Engineering manual Nr. 09

Updated 12/2024

Design of a steel beam

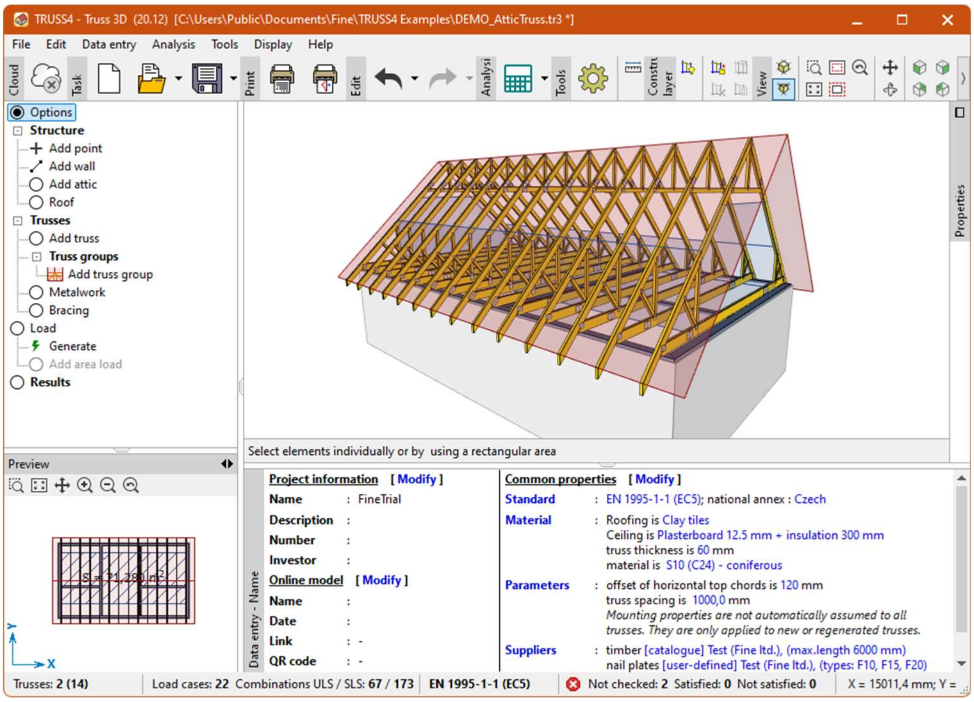
Assignment

Program: Truss 3D, Fin 2D, Steel
File: DEMO_AtticTruss.tr3

In this manual we will show how to support overloaded trusses with a beam. We will show two material options: glued laminated timber and steel. For the design of the steel beam, we will use the "**Fin EC**" programs.

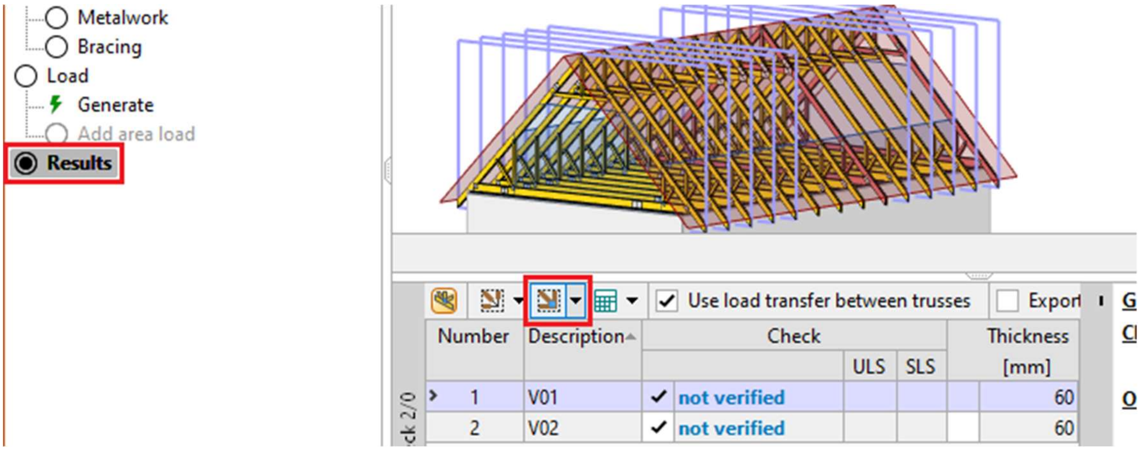
Design of a timber beam

Start "**Truss 3D**" program and select the "**Open an existing project**" option. Select the DEMO_AtticTruss.tr3 file from the folder *Fine Online Examples*. This will open a simple project with attic trusses.



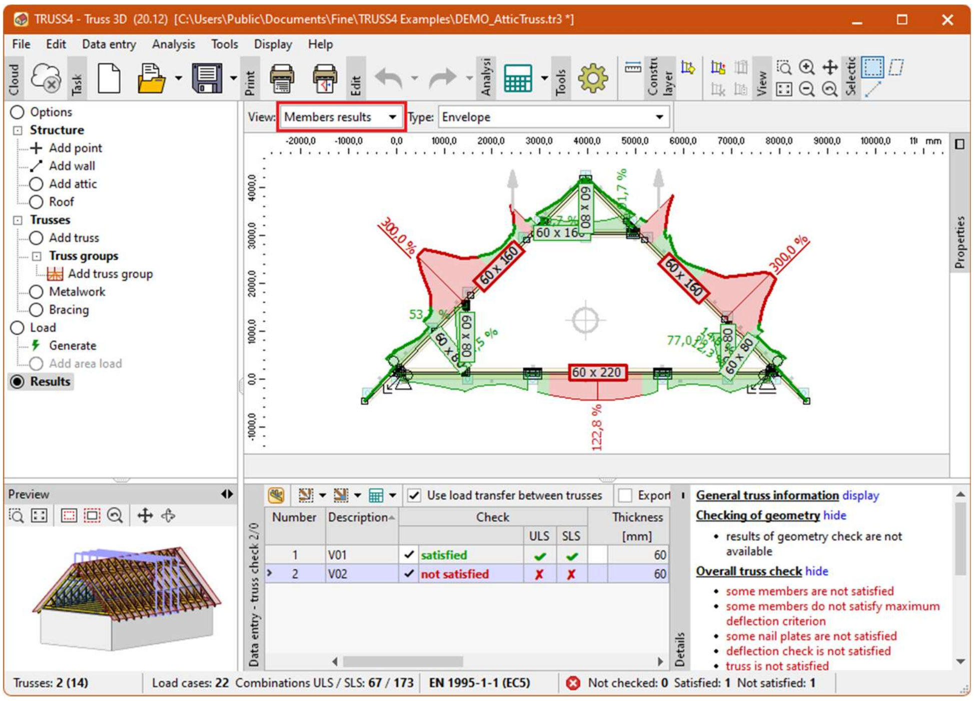
Initial project

The design has not been assessed. So, we go to the "**Results**" section of the tree menu to start the automatic design of the structure. This can be started, for example, with the *F8* key or with the "" button in the toolbar above the truss table.



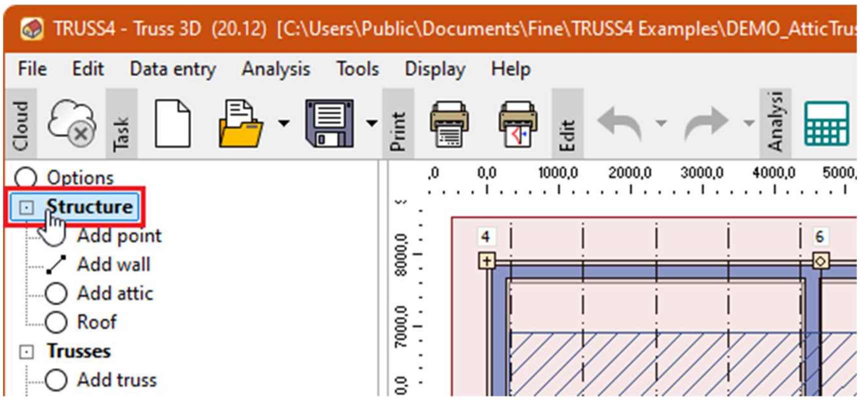
Running automatic design

We see that while truss V01 passes after the design, truss V02, which has one less support, fails significantly.



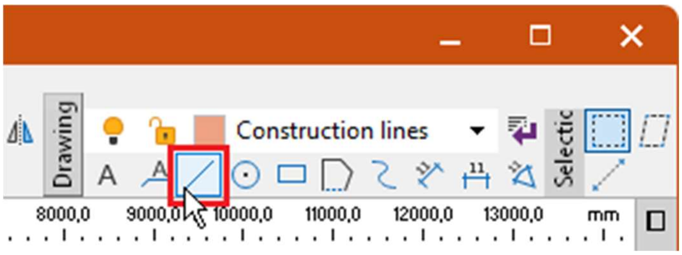
Failing truss V02

There are several solutions. It would be possible to use a greater thickness of timber, double trusses, or scabs. In view of the favourable topology of the walls, we will use a different approach: we will additionally support the V02 trusses with a beam. We proceed to the "**Structure**" section of the control tree. In this section, in addition to the walls and trusses, the point marks that we will use for the input are displayed on the workspace.



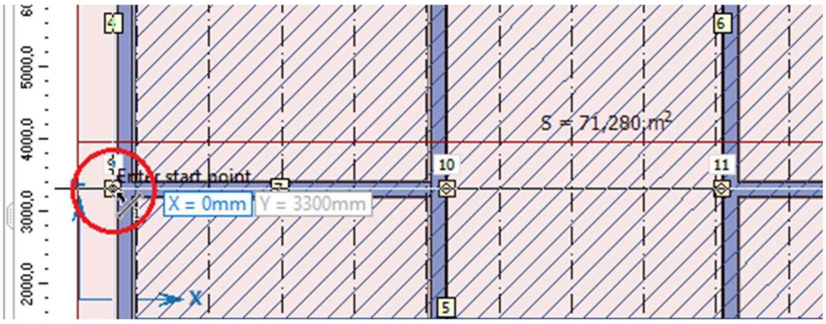
"Structure" section of the control tree

First, we draw an auxiliary line to create the necessary snap points on the walls. In the "**Drawing**" toolbar, select the "**Line**" tool.



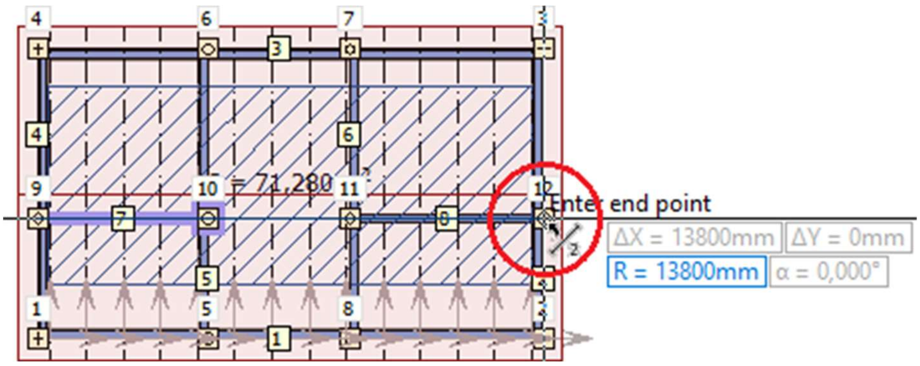
"Line" tool

We choose the point *9* as the origin of the line.



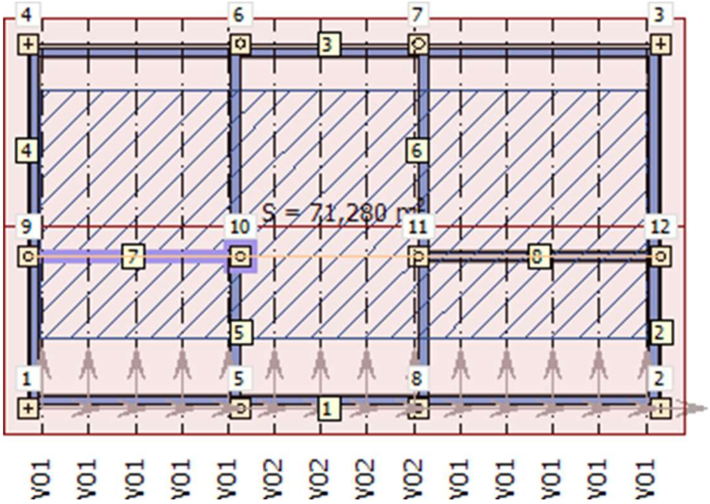
Specifying the beginning of a line

The end point will be point *12*.



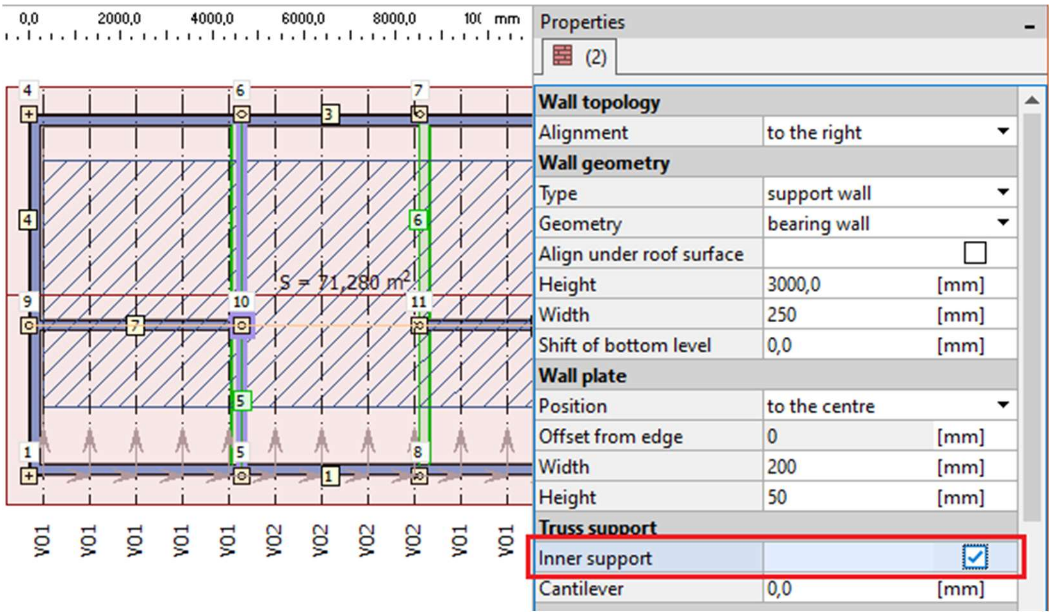
Entering the line end

Now we have a new line on the workspace. Since it is specified in the default "**Construction lines**" layer, it will not appear later in the print reports.



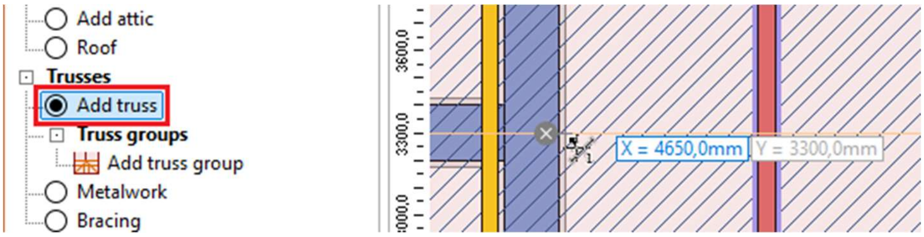
Structure with entered line

Now we will edit the properties of the inner walls *5* and *6*. For editing, we will use the "**Properties**" sidebar, which is used to edit the properties of selected objects in bulk. The properties panel can be minimized at the right edge of the workspace. In this case, first enlarge it with the "◻" button. Select the mentioned walls (they will be highlighted in green) and check the "**Inner support**" parameter in the "**Truss support**" section. Thanks to this, the joint with the support will be inserted exactly in the middle of the wall, which will be especially useful when later converting the structural layout to "**Fin EC**" programs.



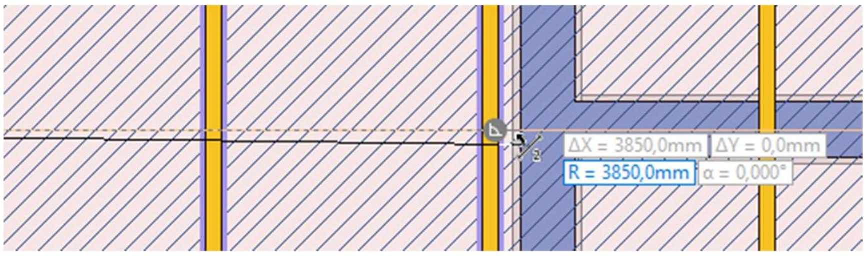
Bulk edit of properties

Let's move on to the actual specification of the beam. Select the "**Add truss**" item in the control tree. First, select the start of the truss. We will place it at the intersection of the construction line and the edge of the wall *5*. When entering, we should approach the intersection with the cursor in the direction from the wall edge (i.e. from the top or bottom). The snap point mark will then turn grey to indicate that the truss is snapping to the wall. If the colour of the mark is different, the truss will be placed without attachment to the wall and will not take on the necessary properties from the wall.



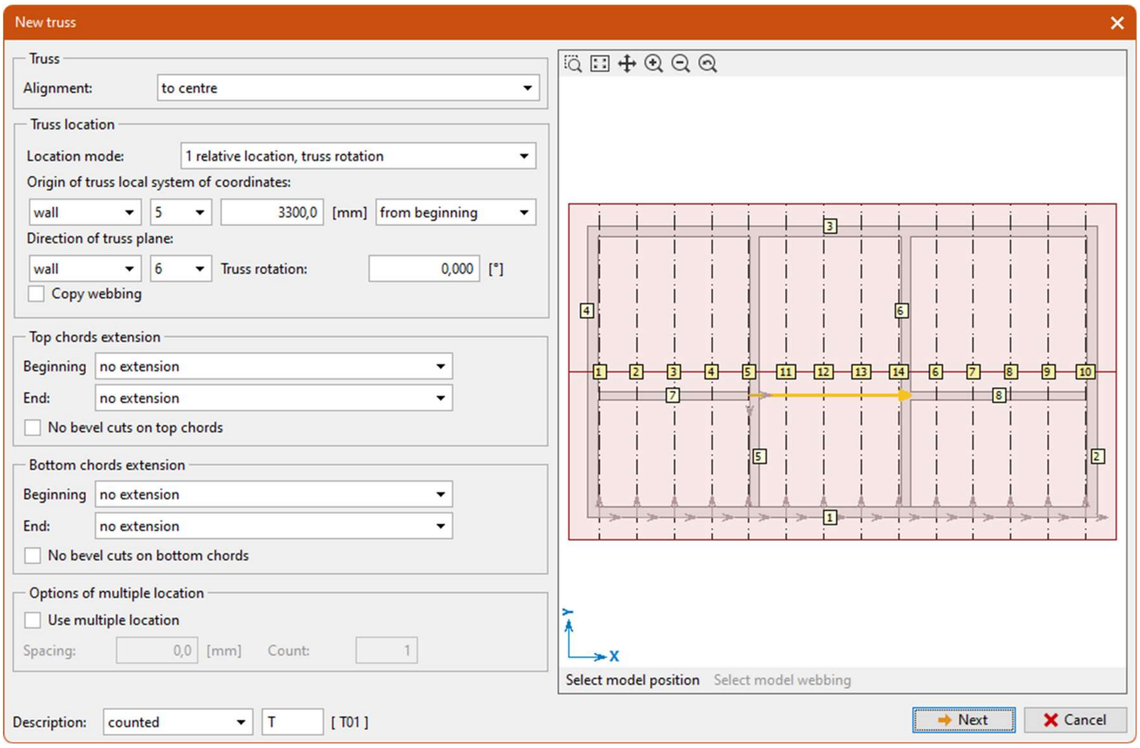
Entering the start point of the beam

The same procedure is then used to specify the end point of the beam. The same rules for selecting the snap point apply. In this case, instead of an intersection, we may also be offered a "**perpendicular**" snap point.



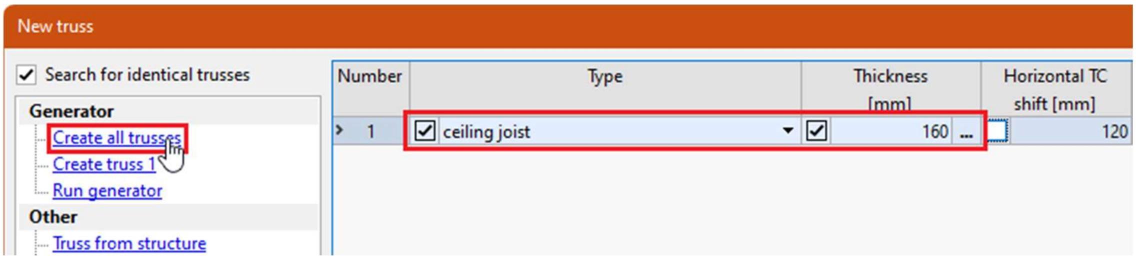
Specifying the end point of the beam

The "**New truss**" window will then appear, where the position of the beam can be modified. We go to the second part of the window using the "**Next**" button.



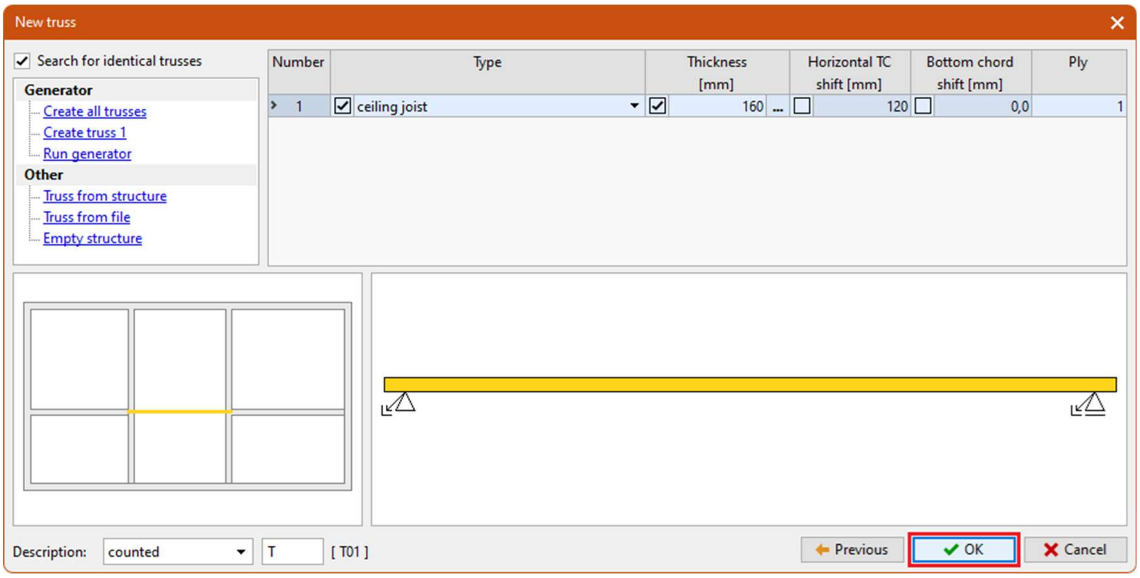
Window "New truss"

In the second part of the "**New truss**" window, we first edit the parameters in the table on the right side of the window. In the "**Type**" column, select the "**Ceiling joist**" entry and in the "**Thickness**" column, enter the timber thickness of *160mm*. Then we can use the item "**Create all trusses**" to generate the beam.



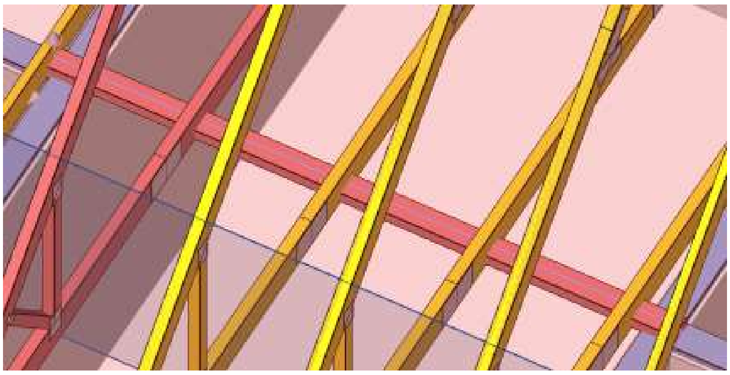
Specifying the parameters of the beam and its creation

At the bottom of the window, we will see a preview of the beam. Close the window with the "**OK**" button.



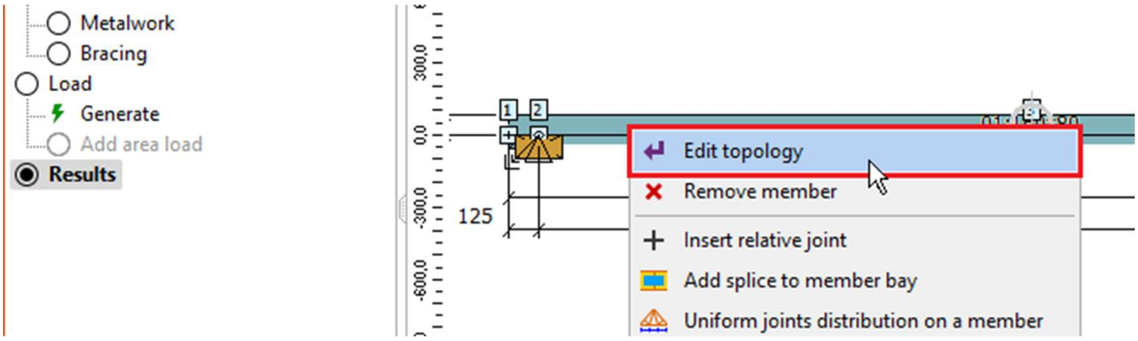
Window "New truss" with the created beam

We can now see the new timber beam in the structure, but it collides in height with the attic trusses.



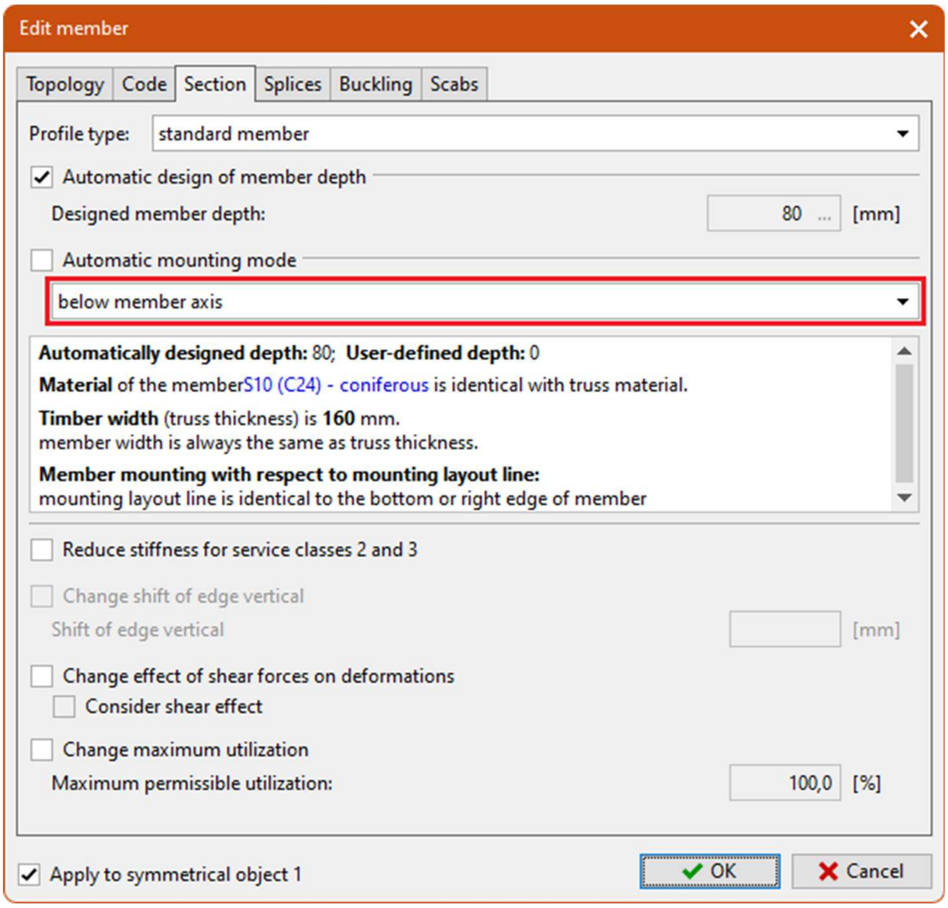
Beam in the structure

The default alignment of the elements in the ceiling plane is above the ceiling reference plane. It is necessary to change this alignment. Go to the "**Results**" section of the control tree. On the main workspace we have a view of the beam. If not, we can view it there using the *Ctrl+Tab* keyboard shortcut. Right-click the local menu for the member and select "**Edit topology**". Alternatively, it would be possible to double-click the member with the left mouse button.



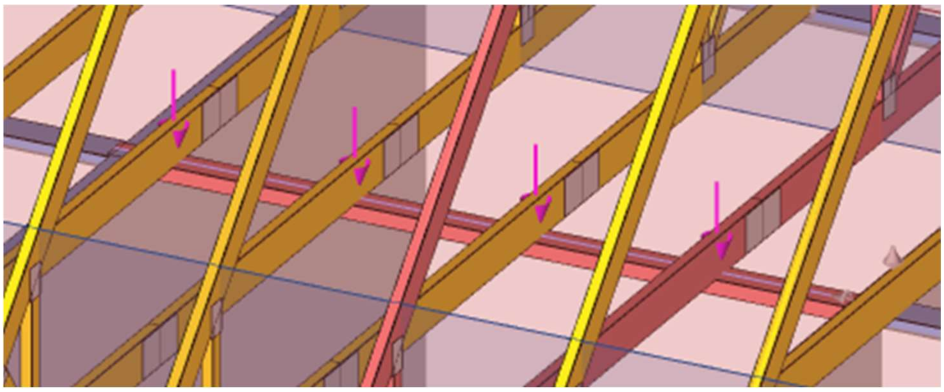
"Edit topology" item in the context menu

The "Edit member" window opens. On the "**Section**" tab, uncheck "**Automatic mounting mode**" and select the option "**below member axis**". Then you can close the window with the "**OK**" button.



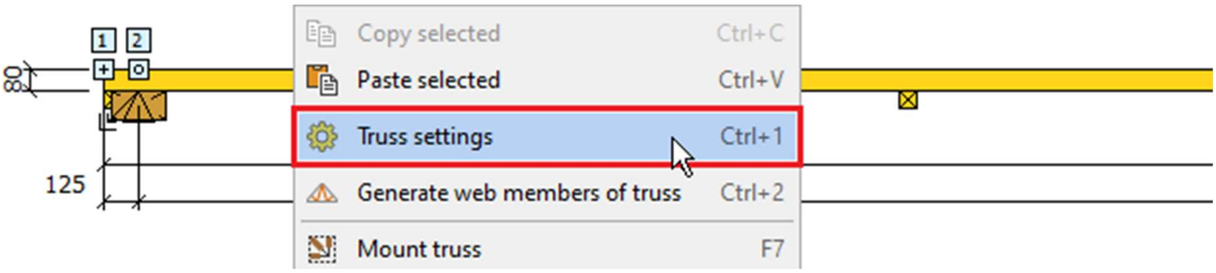
Adjustment of the vertical alignment of the member

The beam is already placed under the trusses in the 3D view, and marks indicating the load transfer from the trusses to the beam have been added.



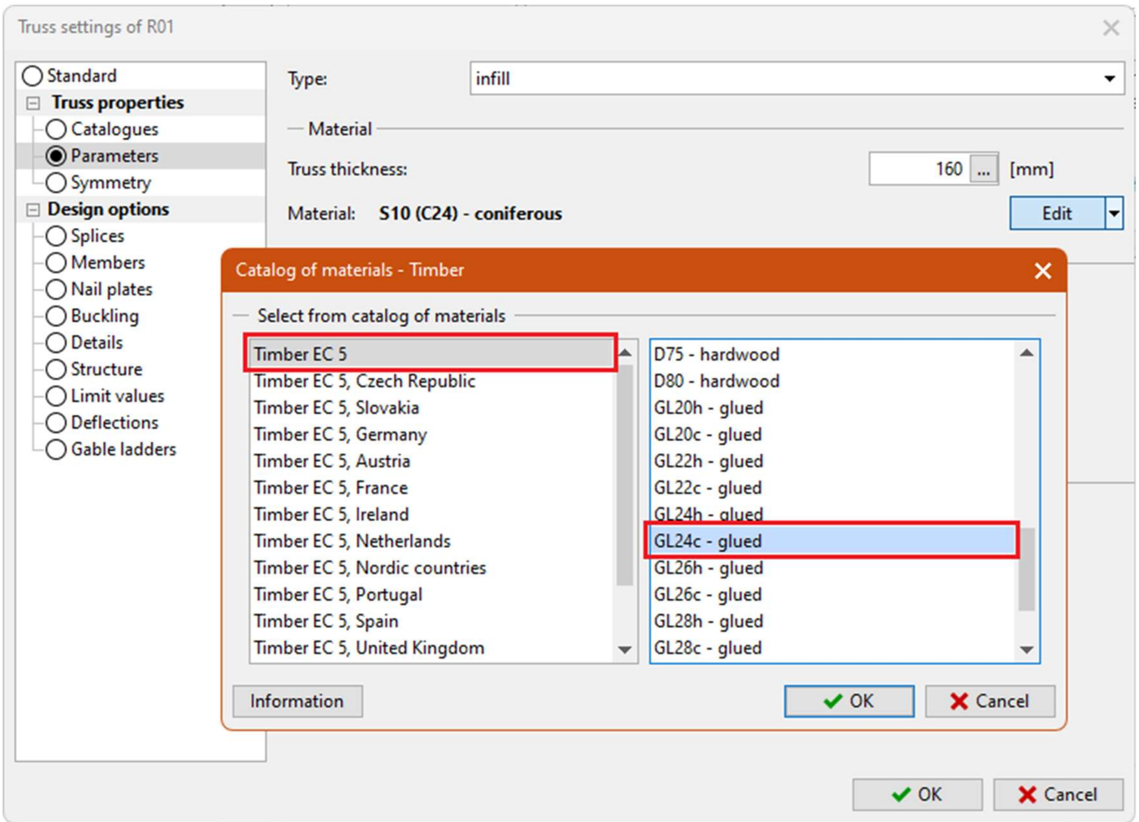
Vertically aligned beam

The next change we need to make is to change the timber grade. We assume the use of glued laminated timber, which has different strength characteristics than the initial material for solid timber. If we right-click anywhere on the workspace outside the beam itself, the local menu for the entire truss will pop up. There we select "**Truss settings**".



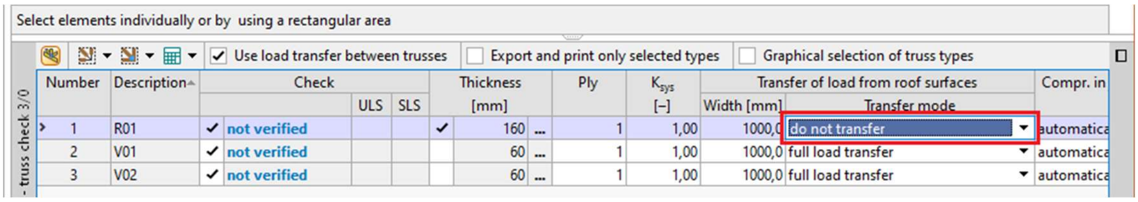
Launching the "Truss settings" window

We go to the "**Parameters**" section in this window, which contains not only properties such as truss type or loading width, but also the timber grade. The "**Edit**" button brings up the " **Catalogue of materials - Timber**" window with a selection of predefined timber grades. Select the grade "**GL24c - glued**" from the "**Timber EC5**" group and confirm the selection with the "**OK**" button. Then close the entire "**Truss settings**" window with the "**OK**" button.



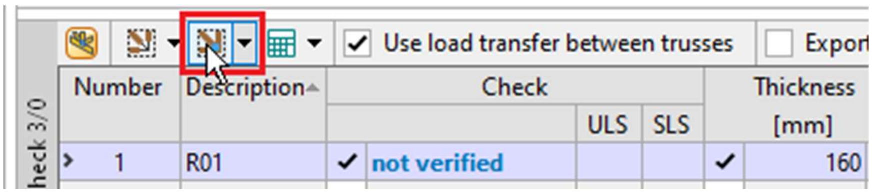
Change of timber grade in the "Truss settings" window

The last adjustment before the calculation is to set the correct mode for the load transfer from the roof planes. The beam will be loaded only by the point forces from the main trusses. Therefore, we do not want any additional load to be taken from the roof planes. Therefore, in the truss table in the column "**Transfer of forces from roof surfaces**" we select the option "**do not transfer**" for the beam.



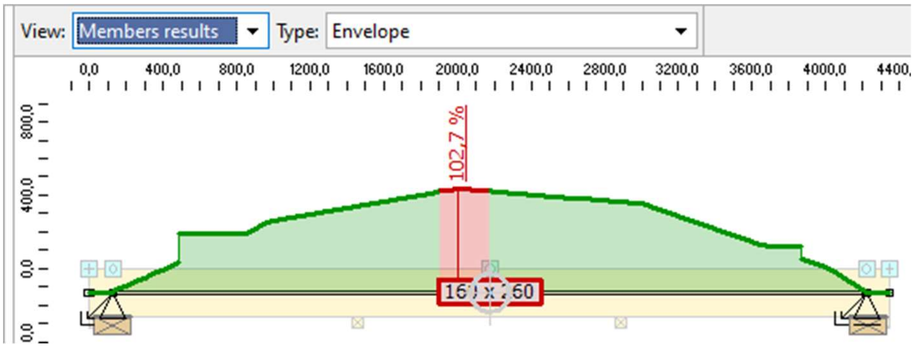
Choosing the method of force transfer from the roof planes

Now we can start the automatic design of the structure. Use the corresponding button in the toolbar above the truss table or the *F8* key.



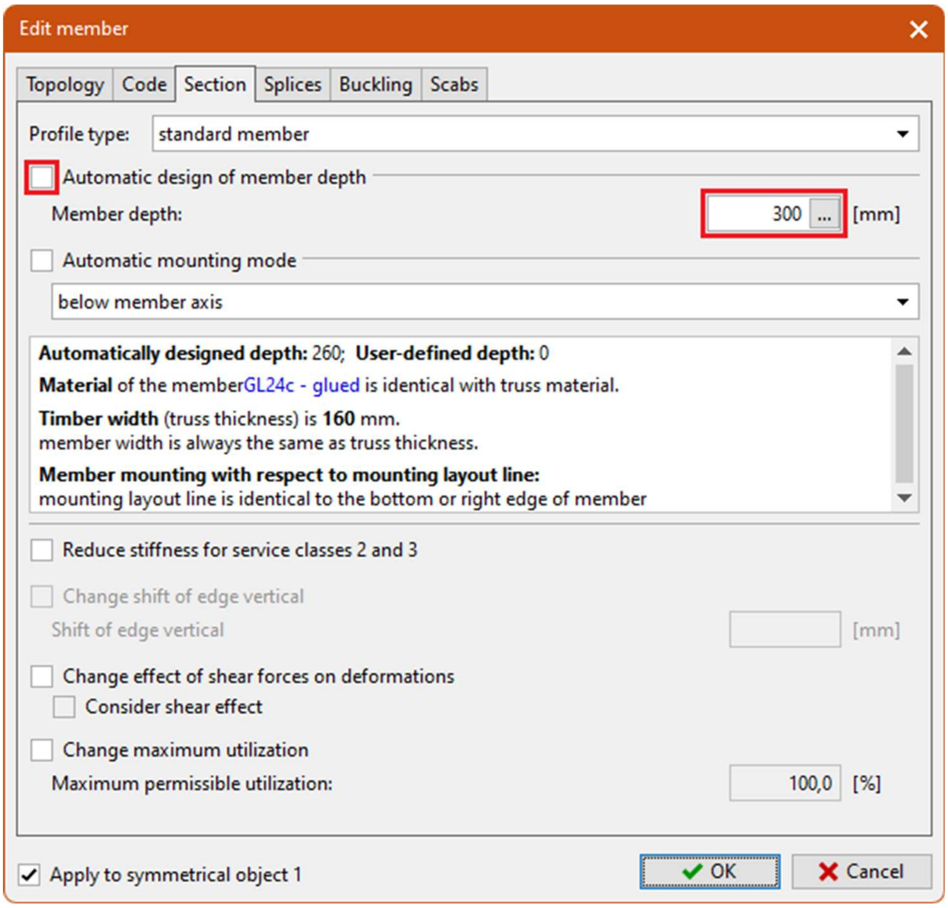
Running the automatic design of the beam

After the calculation we see that the beam fails. Switch the view in the main workspace header to the "**Member results**" mode. We can see that the beam fails in the middle of the span, i.e. at the point of the greatest bending moment.



Results for the beam

The automatic design terminated at a section depth of *260mm*. This corresponds to the default timber database for this project. So, we have to manually enter a higher value in the member properties. We use the local menu of the member to bring up the "**Edit member**" window, as in the case of the vertical alignment option. In the "**Member results**" mode, it is not possible to use the left mouse button double-click, as this brings up a list of the detailed results of the member design. In the "**Section**" tab, disable the automatic design by unchecking the "**Automatic design of member depth**" checkbox and then manually enter a depth of *300mm*.



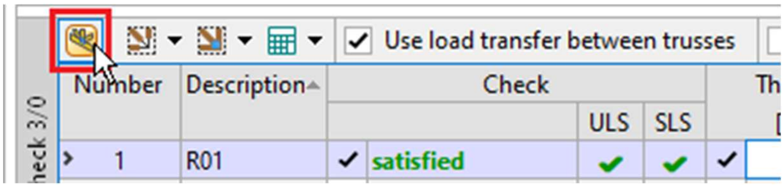
Setting the custom member depth

After changing the cross-section depth and recalculating the structure, the beam is already satisfied. Therefore, a section of *160x300mm* is suitable.

TRUSS4

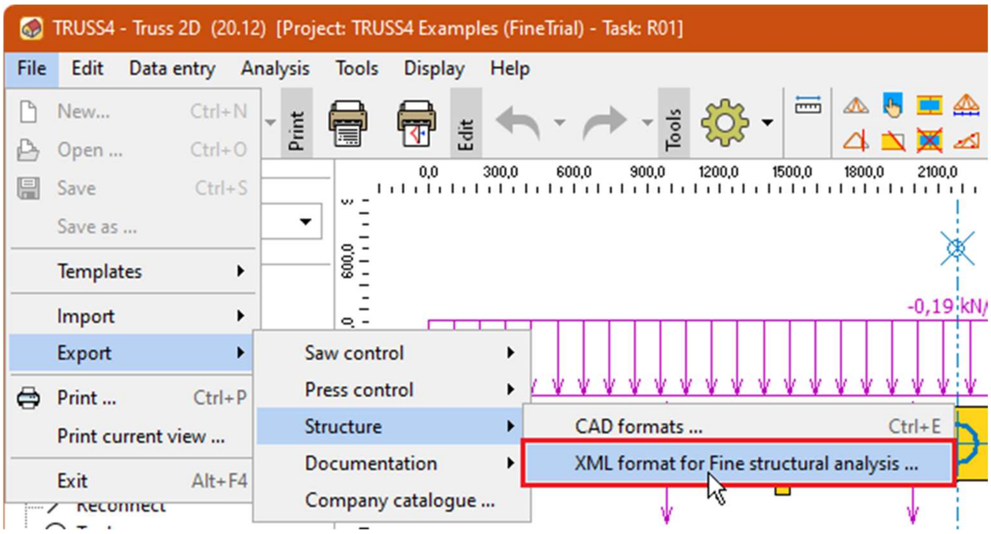
Design of the HEB steel beam

If we are limited by the headroom, it may be more advantageous to use a lower steel beam made of *HEB* profile. The "**Truss 4**" program does not offer direct design of steel elements. However, it allows the export of a structural scheme including loads and load combinations to an XML file that can be loaded into "**Fin EC**" programs for the design of general steel, timber and concrete structures. It is necessary to open the truss in "**Truss 2D**" program to export the structural scheme. This is done by clicking on the appropriate button in the toolbar above the truss table or by double-clicking on the appropriate row in the table.



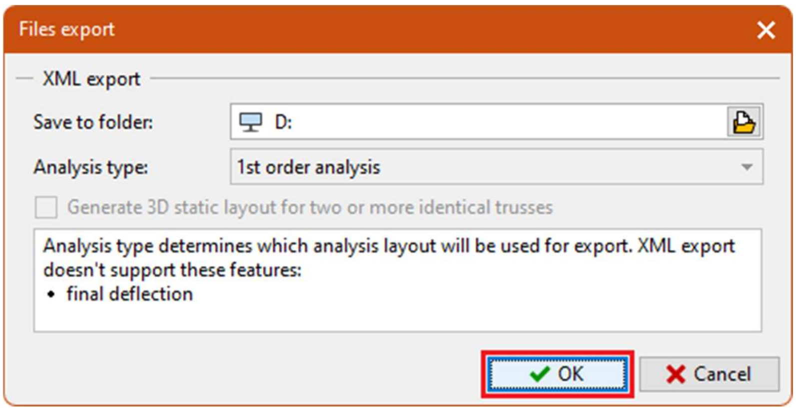
Starting the "Truss 2D" program

Export of an XML file with a structural scheme can be found in the "**File**" "**Export**" "**Structure**" section of the main menu. Since a structural scheme is exported, this operation can only be performed for a structure with results.



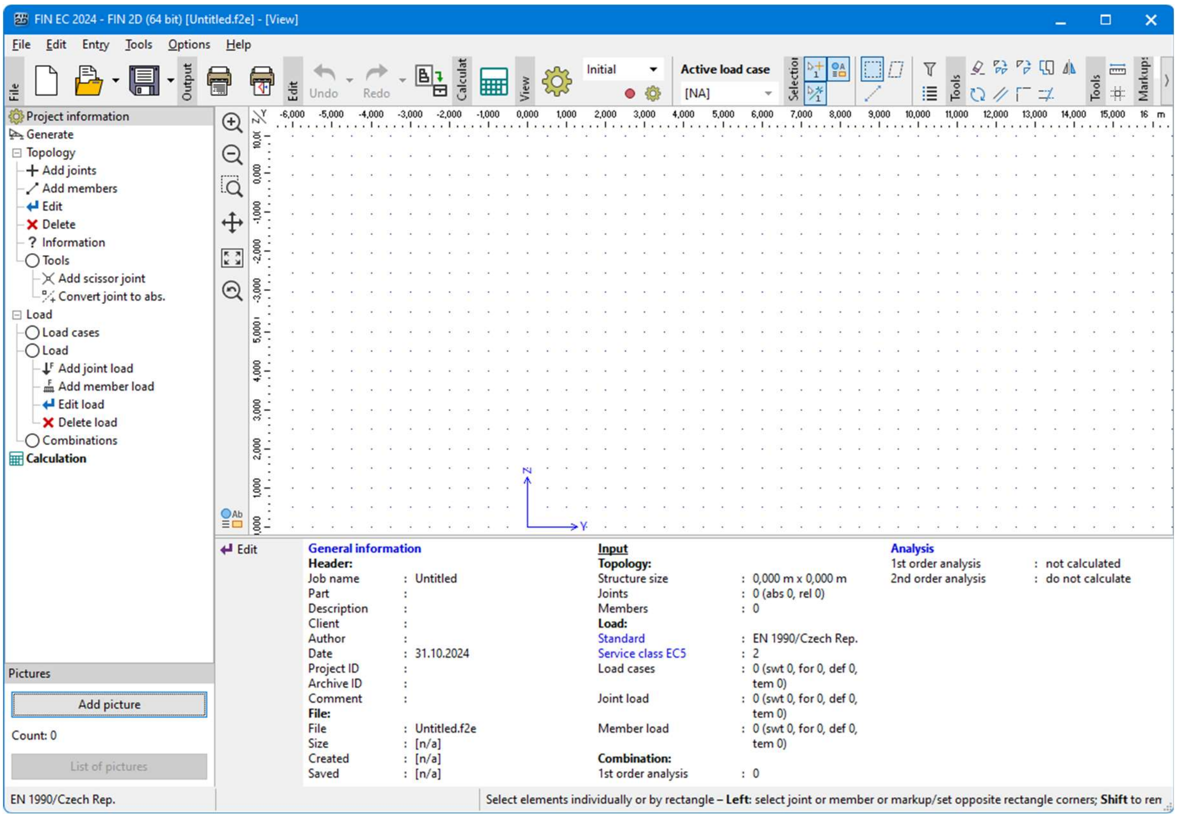
Export XML file with structural scheme

The program displays a window for specifying the folder where the **.xml* file will be saved.



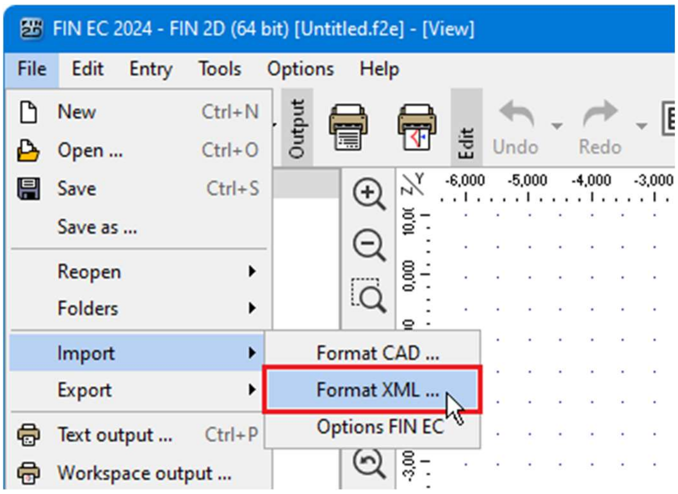
*Choosing a folder to save the *.xml file*

This is the end of our work in "**Truss 4**". Further work will be done in the "**Fin 2D**" program from the "**Fin EC**" package. Let us run this program. The user interface of the program is like the "**Truss 2D**" program. On the left we find the control tree, on the right the workspace and below it the input frame.



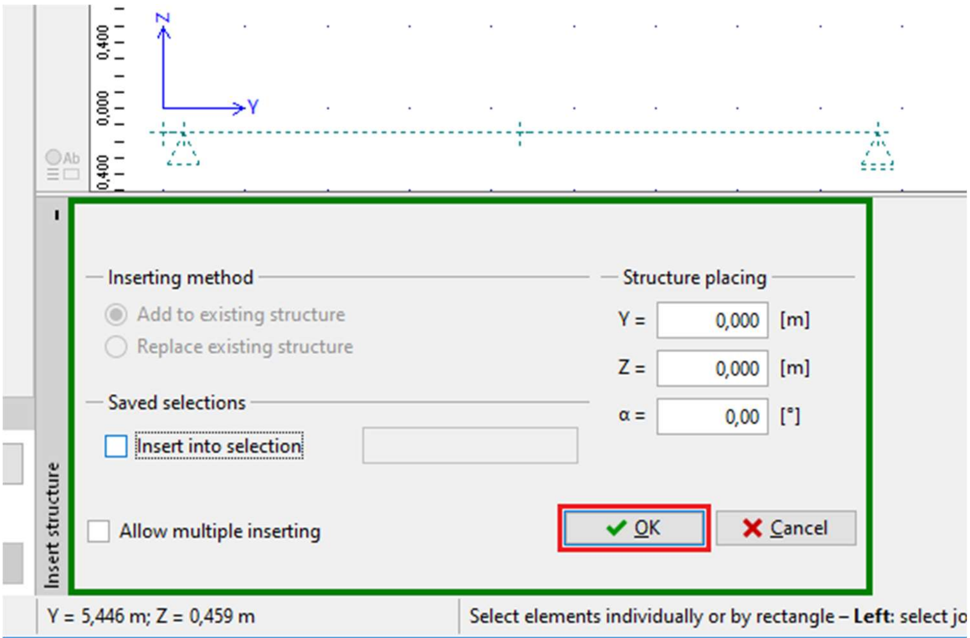
"Fin 2D" user interface

We start working in the program by importing a file exported from the "**Truss 2D**" program. In the main menu under "**File**" select "**Import**" "**Format XML...**" and select the file created in the previous steps.



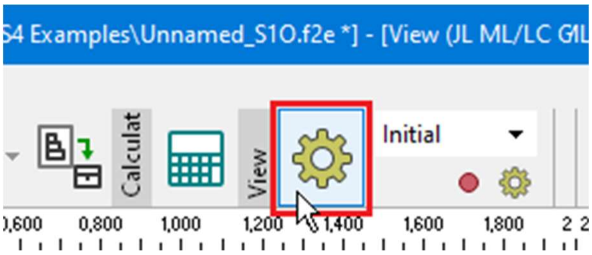
Import of XML file

After selecting the file, options that affect the insertion of the imported structure appear in the input frame. At the same time, the inserted schema is already visible on the workspace. We only confirm the insertion with the "**OK**" button.



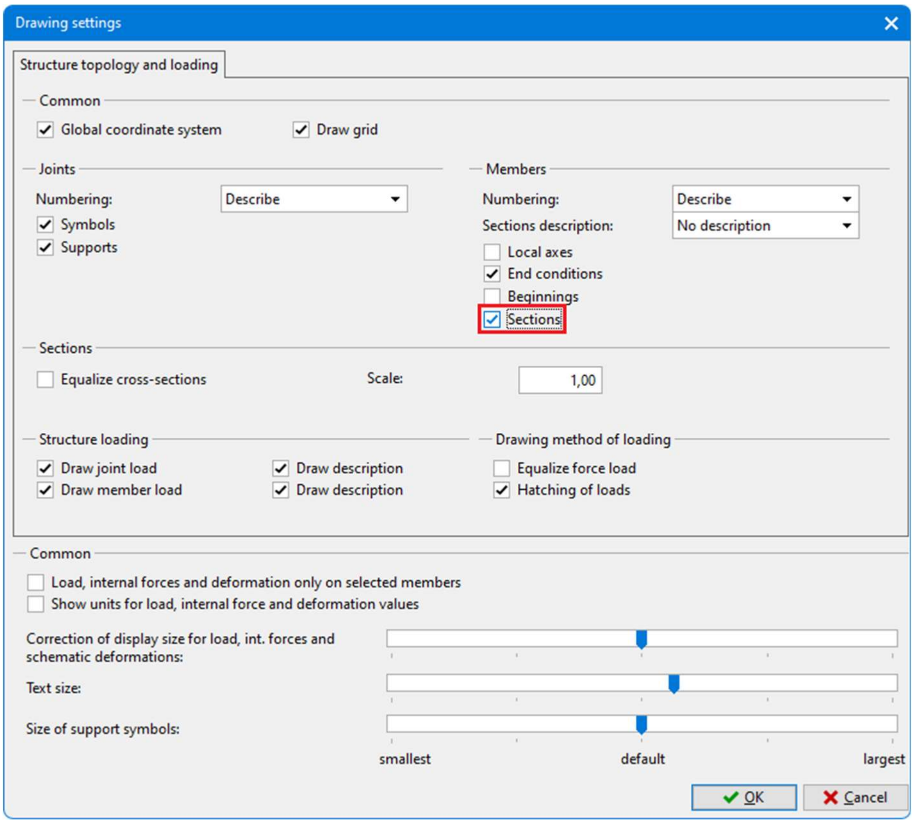
Insertion of imported structure

The structure is inserted, but we can only see the line scheme without any details. We can influence the drawing in the "**Drawing Settings**" window, which can be started by clicking the button with the gear symbol.



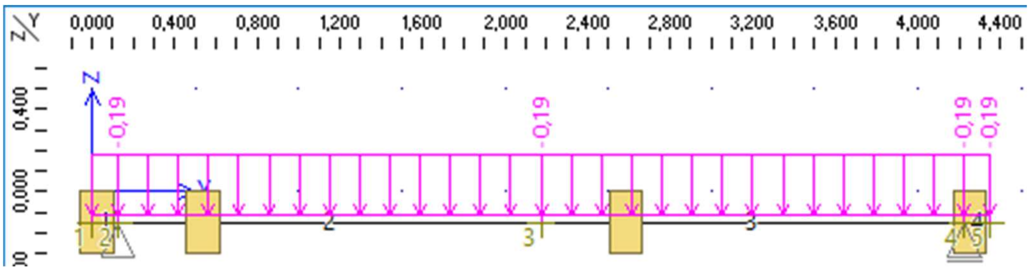
Launching the drawing properties window

In the "**Drawing Settings**" window, check the "**Sections**" box under "**Members**". This will turn on the display of the cross-sections of each member on the workspace. Close the window with the "**OK**" button.



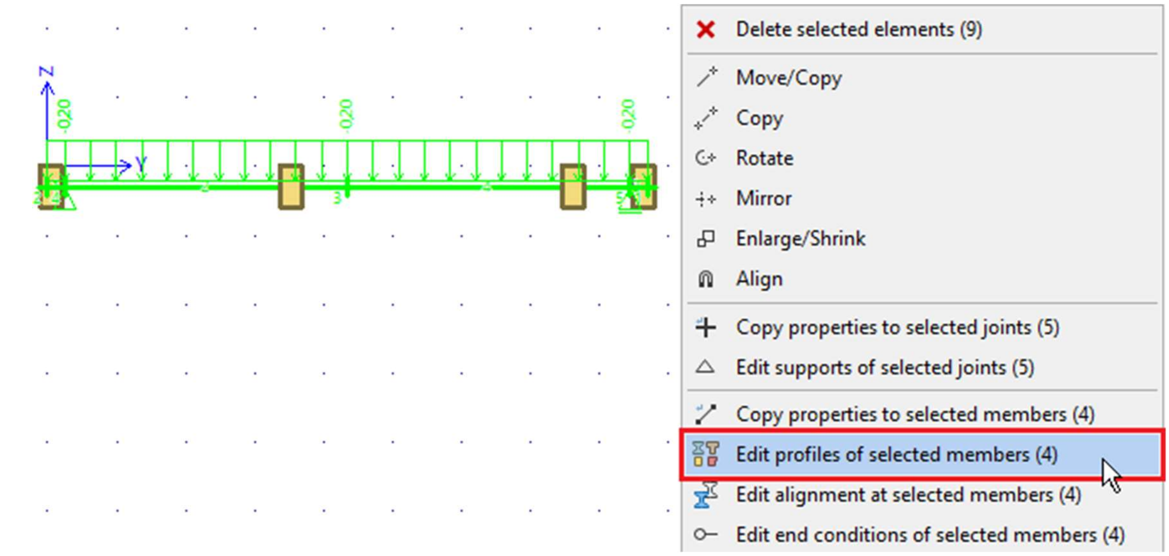
Window "Drawing settings"

Now we have a better overview of the structure on the desktop. We can see that the model is made up of four members, which connect the beam ends, the supports and the empty joint that the "**Truss 4**" program automatically created in the middle of the span. All members have the same cross-section.



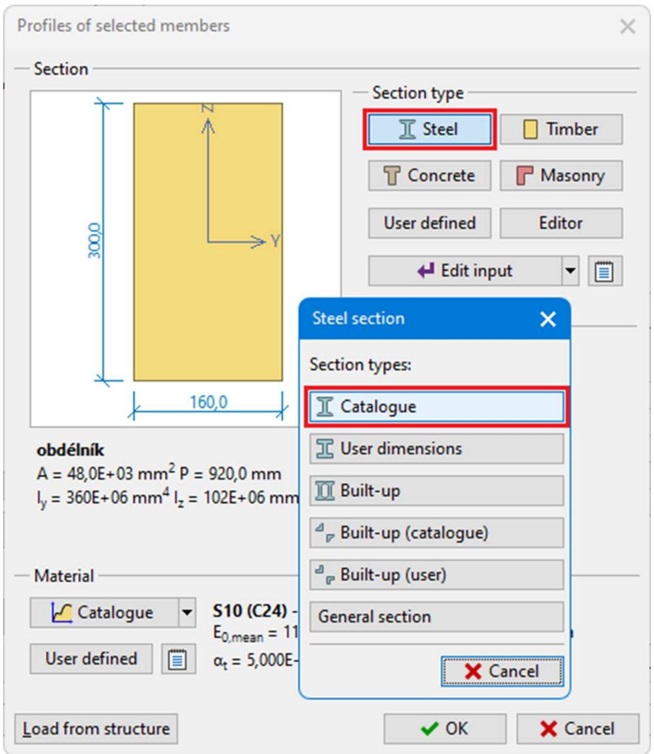
Beam model

The main change to be made is to change the cross section to a rolled steel profile. We select graphically the entire structure and right-click on the workspace to open the pop-up menu. Here we select the "**Edit profiles of selected members**" tool, which can be used to change cross-sections in bulk.



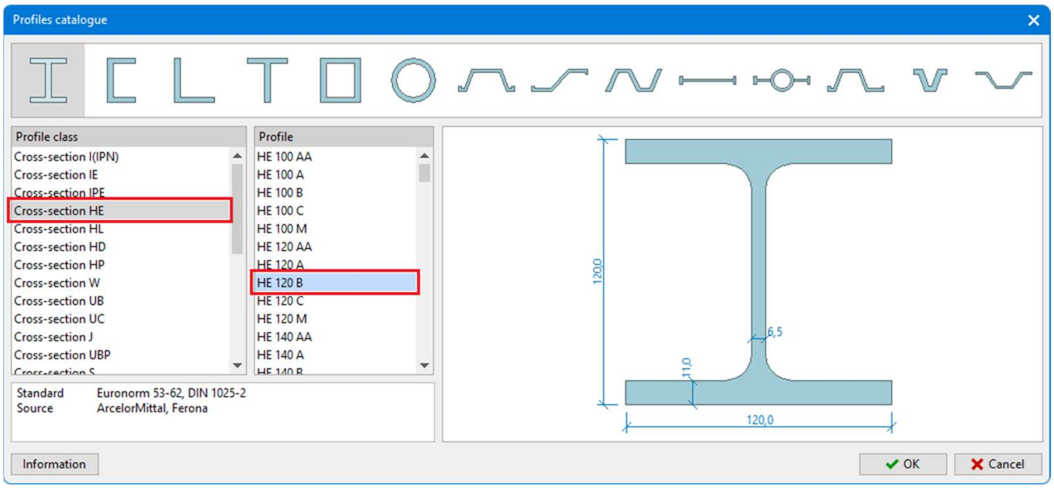
Bulk section editing tool

The "**Profiles of selected members**" window appears. Here, in the "**Section type**" section, click on the "**Steel**" button and then select "**Catalogue**" as the way to enter the section.



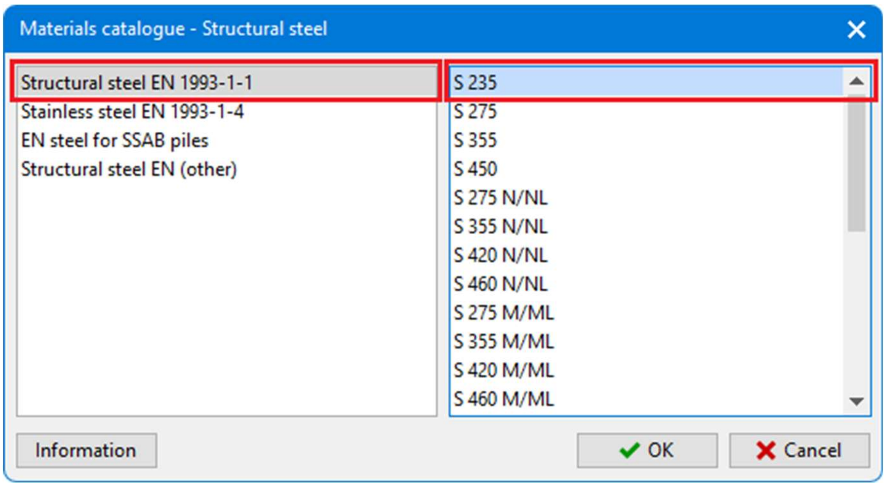
Selection of cross-section type for selected members

The cross-section catalogue will then appear. In the top bar, select the library of I-sections, in the profile class "**Cross-section HE**" and then the profile "**HE 120 B**". Confirm the selection with the "**OK**" button.



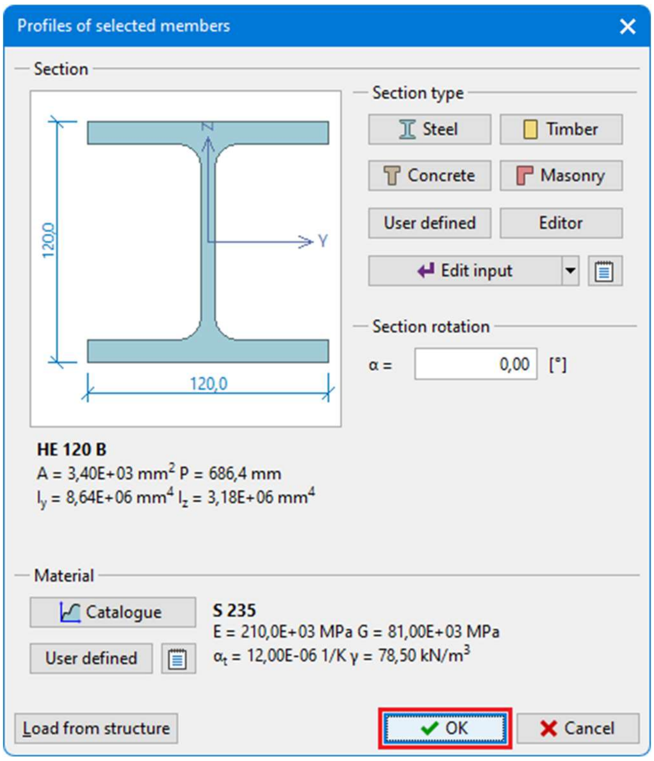
Selecting a profile

Since the previous cross-sections were made of timber, the window for selecting the steel grade will also pop up automatically. Select the library "**Structural steel EN 1993-1-1**" and the class "**S 235**". Confirm again with the "**OK**" button.



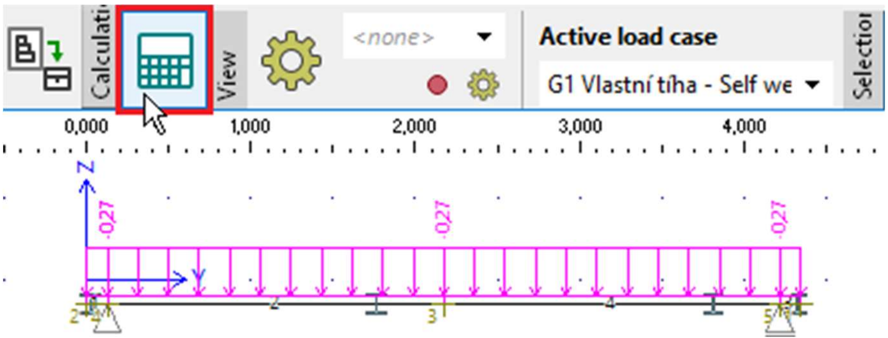
Choice of the steel grade

Now we can see the selected cross-section and material in the "**Profiles of selected members**" window. Close the window with the "**OK**" button.



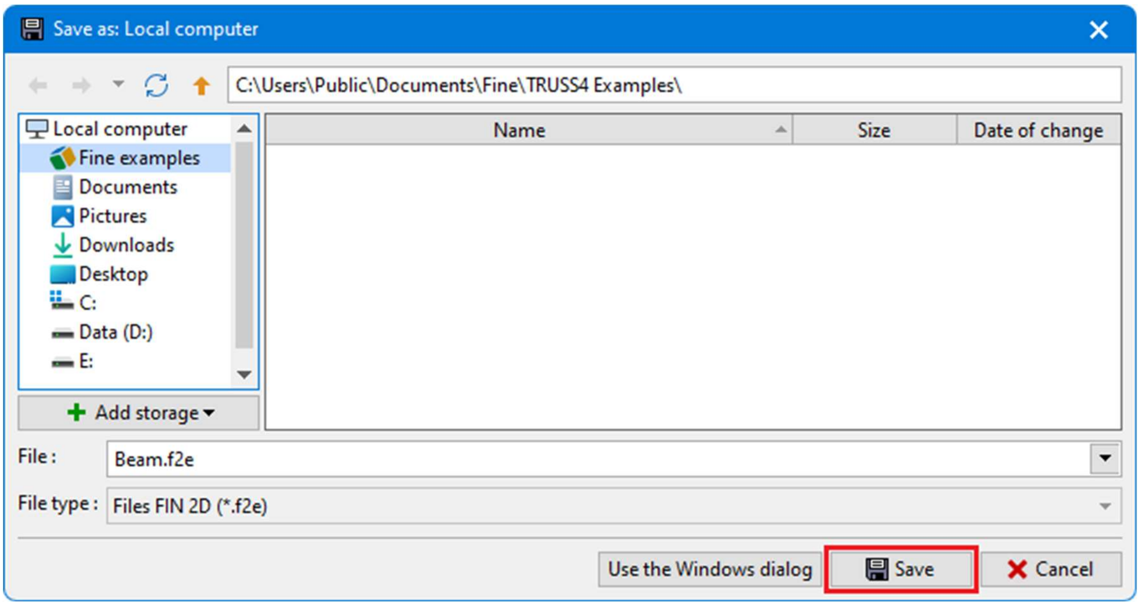
Newly selected cross-section and material

The new cross-sections are now displayed on the workspace after closing the window. We don't need to make any more adjustments, so we start the calculation by clicking the button with the calculator symbol.



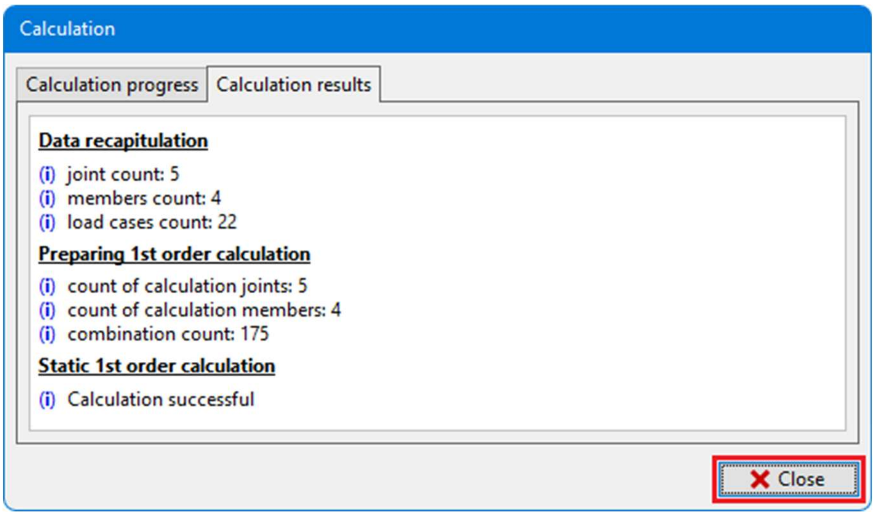
Running the analysis

Before starting the analysis, the program prompts us to save the file. Choose the file name and the folder where it will be saved. Confirm the input by clicking "Save".



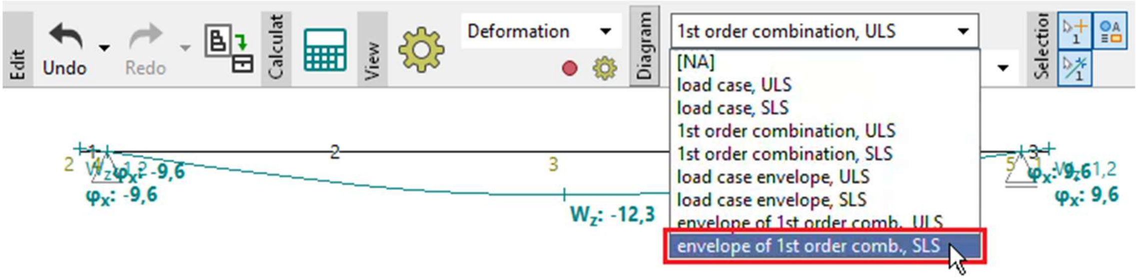
Saving the file

The calculation of internal forces and deformations is then started. At the end of the calculation, the program displays a recapitulation. After reading it, the window can be closed with the "**Close**" button.



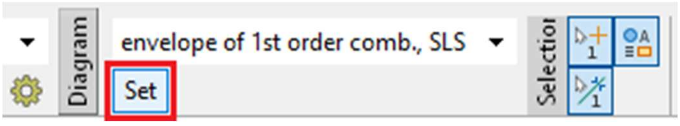
Summary of analysis

The calculation results can be viewed on the workspace after the analysis. Since we are considering a simple beam, we are primarily interested in the deformations, specifically the maximum deflection in the vertical direction. We can see the deformations on the workspace, but if we want to see the maximum value, we need to display the results for the envelope of the load combinations. So, in the "**Diagram**" section of the main toolbar, we select the "**envelope of 1st order comb., SLS**".



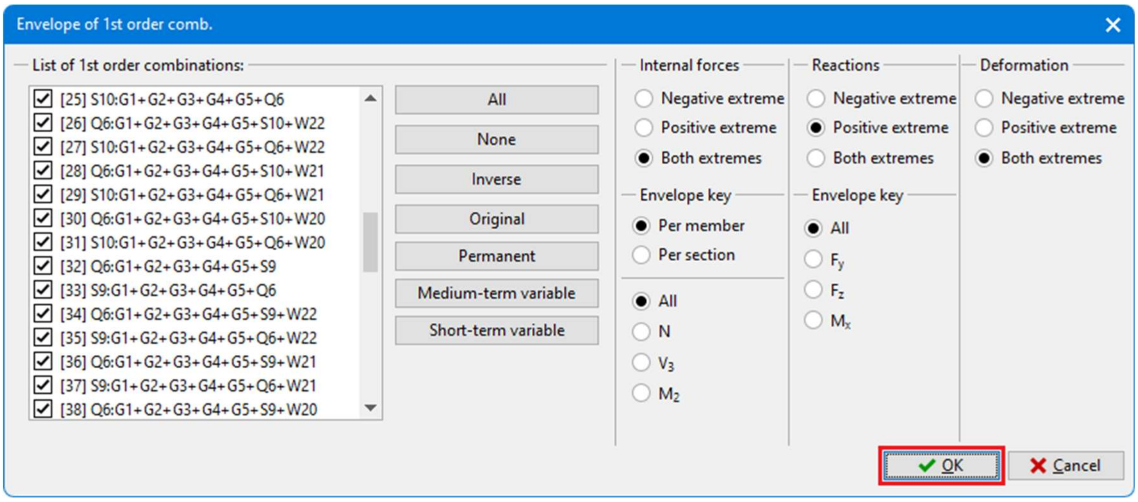
Choice of the envelope of SLS combinations

For the envelope, however, we must first specify what combinations will be included. To do this, use the "**Set**" button.



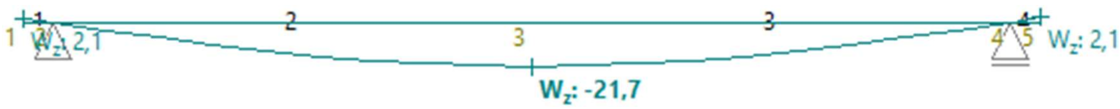
Button to enter the contents of the envelope

The "**Envelope of 1st order combinations**" window opens. Here we check that all combinations in the left column are checked and will therefore be reflected in the envelope. Then just close the window with the "**OK**" button.



Properties of envelope

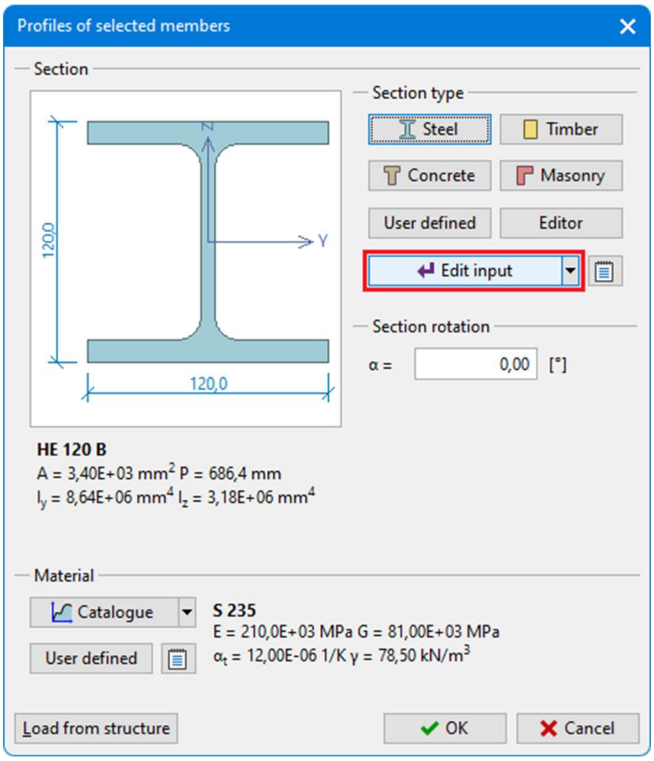
The updated deflection value, *21.7mm*, is then displayed on the workspace.



Deformation for a SLS combination envelope

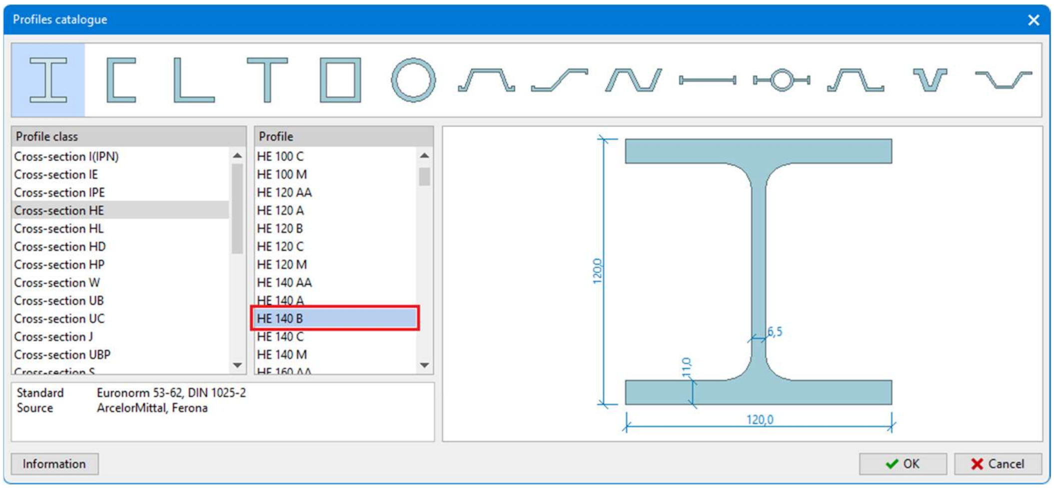
We have a *4100mm* beam span and we want to meet the limit deflection of *1/300* of the span. This is *13.7mm* in this case. We can see that the value on the workspace is higher. Therefore, we must choose a larger cross-section. Again, we select all the members and use the "**Edit profiles of selected members**" tool in the pop-up menu. However, in the "**Profiles of selected members**" window, if we just change the dimension within the same cross-section type, we can use a quicker procedure using the "**Edit input**" button.

TRUSS4



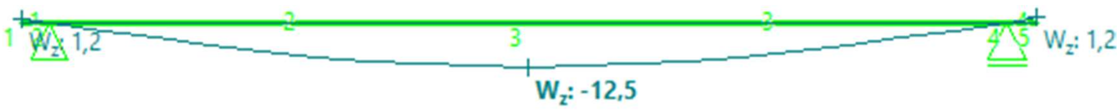
Cross-section changes within the same type

The catalogue of predefined steel cross-sections is displayed straight away. This time we select the cross-section "**HE 140 B**".



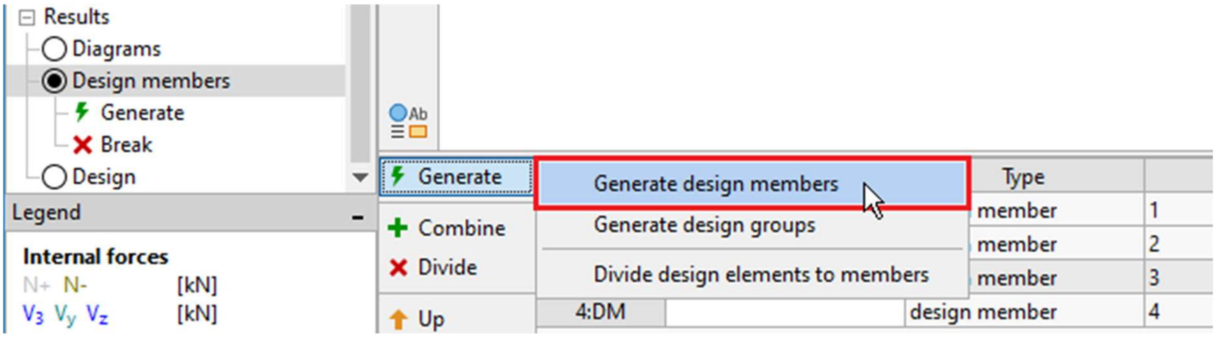
Changing the cross-section dimension

After recalculation, the maximum deformation for the SLS combination envelope is equal to *12.5mm*, which is less than the limit stated above.



Maximum deflection of the beam with modified cross-section

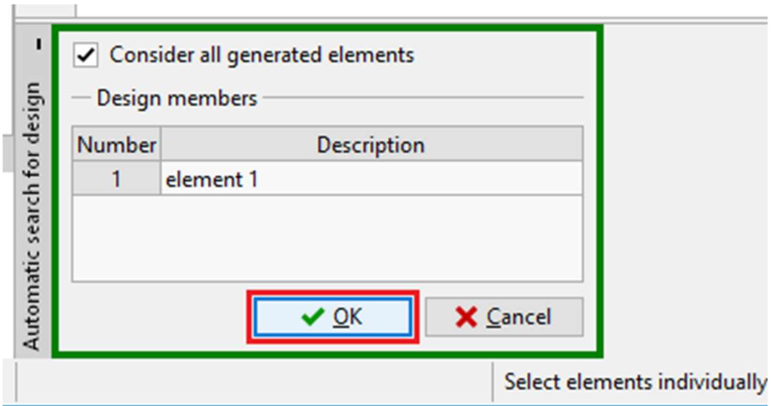
This verified that the criteria given for the serviceability limit states are satisfied. It is now necessary to verify the ultimate limit states, i.e. the load capacity of the beam. In order not to have to assess four interconnected members separately, we first merge them into a so-called "**design member**". Go to the "**Design members**" section of the control tree and press the "**Generate**" button next to the table of design elements. A menu will pop up, from which we select the option "**Generate design members**".



Creating a design member

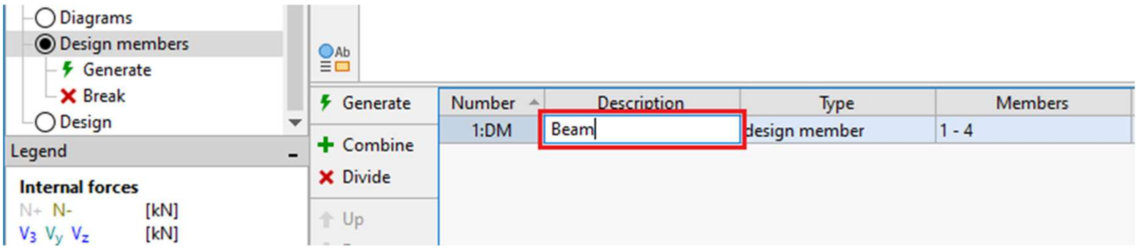
Design members and design groups
The program offers the following options to reduce the number of members to be analysed:
Design members *can be used to merge members of the same material (timber, steel, concrete) that lie in a straight line and are connected to each other. The merged members can have different cross-sections. A single member with a length equal to the sum of the lengths of the individual members is then transferred to the design program instead of several members. The parameters required for the design can then be assigned together to all elements in the design member. Since only the calculated internal force curves are transferred to the dimensioning programs, even members that are connected only by joints or other similarly loose connections can be combined.*
Design group *is used to merge identical members that should be treated as one element during analysis. Members (or design members) having identical cross-section, length and orientation may be combined into a design group. The design group is transferred to the designing program as a single member, but the number of load cases to be considered is multiplied by the number of members in the group.*

The input frame then displays the information that the program has found one possible design member. Confirm the information with the "**OK**" button.



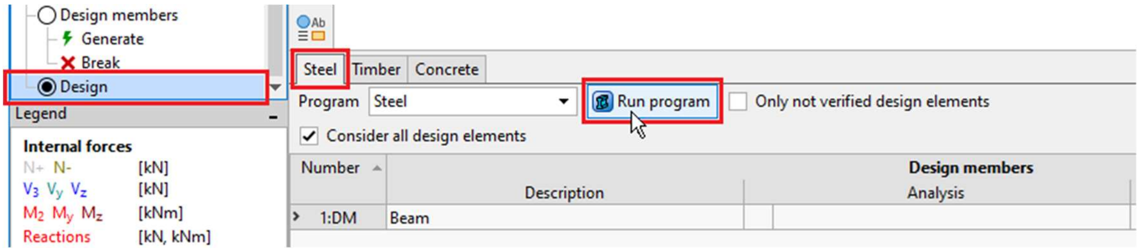
Confirmation of creating a design member

There is now only one row in the element table because the four members have been merged into one. In the table we have the option to enter member name.



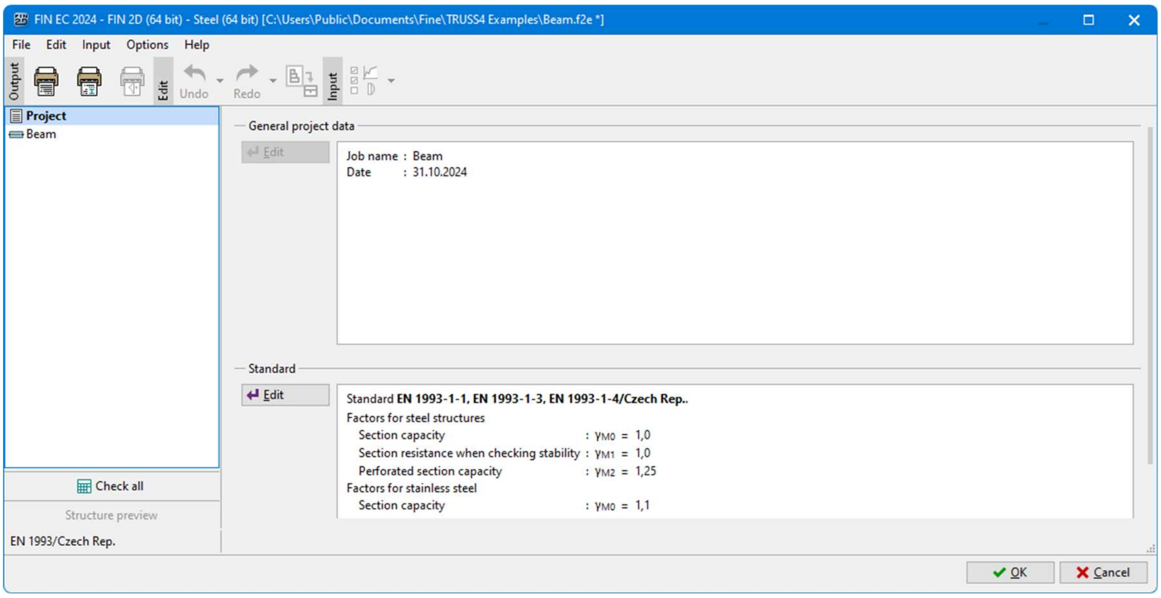
Naming of a member

Now we can move on to the "**Design**" section of the control tree, where the actual member design takes place. In the input frame, select the "**Steel**" tab and then use the "**Run program**" button to transfer the beam to the "**Steel**" program.



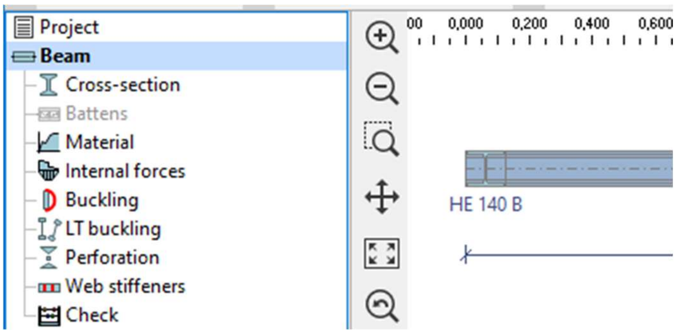
Starting the "Steel" program

In the "**Steel**" program, the members to be checked are arranged in the tree on the left side of the window. The first node "**Project**" contains the possibility to enter project identification data and also to select design standard parameters.



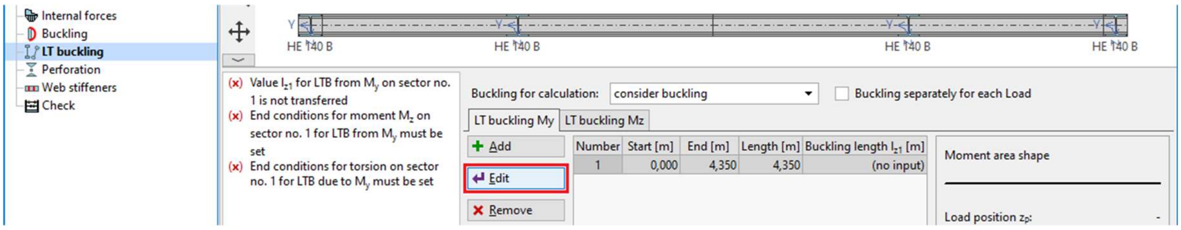
Basic screen of the "Steel" program

After clicking on our beam in the control tree, the list of data important for the verification is expanded. Most of the data ("**Cross-section**", "**Material**", "**Internal forces**") is taken from "**Fin 2D**". Other parameters must be added. Since we are designing a transversely loaded section without axial force loading, these are only the lateral-torsional buckling parameters.



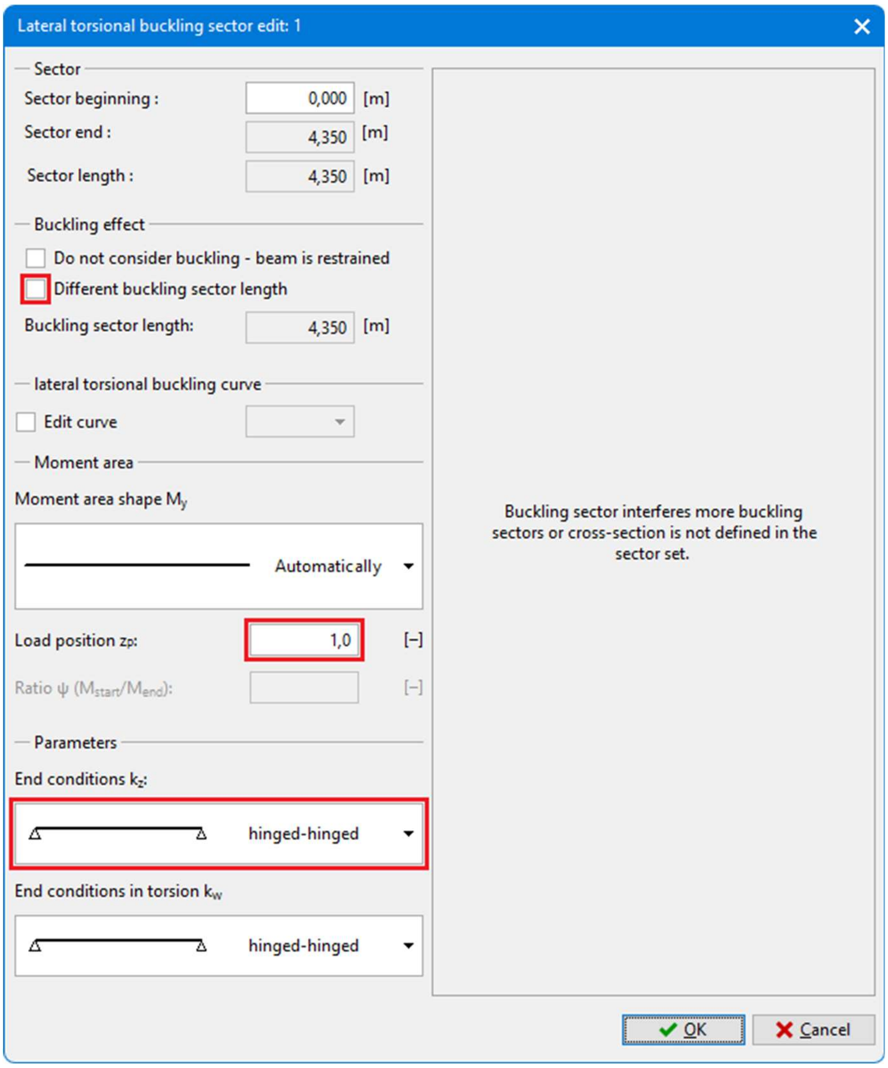
Expanded member properties

We go to the "**LT buckling**" section. Regarding the stresses, we will only be interested in the LT buckling caused by the moment M_y. The buckling properties can be specified differently along the length of the member, but we will specify the same parameters for the whole length. Therefore, we will only use the initial buckling segment, which is already specified in the table. The "**Edit**" button opens the properties window.



Button for modifying LT buckling parameters

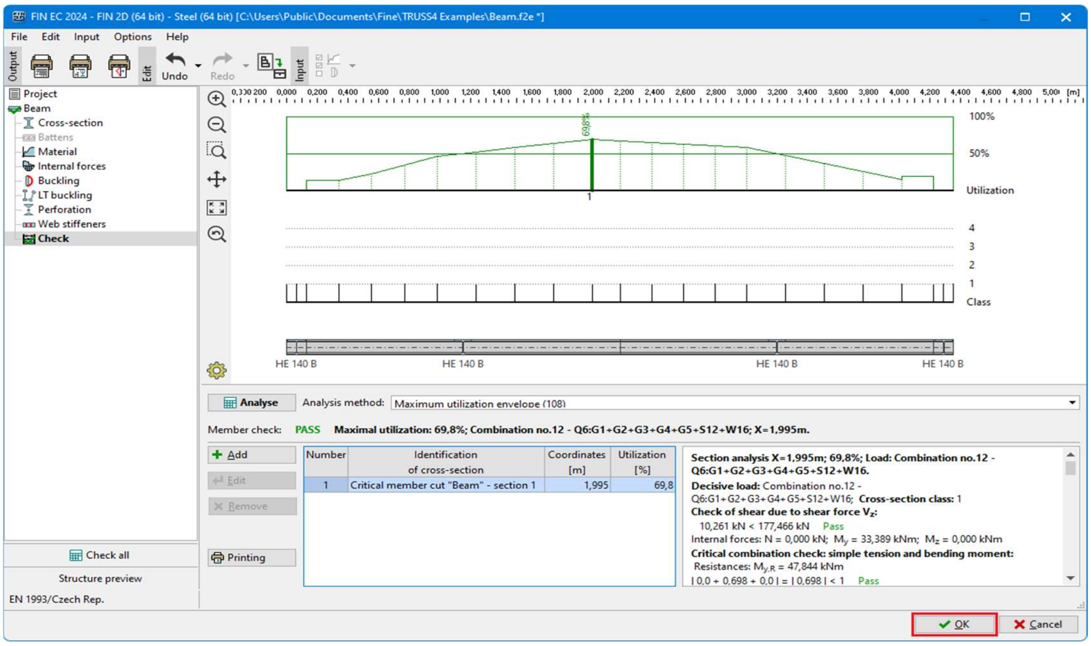
The "**Buckling section length**" represents the basic buckling length over which the member can buckle. We conservatively choose the same length as the length of the member because we are not sure of the stiffness of the connections between the trusses and the beam. In the case of a relatively stiff HEB section, this simplification will not have a significant effect on the result. "**Load position z_p**" specifies the position of the load along the height of the section. In our case, the load is applied to the top flange, so we enter a value of *1.0*. Next, we select the end conditions, selecting the "**hinge-hinge**" option.



Modified buckling parameters

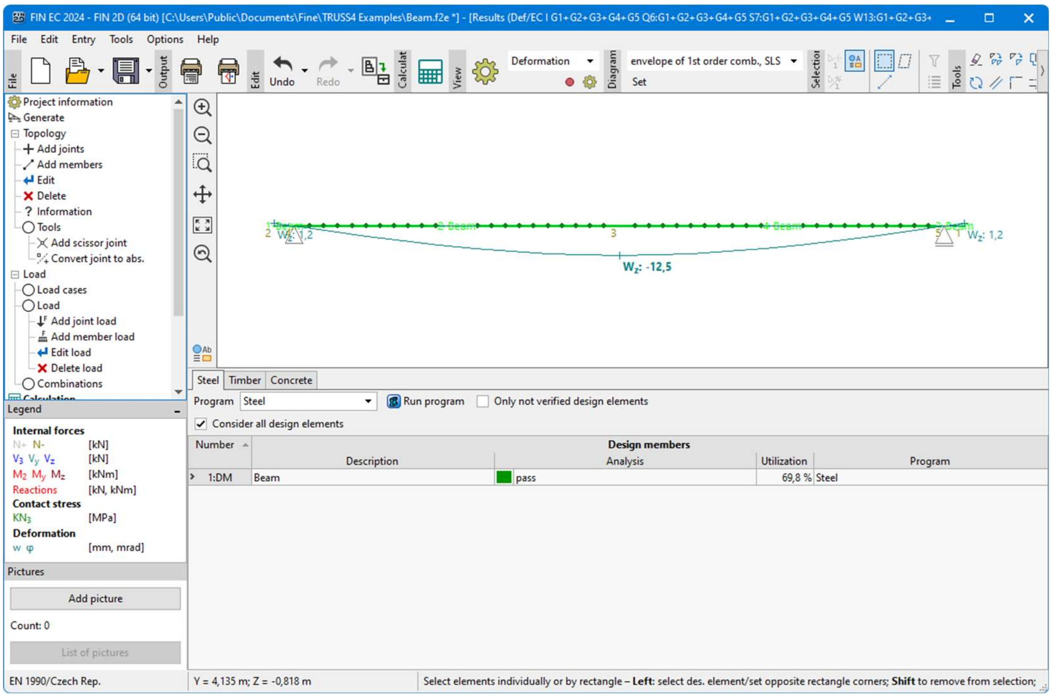
Now we can go to the "**Check**" section where we can see the results of the analysis. Our beam passes at *69.8%*.

TRUSS4



Analysed beam

If you do not want to print the analysis documentation, you can go back to "**Fin 2D**" using the "**OK**" button in the bottom right corner of "**Steel**" program. The basic design results are displayed in the element table.



Verified beam in the "Fin 2D" program

For more engineering manuals visit https://www.finesoftware.eu/.