computed by considering a section of uniform depth, where section height is maximum.



- **Reports**

The Cross-section part of the shape sheet has been updated to feature the values of the $k_{m,\alpha}$ factor.

1) Cross section	
Shape	R150*320 / R150*750
Dimensions(mm)	h = 320.00 b = 150.00 h = 750.00 b = 150.00
Cross sections(cm²)	Area = 480.00 Sy = 320.00 Sz = 320.00 Area = 1125.00 Sy = 750.00 Sz = 750.00
Inertia(cm4)	It = 25411.5 Iy = 40960 Iz = 9000 It = 73745.2 Iy = 527344 Iz = 21093.7
Modules(cm³)	Welyinf = 2560 Welysup = 2560 Welzinf = 1200 Welzsup = 1200 Welyinf = 14062.5 Welysup = 14062.5 Welzinf = 2812.5 Welzsup = 2812.5
Dimension factor	kh(N) = 1.100 khz(My)min = 1.000 khz(My)max = 1.053 khy(Mz)min = 1.100 khy(Mz)max = 1.100 kh(N) = 1.100 khz(My)min = 1.000 khz(My)max = 1.053 khy(Mz)min = 1.100 khy(Mz)max = 1.100
Modification factor (table 3.1)	kmod = 0.900 Duration: Short term Case no 102 kmod = 0.900 Duration: Short term Case no 102
Deformation factor (table 3.2)	kdef = 0.600 kdef = 0.600
Material(MPa)	E = 12600 ν = 0.0
Grade(MPa)	Fmk = 28 Ft0k = 22.3 Fc0k = 28 Fvk = 3.5
Slope Angle (6.4.2)	α = 2.735
Strength reduction factor	Tension: $k_{m\,\alpha}$ = 0.88 Compression: $k_{m\,\alpha}$ = 0.97

Note that the value of $k_{m,\alpha}$ is not constant along the member. It varies due to the size effect k_h factor (k_h) being involved in the $k_{m,\alpha}$ equation by increasing the $f_{m,d}$ strength as long the height of the section is small.

For compressive stress parallel to the tapered edge:

$$k_{m,\alpha} = \frac{1}{\sqrt{1+\left(\frac{f_{m,d}}{1,5\,f_{v,d}}\tan\alpha\right)^2+\left(\frac{f_{m,d}}{f_{c,90,d}}\tan^2\alpha\right)^2}}$$

$$f_{m,d} = \frac{k_{mod}\cdot k_h \cdot k_{sys}\cdot f_{m,g,k}}{\gamma_M}$$

Therefore, keep in mind that the $k_{m,\alpha}$ values featured in this section of the shape sheet are only relevant to the beginning of the member (at x = 0m).

The strength verifications have also been updated to feature the factor when relevant:

3) Cross sections strength	
Tension Compression (6.1.1/6.1.4)	not done (Fx = 0)
Shear on y / z direction (6.1.7)	Case no102: 1.35x[1 D]+1.5x[3 S], Mesh No. 1.1 0/4 $\tau_d \le f_{vd}$ (6.13) 0.89 ≤ 2.52 MPa (35%)
Single tapered beams (6.4.2)	Case no102: 1.35x[1 D]+1.5x[3 S], Mesh No. 1.3 4/4 $\sigma_{m\,\alpha d} = \frac{M}{W} \le k_{m\,\alpha}\, f_{md}$ (6.38) $10.92 = \frac{56.98\ kN*m}{5217.07\ cm^3} \le 0.97 \cdot 20.72$ 10.92 ≤ 20.04 MPa (55%)
Oblique bending (6.2)	Case no102: 1.35x[1 D]+1.5x[3 S], Mesh No. 1.3 4/4 $\sigma_{myd} / (k_{m\,\alpha}\, f_{myd}) + k_m\, \sigma_{mzd} / f_{mzd} \le 1$ (6.11) 0.55 ≤ 1.00 (55%) Case no102: 1.35x[1 D]+1.5x[3 S], Mesh No. 1.3 4/4 $k_m\, \sigma_{myd} / (k_{m\,\alpha}\, f_{myd}) + \sigma_{mzd} / f_{mzd} \le 1$ (6.12) 0.38 ≤ 1.00 (38%)

The presence of the $k_{m,\alpha}$ factor can also be noticed in the stability verifications:

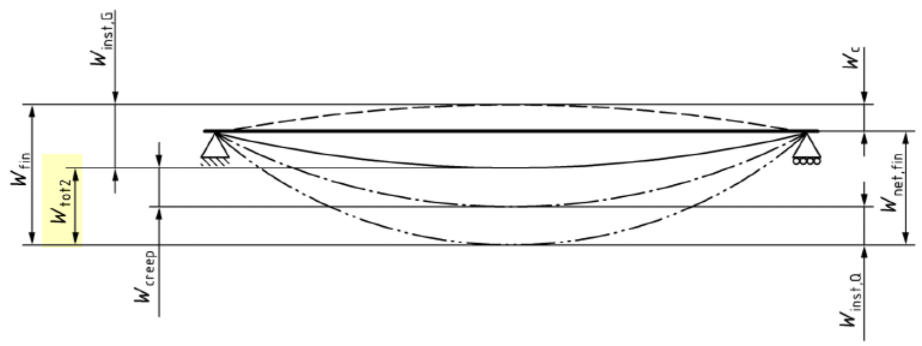
4) Elements stability	
Compression or combined compression and bending (6.3.2) Bending or combined bending and compression (6.3.3)	Case no101: 1x[1 L], Mesh No. 1.6 4/4 L_{fy} = 9.00 m L_{fz} = 9.00 m λ_y = 77.942 λ_z = 271.104 k_y = 0.774 k_{cy} = 0.532 k_z = 1.000 k_{cz} = 0.048 k_m = 0.700 $k_{m\,\alpha}$ = 0.748 $\lambda_{rel,y}$ = 1.048 $\lambda_{rel,z}$ = 0.162 k_{crit} = 0.524 $\lambda_{rel,m}$ = 1.381 L_{di} = 9.00 m L_{ds} = 9.00 m *Work ratio verification:* Case no101: 1x[1 L], Mesh No. 1.1 0/4 σ_{c0d} /(k_{cy} f_{c0d}) + σ_{myd} / ($k_{m\,\alpha}$ f_{myd}) + k_m σ_{mzd} / f_{mzd} ≤1 (6.23) 0.31 ≤ 1.00 (31%) Case no101: 1x[1 L], Mesh No. 1.1 0/4 σ_{c0d} /(k_{cz} f_{c0d}) + k_m σ_{myd} / ($k_{m\,\alpha}$ f_{myd}) + σ_{mzd} / f_{mzd} ≤1 (6.24) 0.22 ≤ 1.00 (22%) Case no101: 1x[1 L], Mesh No. 1.6 4/4 σ_{md} ≤ k_{crit} $k_{m\,\alpha}$ f_{md} (6.33) 7.84 ≤ 11.17 MPa (70%)

6.4. Deflection for brittle finishes criterion (France)

Thorough design of timber floors and ceilings at the SLS according to the French National Annex to EN 1995-1-1.

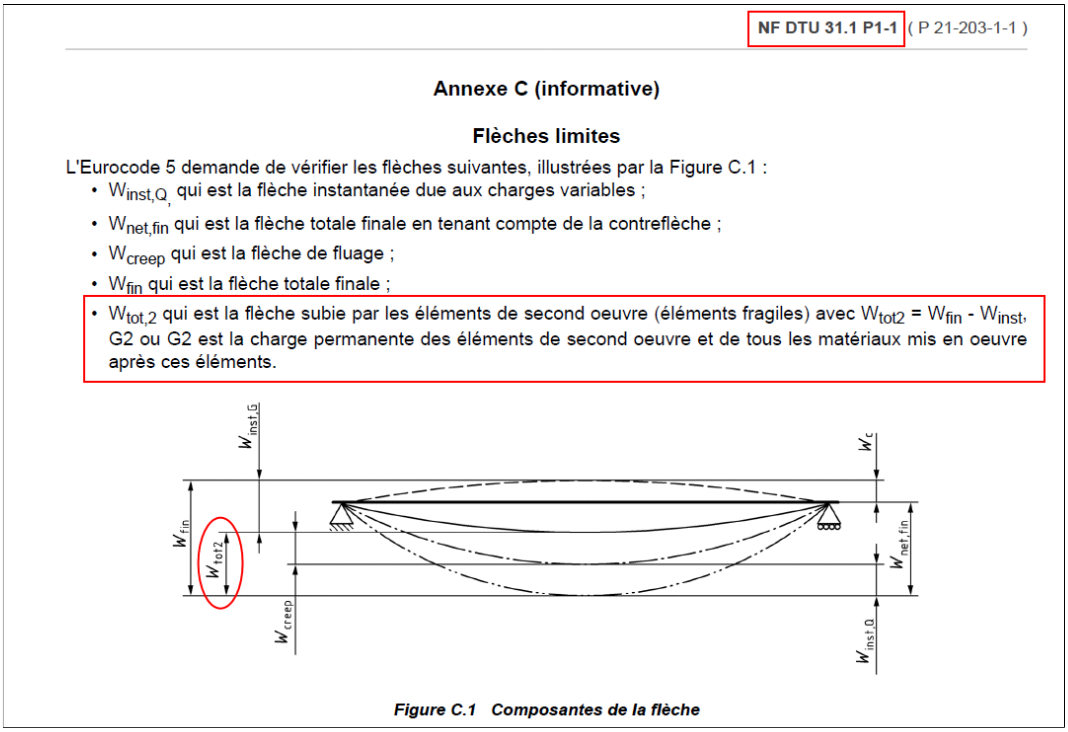
Advance Design 2025 now lets the user check the deflection sustained by the floor finishes, by measuring the $w_{tot,2}$ deflection and comparing it to the limit value imposed by the users.

- $W_{tot,2}$ qui est la flèche subie par les éléments de second oeuvre (éléments fragiles) avec $W_{tot2} = W_{fin} - W_{inst,}$ G2 ou G2 est la charge permanente des éléments de second oeuvre et de tous les matériaux mis en oeuvre après ces éléments.



- **Theory**

The $w_{tot,2}$ deflection is briefly mentioned in the French appendix of the EN 1995-1-1 standards (Clause 7.2 (2)). More information is provided in Annex C from DTU 31.1, part 1-1).



NF DTU 31.1 P1-1 (P 21-203-1-1)

Annexe C (informative)

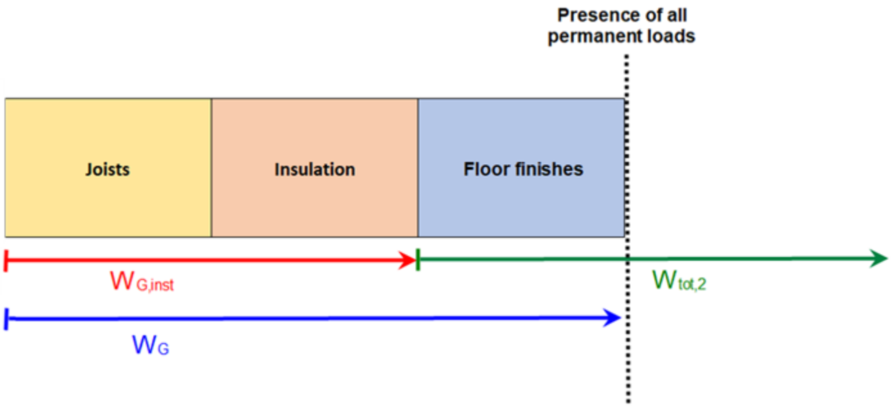
Flèches limites

L'Eurocode 5 demande de vérifier les flèches suivantes, illustrées par la Figure C.1 :

- $W_{inst,Q}$ qui est la flèche instantanée due aux charges variables ;
- $W_{net,fin}$ qui est la flèche totale finale en tenant compte de la contreflèche ;
- W_{creep} qui est la flèche de fluage ;
- W_{fin} qui est la flèche totale finale ;
- $W_{tot,2}$ qui est la flèche subie par les éléments de second oeuvre (éléments fragiles) avec $W_{tot2} = W_{fin} - W_{inst,}$ G2 ou G2 est la charge permanente des éléments de second oeuvre et de tous les matériaux mis en oeuvre après ces éléments.

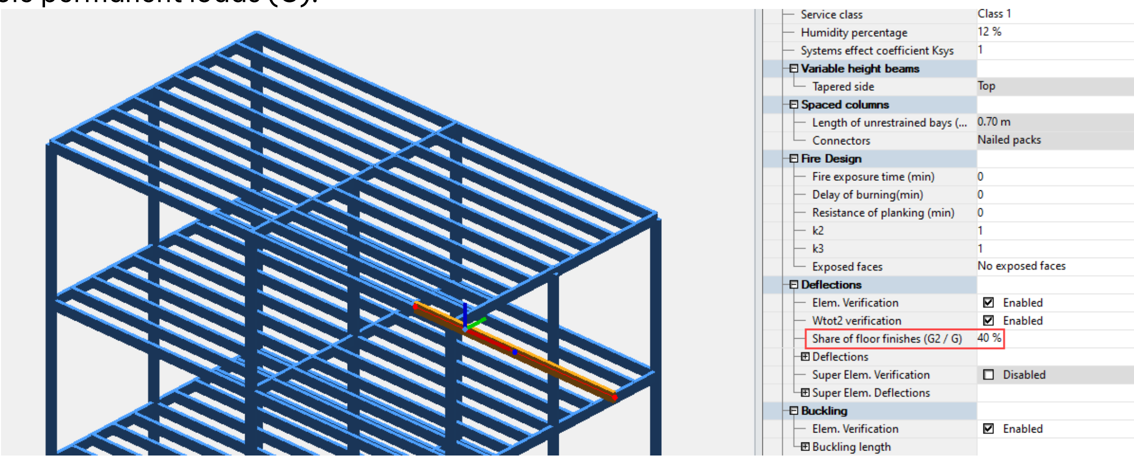
Figure C.1 Composantes de la flèche

$w_{tot,2}$ is the deflection that occurs once the floor finishes have been introduced.
The deformation measured prior to the introduction of the floor finishes (noted $W_{G,inst}$ below) should be excluded.

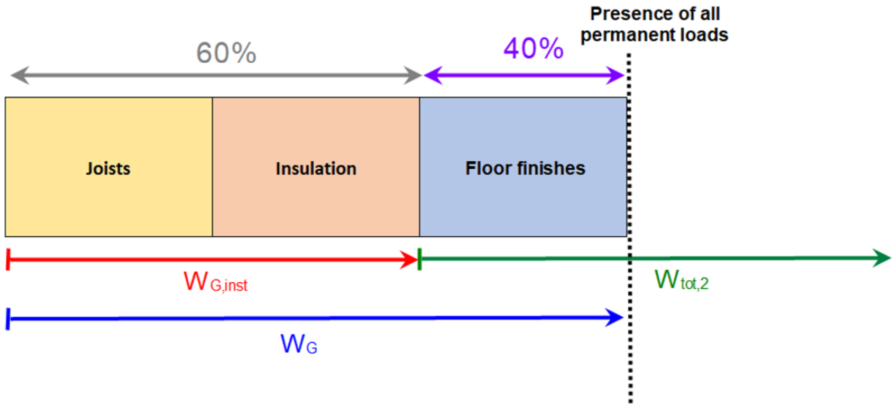


- **Definition**

In Advance Design, the users are expected to define the share of the G2 load (floor finishes) relative to the whole permanent loads (G).



The default amount (40%) means that only 40% of the deformation measured on the member for the G load case (all permanent loads acting) actually impacts the floor finishes.



In this case, to estimate the w_{tot2} deflection sustained by the floor finishes on a given floor, Advance Design will:

- Compute the final deflection at the SLS (w_{fin})
- Subtract the instantaneous deflection from permanent loads that does not impact the floor finishes (60% of w_G)

$$W_{tot,2} = W_{fin} - 60\% \cdot W_G$$

- **Limits**

Limit values should be defined in the project requirements, as stated in the DTU 31.1:

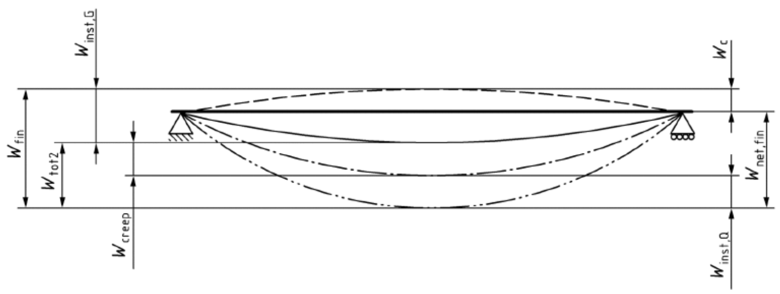


Figure C.1 Composantes de la flèche

L'Annexe nationale de l'Eurocode 5 donne les flèches limites pour $W_{inst,Q}$, $W_{net,fin}$ et W_{fin} au Tableau 7.2 de NF EN 1995-1-1/NA.

Par contre les flèches limites $W_{tot,2}$ doivent être données par les corps d'état des éléments de second oeuvre concernés ou par les documents du marché.

A défaut les valeurs du Tableau C.1 de $W_{tot,2}$ peuvent être utilisées.

Else, typical limit values can be found in the DTU31.1.
The limits depend:

- On the type of member (classic span or cantilever)
- On the nature of the fragile elements (tiles, plasterboards…)

Spans | **Cantilevers**

		Poutres de portée entre appuis L				Consoles et porte-à-faux			
		Rappel de l'Eurocode 5			Utiliser en complément	Rappel de l'Eurocode 5			Utiliser en complément
		$W_{inst,Q}$	$W_{net,fin}$	W_{fin}	W_{tot2}	$W_{inst,Q}$	$W_{net,fin}$	W_{fin}	W_{tot2}
Bâtiments agricoles et similaires	Chevrons ne supportant pas de matériaux fragiles	-	L/150	L/100	-	-	Maximum entre 5 mm et L/75	Maximum entre 5 mm et L/50	-
	Elements structuraux ne supportant pas de matériaux fragiles	L/200	L/150	L/100	-	Maximum entre 5 mm et L/100	Maximum entre 5 mm et L/75	Maximum entre 5 mm et L/50	-
Bâtiments courants	Chevrons ne supportant pas de matériaux fragiles	-	L/150	L/125	-	-	Maximum entre 5 mm et L/75	Maximum entre 5 mm et L/63	-
	Elements structuraux ne supportant pas de matériaux fragiles	L/300	L/200	L/125	-	Maximum entre 5 mm et L/150	Maximum entre 5 mm et L/100	Maximum entre 5 mm et L/63	-
	Elements structuraux supportant des plafonds en plaques de plâtre ou similaire	L/300	L/200	L/125	**L/350**	Maximum entre 5 mm et L/150	Maximum entre 5 mm et L/100	Maximum entre 5 mm et L/63	Maximum entre 5 mm et L/175
	Elements structuraux supportant des plafonds en plâtre projeté sur briquette	L/300	L/200	L/125	**L/400**	Maximum entre 5 mm et L/150	Maximum entre 5 mm et L/100	Maximum entre 5 mm et L/63	Maximum entre 5 mm et L/200
	Elements structuraux supportant du carrelage sur chape	L/300	L/200	L/125	**L/400**	Maximum entre 5 mm et L/150	Maximum entre 5 mm et L/100	Maximum entre 5 mm et L/63	Maximum entre 5 mm et L/200
	Elements structuraux supportant une chape humide sans couche de désolidarisation	L/300	L/200	L/125	**L/500**	Maximum entre 5 mm et L/150	Maximum entre 5 mm et L/100	Maximum entre 5 mm et L/63	Maximum entre 5 mm et L/250

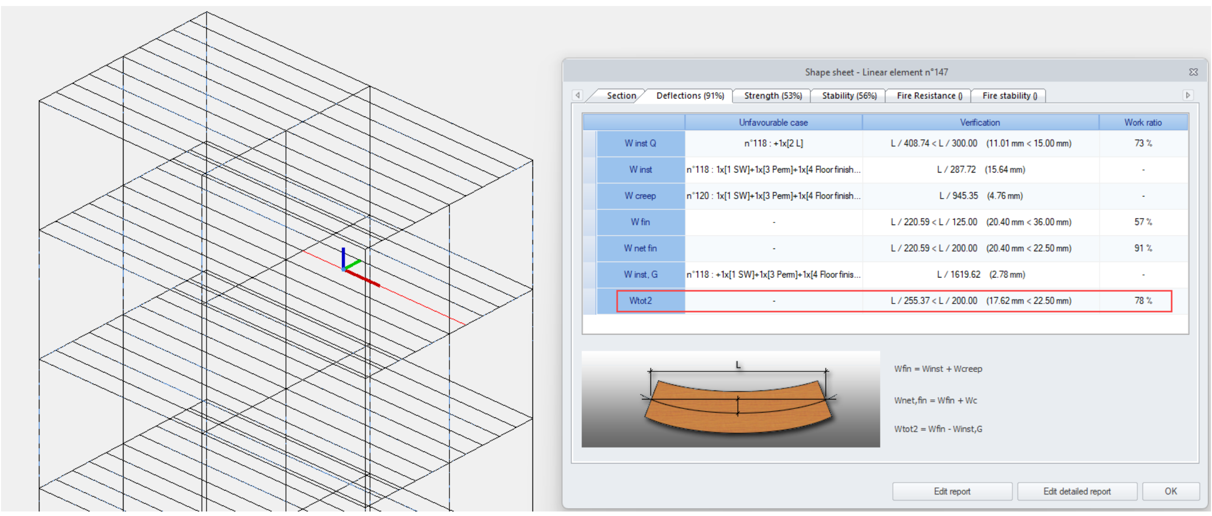
Tableau C.1 Flèches limites

In Advance Design, the default limit for w_{tot2} is set to L/400.

⊟ Deflections	
Elem. Verification	☑ Enabled
Wtot2 verification	☑ Enabled
Share of floor finishes (G2 / G)	40 %
⊟ Deflections	
Constructive defl. Wc	0.0 mm
Allowable defl. Winst = 1/	300
Allowable defl. Wnet,fin = 1/	200
Allowable defl. Wfin = 1/	125
Allowable defl. Wtot2 = 1/	400
Verif. Location	Deflection on span

- **Results**

The verification of the w_{tot2} deflection is available in the Deflection tab of the shape sheet.



These results are also available in the detailed version of the shape sheet.

2) Deflections	
	Case no 118, Mesh No. 147.2 WinstQ: L/408.74 < L/300.00 (11.01 mm < 15.00 mm) (73 %) Winst: L/287.72 (15.64 mm) Wcreep: L/945.35 (4.76 mm) Wfin: L/220.59 < L/125.00 (20.40 mm < 36.00 mm) (57 %) Wfin = Winst + Wcreep Wnetfin: L/220.59 < L/200.00 (20.40 mm < 22.50 mm) (91 %) Wnet,fin = Wfin - Wc WinstG: L/1619.62 (2.78 mm) Wtot2: L/255.37 < L/200.00 (17.62 mm < 22.50 mm) (78 %) Wtot2 = Wfin - WinstG

6.5. Faster timber design (Eurocode)

Better performance of the timber design engine

Advance Design 2025 offers a significant speed increase of the timber engine due to multiple optimizations of the operations occurring during the Eurocode 5 design.

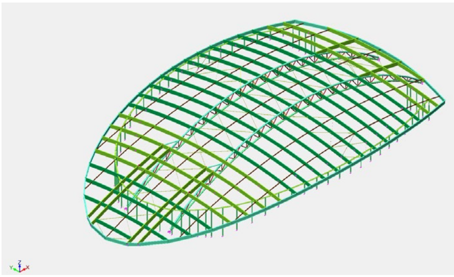
On several models, the timber design engine performed up to 70% faster than in previous versions.

Example 1

1598 linear elements
125 combinations

Calculation time
- Version 2024: 2m20s
- Version 2025: 0m45s

68% faster

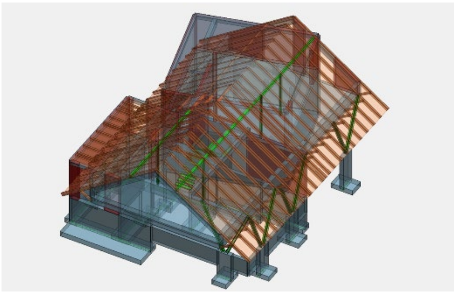


Example 2

342 linear elements
9 combinations

Calculation time
- Version 2024: 10s
- Version 2025: 3s

70% faster

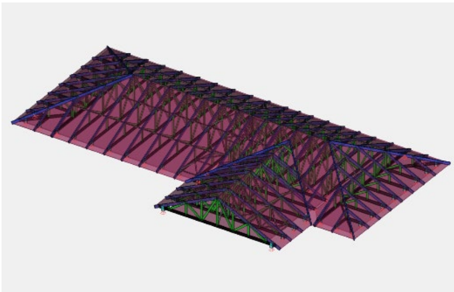


Example 3

671 linear elements
213 combinations

Calculation time
- Version 2024: 3m50s
- Version 2025: 1m49s

53% faster



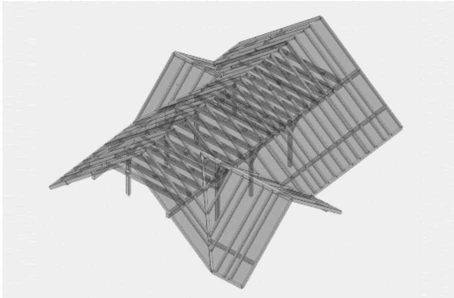
Example 4

205 linear elements
82 combinations

Calculation time

- Version 2024: 26s
- Version 2025: 9s

65% faster



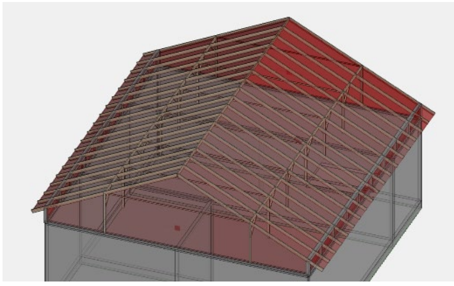
Example 5

122 linear elements
83 combinations

Calculation time

- Version 2024: 7s
- Version 2025: 3s

57% faster



7. Enhancing concrete structure design capabilities

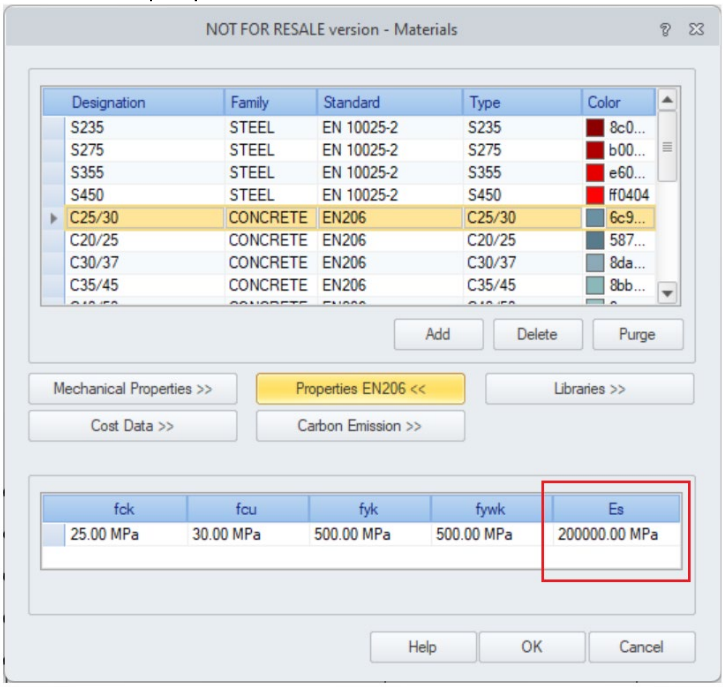
New features related to the analysis of reinforced concrete structures. Note that further news and improvements regarding reinforced concrete structures are described in the section on changes in RC design modules.

7.1. Possibility for editing Young modulus for reinforcing steel

The possibility for editing the Young modulus (Es) value for steel used for reinforcement. This will allow to simulate the reinforcement made of composites when analysing the theoretical reinforcement of elements.

Now in Advance Design 2025, it is possible to edit the Young modulus (Es) value for steel rebars in concrete material. The edited value is now considered for theoretical reinforcement for beam, columns, and planar elements, including interaction curves, and for detailed calculation with beam and column under Advance Design. This also includes the case of calculating an open/exported element to RC Beam and RC Colum modules. The impact of changing Young's modulus applies to calculations according to Eurocode and NAMER standards.

Since the strength parameters of the reinforcement are defined in Advance Design along with the concrete parameters, editing the Young's modulus for the reinforcement is done in the material definition window, in the concrete properties.



The reinforcement Young modulus is impacting several areas:

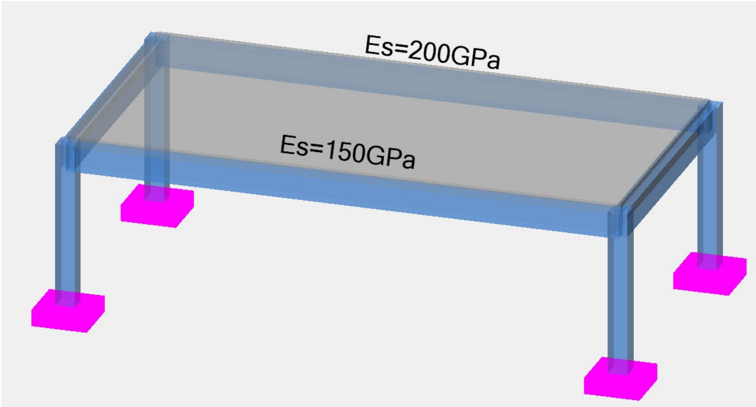
- In general, all SLS calculations, via equivalence coefficient in cracking, stresses, deflections, reinforcement.
- For ULS calculations, is the Steel strain affected and then Steel stress when using inclined steel law.
- For columns design methods Nominal stiffness, Curvature and General method.
- Interaction curves for columns.

The edited Young modulus can be used several specific cases, especially for different quality black carbon steel and stainless steel. In a more limited sense, as an approximation, we can use it to model non-steel reinforcement such us carbon or glass or polypropylene rebars.

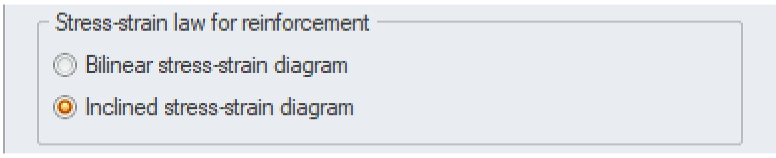
Special attention should be paid to the use of low Es stiffness values in calculations, as this has a major impact on SLS verification, and reinforcement from SLS becomes determinative.

Example

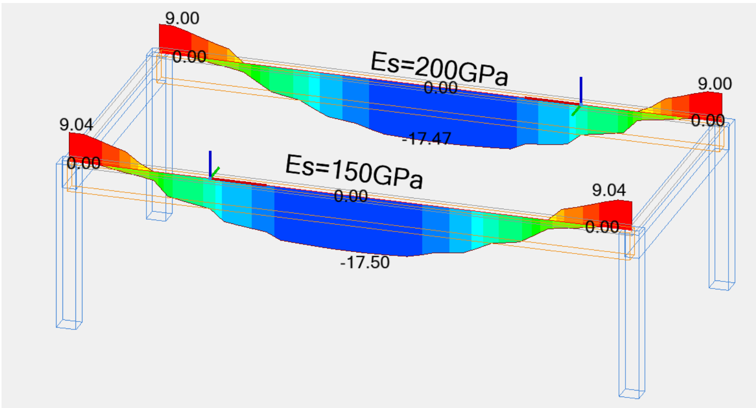
Let us consider a simple example with two beams having different Youngs modulus - with a value of Es=200GPa (29,000 ksi) typical for regular reinforcement, and with a modified value of Es=180GPa (26,100 ksi) .



For calculations we use inclined stress-strain law for reinforcement:

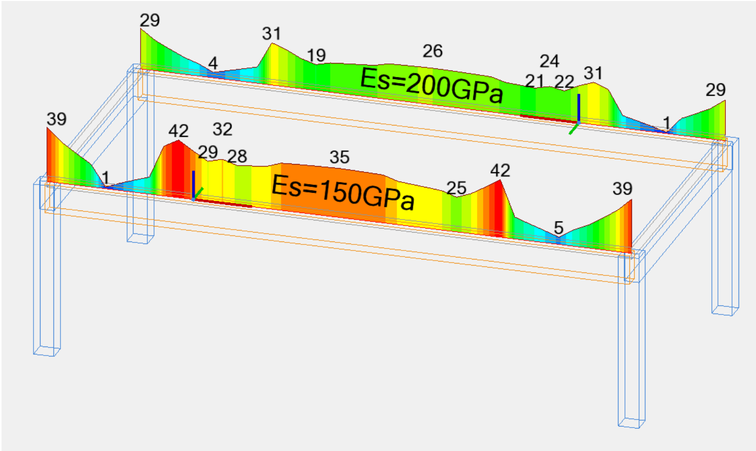


If we check the area of theoretical reinforcement Az we see a small impact of the change in modulus, mainly because of inclined law .

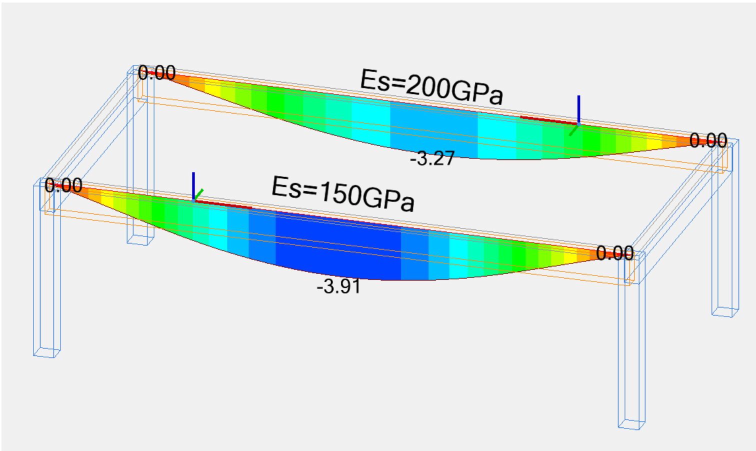


Theoretical area of longitudinal reinforcement (Az)

However, for the same area of reinforcement, we see a significant effect on cracking and on the total deflection of the beams.



Cracking (work ratio z)

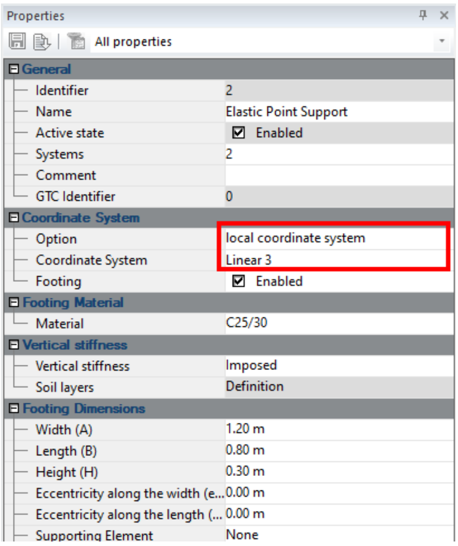


Total deflection

7.2. Considering the local system of support for foundations

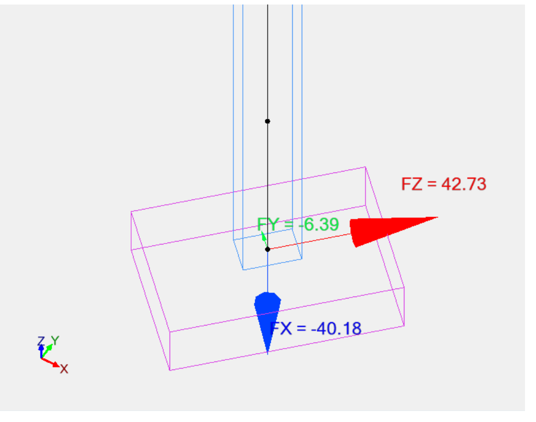
Consideration of the local support layout (set according to the supported element) when transferring foundation dimensions and reaction forces to RC Footing module.

In the case of supports below a column, whose reactions we want to read in the local axis system, rotated relative to the global system, we define such a support as elastic in Advance Design and set the local coordination system to that column.

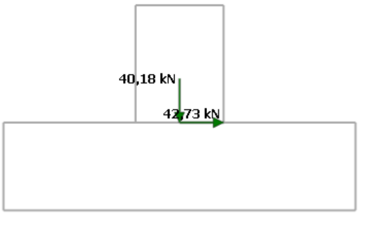


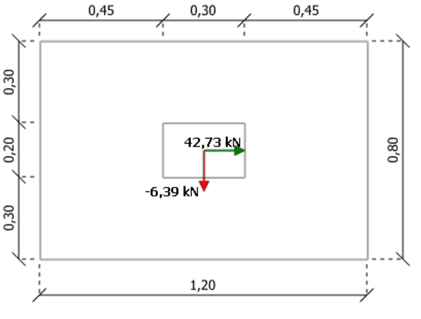
In the latest version of the program in this case, both the support reactions and the lateral dimensions of the foundation are transferred considering the rotated axis system.

Let us see this behavior with a simple example. Consider an elastic support defining a footing with dimensions A=1.20m and B=0.80m. Dimension A would be parallel to the global X axis if the axis system of the support were set to the global or user coordination system. However, in this example, the support has a local coordinate system set up according to the column, which is rotated relative to the global system by an angle of 45 degrees. In this case, dimension A is parallel to the Z-axis of the column's local system. Also, the support reactions are then in line with the column's local coordinate system - for example, the reaction Fz is parallel to the Z-axis of the column's local system.



When this foundation is then transferred to the RC Footing module, appropriately rotated support reactions according to the local column axis system are taken into the calculations.





It should also be mentioned that, as for elastic point supports, the transfer of reaction and foundation dimensions for linear elastic supports has also been updated.

7.3. Reduction of reinforcement calculation time for surface elements

Accelerate calculation time for calculating reinforcement for surface elements by optimizing the program code.

The algorithm for determining reinforcement in surface elements in the latest version of the program has been improved and modernized, bringing a reduction in calculation time.

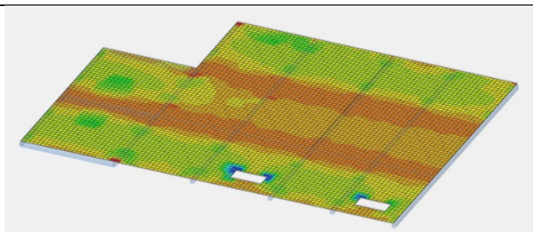
Although the acceleration should be visible for every example and standard, the biggest differences are visible for large models with a large number of combinations, especially during verifications according to Eurocode.

Examples of the acceleration of the reinforcement analysis time relative to versions 2024.1 vs. 2025:

Example 1

6744 finite elements

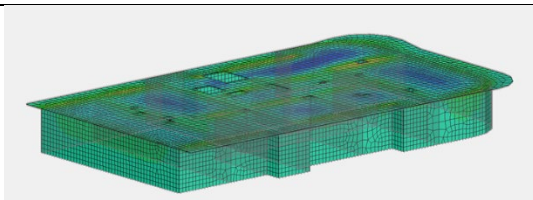
- CSA code **10% faster**
- Eurocode **68% faster**



Example 2

7166 finite elements

- Eurocode **35% faster**



8. Results

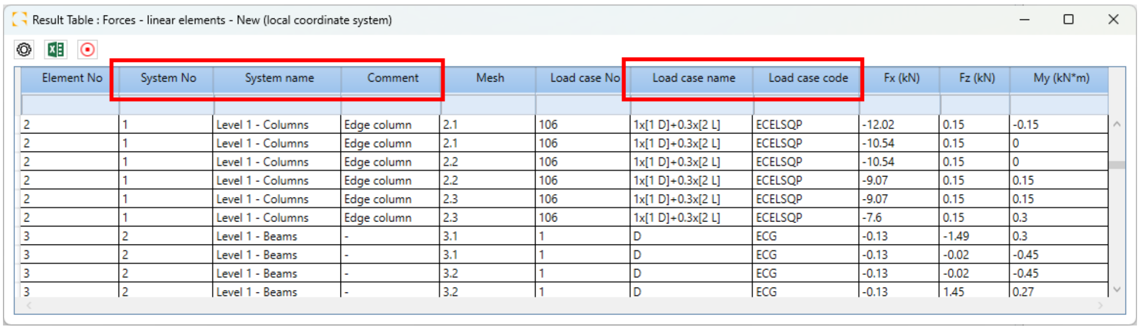
A series of novelties and improvements related to the presentation of results.

8.1. Additional data properties on Result tables

A set of new fields with properties to be selected when creating custom tables with results.

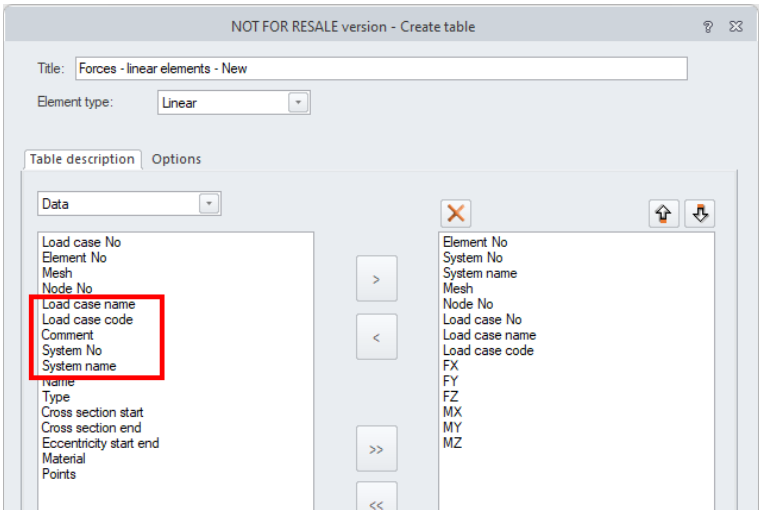
To make it easier to verify the results with tables, in Advance Design 2025, additional data properties are available in user result tables.

For **linear** and **planar elements** in addition to **point**, **linear** and **planar supports**: the user can now display the name and code of the load cases, the name and number of structural systems containing the elements and the comment placed for each of them.

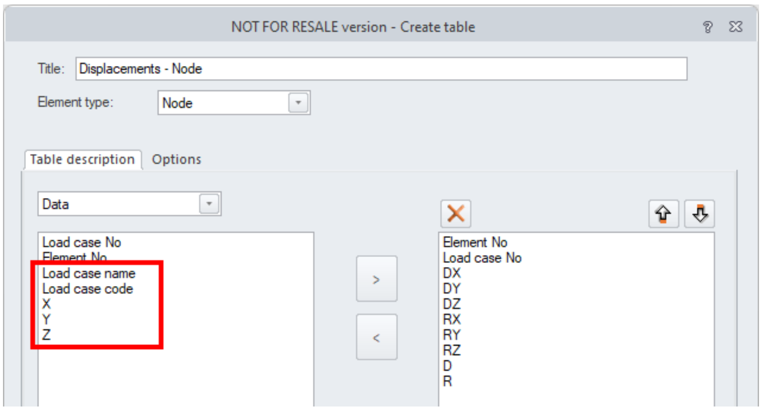

Result Table : Forces - linear elements - New (local coordinate system)

Element No	System No	System name	Comment	Mesh	Load case No	Load case name	Load case code	Fx (kN)	Fz (kN)	My (kN*m)
2	1	Level 1 - Columns	Edge column	2.1	106	1x[1 D]+0.3x[2 L]	ECELSQP	-12.02	0.15	-0.15
2	1	Level 1 - Columns	Edge column	2.1	106	1x[1 D]+0.3x[2 L]	ECELSQP	-10.54	0.15	0
2	1	Level 1 - Columns	Edge column	2.2	106	1x[1 D]+0.3x[2 L]	ECELSQP	-10.54	0.15	0
2	1	Level 1 - Columns	Edge column	2.2	106	1x[1 D]+0.3x[2 L]	ECELSQP	-9.07	0.15	0.15
2	1	Level 1 - Columns	Edge column	2.3	106	1x[1 D]+0.3x[2 L]	ECELSQP	-9.07	0.15	0.15
2	1	Level 1 - Columns	Edge column	2.3	106	1x[1 D]+0.3x[2 L]	ECELSQP	-7.6	0.15	0.3
3	2	Level 1 - Beams	-	3.1	1	D	ECG	-0.13	-1.49	0.3
3	2	Level 1 - Beams	-	3.1	1	D	ECG	-0.13	-0.02	-0.45
3	2	Level 1 - Beams	-	3.2	1	D	ECG	-0.13	-0.02	-0.45
3	2	Level 1 - Beams	-	3.2	1	D	ECG	-0.13	1.45	0.27

For on the Result Table Settings dialog corresponding new fields are available.



For **nodes**: the user can now display additionally the global coordinates of the nodes, the name and code of the load cases.

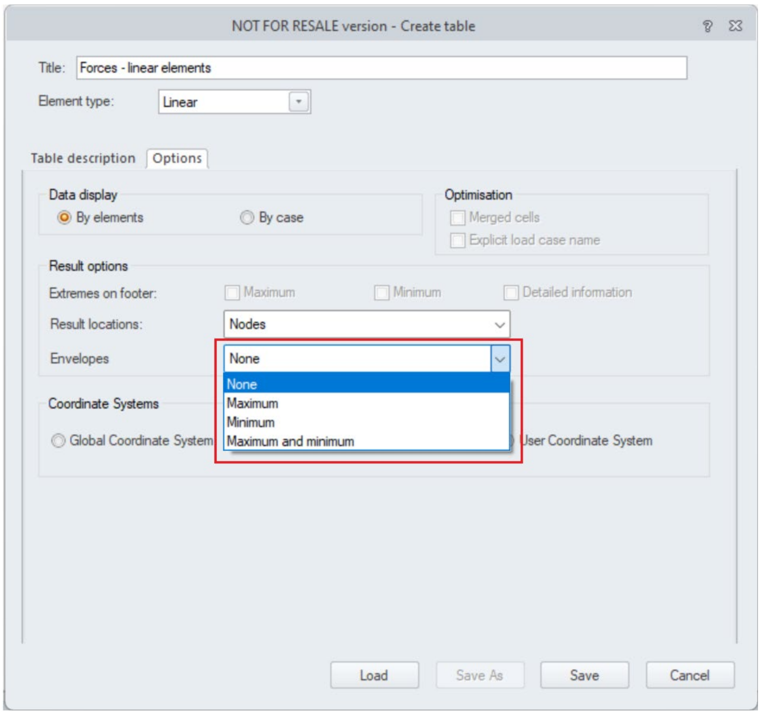


8.2. Display of extreme values on Result tables

New modes of displaying values in tables with results with displaying extreme results (envelope - min/max) . This allows for easy looking for only an extreme value of a force and its localization.

In the result tables of Advance Design 2025, it is now possible to uniquely display the loads/combinations that generate the extreme results (displacement, internal forces, and stresses).

Instead of displaying the results of each load/combination and getting tables with a lot of rows, the user has now the option to only view the load/combination that causes extreme value for each component of results in every element. To activate this display mode, the user needs to choose an envelope from the result table settings.



By default, envelopes are set to none and all loads/combinations are displayed. The user can change it to visualize the maximum only, minimum only or maximum and minimum results.

According to the user's choice and for every element, the table will display only the load/combination that gives extreme values for every component of the result (extreme values are in red). In the following is an example of maximum and minimum internal forces values for line elements of a structure

Result Table : Forces - linear elements - Min and Max (local coordinate system)

Element No	Load case No	Mesh	Node No	Fx (kN)	Fy (kN)	Fz (kN)	Mx (kN*m)	My (kN*m)	Mz (kN*m)
7	103	7.1	13	-35.75	0	-99.71	0	71.82	0
7	107	7.6	26	-1.4	0	4.41	0	2.81	0
7	1	7.1	13	-1.4	0	-4.41	0	2.81	0
7	109	7.6	26	-8.17	0	23.16	0	16.42	0
7	102	7.1	13	-30.28	0	-107.21	0	78.11	-0.03
7	102	7.6	26	-30.28	0	107.21	0	78.11	-0.03
7	1	7.1	13	-1.4	0	-4.41	0	2.81	0
7	109	7.6	26	-8.17	0	23.16	0	16.42	0
7	103	7.3	20	-35.75	0	0	0	-99.63	0
7	102	7.6	26	-30.28	0	107.21	0	78.11	-0.03
7	102	7.1	13	-30.28	0	-107.21	0	78.11	-0.03
7	109	7.6	26	-8.17	0	23.16	0	16.42	0
8	103	8.1	15	-35.75	0	-99.71	0	71.82	0
8	107	8.6	28	-1.4	0	4.41	0	2.81	0
8	1	8.1	15	-1.4	0	-4.41	0	2.81	0
8	109	8.6	28	-8.17	0	23.16	0	16.42	0
8	103	8.1	15	-35.75	0	-99.71	0	71.82	0

In this table we can see for example that for the element 7 the minimum moment My is -99.63 kNm and is occurring for load case 103 at mesh 7.3 (node 20). While the maximum moment My is 78.11 kNm and is occurring for load case 102 at mesh 7.6 (node 26).

Another example of the table is for the element 8, the minimum axial force Fx is -32.75 kN and is occurring for load case 103 at mesh 8.1 (node 15). While the maximum force Fx is -1.4 kN and is occurring for load case 107 at mesh 8.6 (node 28).

8.3. New commands on postprocessing ribbon

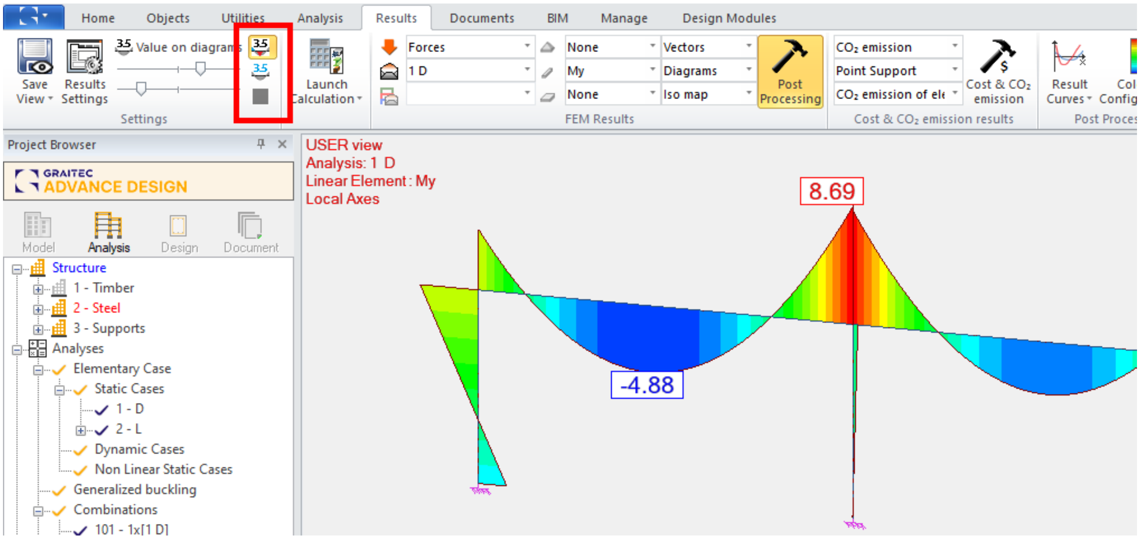
The ability to easily access from the Ribbon some frequently used postprocessing options.

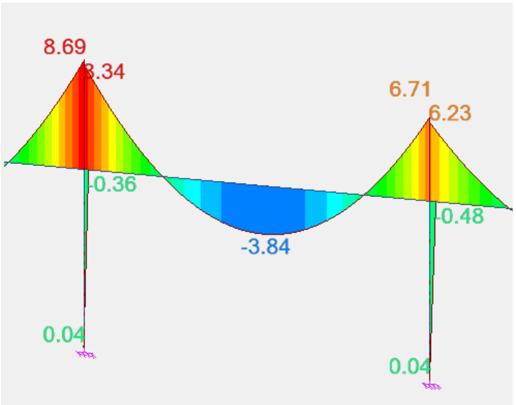
The latest version of AD brings further improvements to the graphic post-processing of the results, consisting in making it easier to select the most used options by placing them on the Results ribbon. There are 3 types of changes: new commands on the ribbon for displaying values, quick selection of Display mode for FEM results from the ribbon, and a new Automatic postprocessing option.

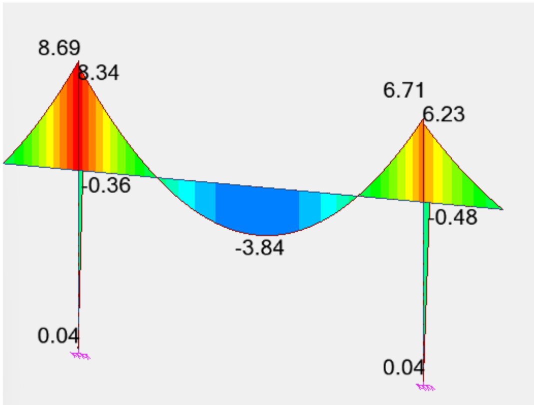
New commands on the ribbon for displaying values

On the **Results** ribbon, in the **Settings** group, 3 new commands for displaying values were added:

- The top command is used to activate the display of extreme values.
- The middle command is used to activate the display values in a solid color.
- The lower command opens a color selection window for displayed values.

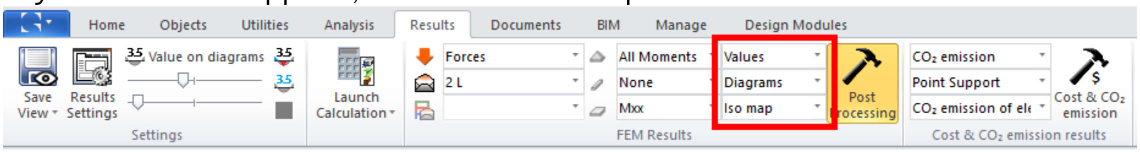


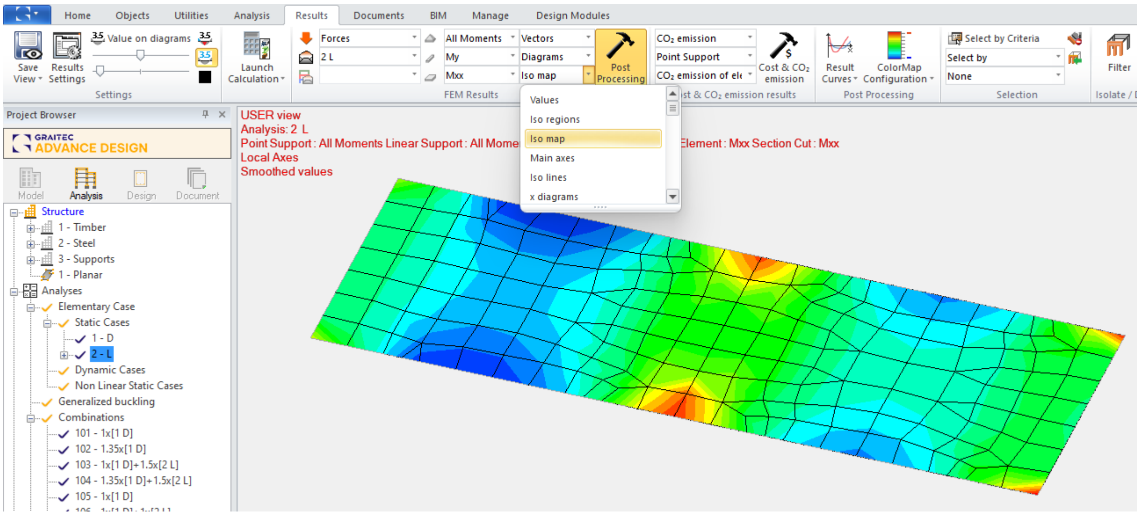




Quick selection of Display mode for FEM results

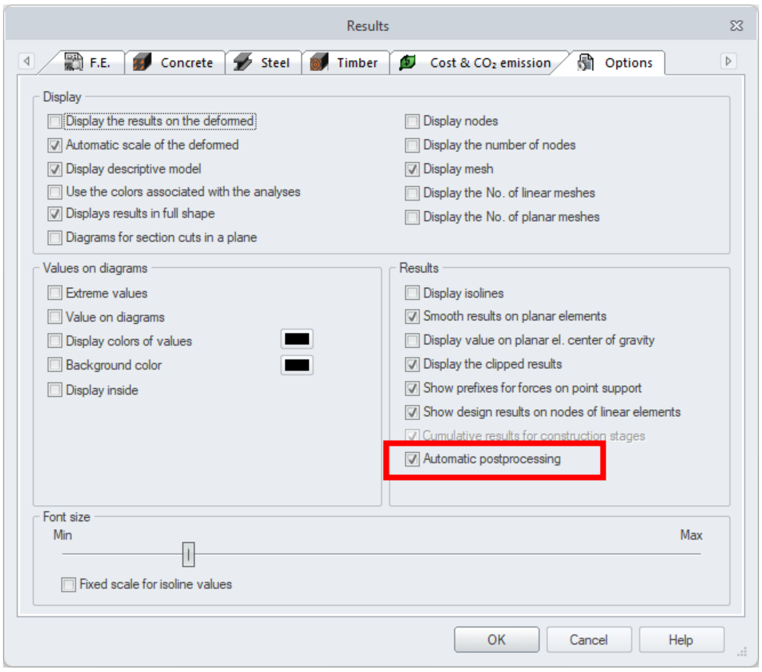
On the **Results** ribbon, in the **FEM Results** group, Display mode selection lists have been added, separately for results for supports, linear elements and planar elements.





Automatic postprocessing

A new option 'Automatic postprocessing' has been added to the Options tab of the Results dialog. It is enabled by default and its function is quite simple - after selecting any result from the ribbon, the result will be automatically accepted and displayed graphically, thus there is no need to use the 'Postprocessing' button. This small change increases the comfort of everyday work.



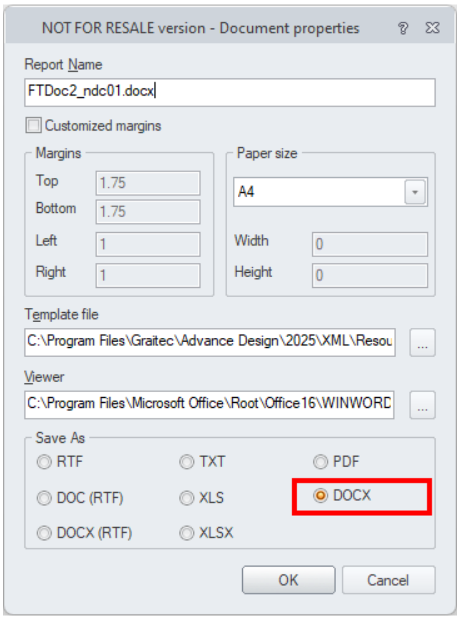
8.4. New mechanism for creating reports

A new report creation mechanism that allows direct generation of content in Microsoft Word (docx) format.

Previously, Advance Design reports used a content generation mechanism based on the RTF format, which could be further displayed/converted to other formats, including DOC/DOCX. Unfortunately, limitations of the RTF format, such as lack of content compression, sometimes result in the inability to generate a report when the RTF file is larger than 512 MB.

In Advance Design 2025, an additional parallel mechanism has been introduced that allows you to define content directly using DOCX native report file. The mechanism is based on OpenXML (by Microsoft), which has excellent speed and quality, and is devoid of the limitations of the old RTF format. The new report mechanism will allow for a number of improvements in future versions of the program, including a new way of previewing.

A new mechanism using the native DOCX format can be selected in the report configuration window.



Since the process of adapting report templates to the new format is being introduced in stages, therefore in version 2025 reports can be generated for all types of tables except those containing formulas, for example, Shape Sheets.

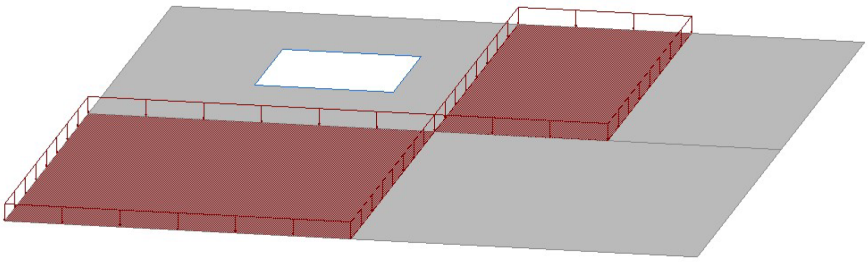
9. Enhanced user experience and the comfort of program operation

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

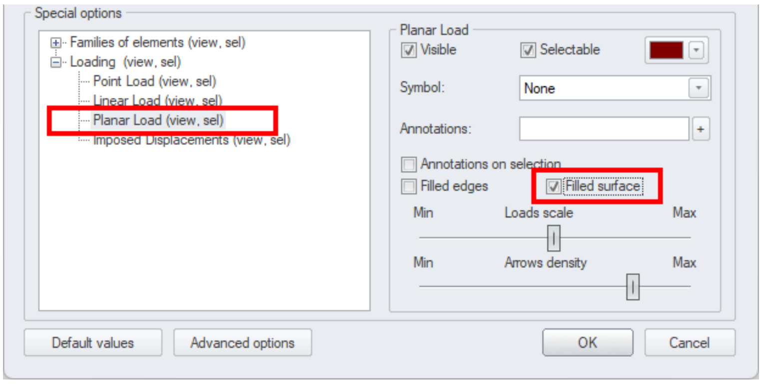
9.1. Infill for planar loads presentation

A new option to the Planar loads to fill surface by color. Helpful both when working and creating documentation, especially to present loads in a top view.

When defining or verifying a defined load on surface elements, we often operate with a view perpendicular to the surface of these elements. This view is also often used to prepare documentation that includes a presentation of the distribution of surface loads. In order to make the surface load visible in such cases, the latest version 2025 introduced the possibility to fill the area of the load with color.



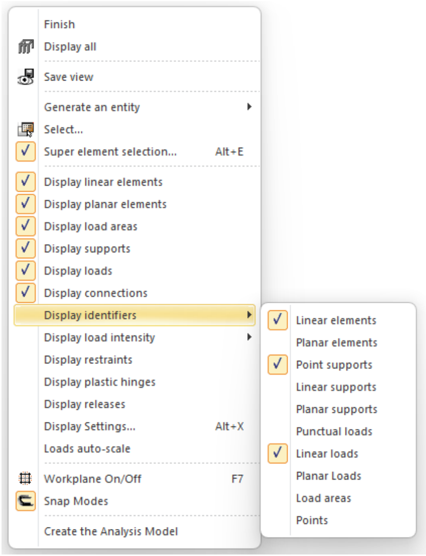
Activation of the new 'Filled surface' option is available for surface loads in the Display settings window.

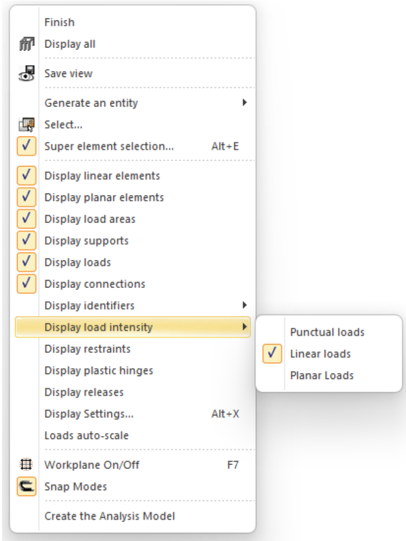


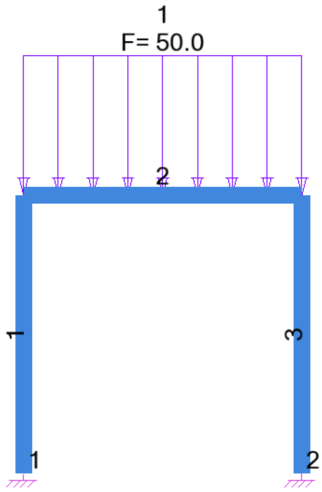
9.2. Quick display of object identification numbers and load values

Quick display of object identification numbers and load values using the right click menu. This makes it faster and easier to manage the display of components.

In Advance Design 2025, it is now easier in the model phase to display objects ID number without having to go through the display settings dialog. From the right click menu, the user can now directly activate the display identifiers for linear and planar elements, supports, loads load areas and points. In addition, the quick display of load intensities can be activated in a similar way.



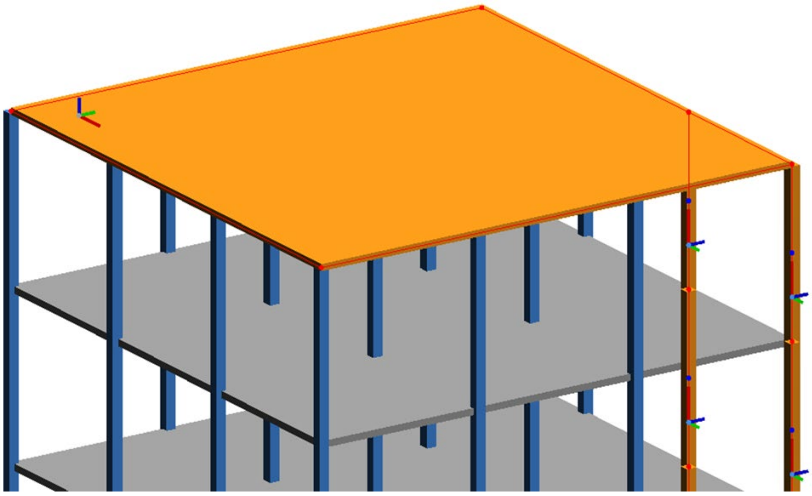


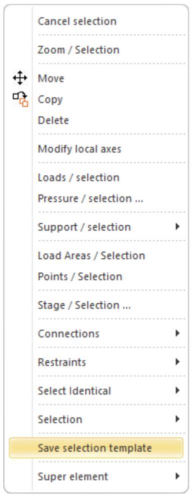


9.3. Easier and faster creation of element selection templates

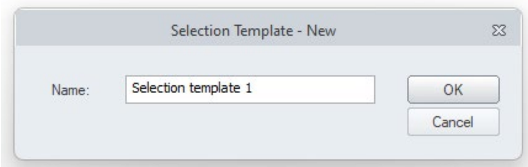
Easier and faster creation of element selection templates by saving selection templates from the right click menu.

In Advance Design 2025, it is now easier to create a selection template without having to go through the criteria selection dialog. The user can now select the elements then save them in a selection template via the new command "Save selection template" of the right click menu.





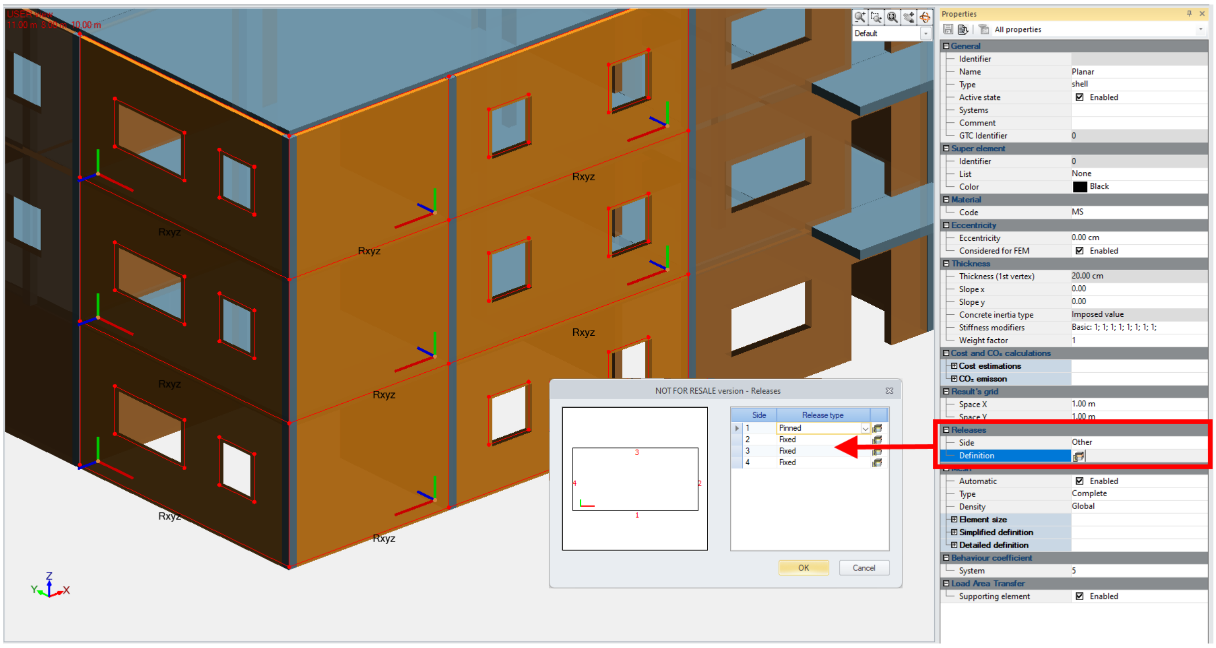
This will open a small dialog for the user to name the selection template and save it.



9.4. Defining linear releases for selected edges for multiple planar elements

The possibility to set linear releases on one (or more) selected edges for multiple elements having the same geometry.

In order to simplify the definition of linear releases on the edges of surface elements, it is now possible to call the window for setting the releases on selected edges also in case of selection of multiple surface elements. Thanks to this, we can very quickly set the same release on selected edges in multiple elements simultaneously.



In order to bring up a window for setting edge releases, the selection should contain surface elements with the same number of outer edges.

As the edges of the surface elements are oriented with respect to the local axis of the element, so for better control, it is recommended to set the same local axis orientation for the selected elements in advance.

9.5. Ability to sort the elements from a system

New options to easily sort the elements in a system using different criteria.

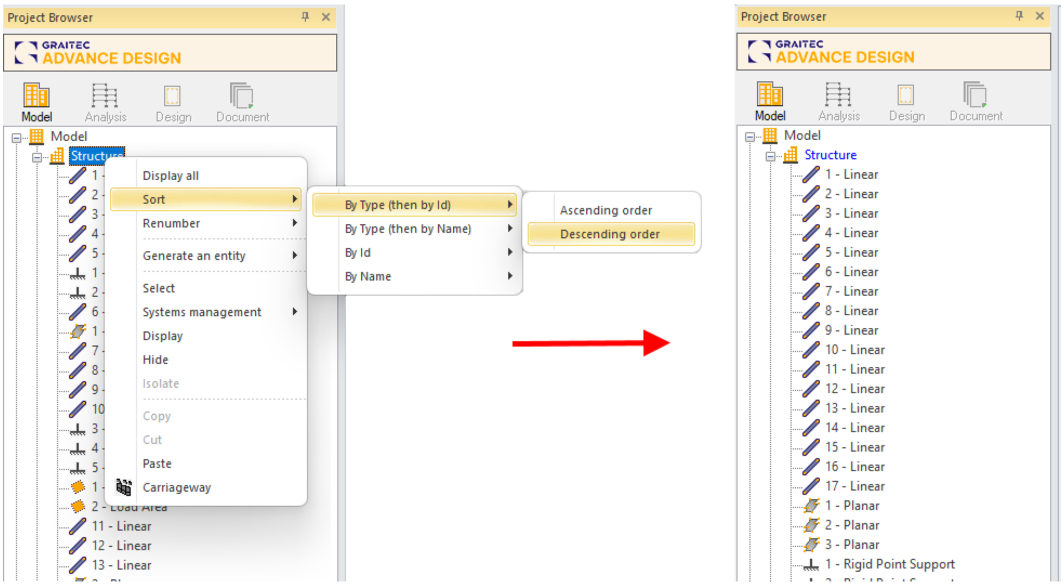
When creating and editing structures, we often ignore the order of elements entered and their assignment to systems. This can result in an unclear list of elements visible in the Project Browser. In order to make work easier, including finding and selecting elements, a set of new commands has been introduced for sorting content of object list.

New commands are available in the context menu opened on any system.

Four types of sorting are available:

- By type and then by Id of elements
- By type and then name of elements
- By Id
- By name

In all types we have the option of sorting the numbering/alphabetical ascending or descending.

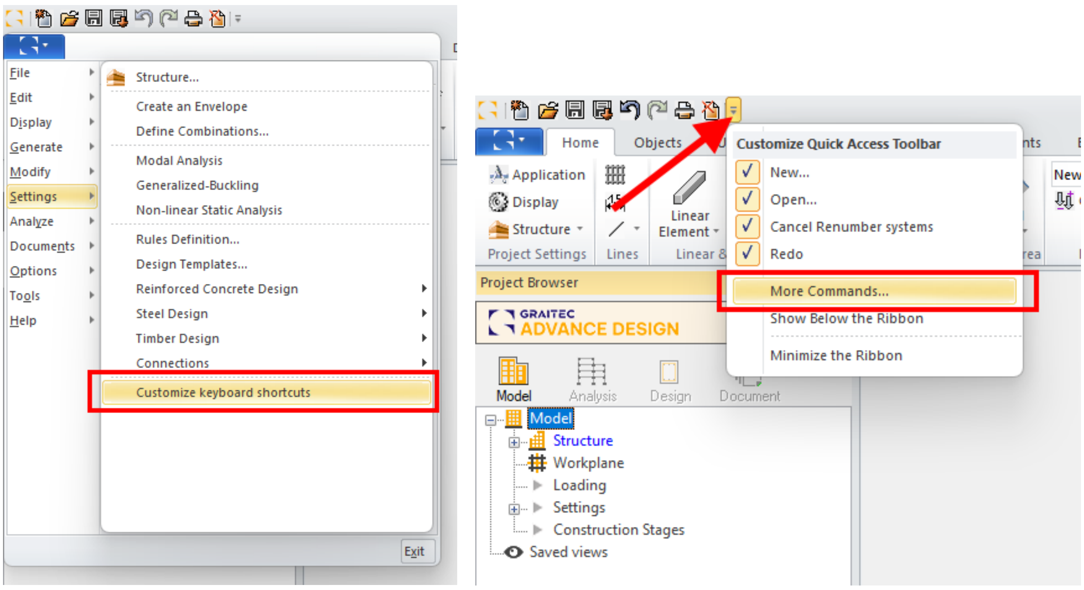


9.6. Configuration of keyboard shortcuts

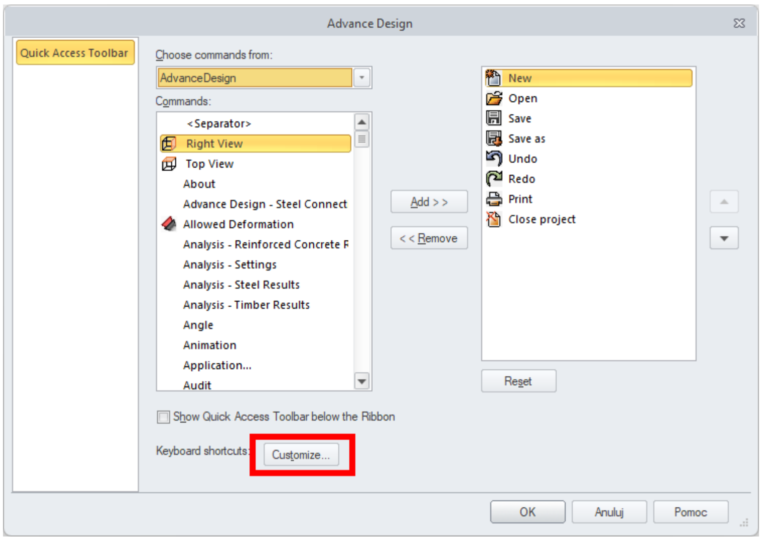
Possibility to configure custom keyboard shortcuts in Advance Design environment. You can check the current mapping of keyboard shortcuts and add your own.

In Advance Design 2025, the user can now customize keyboard shortcuts and add new shortcuts to any command. To open the keyboard shortcuts customization dialog, first we need to open the dialog for managing the Quick Access Toolbar using one of two methods:

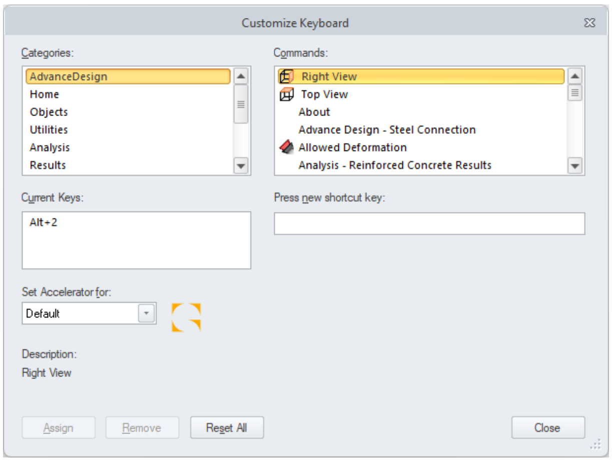
- by selecting the *Customize keyboard shortcuts* command available on the Settings menu
- or by clicking on *More commands…* option in the *Customize Quick Access Toolbar*.



Then next to the Keyboard shortcuts we can use the *Customize* button.



In the keyboard shortcuts customization dialog, the user can select the command from each category and visualize its current shortcut keys (if any), replace the current keyboard shortcut, create a new one or add an additional shortcut for the same command.



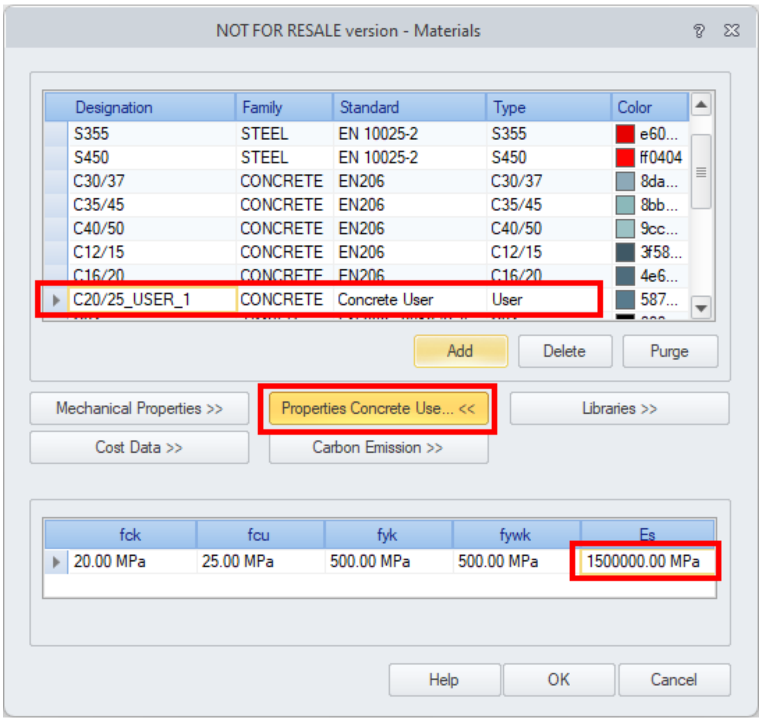
9.7. Improvement to editing material properties

Automatically creating a new user material if a manual change of parameters has been made for an existing material.

Normally, materials sourced from catalogs, such as types of steel, wood, or concrete, took on parameters according to catalog data during analysis. However, it is always possible to edit material parameters.

In order to avoid the problem of the possibility of using materials with a given name with different parameters than the catalog, now if you modify the parameters for an existing material in the properties window, a user material is automatically created.

This functionality will avoid errors and enable clear marking of modified materials.

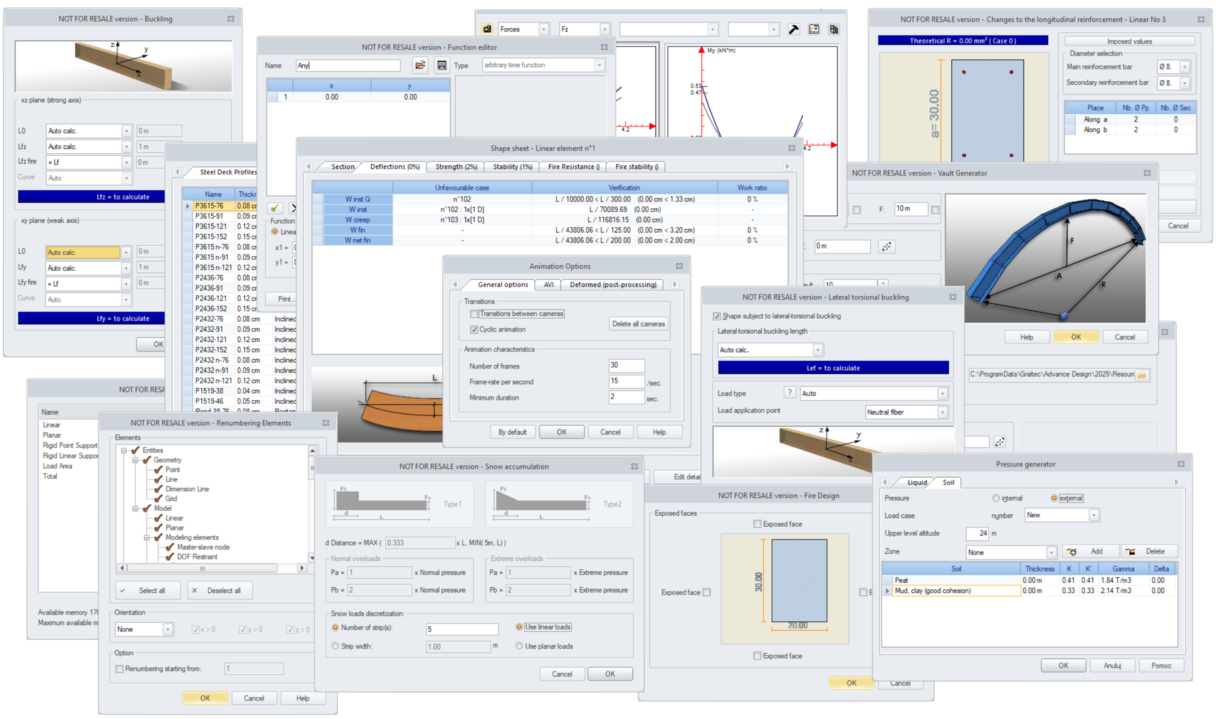


Automatic creation of C20/25_USER_1 material after modification of one of concrete properties

9.8. 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 (42 dialogs) 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.

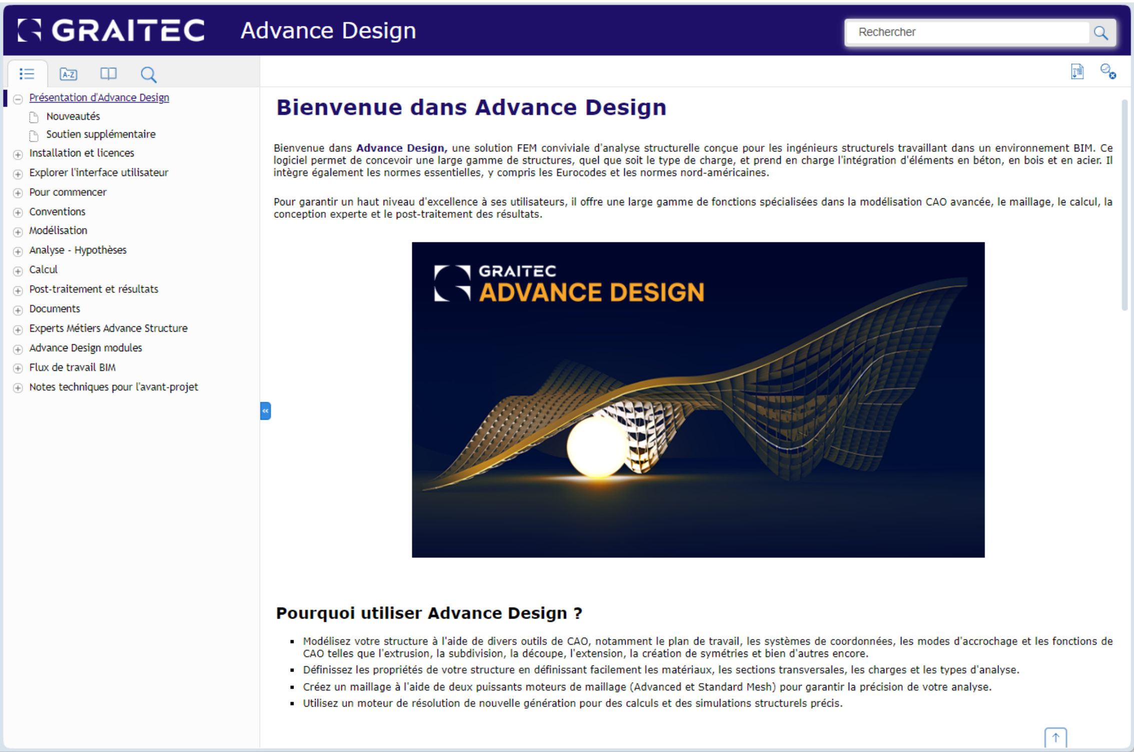


9.9. Online help in local languages

Easy access to local language help content for Advance Design and design modules.

The online help for Graitec software (including Advance Design and Design modules) is now available in local languages. The current online help, which was only available in English, has been automatically translated into selected local languages (French, Polish, Spanish, Portuguese, Romanian, German, Italian, Czech).

Note. Despite the sophistication of automated translation tools, due to the nature of the technical language, the automatically translated texts are gradually being revised to ensure the best possible quality. Therefore, it is possible that at the time of the release of the new version of the software not all content will be verified for selected languages. In this case, we ask for your understanding and patience.



10. RC Design Modules

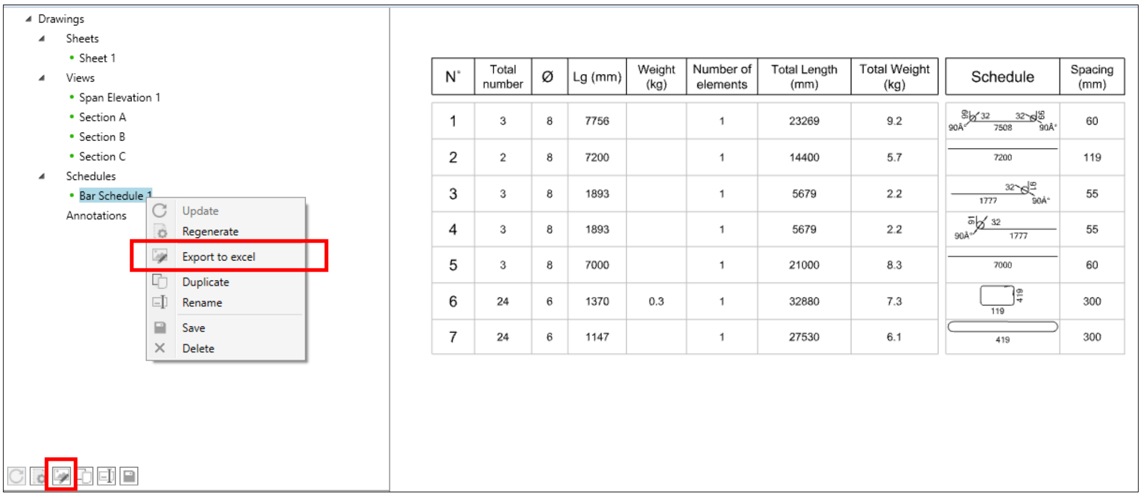
New features and improvements implemented in the latest version of the RC Beam module.

10.1. Export reinforcement schedules to excel files

Easily transfer detailed reinforcement information from bar schedules directly to the Excel sheet.

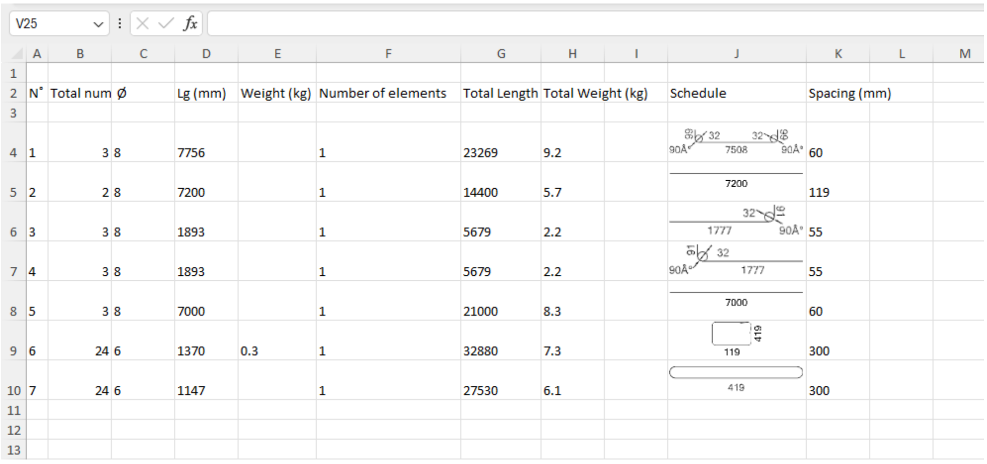
Since version 2025 of RC modules of Advance Design, it is possible to easily export the contents of reinforcement bar schedules generated on drawings directly to an Excel spreadsheet. This allows, for example, easy use of the contents of the bar schedules to prepare various types of summary tables.

For this purpose, there is a new Export to excel option available in the content tree of the drawing after selecting the selected bar schedule, visible on the icon bar as well as in the context menu.



N°	Total number	Ø	Lg (mm)	Weight (kg)	Number of elements	Total Length (mm)	Total Weight (kg)	Schedule	Spacing (mm)
1	3	8	7756		1	23269	9.2		60
2	2	8	7200		1	14400	5.7		119
3	3	8	1893		1	5679	2.2		55
4	3	8	1893		1	5679	2.2		55
5	3	8	7000		1	21000	8.3		60
6	24	6	1370	0.3	1	32880	7.3		300
7	24	6	1147		1	27530	6.1		300

The contents of the schedules, including numerical values, column descriptions and optional bar diagrams, are then generated directly to an xlsx file of an Excel spreadsheet.



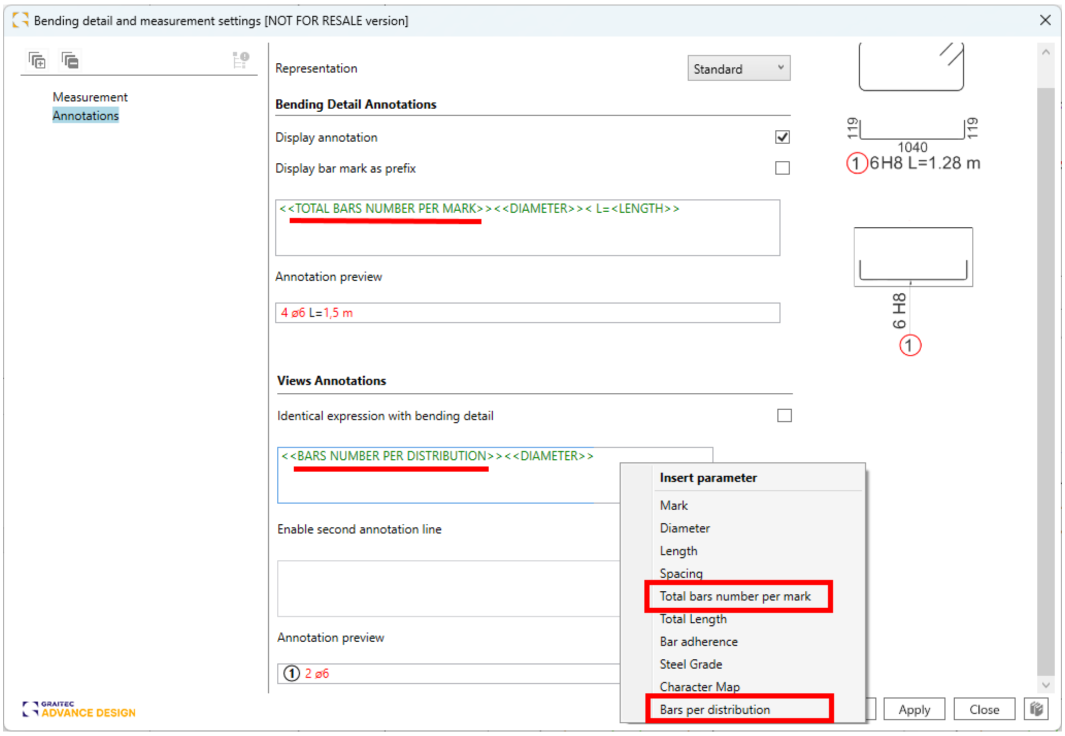
N°	Total num	Ø	Lg (mm)	Weight (kg)	Number of elements	Total Length	Total Weight (kg)	Schedule	Spacing (mm)
1	3	8	7756		1	23269	9.2		60
2	2	8	7200		1	14400	5.7		119
3	3	8	1893		1	5679	2.2		55
4	3	8	1893		1	5679	2.2		55
5	3	8	7000		1	21000	8.3		60
6	24	6	1370	0.3	1	32880	7.3		300
7	24	6	1147		1	27530	6.1		300

10.2. New parameter for displaying the number of bars per distribution

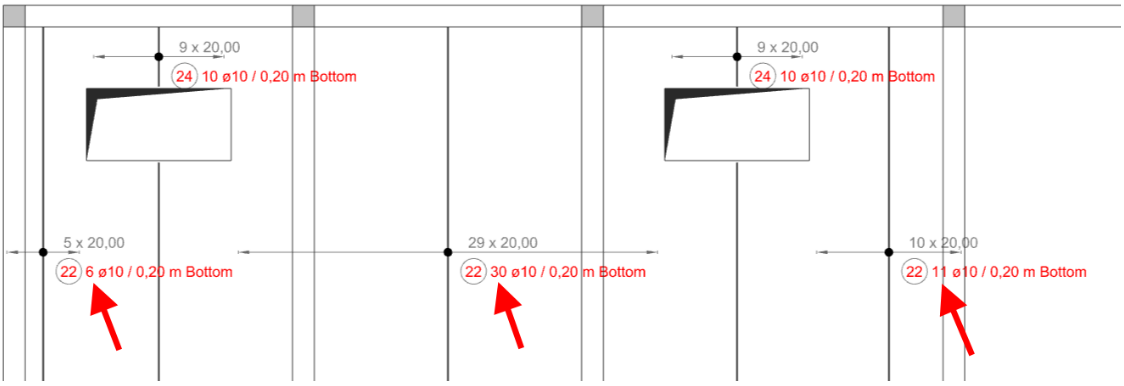
The ability to describe bars on drawings of RC elements by the total number of bars of a given bar mark or the number of bars occurring in a given distribution.

One of the parameters used when describing reinforcement on drawings is the number of bars for a given bar mark number. However, in some cases we want to see the number of all bars with the same bar mark number (for example when describing bar bending detail), and in some situations we want to see the number of bars with a given bar mark number for a given occurrence (for example when describing bar distribution). As an example, let us have a plate in which a given bar with the same bar mark number can occur in several places / several distributions. Thus, for each bar distribution, we want to see in the bar description the number of bars only from this given distribution.

Therefore, to make it easier to choose how to describe bars since the latest version of all RC modules , in the drawing settings are available two parameters for determining the number of bars: **Total bars number per mark** and **Bars per distribution**.



Selection of bar numbering types for drawing annotation



Bars per distribution in the annotation of different distributions of the same bar mark

10.3. RC Beam – Set of improvements to facilitate the daily work

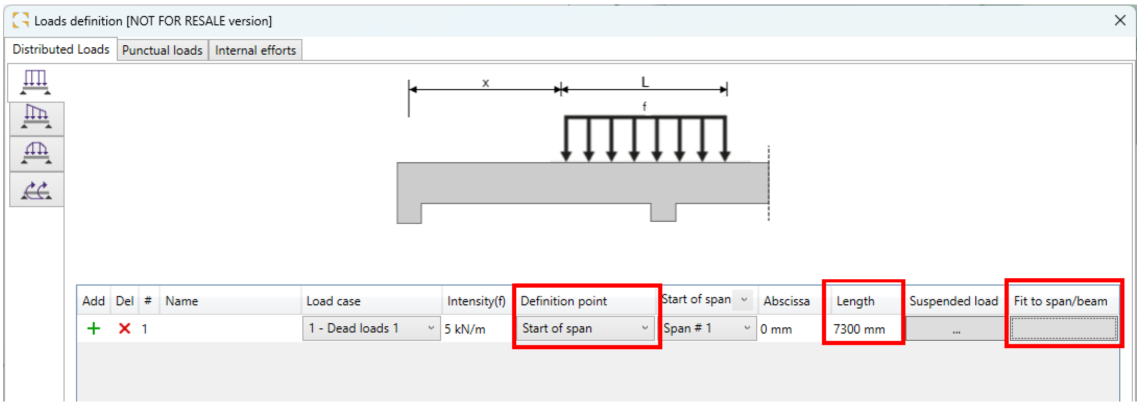
Set of small improvements to all RC modules designed to work more efficiently with the module.

Easier definition of distributed load

To make it easier and faster to define distributed loads, three minor changes have been made to the Loads/Distributed loads window:

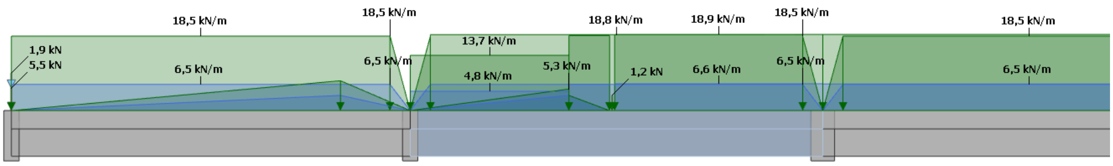
- The default *Definition point* setting has been changed to *Start of span*. This increases the ease of adjusting loads especially the ability to copy loads when cloning spans.

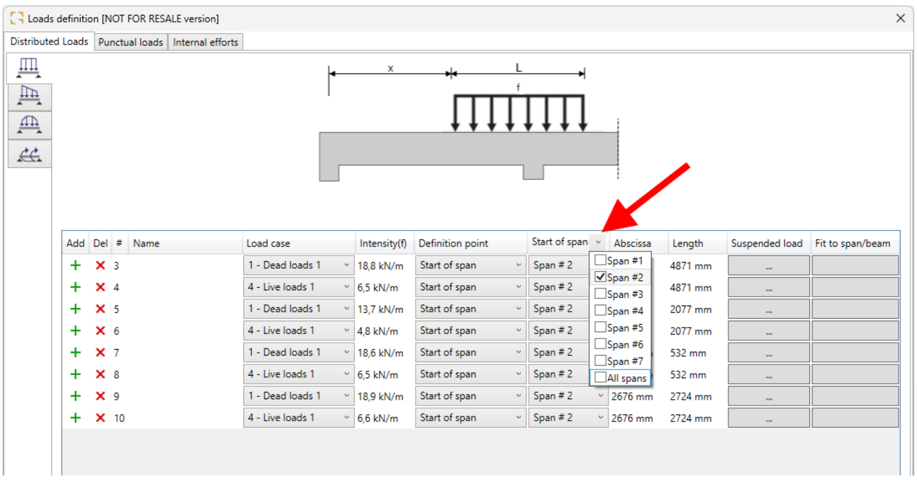
- Added automatic completion of load length to span/beam length. This value can be edited if the load on a shorter span.
- The button in the last column now functions to adjust the length of the load action to the length of the span or beam, depending on the Definition point setting.



Filtering the load table to the selected span

To make it easier to view and edit loads for multi-span beams, there is now a quite easy way to filter the table contents in the Loads definition window. To do so, expand the list of available spans in the Start of span column heading and select the desired span.



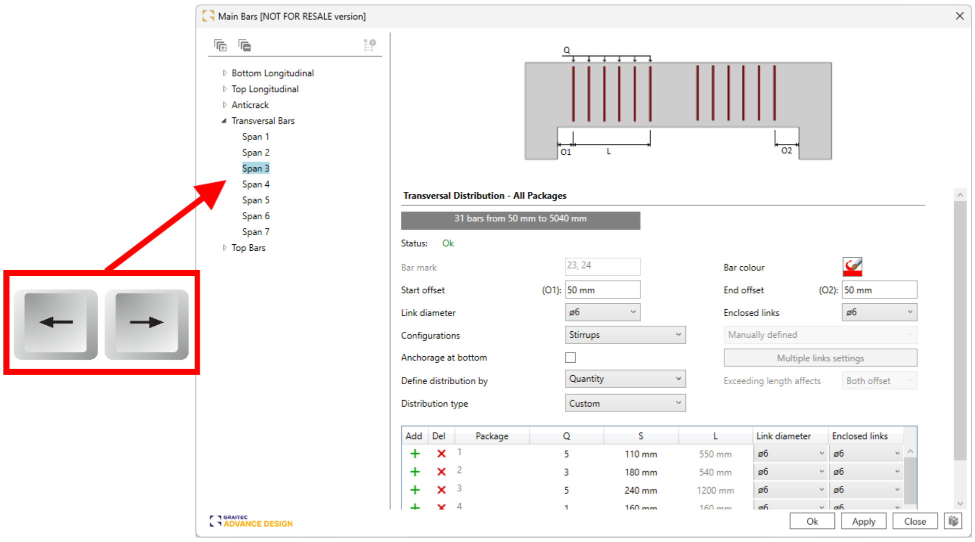


Moving between beam spans /walls in a group using the Left and Right Arrow keys

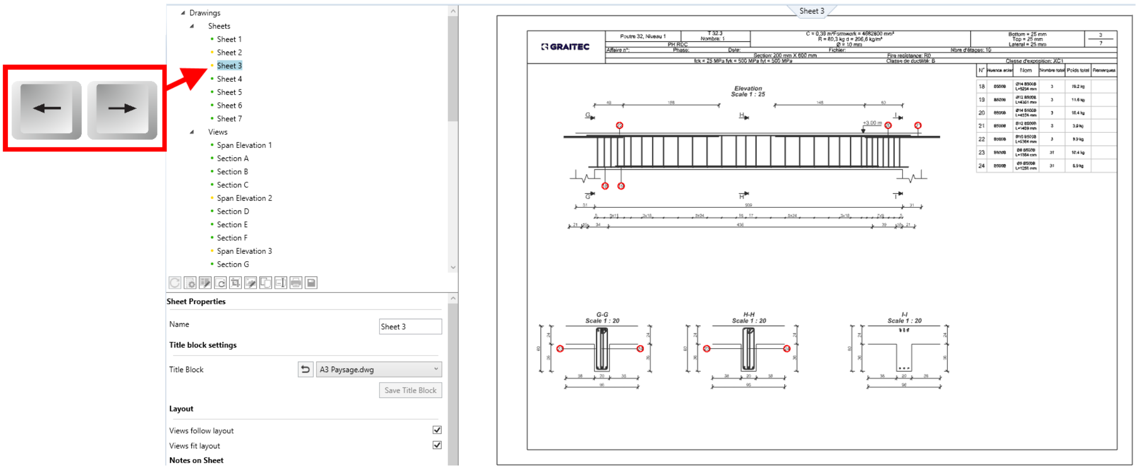
In order to make it easier to check and edit data in the dialog windows in which the data is presented separately for each span / wall in the group, in RC Beam and RC Wall modules you can navigate between the spans / walls by using the left and right arrow keys on the keyboard.

For Beam, this functionality is available in all dialogs which contain a tree structure with navigation between spans: Geometry, Reinforcement Assumptions - Special Features, Main , Bottom on Supports, Opening, Additional.

For Wall, this functionality is available in all dialogs which contain a tree structure with navigation between walls: Geometry, Design Assumptions - Additional Stiffeners + Interface Verification +Buckling Assumptions + Concrete Covers , Main , Opening, Linkage.

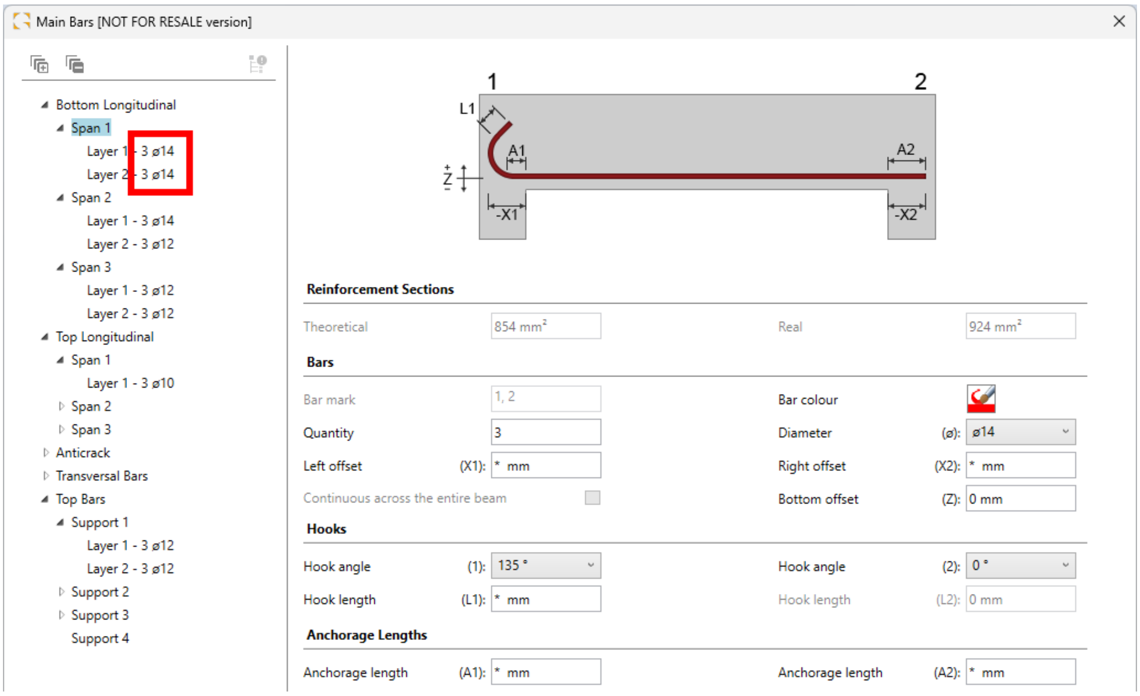


In addition, the same mechanism has been implemented for viewing the generated drawing sheets. This is useful for easily switching between drawings generated for individual spans of a multi-span beam.



Quick info on reinforcement in the tree

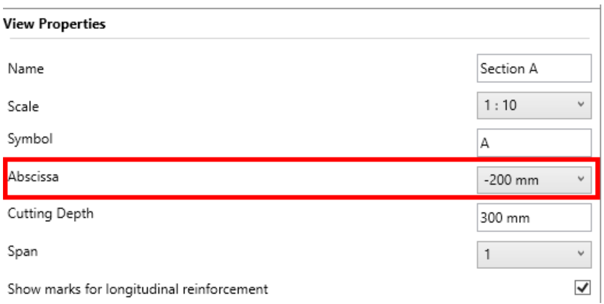
To make it easier to check the number and diameter of bars of the accepted reinforcement directly from the reinforcement editing windows, the tree with the list of reinforcement shows information about the number and diameter of bars for a given layer.



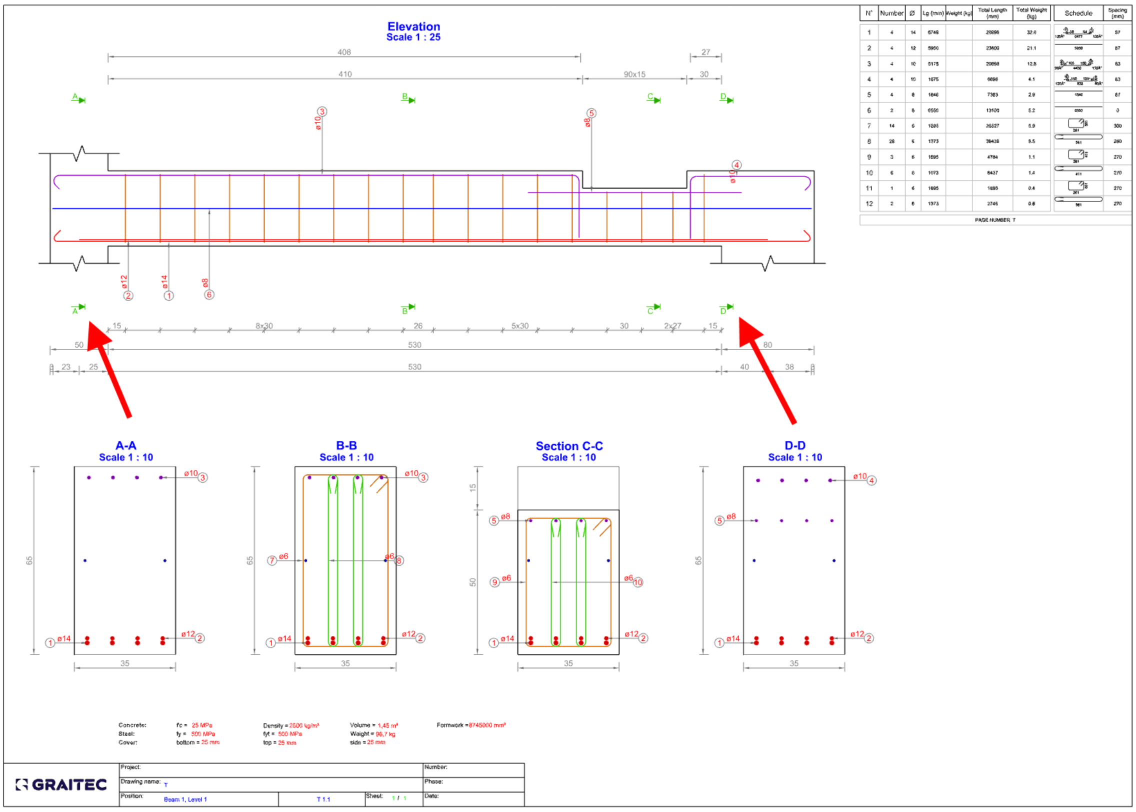
10.4. RC Beam – Drawings with cross sections on supports

Possibility for creating sections not only along the clear span, but also on supports.

When defining cross-sections on beam drawings, until now it was possible to generate them along the span length, i.e., between the edges of the supports. Since the latest version, cross-sections can also be generated along the length of the supports. For this purpose, it is now possible to set abscissa values smaller or larger than the span length range in the section view properties. For the first support, negative values must be entered for this purpose.



Note that in order to explicitly be able to assign a cross-section to a given span of multi-span beams, for a given span we can define cross-sections up to half of the support width.

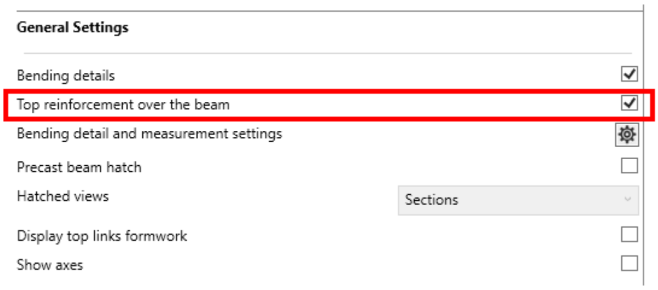


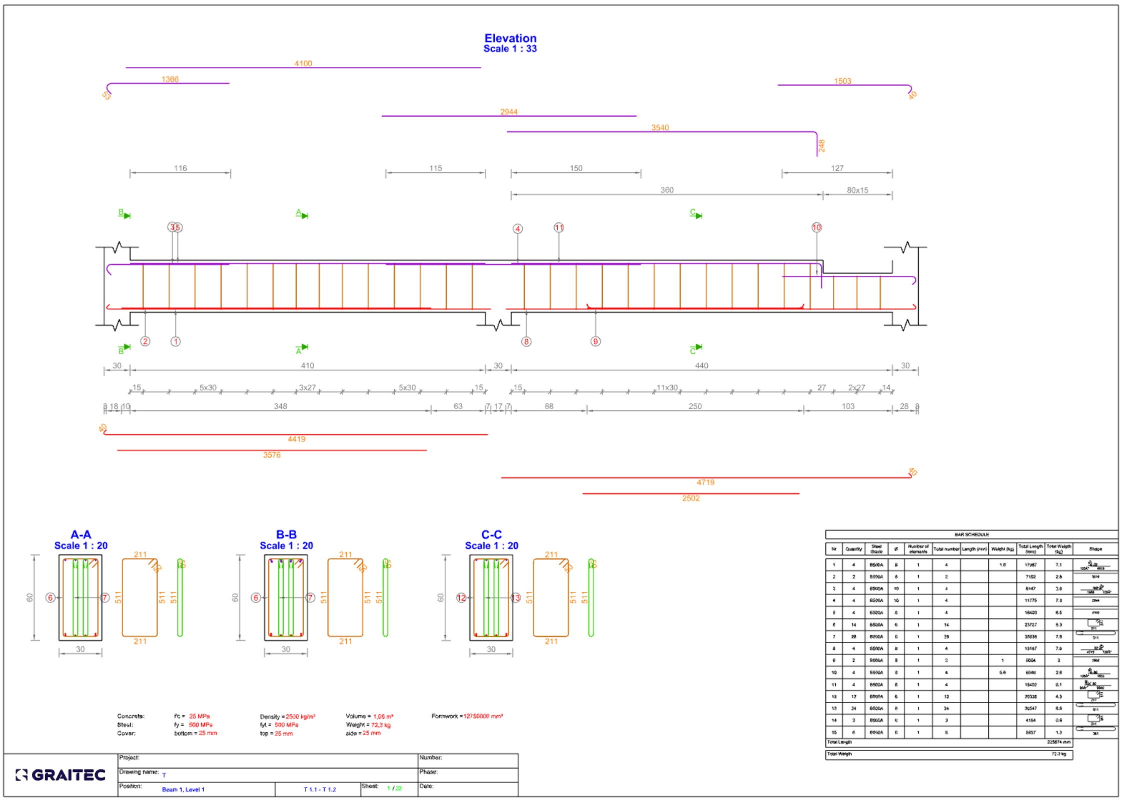
10.5. RC Beam - Displaying bending details of top bars over a beam

The possibility for generating on drawings bending details for top reinforcement of a beam above the beam elevation.

When generating drawings for reinforced concrete beams, we have the option to generate bending details of rebars directly on the drawing. In the case of the beam elevation view, so far the bar diagrams. generated and described were always below the beam.

In the latest version, a new option is available in Drawings properties, which allows splitting the bending detail schemes on the elevation view into two groups: for the bottom bars they are generated under the beam rotor, while for the top bars they are generated above the beam.



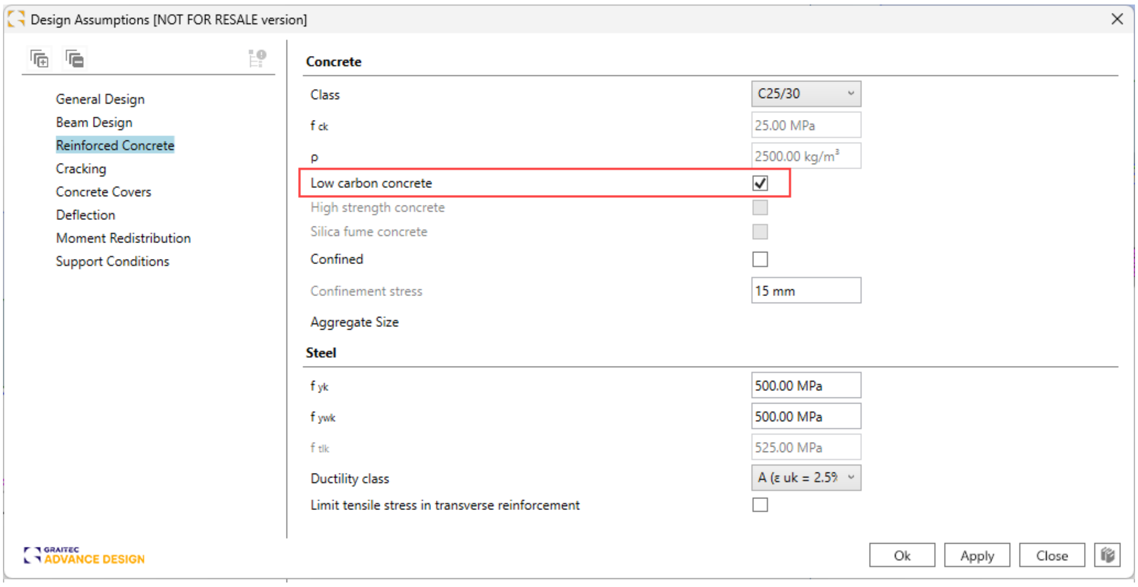


10.6. RC Beam - Low carbon concretes (Eurocode)

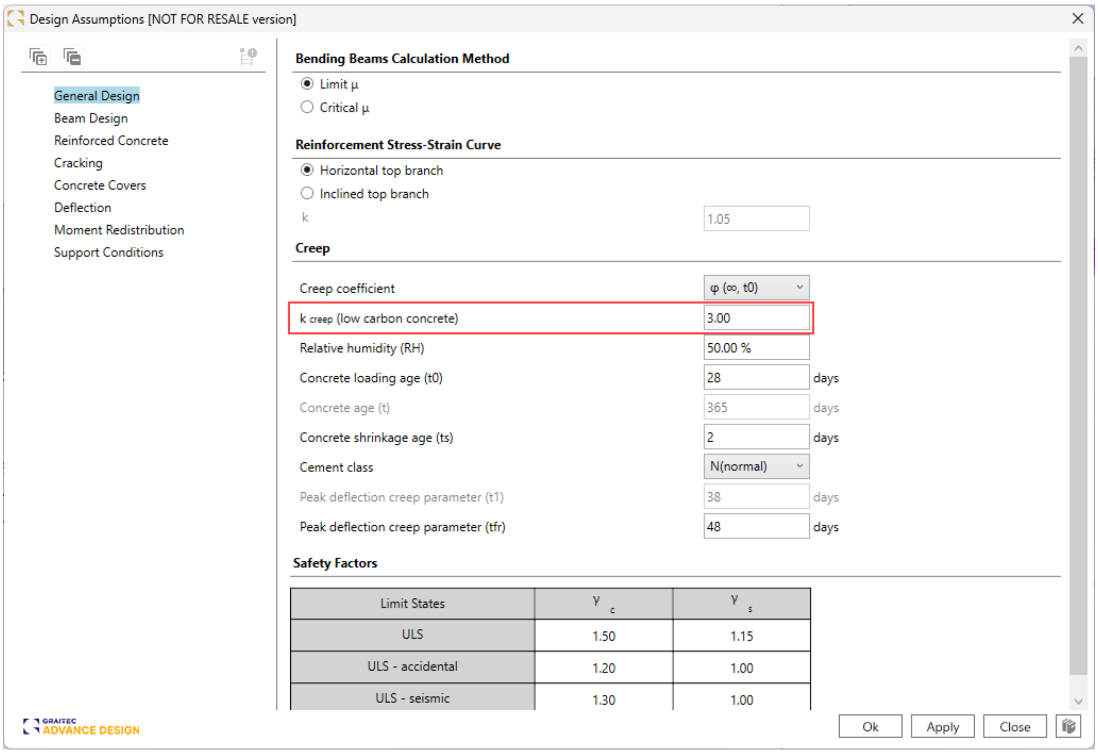
Ability to include low carbon concrete in calculations allowing to reduce carbon footprint by using environmentally friendly materials.

Advance Design RC Beam 2025 now enables definition of low-carbon concretes. Low carbon concrete is concrete produced with a lower carbon footprint than traditional concrete.

For this a new checkbox has been introduced in the Reinforced concrete assumptions:



Since low-carbon concrete has different creep characteristics, the creep factor is usually used to account for the differences. That is why enabling this option gives access to the k_{creep} coefficient in the General Design section:



This k_{creep} coefficient amplifies the creep coefficient:

$$\varphi_{(t,t_0)} \rightarrow k_{creep} \cdot \varphi_{(t,t_0)}$$

This will have an impact on deformations by significantly reducing the effective modulus of elasticity for concrete:

$$E_{c,eff} = \frac{E_{cm}}{1 + k_{creep} \cdot \varphi_{(t,t_0)}}$$

In the report, the Creep Coefficient chapter has been updated to show the influence of the k_{creep} factor:

1 Creep coefficient

The creep coefficient calculation is done according to Annex B of EN1992-1-1.

Relative Humidity	$RH = 50.000\%$
Time at initial loading (in days)	$t_0 = 28$
Coefficient to describe the development of creep with time after loading	$t = \infty \rightarrow \beta_c(t,t_0) = 1.0$
Span 1	
Notional size of the member in mm (B.6)	$h_0 = \frac{2Ac}{u} = \frac{2\times1500.00\ cm^2}{1700\ mm} = 176.471\ mm$
Humidity influence (B.3a)	$\varphi_{RH} = 1 + \frac{1-RH}{0.1\sqrt[3]{h_0}} = 1 + \frac{1-50.00\ \%}{0.1\times\sqrt[3]{176.471}} = 1.891$
Influence of concrete resistance (B.4)	$\beta(f_{cm}) = \frac{16.8}{\sqrt{f_{cm}}} = \frac{16.8}{\sqrt{33.000}} = 2.925$
Age of loading t_0, considering the effect of the cement type (B.9), in days	$t_{0,c} = t_0 \cdot \left[\frac{9}{2+(t_0)^{1.2}}+1\right]^{\alpha} = 28.000 \times \left[\frac{9}{2+28.000^{1.2}}+1\right]^{0.000} \geq 0.5$ $t_{0,c} = 28.000$
Influence of concrete maturity (B.5)	$\beta(t_0) = \frac{1}{0.1+(t_{0,c})^{0.2}} = \frac{1}{0.1+(28.000)^{0.2}} = 0.488$
Long term creep coefficient (B.2)	$\varphi_0 = \varphi_{RH} \cdot \beta(f_{cm}) \cdot \beta(t_0) = 1.891\times2.925\times0.488 = 2.702$
Low carbon concrete coefficient	$k = 3.000$
Creep coefficient (B.1)	$\varphi(t,t_0) = \varphi_0 \cdot \beta_c(t,t_0) \cdot k = 2.702\times1.000\times3.000 = 8.105$

On the example below, one can see the impact of k_{creep} = 3 on the long-term modulus of a low-carbon C25/30 concrete, leading to:

$$E_{c,eff} = \frac{E_{cm}}{1 + k_{creep} \cdot \varphi_{(t,t_0)}} = \frac{31476}{1 + 3 \times 2,702} = 3457\,MPa$$

Combination	**110: 1x[1 G]+0.3x[2 Q]**
Maximum deflection abscissa	x = 3500 mm
Bending moment	M = 106.58 kN·m
Cracking moment	M_{cr} = 107.39 kN·m
Distance between support axes	L = 7300 mm
Modular ratio	α_e = 57.86
Mean flexural tensile strength of reinforced concrete	$f_{ctm,fl}$ = 2.56 MPa
Modulus of elasticity of concrete	$E_{c,eff}$ = 3456.79 MPa
Neutral axis position (uncracked)	υ_h = 403 mm

The deflection is then larger for a low carbon concrete than it is for a regular concrete equivalent:

Total deflection				
Span	**f**	f_{max}	**Ratio**	**Status**
	(mm)	**(mm)**	**(%)**	
1	-21 mm	29 mm	**72.98 %**	**Passed**

Low carbon concrete (k_{creep} = 3)

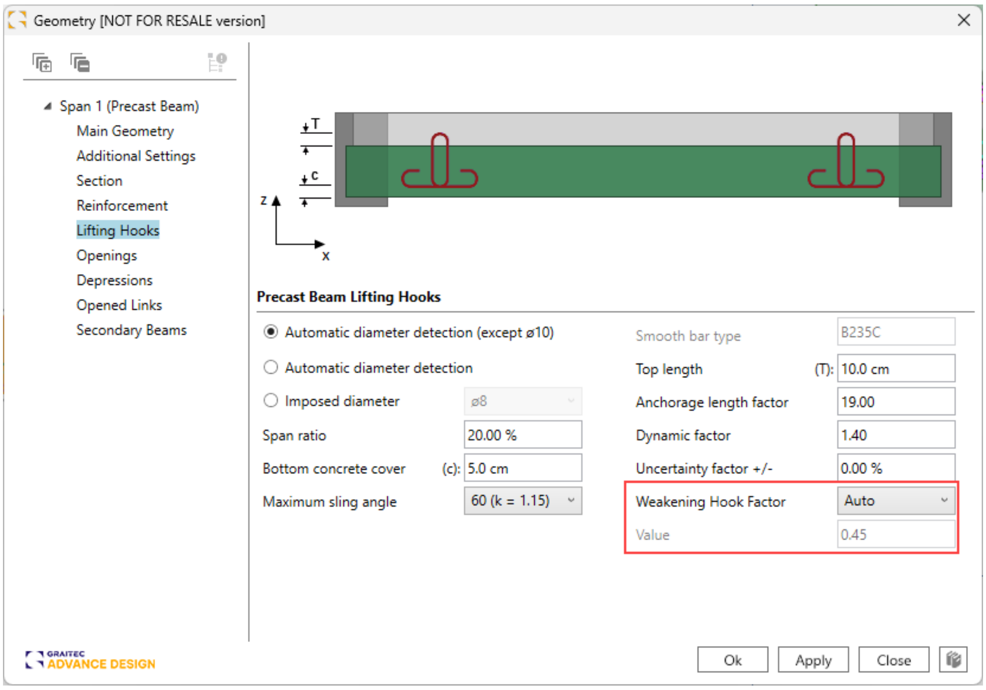
Total deflection				
Span	**f**	f_{max}	**Ratio**	**Status**
	(mm)	**(mm)**	**(%)**	
1	-14 mm	29 mm	**47.30 %**	**Passed**

Regular concrete

10.7. RC Beam - Weakening Hook Factor for precast beams

A possibility for imposing the value of the weakening hook factor in case of precast beams.

When it comes to precast beams, Advance Design RC Beam 2025 now offers more control over the weakening hook factor.

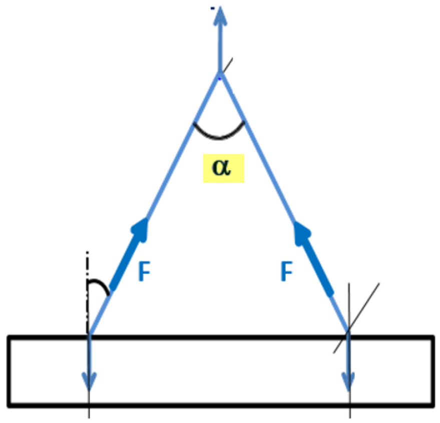


This γ_w safety coefficient is used when computing the required reinforcement diameters:

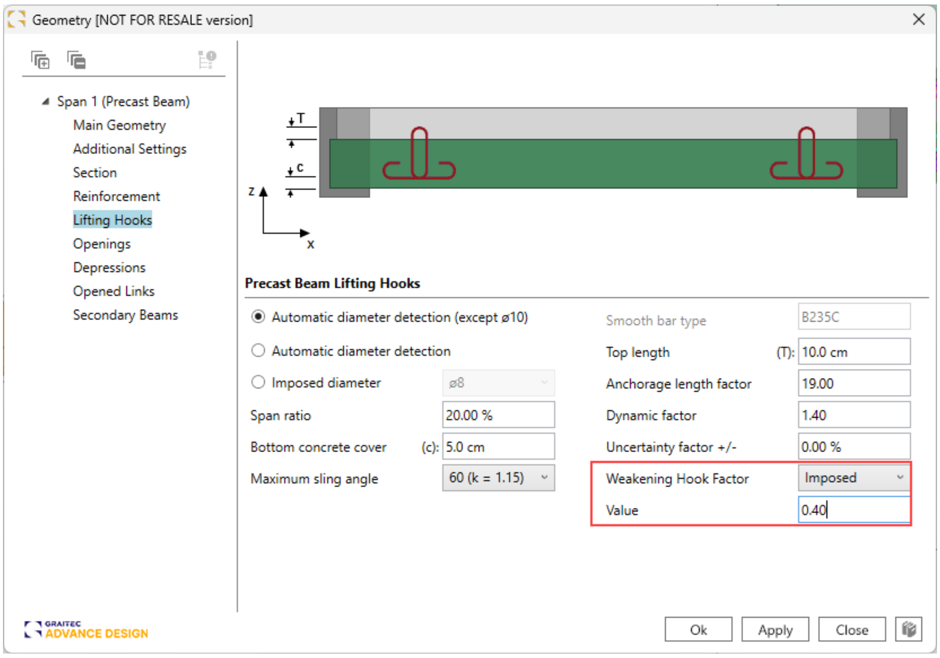
Required hooks diameter	$\frac{F_{Max}}{9.81 \cdot (1-\gamma_w)} = \frac{7.12\text{ kN}}{9.81 \cdot (1-0.45)} = \frac{7.12\text{ kN}}{5.39\text{ kN}} = 1.32 < 2.26 \rightarrow$ $\rightarrow D_{rqrd} = 12\text{ mm}$

The Auto determination remains available, where the γ_w coefficient depends on the α angle at the lifting hook:

$$\gamma_w = \frac{\alpha}{20} \times 15\%$$



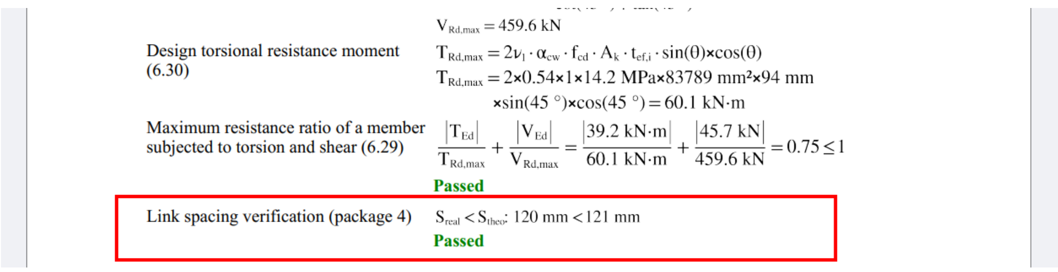
In addition, version 2025 now offers the ability for the users to impose the γ_w value or to ignore it entirely.



10.8. RC Beam – Expansion of torsion reinforcement report chapter

The update of the torsional reinforcement chapter on the report with a link spacing verification.

One of the additional verifications conducted during the torsional reinforcement analysis is the verification of stirrup spacing, which checks if the real spacing is lower than the theoretical one. Now the details of this verification are presented in the detailed report at the end of the section on torsional reinforcement. Verification is available for all supported design standards.



	$V_{Rd,max} = 459.6$ kN
Design torsional resistance moment (6.30)	$T_{Rd,max} = 2\nu_1 \cdot \alpha_{cw} \cdot f_{cd} \cdot A_k \cdot t_{ef,i} \cdot \sin(0)\times\cos(0)$ $T_{Rd,max} = 2\times0.54\times1\times14.2$ MPa$\times83789$ mm²$\times94$ mm $\times\sin(45°)\times\cos(45°) = 60.1$ kN·m
Maximum resistance ratio of a member subjected to torsion and shear (6.29)	$\frac{\lvert T_{Ed}\rvert}{T_{Rd,max}} + \frac{\lvert V_{Ed}\rvert}{V_{Rd,max}} = \frac{\lvert 39.2 \text{ kN·m}\rvert}{60.1 \text{ kN·m}} + \frac{\lvert 45.7 \text{ kN}\rvert}{459.6 \text{ kN}} = 0.75 \leq 1$ Passed
Link spacing verification (package 4)	$S_{real} < S_{theo}$: 120 mm < 121 mm Passed

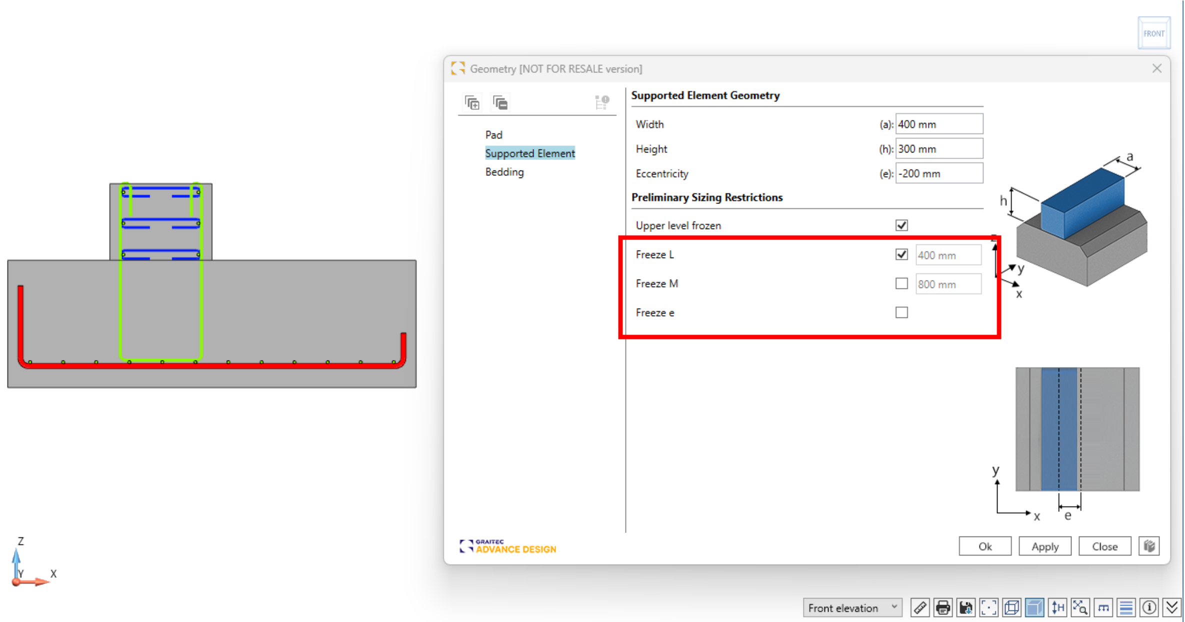
10.9. RC Footing – Improvements to preliminary sizing of continues footing

The ability to perform preliminary sizing for continuous foundations, considering its limitations with respect to the width of the pad in each direction, as well as with the ability to specify a specific eccentricity.

During the analysis of strip footings there are situations in which the designer does not have full freedom when determining the dimensions of the foundation and must use an asymmetric section shape. This is usually caused by restrictions related to, for example, the boundary of the parcel/other foundation. Also, sometimes an asymmetrical shape of the foundation may be more suitable due to the effect of eccentricity.

To facilitate automatic sizing of the foundation in RC Footing module, starting from the latest version we have a set of additional new options to facilitate the introduction of dimension constraints. They allow sizing of the foundation while maintaining an asymmetrical/eccentric shape of the profile. The new options are available in *Supported Element* tab of the *Geometry* window.

[The picture below should be replaced with a new one, where L and M dimensions will be visible on a picture]



There are three new options:

- **Freeze L** - Freezes for preliminary resizing the distance between the edges of the wall and the foundation on the left side.
- **Freeze M** - Freezes for preliminary resizing the distance between the edges of the wall and the foundation on the right side
- **Freeze e** - Freezes for preliminary resizing the eccentricity between the wall and pad axes.

Note that only one of 3 new options can be activated at the same time.

10.10. RC Column - Improvements related to the fire verification (Eurocode)

Set of improvements related to the fire verifications of RC Columns, including displaying on the Info panel a set of additional results coming from the fire verification, additional check, more detailed reports, as well as providing additional warning messages.

To better control the scope and parameters of fire verification according to Eurocode, a number of improvements related to verification and presentation of results have been introduced in the latest version of RC Column.

Extension of the information panel with the result of fire verification

In order to make the work easier and speed up the verification of the performed calculations, the info panel visible immediately after the column analysis now provides summarized results from the fire analysis.

Buckling length	2170 mm	2170 mm
Slenderness	25,06	25,06
	Second order effects are ignored	

Reinforcement	Real	Theoretical	Ratio	Combination	Amin	Amax
Longitudinal top	452 mm²	254 mm²	56,25%	104: 1.35x[1 G]+1.5x[2 Q]	180 mm²	3600 mm²
Longitudinal bottom	452 mm²	254 mm²	56,25%	104: 1.35x[1 G]+1.5x[2 Q]	180 mm²	3600 mm²
Transversal along X	259 mm²/m	0 mm²/m	0.0%	-		
Transversal along Y	259 mm²/m	0 mm²/m	0.0%	-		

Fire design	Fire Resistance	Required fire resistance	WR
	238,71	90	37,7%

Verification of minimum dimension/minimum concrete cover with table 5.2 a

The verification of minimum dimension/minimum concrete cover with Table 5.2a, 5.3.2 Method A, EN 1992-1-2 was implemented along with the appropriate warning messages in info-panel and report.

Type	Details	Value	Limit
✖	Fire: column resistance too small (§5.3.2 - Method A from EN1992-1-2)	156,91	240
⚠	Fire: The size of the column is smaller than the minimum value. (EN 1992-1-2, 5.3.2, Table 5.2a)	300 mm	350 mm
⚠	Fire: The concrete cover is smaller than the minimum value. (EN 1992-1-2, 5.3.2, Table 5.2a)	37 mm	61 mm
⚠	Fire: The number of reinforcement bars is smaller than the minimum value. (EN 1992-1-2, 5.3.2, Table 5.2a)	4	8

Accordingly, the content of the report has been expanded to include a new section that presents if the conditions from Table 5.2a (minimum dimension, minimum concrete cover) are met:

Verification of minimum dimension/minimum concrete cover with table 5.2 a

Minimum dimensions verification	$\min(b, h) \geq b_{min}$: 300 mm $\geq$ 250 mm **Passed**
Minimum concrete cover verification	$c \geq c_{min}$: 37 mm $\geq$ 40 mm **Failed**

On the other hand, if the verification from Table 5.2a cannot be used, the conditions that prevent it are presented (only the conditions which are not met are displayed here) .

Verification of minimum dimension/minimum concrete cover with table 5.2 a

The table cannot be used to verify the minimum dimensions/minimum concrete cover because the following conditions are not satisfied:

Effective length condition	$l_{0,fi} \leq 3$ m : 3.15 m $\leq$ 3 m
Eccentricity condition	$e_{0,fi} \leq e_{max}$: 84 mm $\leq$ 30 mm

Additional information in report

In the detailed report, the fire verification results section was expanded to include additional information, including:

- The value of the distance between longitudinal bars axis and column face

Distance between longitudinal bars axis and column face	$a = 37$ mm $b' = 300$ mm

- Information on the condition for the effective length of the column under fire conditions

Column effective length used in fire design	$2 \text{ m} \leq l_{0,fi} \leq 6 \text{ m}$ $l_{0,fi} = 2170$ mm

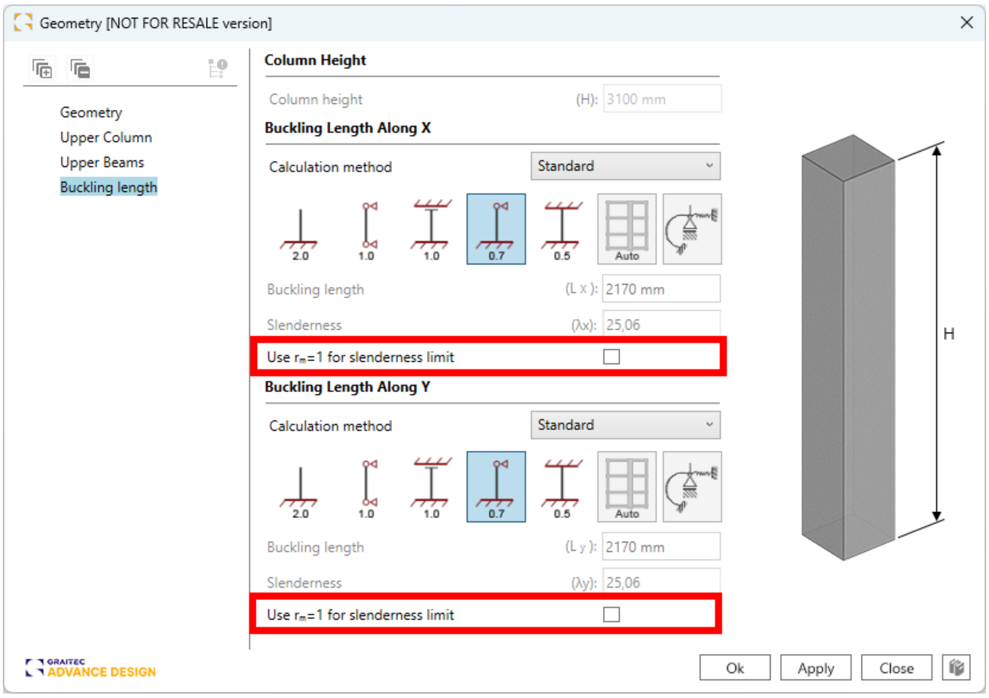
10.11. RC Column – Possibility for imposing the moment ratio for slenderness limit

The ability to impose the moment ratio value used in slenderness limit calculations according to Eurocode.

In the latest version of the RC Column module, in the Buckling Length tab of the geometric data window, an additional option has been introduced to impose the value of moment ratio rm = 1, separately for both buckling directions.

According to EN 1992-1-1, Section 5.8.3.1, the rm value should not be calculated as a ratio of first order end moments but taken as 1 in two cases:

- For braced members in which the first order moments arise only from or due to imperfections or transversal loading
- For unbraces members in general.



The new options are not available for the German national annex (whose provisions do not include this condition) and other standards than Eurocode.

10.12. RC Slab - Improved module performance

Increase the comfort of work by significantly increasing the speed of the module, including flat work during editing and generation of reinforcement for models with a large number of finite elements.

In previous versions of RC Slab module, analyzing certain slab models, particularly those with extensive finite elements and numerous combinations, often posed significant operational challenges. Users experienced delays in processing and occasionally suffered long loading times for results, impending overall efficiency. These issues arose from the data storage methods used for the finite elements in the module.

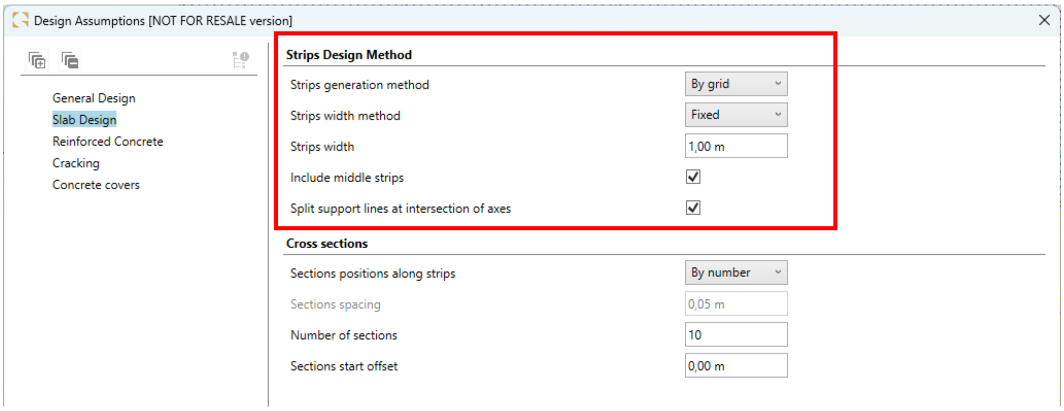
However, in the latest version 2025 of RC Slab module, substantial enhancements have been made to data storage and handling mechanisms. Consequently, the module's performance has undergone a remarkable boost. This improvement is particularly notable when dealing with large-scale models featuring numerous finite elements. Tasks such as data loading, result visualization, and defining reinforcement areas now operate significantly faster - up to several dozen times quicker compared to previous versions.

10.13. RC Slab – Enhancement of the automatic strip generation mechanism

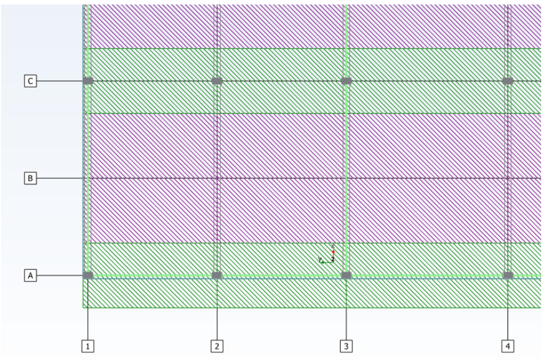
Several updates addressing strip generation for RC slabs, implementing automatic strip definition in accordance with the standards laid out in the American ACI code.

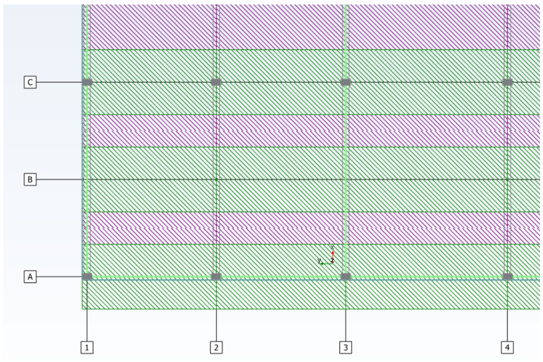
In the latest version of the RC Slab module, a number of improvements have been made to the automatic generation of strips on slabs. The improvements include the addition of new parameters as well as new mechanisms that consider the length and width of spans to determine the width of strips, in accordance with the rules per the American ACI standard..

Modification of the settings can be done using the new parameters on the *Slab Design* tab of the *Design Assumptions* window.



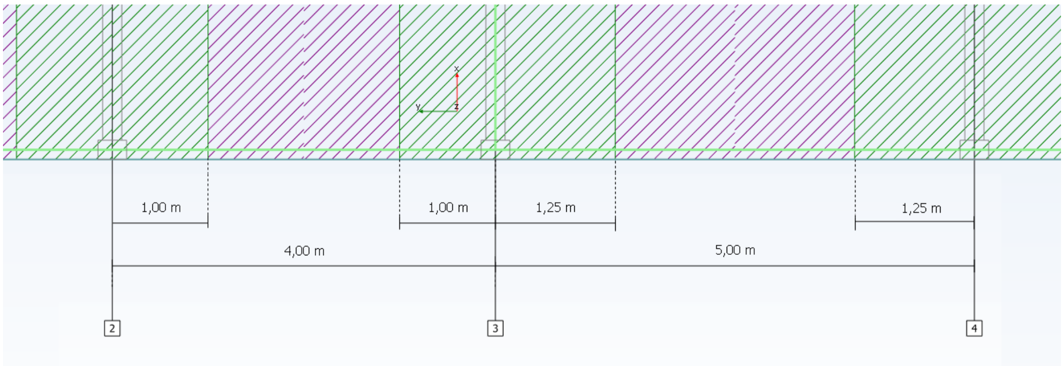
The *Strips generation method* option allows you to decide whether, during automatic strips generation, strips are to be defined along structural axes or along automatically detected lines created by elements supporting the slab. To increase control over the generation based on the axes , in the window for managing the axes of the structure, we can temporarily disable selected axes so that they are not considered during the definition of strips.



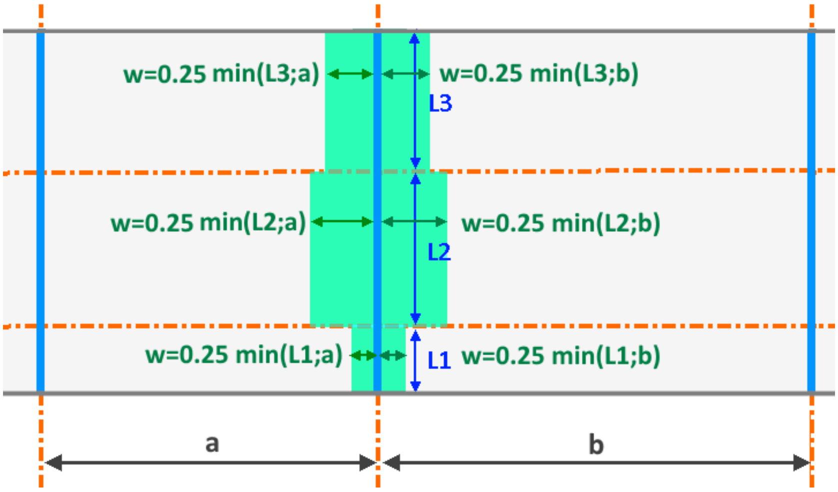


The *Strips width method* parameter allows you to choose one of 3 methods of strip generation:

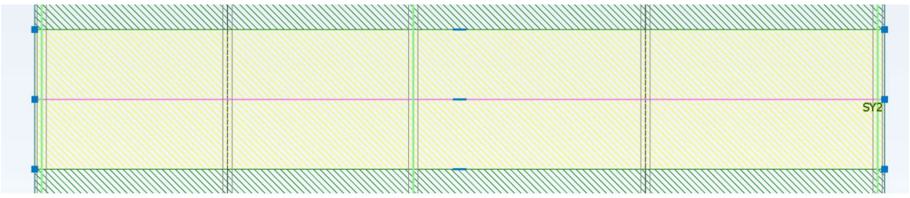
- **Fixed** - all strips have the same imposed width, which by default is used for the support strips.
- **Auto (one-way spacing)** – the strip widths are defined based of grid spacing for the selected direction. The left and right widths of Support strips are calculated separately with using spacing to the next axis on the left and right, as half of distance to the next axis (when middle strips are not created) or as a fourth of the distance to the next axis (when middle strips are created).

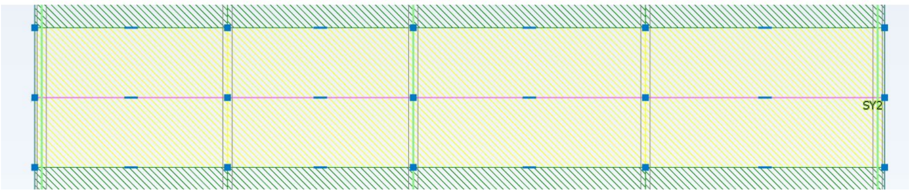


- **Auto (two-ways spacing)** - the strip widths are defined based of grid spacing for both directions. The rules for automatic definition of strips withs are following ACI code provisions. In this method, each of the support strips is divided into segments as long as the spans, and the widths of each segment are calculated separately for the left and right sides. The width of a given strip segment on one side is calculated as a fourth of the smaller of: the distance to the next axis or the length of the segment.



The *Include middle strips* option allows you to decide whether to generate only support strips or also middle strips. Meanwhile, the *Split support lines at intersection of axes* option allows you to decide whether the strip is generated as a single object or is divided into spans, which affects the possibility of further manual modifications to their geometry.





11. Masonry Wall

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

11.1. Confined masonry

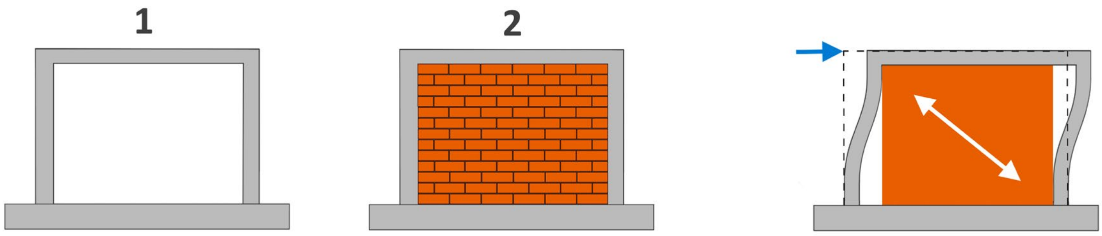
Possibility for performing verifications of masonry walls bound with reinforced concrete columns, and verifications of masonry walls that includes reinforcement.

Advance Design Masonry Wall 2025 now enables definition of reinforced and confined masonry. The implementations are based on the design codes EN 1996-1, CR6-2013 and the proposals regarding confined masonry from the future Eurocode 1996-1.

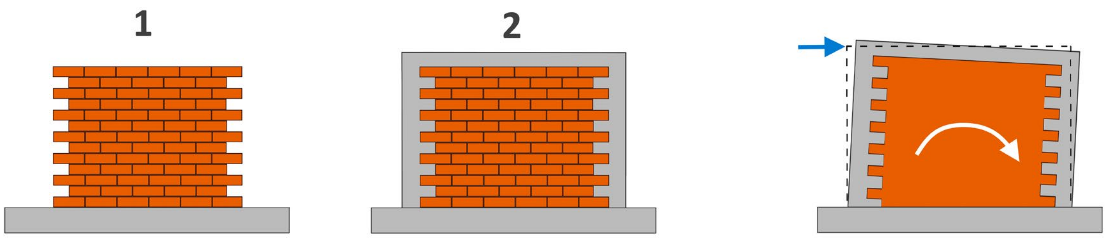
Reinforced masonry walls are structures in which reinforcing bars are placed inside the masonry to increase their load-bearing capacity and to reduce their tendency to crack. They are more stable and can manage greater loads than unreinforced masonry walls.

Confined masonry walls are structures whose deformations have been limited vertically and horizontally by an adjacent reinforced concrete structure or reinforced masonry. This is ensured by the proper bonding of the walls to the frame, usually with the use of reinforcement as well as with the use of a toothed wall edge.

It is worth mentioning that confined masonry is distinct from masonry filling within a reinforced concrete frame. In confined masonry construction, the masonry walls carry the loads (including seismic loads) and the concrete is used to confine the walls, while in concrete frame buildings with infills the concrete frames need to carry the load. In ensuring the interaction of the confined wall with reinforced concrete elements, a different method of erection is important. In frame structures with infill walls, the reinforced concrete frame is built first, followed by the infill. In the case of confined walls, the order is reversed - the walls are built first, and the columns and beams are poured in afterwards to enclose (confine) the wall.



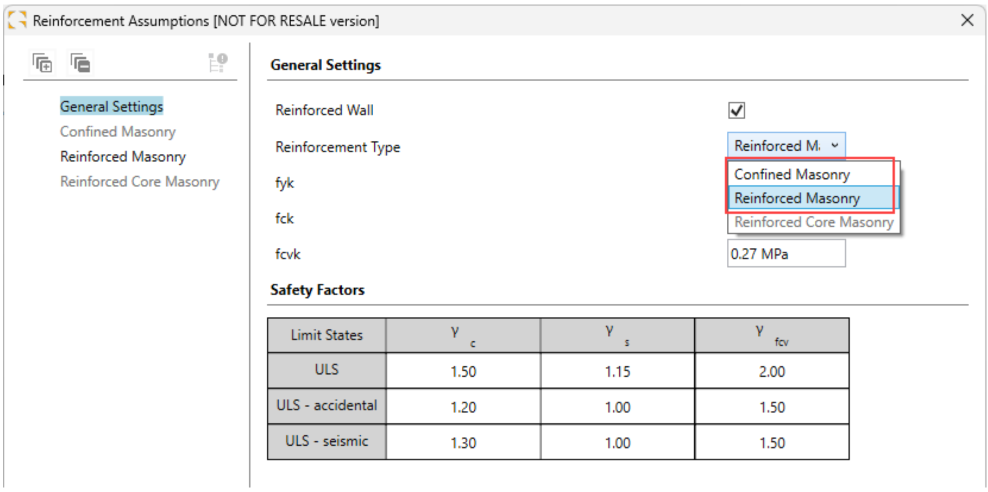
Steps of construction and work of reinforced concrete frame filled with masonry wall



Steps of construction and work of confined masonry

General settings

Activation of reinforcement verification and type selection is available in the new Reinforcement Assumptions window:

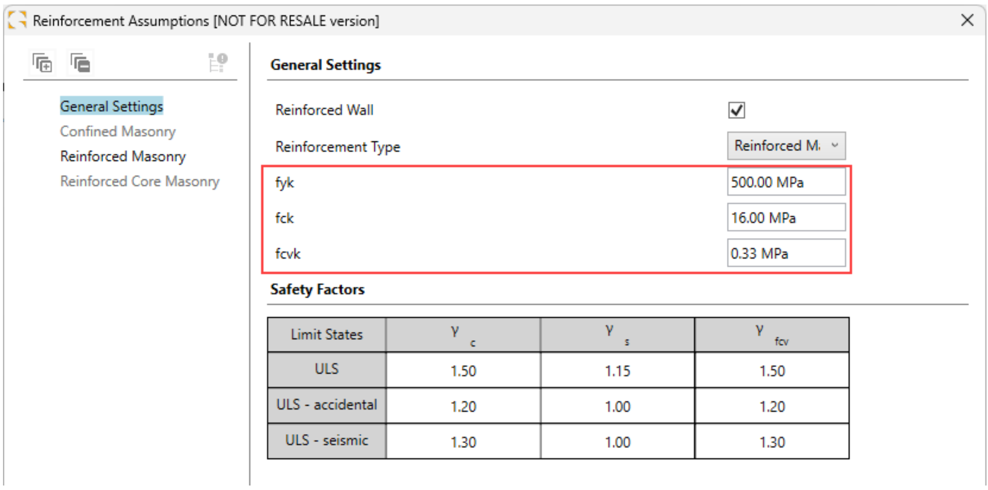


The availability of a particular type depends on the design standard chosen and the type of section:

- The *Confined Masonry* is available for single-leaf walls and the EC6 or CR6-2013 code.
- The *Reinforced Masonry* is available for single-leaf walls and the EC6 code.
- The *Reinforced Core Masonry* is available for grouted cavity walls and the CR6-2013 code.

In the reinforcement assumptions dialog, we can also define:

- f_{yk}: Characteristic yield strength of reinforcement
- f_{ck}: Characteristic compressive strength of concrete infill
- f_{cvk}: Characteristic shear strength of concrete infill

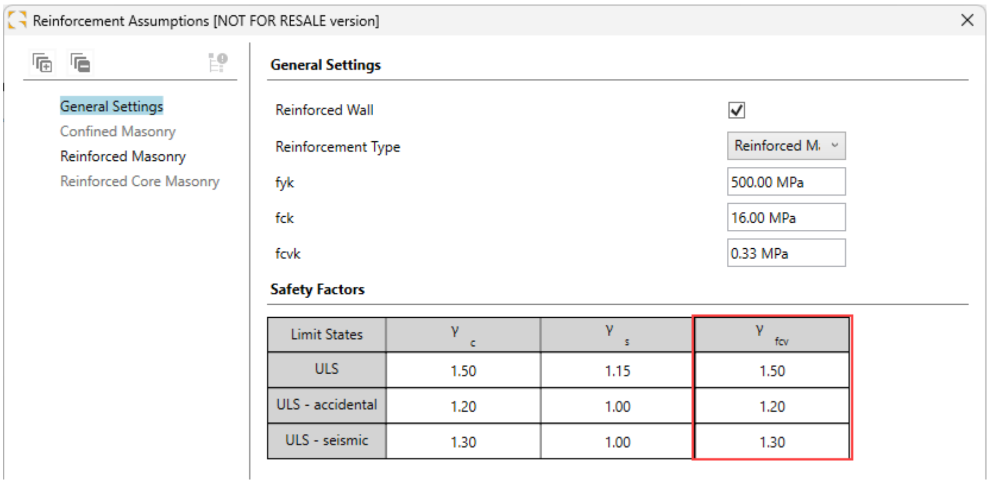


Typical values of f_{ck} and f_{cvk} for concrete infill may be taken from Table 3.2 from EN 1996-1-1:

Table 3.2 — Characteristic strengths of concrete infill

Strength class of concrete	C12/15	C16/20	C20/25	C25/30, or stronger
f_{ck} (N/mm²)	12	16	20	25
f_{cvk} (N/mm²)	0,27	0,33	0,39	0,45

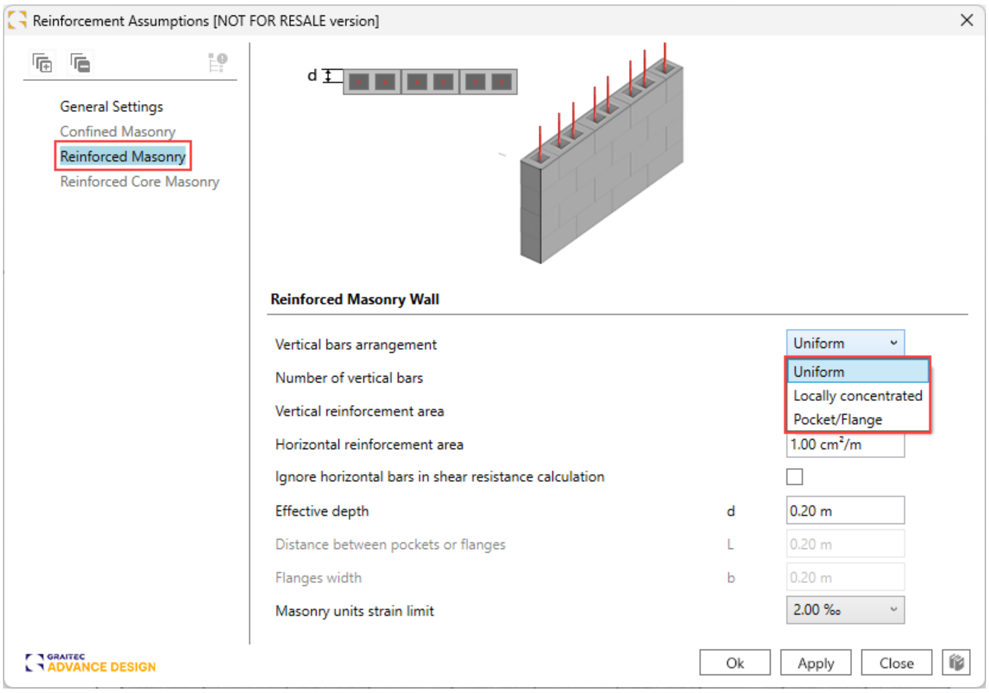
The safety factors section sees the addition of a γ_{fcv} column related to the partial factor for the concrete infill to be considered for the ULS, accidental and seismic combinations:

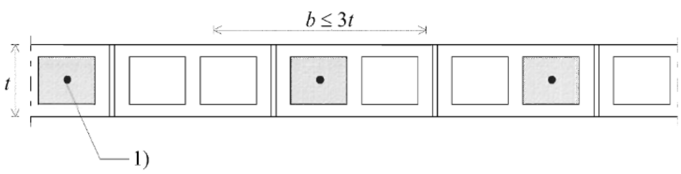


Reinforced masonry

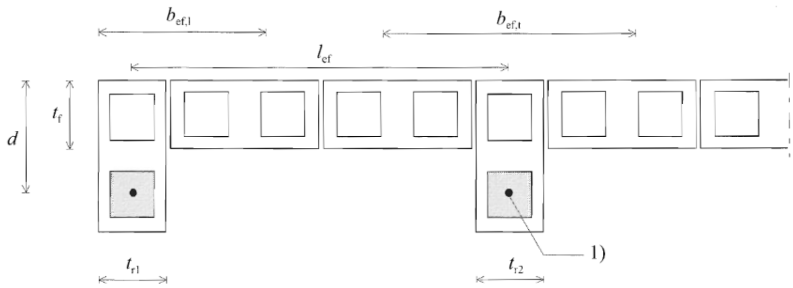
When set on *Reinforced masonry*, vertical reinforcement may be:

- Uniformly distributed along the wall
- Locally concentrated (as per Fig. 6.5 from EN 1996-1-1)
- Arranged in flanges (as per Fig. 6.6 from EN 1996-1-1)



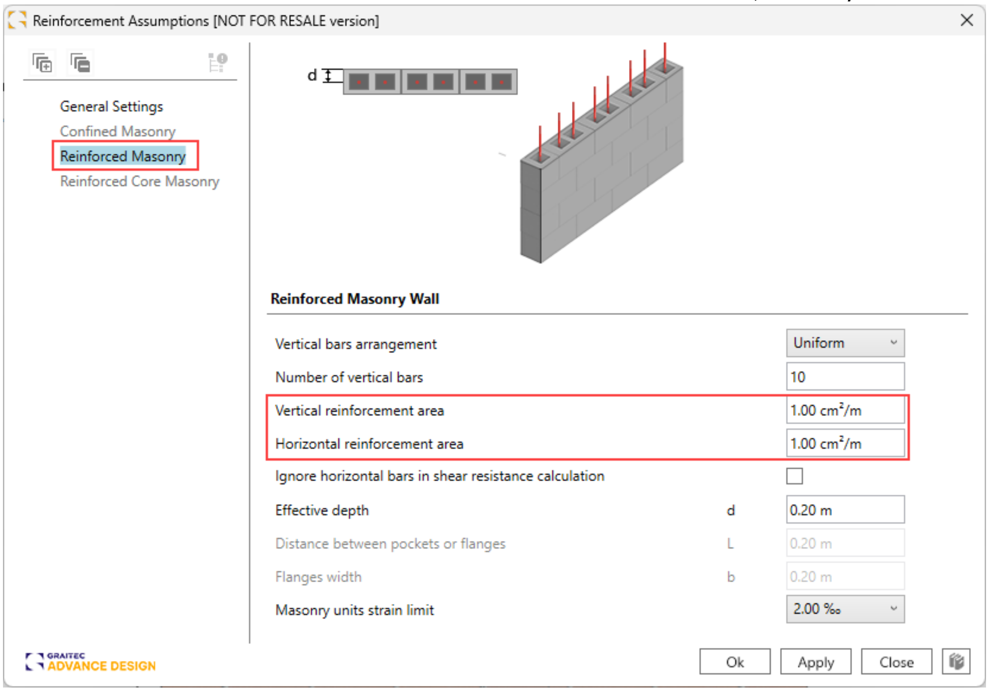


Wall section with locally concentrated reinforcement



Wall section treated as a flanged reinforced member

The users can define the areas of vertical and horizontal reinforcement, in cm²/m.

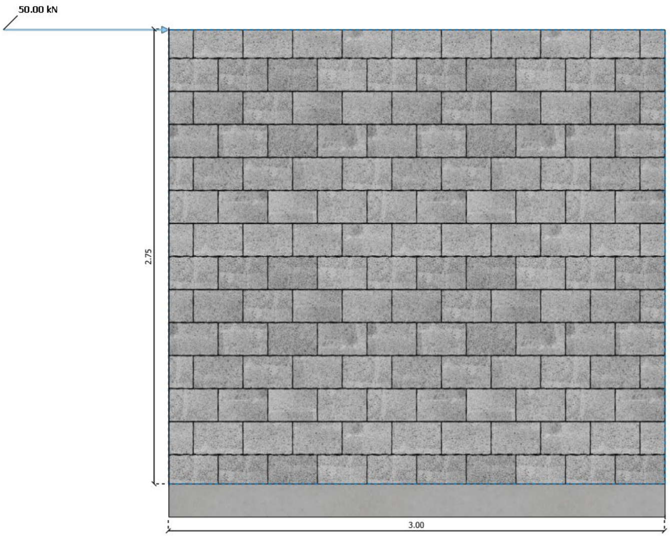


The reinforcement defined by the users will have a beneficial effect on various verifications, including a verification of wall subjected to in-plane shear loading and subjected to bending.

For example, for a masonry wall subjected to in-plane shear loading:

- Vertical reinforcement will prevent any uplift from the wall, increasing the L_c compressed length
- Horizontal reinforcement will contribute to the V_{Rd} shear resistance

On the example below, one can note the resistance to shear force can be enhanced by considering a minimum area of reinforcement (0,05% of the cross-sectional area of the wall).



Shear Loads (in-plane)	Combination	Critical Section	VEd	VRd	WR
VEd < VRd	103: 1x[1 G]+1.5x[2 Q]	Middle section	75.00 kN	32.71 kN	229.26%

Unreinforced masonry

Shear Loads (in-plane)	Combination	Critical Section	VEd	VRd	WR
VEd < VRd	103: 1x[1 G]+1.5x[2 Q]	Middle section	75.00 kN	140.32 kN	53.45%

Reinforced masonry

This verification is properly documented in the calculation report:

5 Verification of wall subjected to shear loading (in-plane)

The verification of reinforced masonry walls subjected to shear loading is done according to (6.7.2 (2)) from EN 1996-1-1.

Verification of wall subjected to shear loading (in-plane)						
Leaf	Comb.	Critical Section	V_{Ed}	V_{Rd}	WR	Status
			(kN)			
-	103	Top	75.00	134.88	55.60 %	Passed

Critical Section **Bottom of the wall**

Combination **103: 1x[1 G]+1.5x[2 Q]**

Verification $V_{Ed} < V_{Rd}$

Design value of the shear load applied to the masonry wall $V_{Ed} = 75.00$ kN

Design value of the shear resistance of the masonry wall $V_{Rd} = f_{vd} \cdot t \cdot l_c = 0.09\ \text{MPa} \times 200\ \text{mm} \times 1500\ \text{mm} = 0.00\ \text{kN}$

Shear resistance of the masonry wall limit $V_{Rd,lim} = V_{Rd} - 2 \cdot A_w = 0.00\ \text{kN} - 2 \times 0.60\ \text{m}^2 = 0.00\ \text{kN}$

(EN 1996-1-1 (eq. 6.38))

(EN 1996-1-1 (eq. 6.37))

$$V_{Rd2} = \min\begin{cases} 0.9 \cdot h \cdot A_h \cdot f_{yd} \\ V_{Rd,lim} \end{cases}$$

$$V_{Rd2} = \min\begin{cases} 0.9 \times 2750\ \text{mm} \times 1.00\ \text{cm}^2 \times 434.78\ \text{MPa} \\ 107.61\ \text{kN} \end{cases} = 107.61\ \text{kN}$$

(EN 1996-1-1 (eq. 6.35)) $V_{Rd1} = f_{vd} \cdot t \cdot l_c = 0.09\ \text{MPa} \times 200\ \text{mm} \times 1500\ \text{mm} = 27.27\ \text{kN}$

Design value of the shear resistance of the masonry wall $V_{Rd} = V_{Rd1} + V_{Rd2} = 0.09\ \text{MPa} + 0.00\ \text{m}^2 = 134.88\ \text{kN}$

(EN 1996-1-1 (eq. 6.36))

Design value of the shear strength of masonry

(EN 1996-1-1 (2.4.1) and (3.6.2))

$$f_{vd} = \frac{f_{vk}}{\gamma_M} = \frac{f_{vk0} + 0.4 \cdot \sigma_D}{\gamma_M}$$

$$f_{vd} = \frac{0.20\ \text{MPa} + 0.4 \times 0.00\ \text{MPa}}{2.20} = 0.09\ \text{MPa}$$

Characteristic shear strength of masonry $f_{vk} = 0.20$ MPa

Thickness of the wall $t = 200$ mm

Compressed length $l_c = 1500$ mm

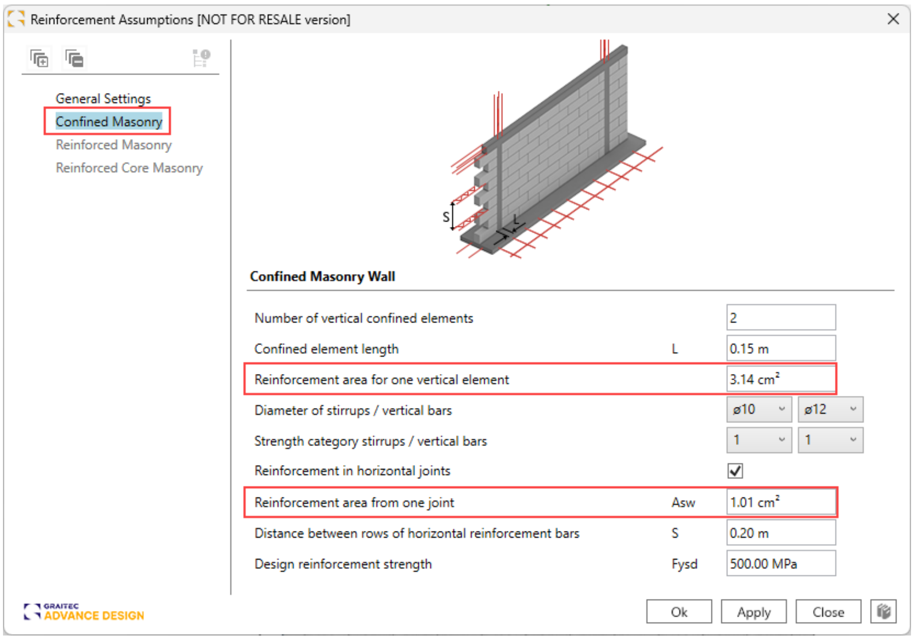
Design resistance of the reinforcement in the compressed column $f_{yd} = 434.78$ MPa

Verification $V_{Ed} < V_{Rd}$: 75.00 kN < 134.88 kN

55.60 % (Passed)

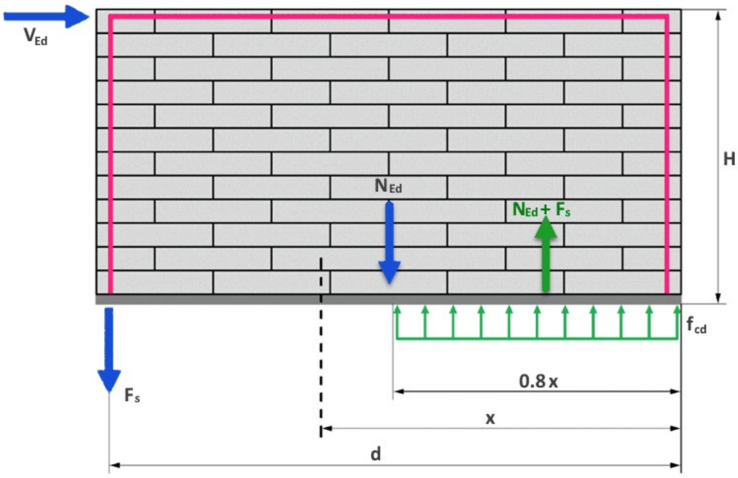
Confined masonry

When set on *Confined masonry*, the users define the reinforcement in one confining element, in the vertical and horizontal directions.



The reinforcement defined by the users has a beneficial effect on various verifications, including a verification of wall loaded mainly vertically, a verification of wall subjected to in-plane shear loading and to bending.

As for the overturning verification, it is conducted based on the diagram below:



The equilibrium of forces serves to determine the position of the neutral axis (x).
Then, the equilibrium of moments serves to determine the resisting moment (M_{Rd}).

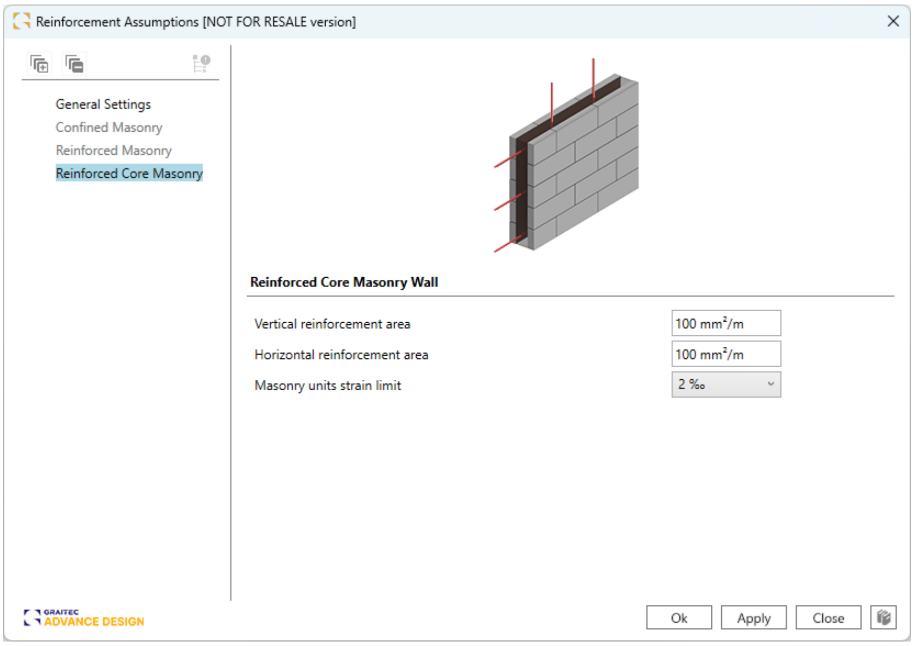
6 Verification of wall subjected to in-plane bending

Verification of wall subjected to in-plane bending						
Leaf	**Comb.**	**Critical Section**	M_{Ed}	M_{Rd}	**WR**	**Status**
			(kN·m)			
-	107	Bottom	292.50	342.03	85.52 %	Passed

Critical Section	**Bottom of the wall**
Combination	**107: 1x[1 G]+1x[101 COMB]**
Verification	$M_{Ed} < M_{Rd}$
Design value of the applied bending moment	$M_{Ed} = 292.50$ kN·m
Design value of the resistant bending moment	$M_{Rd} = M_{Rd}(z_{na}) + M_{Rd}(A_s)$ $M_{Rd} = 236.98$ kN·m $+ 105.05$ kN·m $= 342.03$ kN·m
Design value of the resistant bending moment for unreinforced section	$M_{Rd}(z_{na}) = N_{Ed} \cdot (0.5 \cdot l_w - 0.4 \cdot x)$ $M_{Rd}(z_{na}) = 175.00 \text{ kN} \times (0.5 \times 3000 \text{ mm} - 0.4 \times 365 \text{ mm}) = 236.98$ kN·m
Compressed area	$A_c = \frac{N_{Ed}}{1.00 \cdot f_d} = \frac{175.00 \text{ kN}}{1.00 \times 3.67 \text{ MPa}} = 0.05 \text{ m}^2$
Axial load	$N_{Ed} = 175.00$ kN
Design compressive stress of masonry	$f_d = 3.67$ MPa
Wall length	$l_w = 3000$ mm
Neutral axis	$x = \frac{N_{Ed} + A_s \cdot f_{yd}}{0.8 \cdot \eta \cdot f_d \cdot t_l}$ $x = \frac{175.00 \text{ kN} + 1.01 \text{ cm}^2 \times 384.62 \text{ MPa}}{0.8 \times 1.00 \times 3.67 \text{ MPa} \times 0 \text{ mm}} = 365 \text{ mm}$
Reinforcement area	$A_s = 1.01 \text{ cm}^2$
Equivanence factor for compressed zone diagram	$\eta = 1.00$
Leaf thickness	$t_l = 0$ mm
Design value of the resistant bending moment from reinforcement	$M_{Rd}(A_s) = A_s \cdot f_{yd} \cdot (d - 0.4 \cdot x)$ $M_{Rd}(A_s) = 1.01 \text{ cm}^2 \times 384.62 \text{ MPa} \times (2850 \text{ mm} - 0.4 \times 365 \text{ mm})$ $M_{Rd}(A_s) = 105.05$ kN·m
Lever arm	$d = 2850$ mm
Verification	$M_{Ed} < M_{Rd}$: 292.50 kN·m $<$ 342.03 kN·m **85.52 % (Passed)**

Reinforced core masonry

When set on *Reinforcement core masonry*, the users define the reinforcement area in the concrete core, in the vertical and horizontal directions.



Note, this reinforcement type is available only when doing the verifications according to Romanian code CR6-1013.

12. Steel Connections

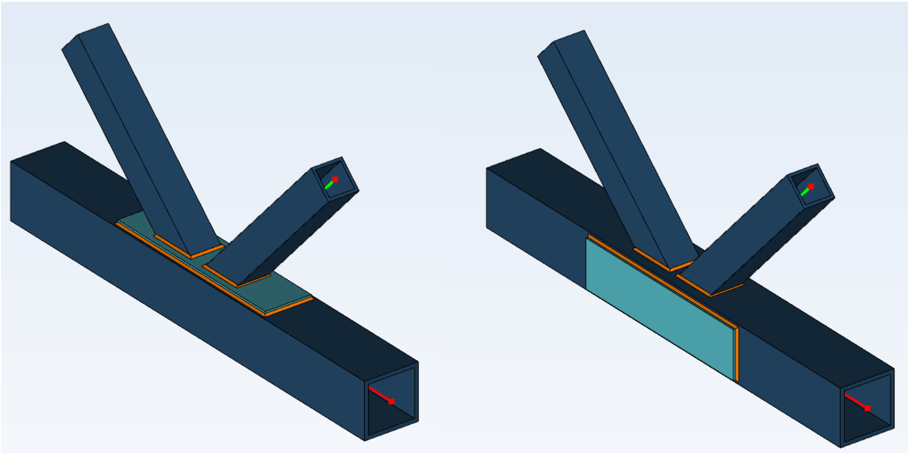
New features and improvements implemented in the latest version of the Steel Connection module.

12.1. Welded tube truss connection – Reinforcement plates

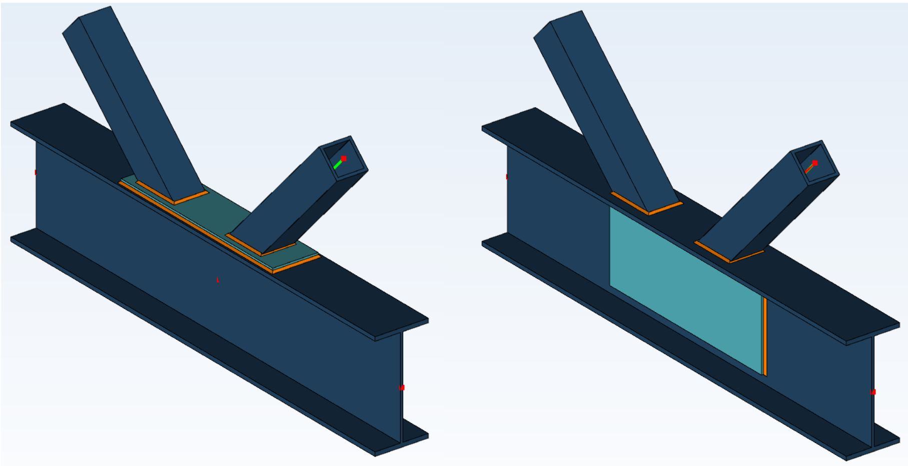
Possibility for defining for the welded tubular truss connections additional plates needed to reinforce the contact area between the chord and the brace members. These plates can be two types: horizontal and lateral.

With the 2025 release of the Advance Design Steel Connection module, for welded tubular connections, it is now possible to add reinforcing side plates or flange plates.

For rectangular hollow chord sections, these reinforcement plates help in resisting chord face failure, brace failure, chord side wall buckling and chord shear.



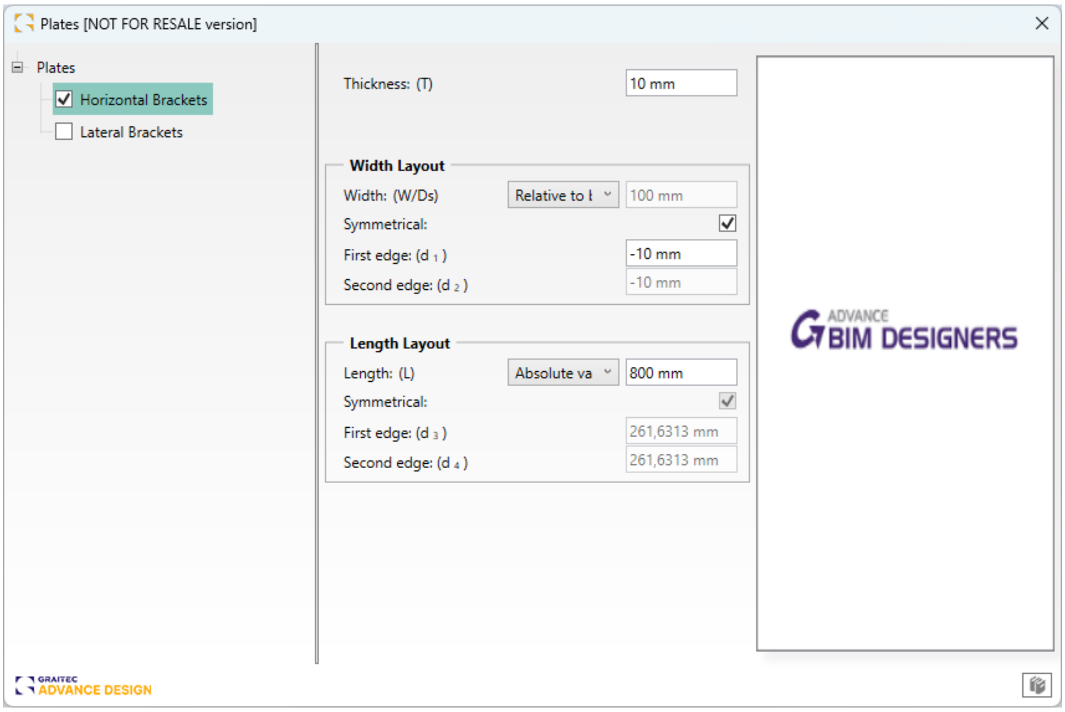
For I / H chord sections these reinforcement plates help in resisting chord web yielding, brace failure and chord shear.



At the same time, you can define either a side or top reinforcement plate, which is welded to the chord along its entire perimeter.

The parameters of the reinforcement sheets are defined in the new Plates window.

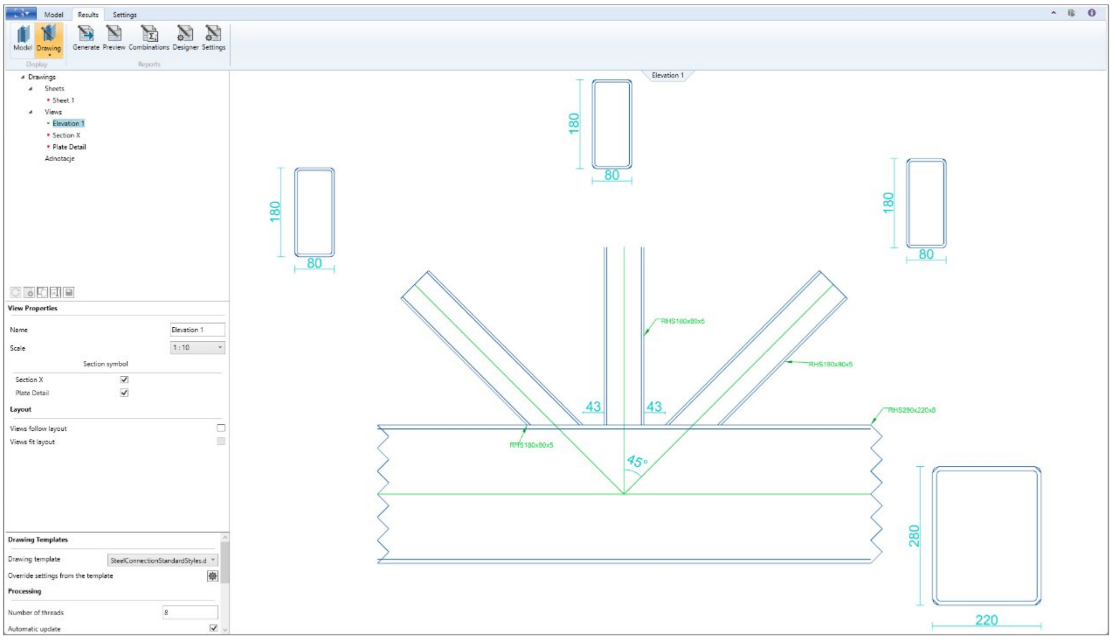
[The picture below should be replaced with a new one that contains pictures on dialog. Still not the case on 13.05.2024]



12.2. Welded tube truss connection – Drawings

Drawing generation capabilities for welded tube truss connections.

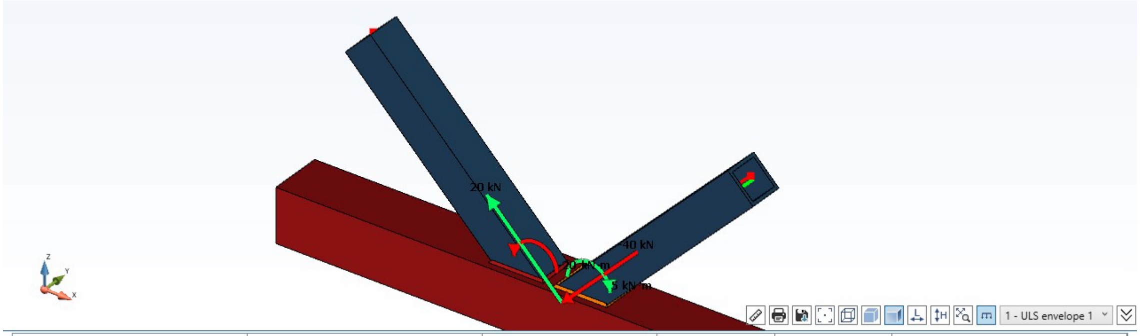
With the latest version of Steel Connection, it is now possible to generate drawings for the latest available connection, the welded tube truss. As in the case of other connections, the drawing contains views with descriptions and dimensions of elements, arranged on a sheet, which can be printed or saved for further editing in CAD programs.



12.3. Welded tube truss connection – Punching shear verification

Implementing a punching shear failure verification for welded tube sections.

In the 2025 version of the Advance Design Steel Connection module, a punching shear verification is now conducted for welded tubular connections made of Rectangular Hollow Sections (RHS) or Square Hollow Sections (SHS). The punching shear of each diagonal against the main chord is checked according to EC 3-1-8 tables 7.11 & 7.12.



Verification type	Objects	Combination	Force	Resistance	Work Ratio
Chord face failure	[First diag.] RHS80x7 S235	[1]: ULS envelope 1	4,84	1	484,45%
Weld seam	[Weld] 5mm On sec. diag.	[1]: ULS envelope 1	2149,08 kN/m	1039,23 kN/m	206,79%
Chord shear	[First diag.] RHS80x7 S235	[1]: ULS envelope 1	-20 kN	299,41 kN	6,68%
Punching shear	[First diag.] RHS80x7 S235	[1]: ULS envelope 1	-20 kN	398,65 kN	5,02%
Brace failure	[First diag.] RHS80x7 S235	[1]: ULS envelope 1	-20 kN	405,14 kN	4,94%

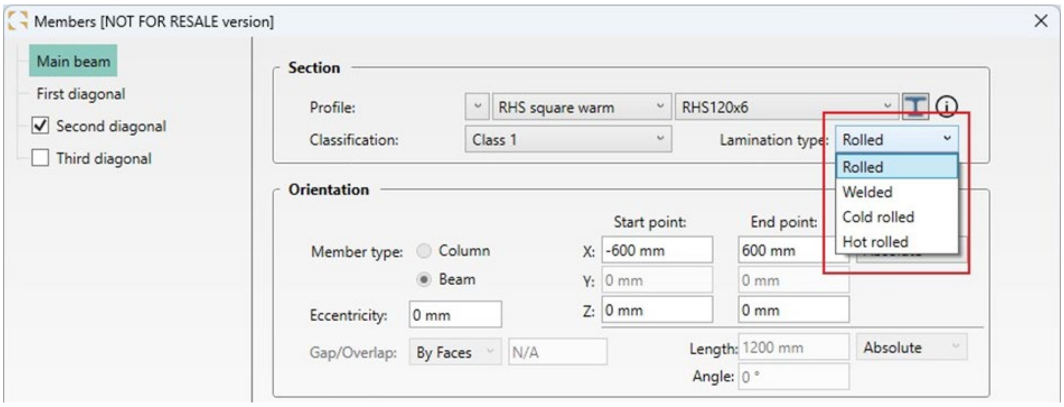
12.4. Welded tube truss connection – Set of improvements

A set of smaller enhancements to the Welded tube truss connection to increase functionality.

The welded tube truss connection is the youngest of the available connections, so a number of smaller improvements have also been made in the latest version to improve its functionality.

- **Filed for defining the lamination type for members**

Starting with the new version of the module the cross-section lamination type can be specified. It can be set to rolled, welded, cold rolled or hot rolled. The lamination type has an impact on the buckling curve used for chord side wall buckling calculation.



- **Standardized coloring of statuses in reports**

In the newest version of the module, the way passing results are displayed in reports is homogenized, and the text showing the verification status for a given check is now colored - the status is *Passed* is written in green, and *Failed* in red.

- **Additional warning about SLS combinations not being used in calculation**

When exporting a welded tubular connection from Advance Design to the Steel Connection module, Service Limit State (SLS) combinations coming from Advance Design are not considered in verifications of connections. Only Ultimate Limit State (ULS) combinations are considered for design. For this purpose, a warning message is displayed in the info panel telling the user that only ULS combinations were considered.

Type	Details	Value	Limit
⚠	Serviceability limit state combinations are excluded from the verification of the design resistances of the joint (7.2.1(2) EN 1993-1-8)!	-	-

- **Improved import of loads from Advance Design**

In the case of welded tube connections as well as gusset connections, the transfer of forces acting in the diagonal members was revised and improved for different types of geometric configurations and settings of local axes arrangements in the chords and diagonal members.

- **Improvements in calculations and report for moment brace failure check**

Revisions have been made to calculations and reports related to moment brace failure verification acc. table 7.15 of EN 1993-1-8 for cases that are not within the scope of the standard. The changes include additional explanatory information in the reports, as well as handling of additional scenarios. In addition, Y, K and N-gap joints are now treated as individual T-joints for moment resistance.

8.1.3 Bending resistance

Check relation for diagonal 1:

$|M_{i,Ed}| \leq M_{ip,i,Rd}$

$M_{i,Ed} = 5\ \text{kN·m}$

Moment resistance is determined as for individual T nodes (5.3.5 from "Design of welded joints Celsius355 and Hybox355" – author: TATA Steel Europe Limited, 2013).

$M_{ip,i,Rd} = f_{yi} \cdot \left[W_{pl,i} - (1 - b_{eff,i}/b_i) \cdot b_i \cdot (h_i - t_i) \cdot t_i\right] / \gamma_{M5}$ EN 1993-1-8 Table 7.14

Bending resistance formula is valid for: $0.85 < \beta \leq 1$ EN 1993-1-8 Table 7.14

where

$\beta = \frac{b_i}{b_0} = \frac{100\ \text{mm}}{180\ \text{mm}} = 0.56$ EN 1993-1-8 1.5(6)

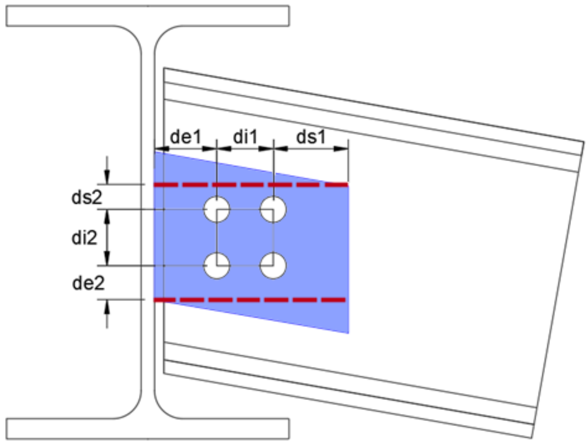
$0.85 < \beta = 0.56 \leq 1$ **Verification failed!**

Warning: Eurocode 3 does not cover this situation!

12.5. Shear plate connection – Improvement to bolts positioning

Adjusted the existing behavior for the bolts positioning in the case when the secondary beam is sloped.

For the case of sloped secondary beam connected to a main beam via a shear plate, this shear plate is skewed to follow the secondary beam inclination. For such skewed plates, special attention should be made for bolts positioning to ensure proper spacing and end distances. Now, the 2025 version of the Advance Design Steel Connection module, conducts additional verifications for bolts in a skewed plate. Instead of considering the actual contour of the skewed plate for bolt positioning, the module calculates the biggest rectangle that can fit inside the skewed plate and it is this rectangle that will accommodate the bolts and will be considered for spacing and end distances verifications.

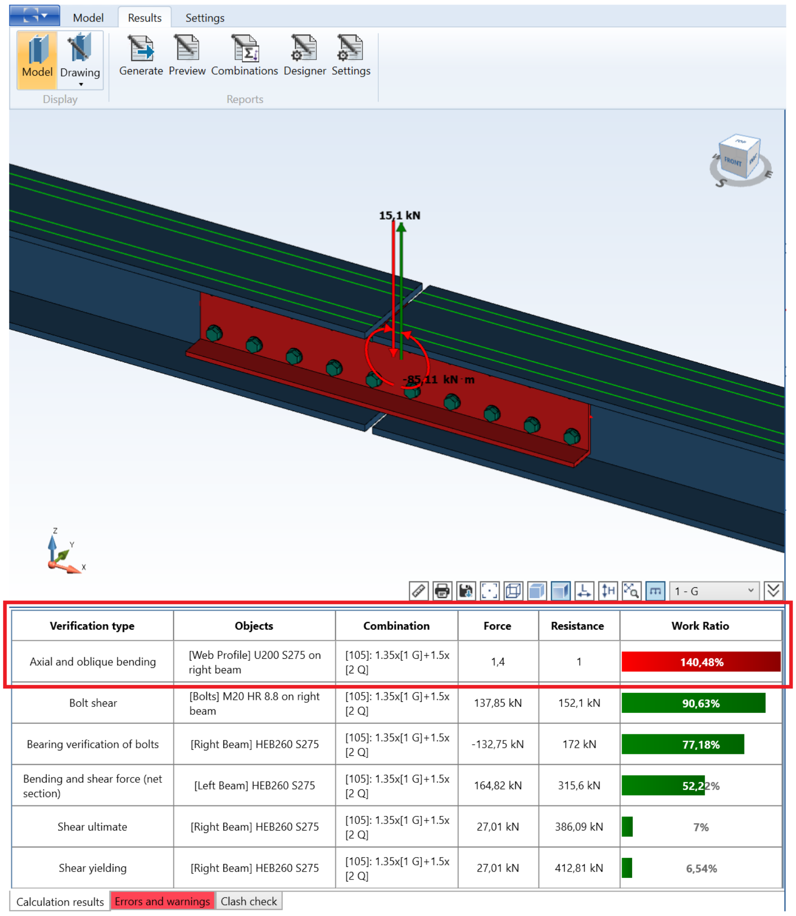


Shear plate real shape (blue) and imaginary boundary line for placing the bolts (red)

12.6. Splice connection – Check for U splice profile on Info panel

The check for U splice profile is now available on the Info panel.

Previously, when beams were connected by a splice connection and the connecting element was a U-profile, the resistance check of the splice was displayed only in the report. Now with the 2025 release, this check is clearly displayed also on the info panel.



12.7. Update default properties of the hooked anchors

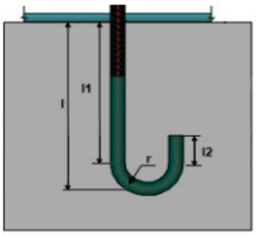
Changes related to properties of hooks to get correct mandrel diameter.

Previously in the Steel Connection module, for the hooked anchors of base plate and tubular base plate joints, the proposed default value of hook diameter was sometimes smaller than the minimum bend diameter of the anchor bar and this produced a warning message requiring the user to change this default diameter and rerun the calculation.

Now, the default value for hook extension length and hook radius diameter were recalculated using CNC2M recommendations (*Recommandations pour le dimensionnement des assemblages selon la NF EN 1993-1-8, tab. 19*) and EN 1992-1-1:

- For the hook extension length, the rule (l2=2d) from the CNC2M is adapted
- For the hook radius for d ≤ 6mm, the rule (R=3d) from the CNC2M is adapted
- For the hook radius for d >16mm, the rule (R=3.5d) from EN 1992-2-2 tab 8.1N is adapted

where d - anchor diameter

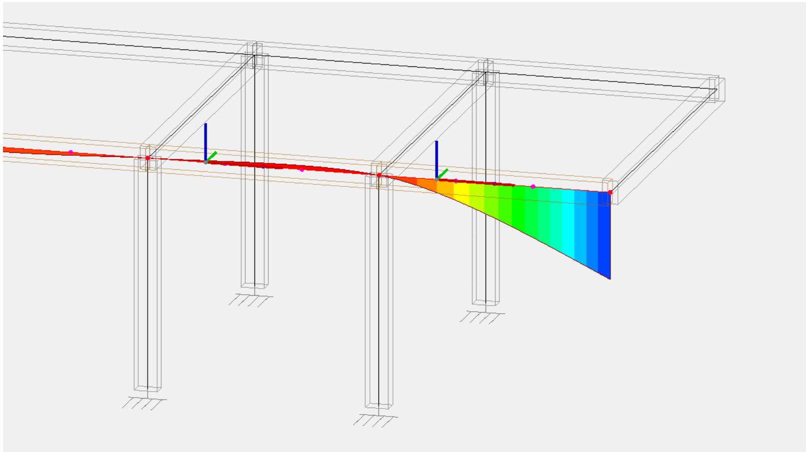


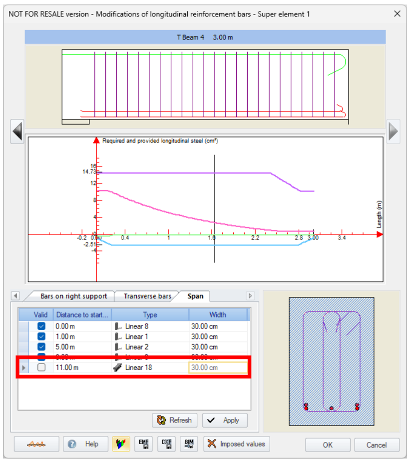
13. Other novelties and small improvements

Selected small improvements introduced in the newest version of Advance Design

- **Reinforcement beams – consideration of the cantilever for the real reinforcement for superelements**

In the case of a multispan reinforced concrete beam defined as a superelement, for which the determination of the real reinforcement is performed, the exclusion of the outermost support in the Reinforcement window allows the determination of the reinforcement and verification of the deflection as for the cantilever.

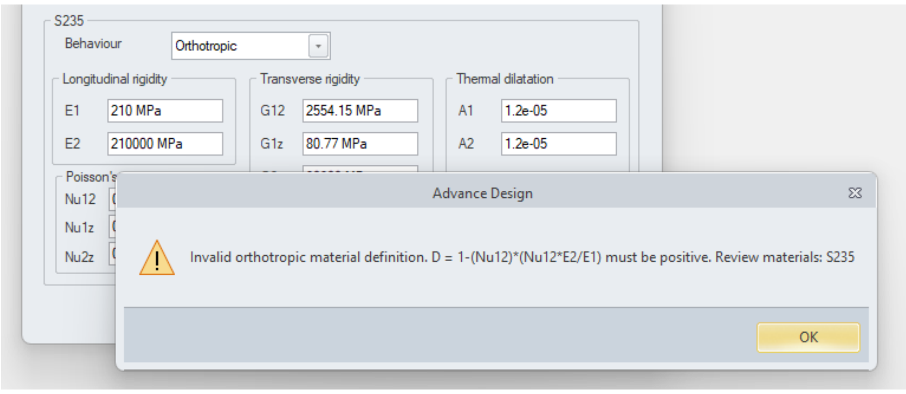




- **Additional verification during the definition of orthotropic materials**

When defining an orthotropic material, an additional data validation is now conducted. A new condition checks that the entered values of stiffness modulus and Poisson's ratio do not give a negative value for the delta factor, which could lead to instability of the element stiffness matrix.

$$\Delta = 1 - \vartheta_{12}\vartheta_{21} = 1 - \vartheta_{12}\vartheta_{12}\frac{E_2}{E_1}$$

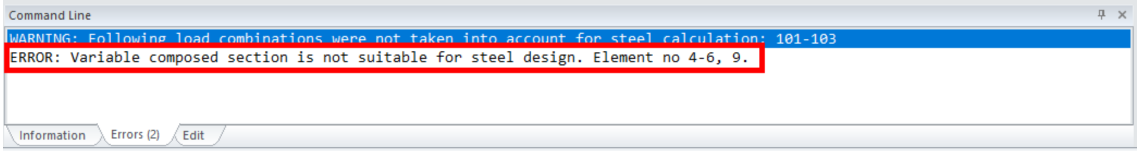


- **Change the default color for displaying result values for selected localizations**

For selected locations (CZ and SK), the default color settings for graphic result descriptions have been modified.

- **Improvements in the display of errors and warnings**

For selected errors and warnings, in order to reduce the number of rows with messages when the same error affects multiple elements, a single row with a list of elements is now displayed. In addition, several new warnings have been added - for example, information about the impossibility of conducting standard verification for steel elements with variable height if they are made of compound profiles.

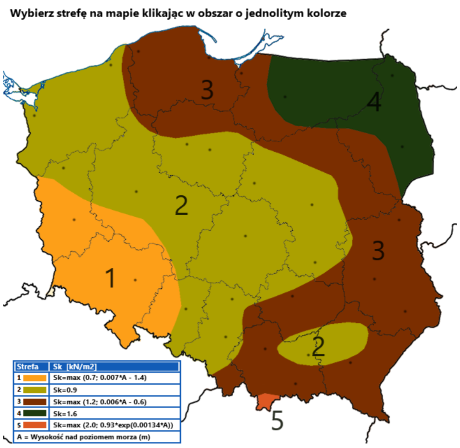


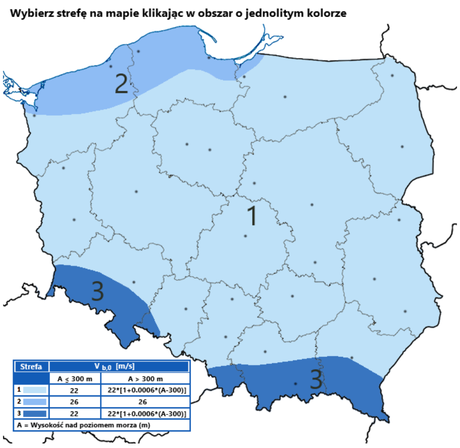
- **Correct reading data from steel design template for LTB**

In this version of the program, the behavior regarding the consideration of LTB (lateral torsional buckling) settings defined in the design template has been improved. Now these settings, in cases where the values were not determined automatically but were imposed directly in the design template, are now correctly considered during calculations for linear elements as well as for superelements.

- **Updated maps for Wind and Snow for Poland**

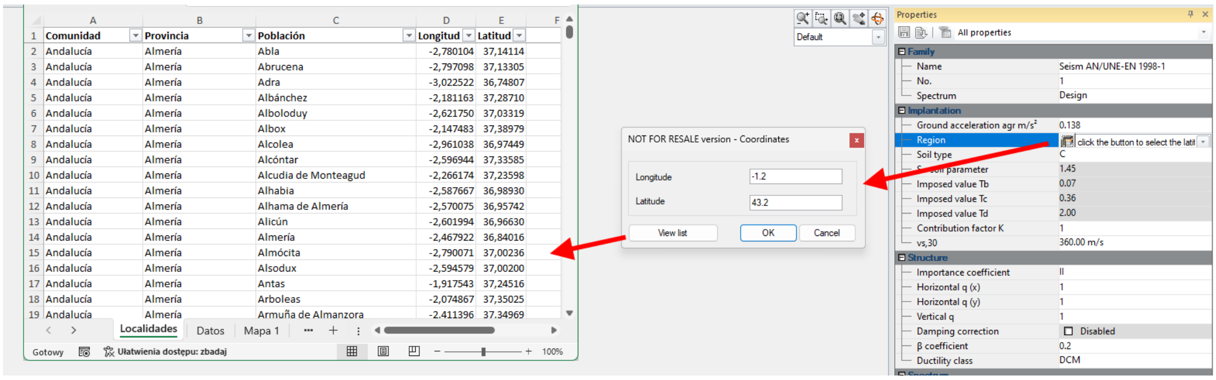
To make it easier to use the maps for wind and snow loads for Poland, a number of small improvements have been made to them, such as adding new comments and descriptions, translating picture texts, improving outlines, and adding zone numbers.





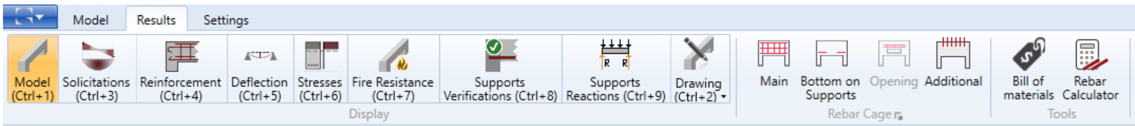
- **Easier seismic data entry for Spain.**

When entering data in the seismic load family for Spain, coordinates must be entered to automatically determine horizontal ground acceleration values. In the latest version of the application you can easily search for coordinates for a given location thanks to an attached list in an Excel file.



- **RC modules - Faster switching between windows**

To make it easier to remember the keyboard shortcuts that allow quick switching between result windows in Advance Design RC modules during standalone work, these shortcuts are now shown on the ribbon under the name of the respective result window type.

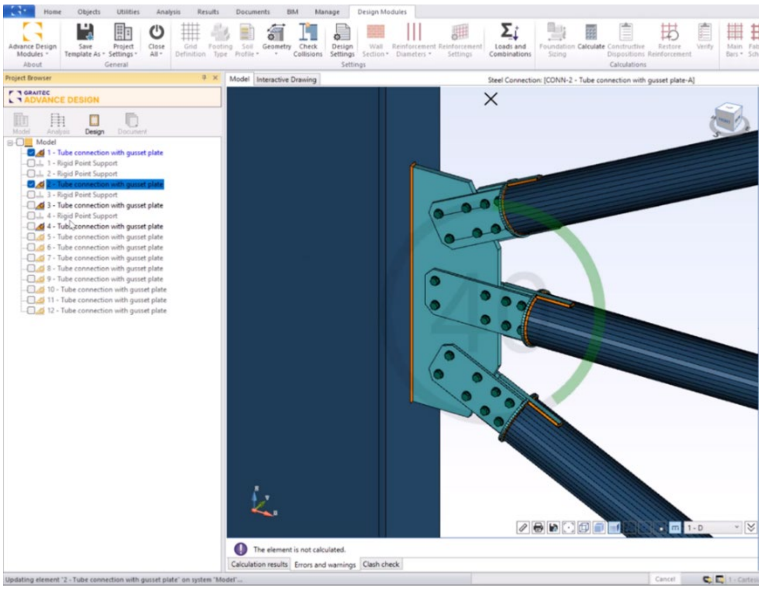


- **RC Wall module– Better description of decisive combinations in the report**

On the report or share walls, when the critical combination involves a Newmark comb, names of load cases involved in this combination are presented.

- **Steel Connection module – Progress bar during the update process**

During operations that require refreshing the connection data, especially when updating the connection from Advance Design model data, the progress of the operation is shown graphically.



- **Steel Connection module - Importing of steel materials that are not available in a database**

When exporting from Advance Design a steel connection with linear elements made of steel material that is not available in the Steel Connection module, the new version of automatically creates this new steel material and assigns it by default to the corresponding linear elements.

- **Steel Connection module – Indicating the combination decisive for rotational stiffness calculation**

For connections that perform rotational stiffness calculations (Base plate, Tubular base plate, Apex, Moment end plate, Gable wall), the report now presents information about the combination that was decisive during these calculations.

13.2 Rotational stiffness calculation

Combination: [111]: 1.35x[1 G]+1.5x[3 V]+1.05x[2 Q]

$$S_j = \frac{S_{j,ini}}{\mu}$$ EN 1993-1-8, 5.1.2 (4)

Stiffness ratio:

$$M_{j,Ed} > \frac{2}{3} \times M_{j,Rd} \rightarrow \mu = \left(1.5 \times \frac{M_{j,Ed}}{M_{j,Rd}}\right)^{\psi} = \left(1.5 \times \frac{130.35\ kN \cdot m}{184.24\ kN \cdot m}\right)^{2.7} = 1.17$$