Engineering Manual Nr 01

Updated 09/2022

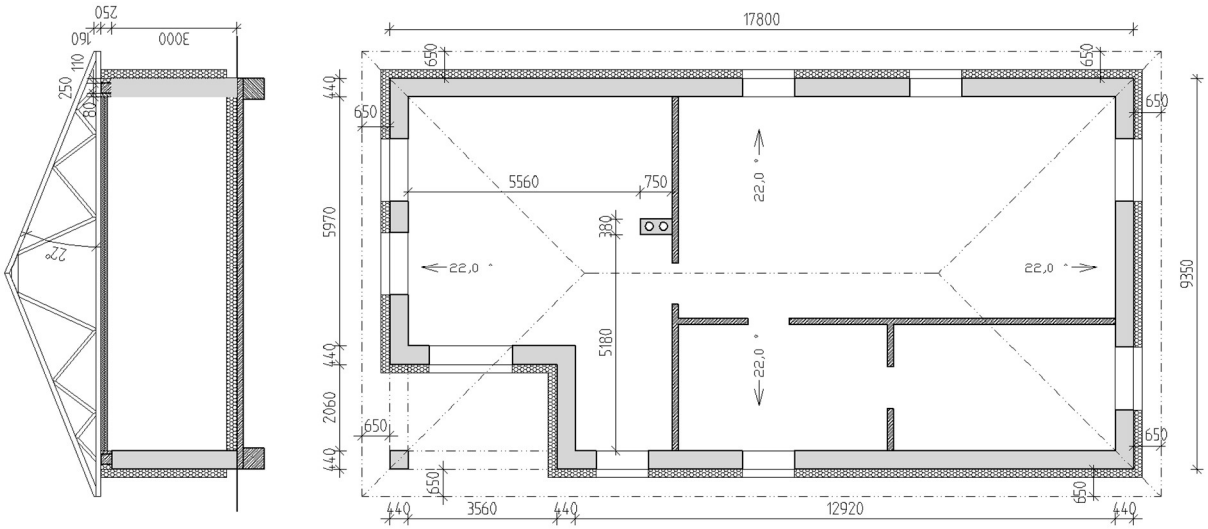
Design of a Hip Roof

Input

Program: Truss 3D

File: FineTrial.tr3

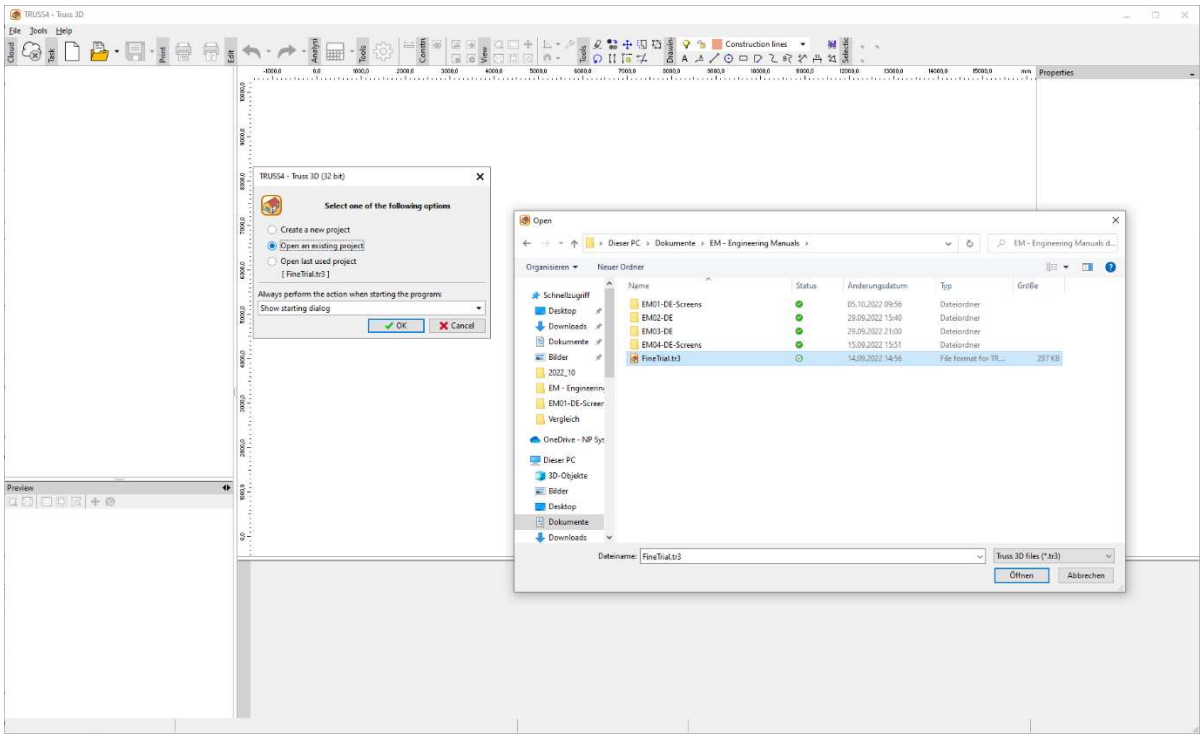
We want to design a roof with trusses acording to the given plans – plan view and section. The roof cover is made up with clay tiles, the ceiling covered with plasterboard. The building site is located in snow zone II and wind zone II. The structure will be modelled in Truss3D and then designed. We will create the documentation and export files for a CNC saw.



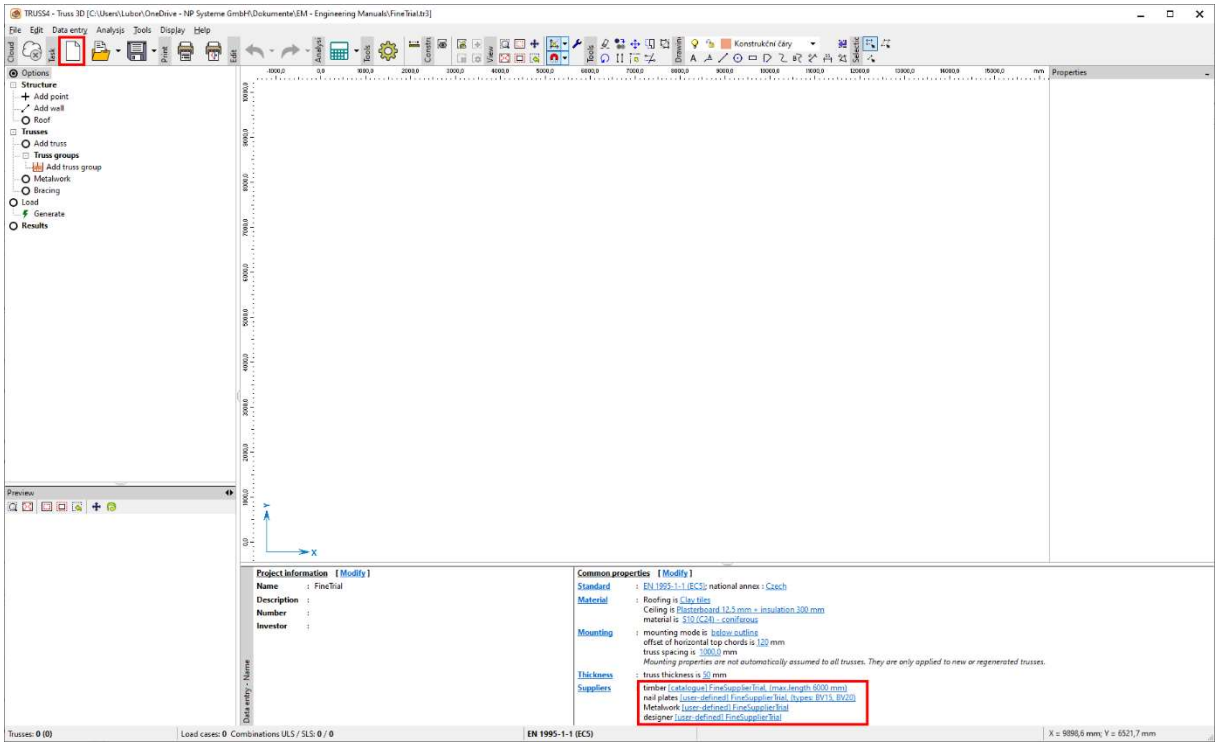
Creating a new project

We run the program Truss3D.

For the good of this example, we first open the prepared file “FineTrial.tr3” so we get corresponding catalogues for timber and plates.

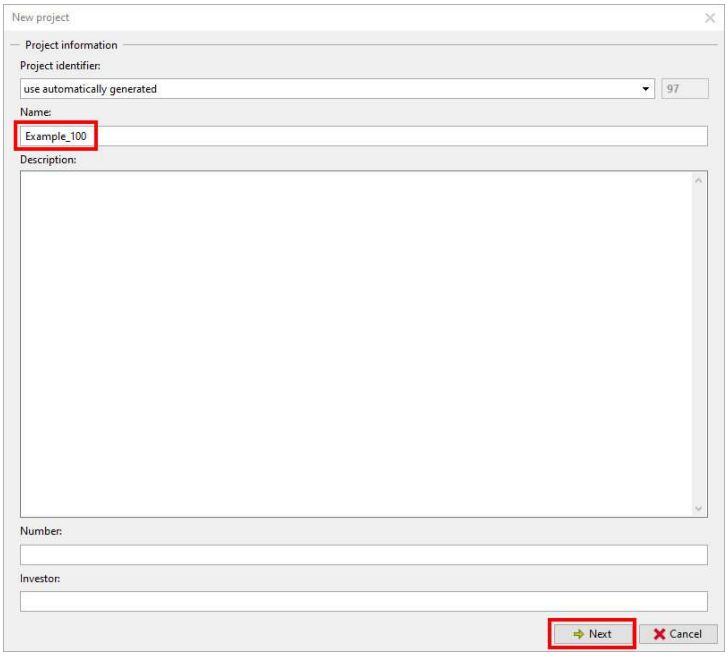


By opening the file “FineTrial.tr3” we received the required catalogues and now can create a new project via the symbol “New”.

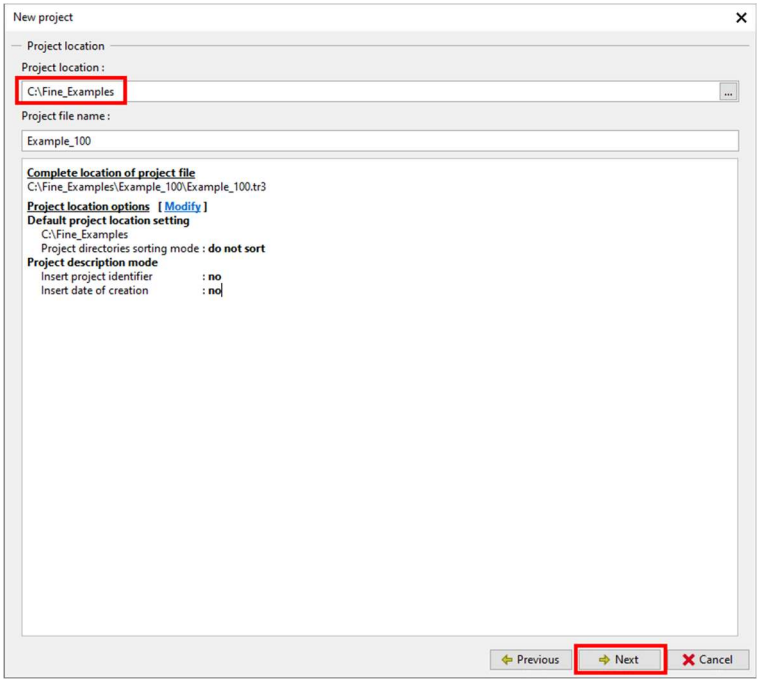


Creating a new project automatically runs the input wizard.

First, we name the project “Example_100” and continue with “Next”.

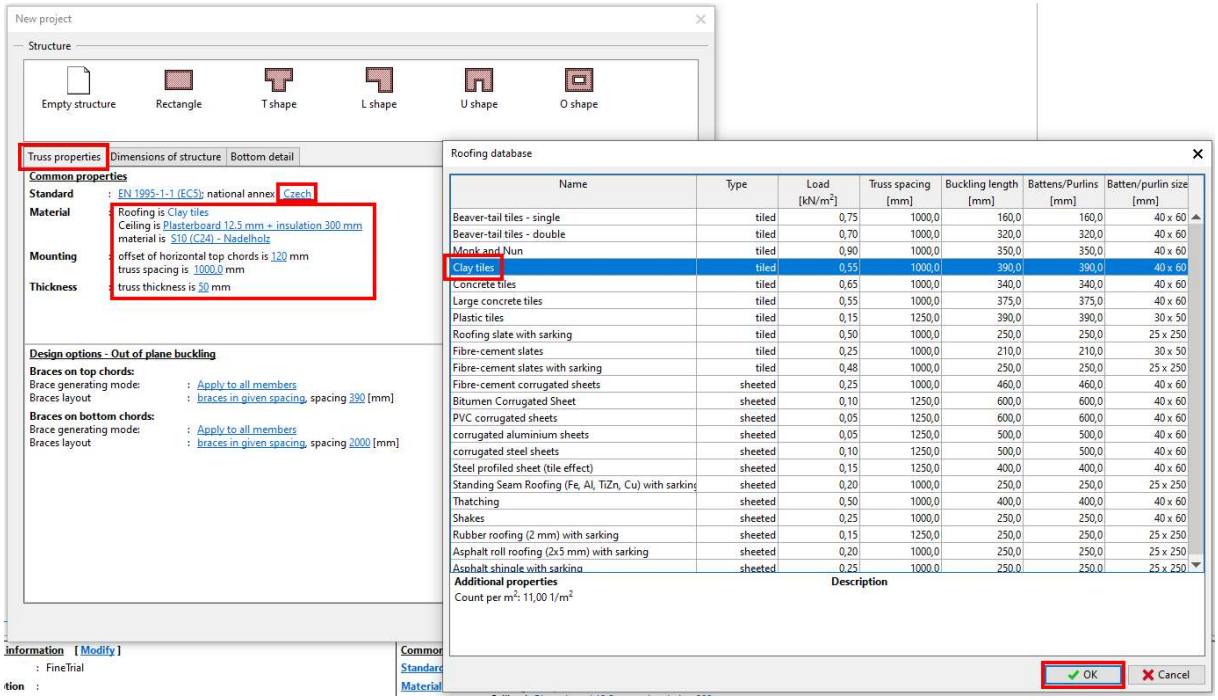


The path to the saving location is being displayed which in case can be modified.



Using a predefined structure from the wizard

On the tab “Truss properties”, we choose the national annex, the material for the roof cover and the ceiling, the truss spacing and the timber thickness. Underlined text in blue colour indicates hyperlinks and reacts upon left mouse clicks.

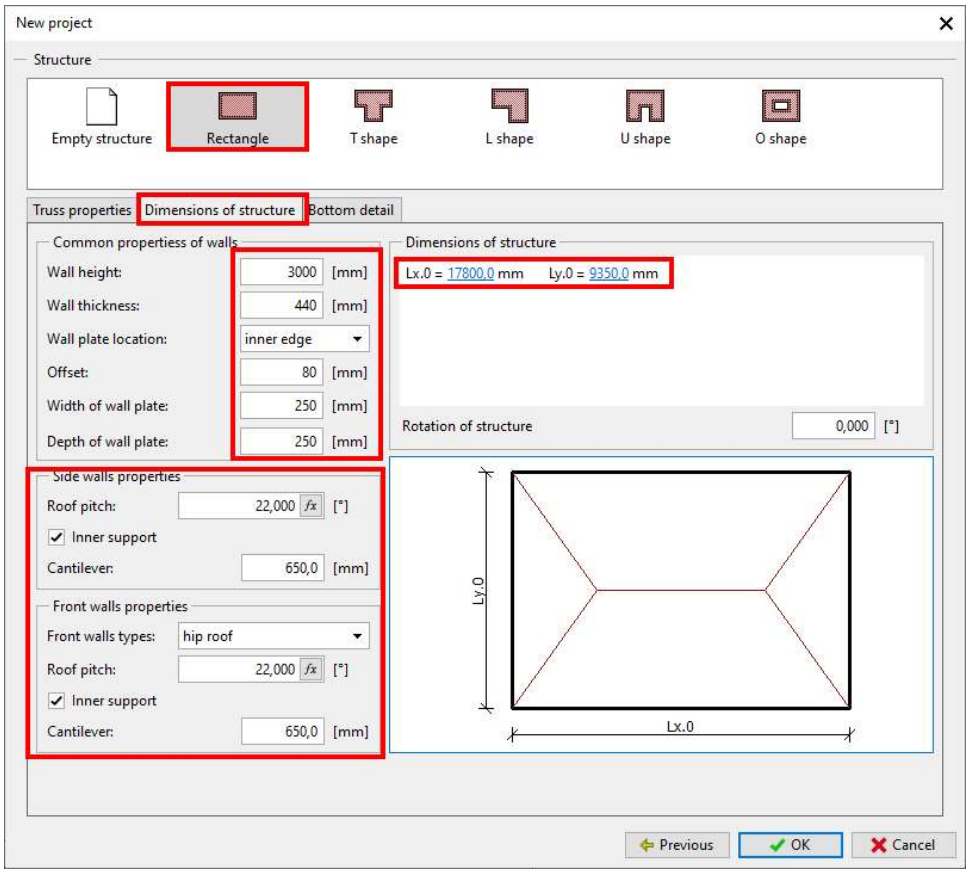


We select “Rectangle” as the structure shape, as it represents the task to be solved best.

On the tab “Dimensions of structure” we enter the geometrical parameters of the object.

NOTE: If you should close the wizard too soon by clicking “OK” (before all input fields were set with the correct values), then you can’t re-open the wizard again. You must enter the required values in the corresponding program parts (Common properties, Structure/Wall properties).

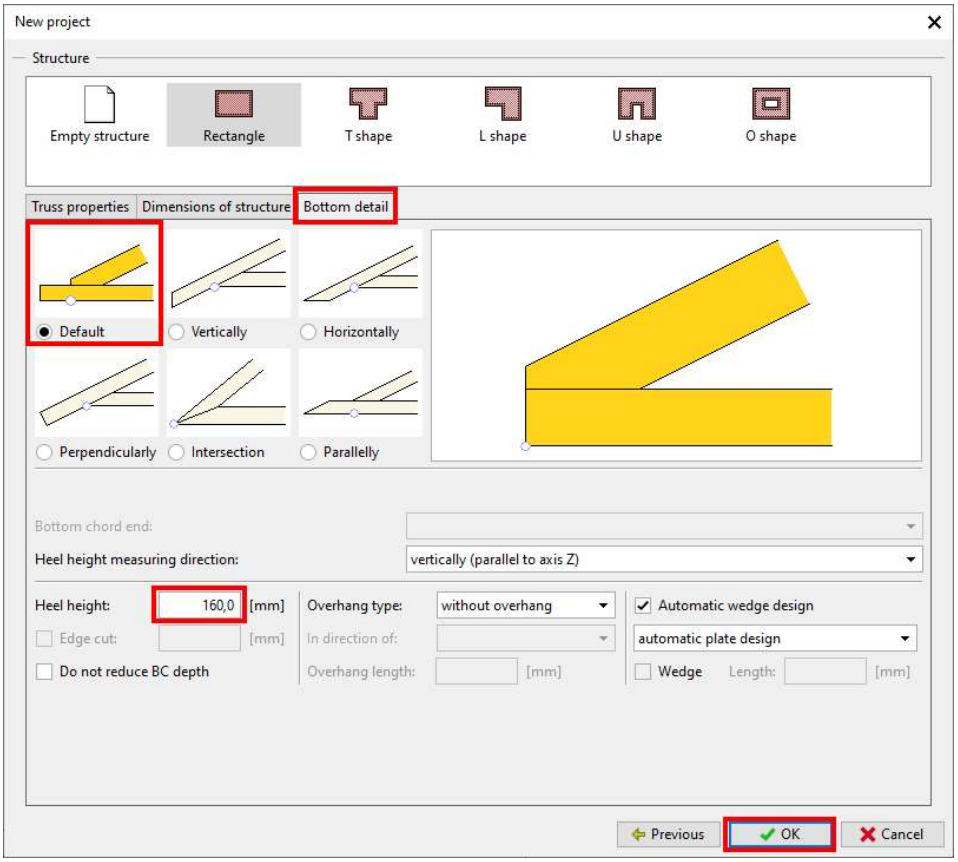
The entered values respect the given ones (plan view and section through building), which in this case is 17600 mm x 9350 mm, roof pitch 22°, cantilever, ... etc. See picture below.



On the tab “Bottom detail” we select the heel type and heel height, here 160 mm.

NOTE: Roof planes are always defined above walls. For that such properties (roof pitch, overhang, cantilever, bottom detail etc.) of individual walls can be edited under wall properties.

We close the wizard by clicking “OK”.



The user interface of TRUSS3D

When the project has been created successfully the program automatically creates the basic shape of the structure.

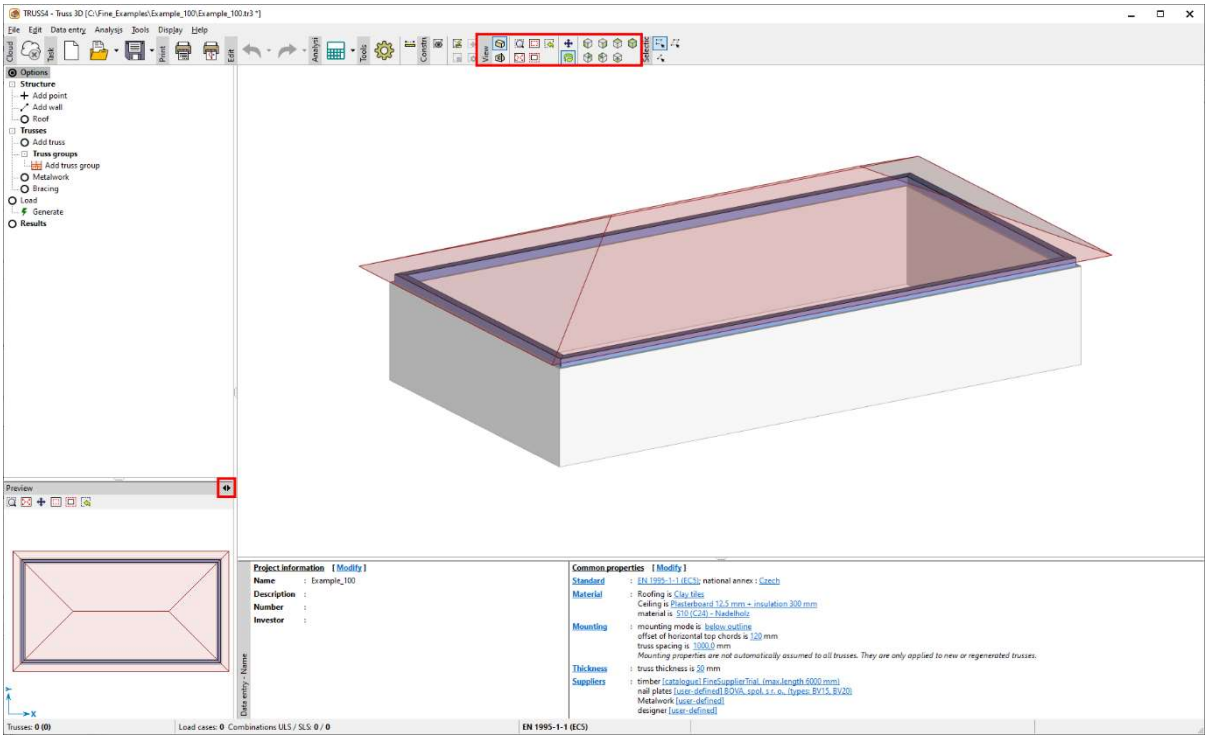
The user interface of TRUSS4 contains the typical menu and tool bar. On the left you will find an options window with a tree structure, whose items approximately correspond with the order of the work flow. Underneath the options window is a secondary work space. On the right you find the main work space with a table below it which can be hidden and re-opened with the function key “F11”.

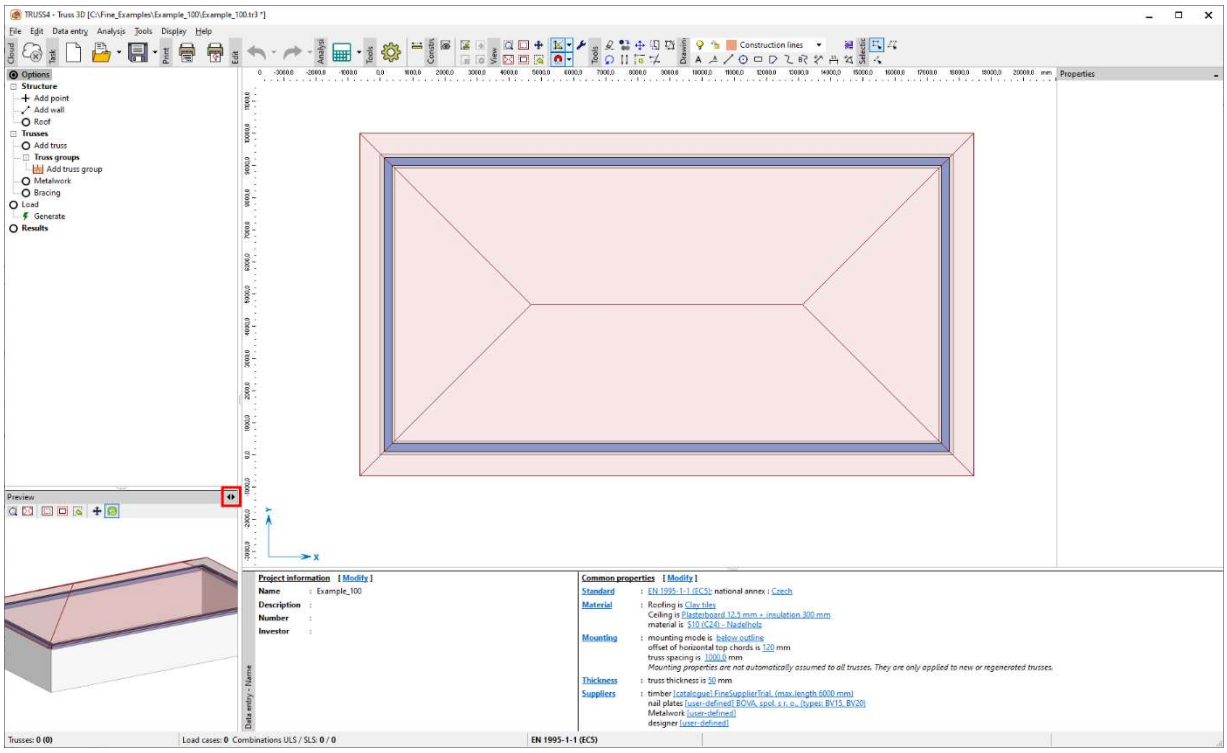
The main and the secondary workspace swap the 2D and 3D display of the structure. This can either be done by clicking on the <> arrow in the upper right corner of the secondary workspace window or by using the key combination <Ctrl+Tab>.

Both work spaces are fully active, in both of them objects can be selected and edited.

The view at the structure can be changed by using the symbols in upper icon bar 

and the mouse, respectively: zoom – mouse wheel, hover – hold down mouse wheel and move the mouse, rotate view – hold down mouse wheel and <Ctrl> key and move the mouse.





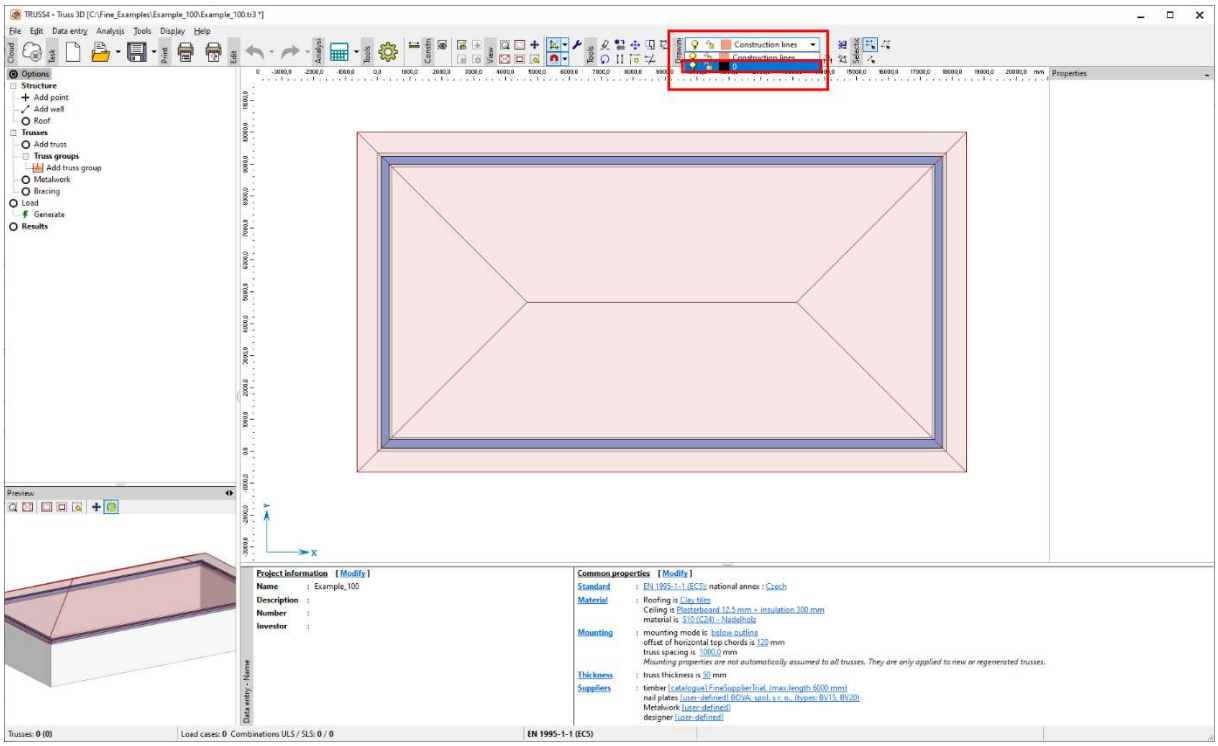
Graphic input and construction lines

We will create the further parts of the structure according to the specification by adding them to the existing rectangular shape. The input and editing can both be done in a numerical/tabular entry of coordinates and graphically in the “Structure” section of the options windows. In this example we will use construction lines.

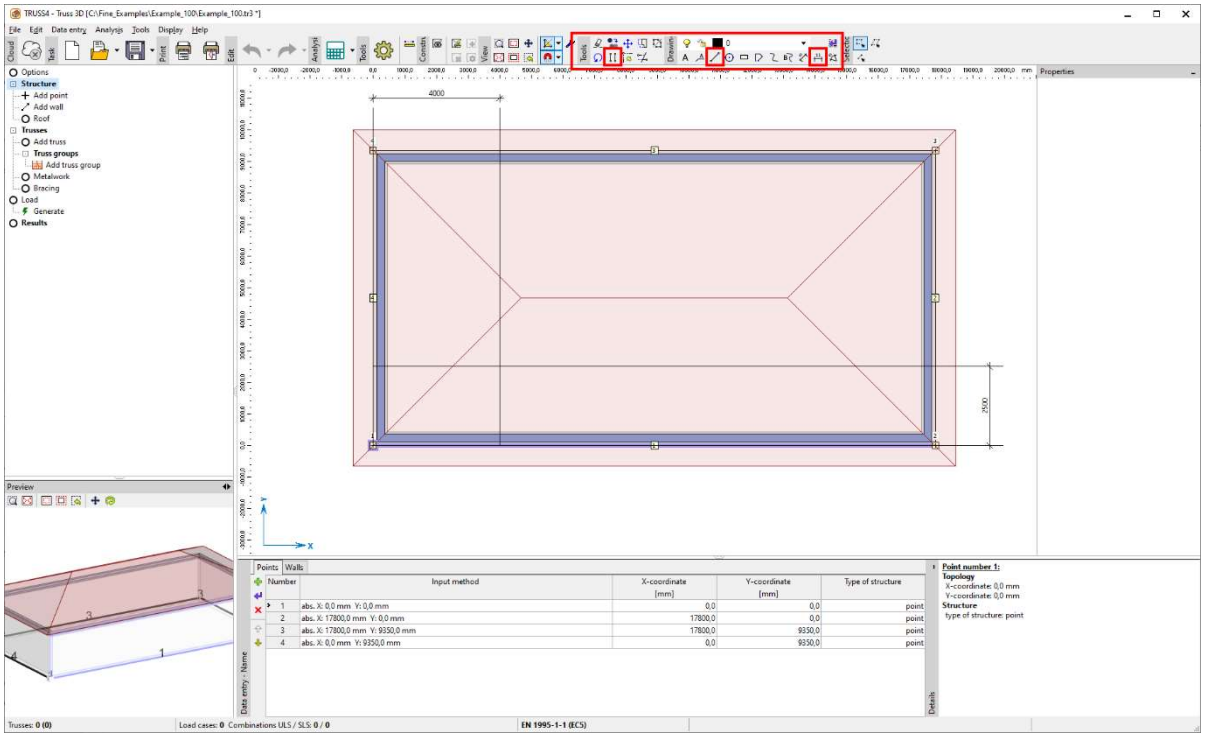
The graphical input is available in the 2D view only!

The tools of the graphical input meet the standard CAD programs – line, circle, text, dimensions etc. are available. Further you can use individual layers with different properties (printable yes/no, colour, line type, etc.) and snap points and polar direction snap.

NOTE: Objects such as construction lines, walls and trusses can be deleted by selecting them and hitting the <Del> key.



For the lines we want to draw we choose the black colour by selecting the corresponding layer. We draw lines and parallels which meet the specification.



Structure – Points, Walls and Roofs

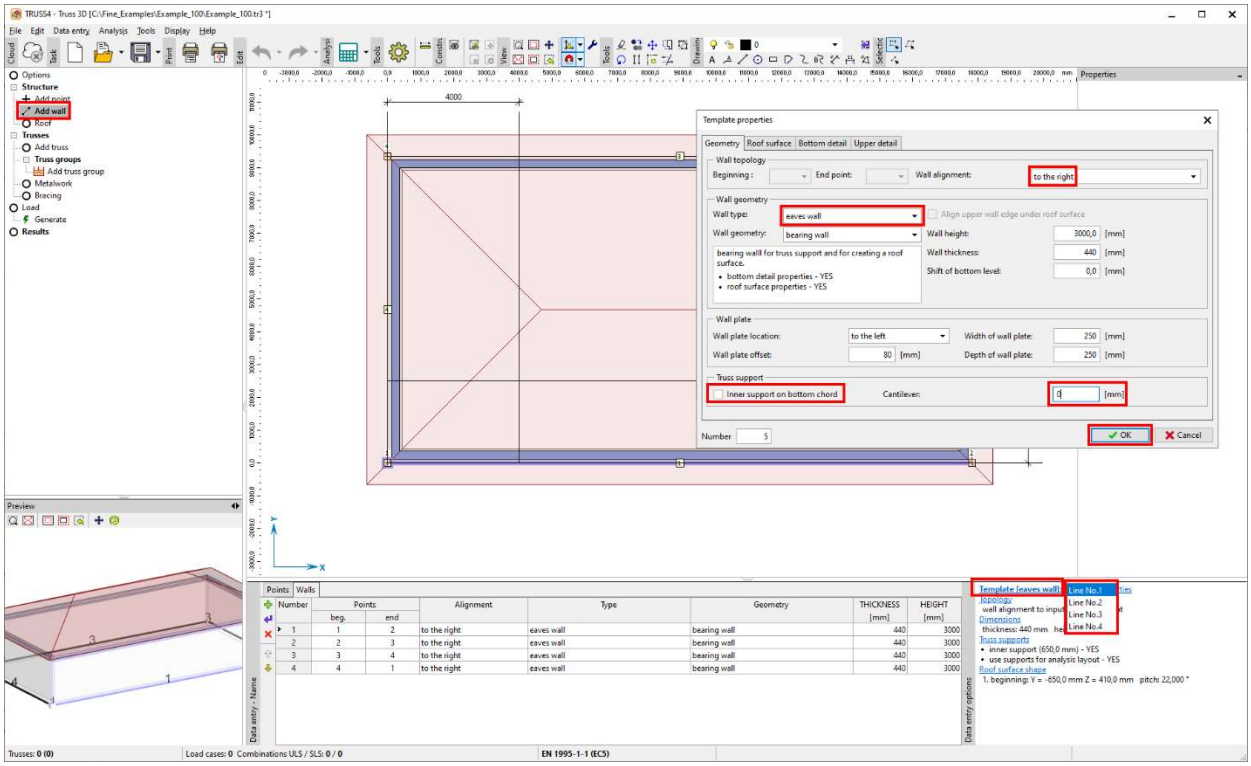
In the options window in section “Structure” select “Add wall”.

Each wall has their own parameters which define their geometry and eventually that of the roof planes.

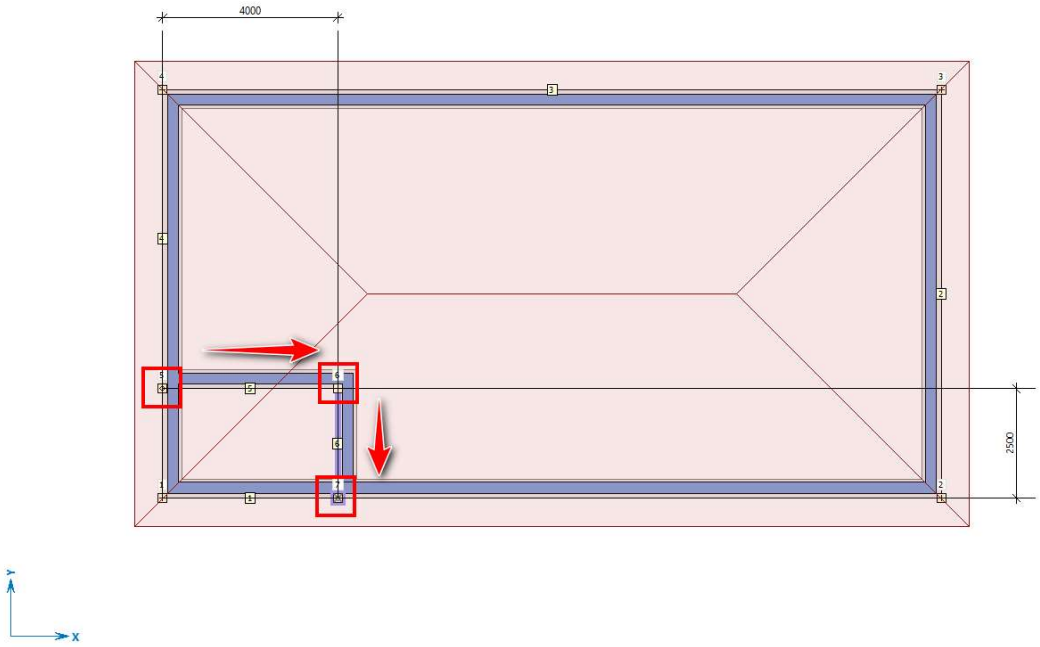
NOTE: The wall type §eaves wall" defines a roof plane, "support wall" on the other side doesn't (e.g., gable wall or inner supporting wall).

In order to avoid entering all wall properties again, in the right part of the lower table window we can use the properties of existing wall and eventually adjust those later.

In the given case the wall type is "eaves wall". We close the template for the wall to be entered with "OK".

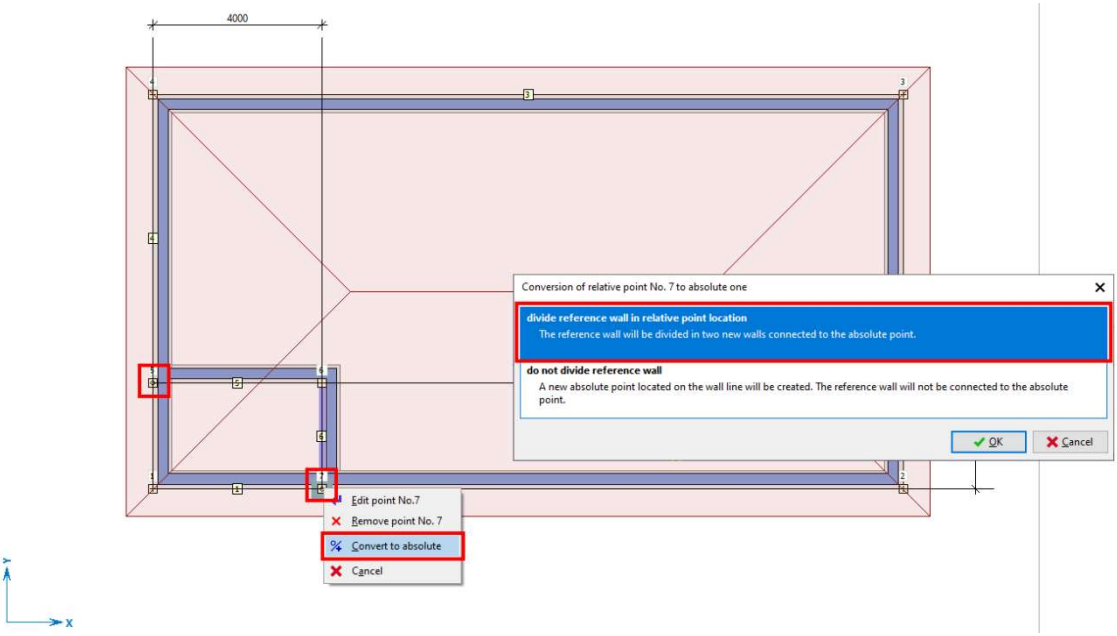


We add the walls graphically by using the mouse, click on the points of intersection bewteen the walls and the construction lines. Watch out for the input direction, the wall alignment meets "to the right".



Just for the reason of graphical display, we split the outer walls by changing relative points into absolute ones and then assign them different properties. This has no influence on the design and analysis.

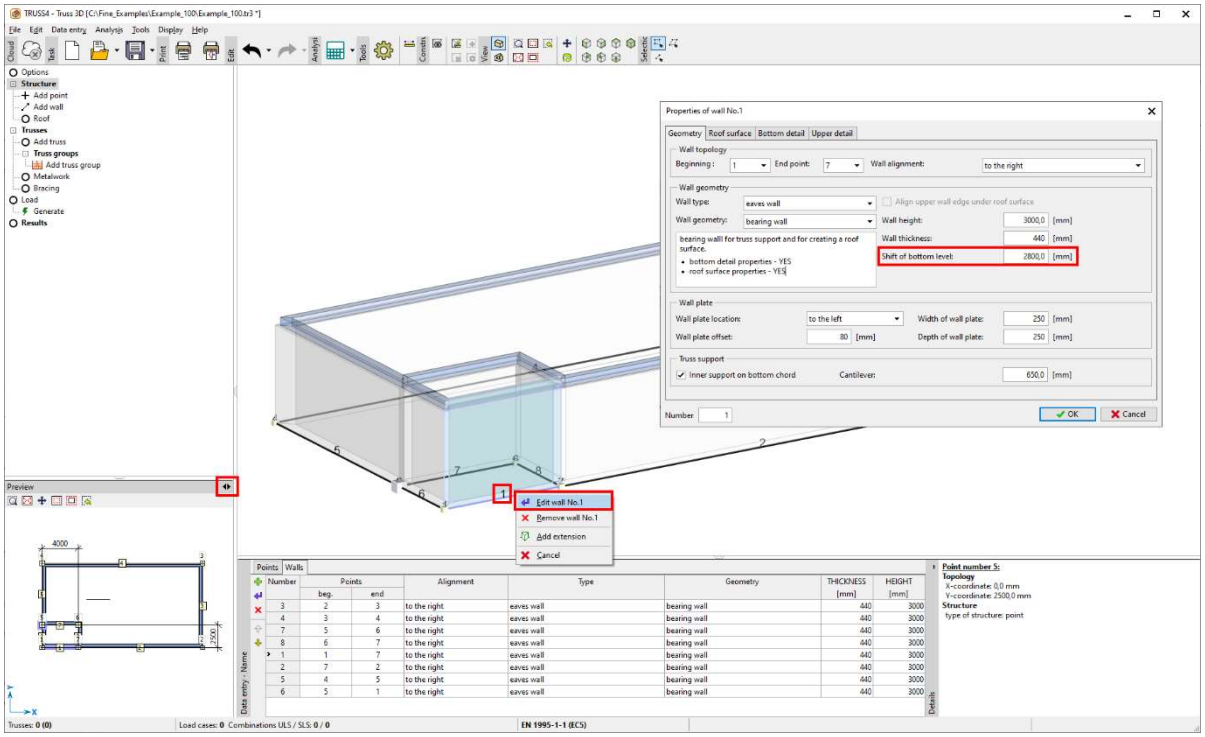
NOTE: There are two types of points: absolute is defined through the coordinates and relative is defined through the wall (it is connected with it). This rule is also being applied for joints in trusses.



For visualizing purposes, we shift the bottom level of the walls so they appear as girders – to the level of the absolute coordinate Z = 2800 mm.

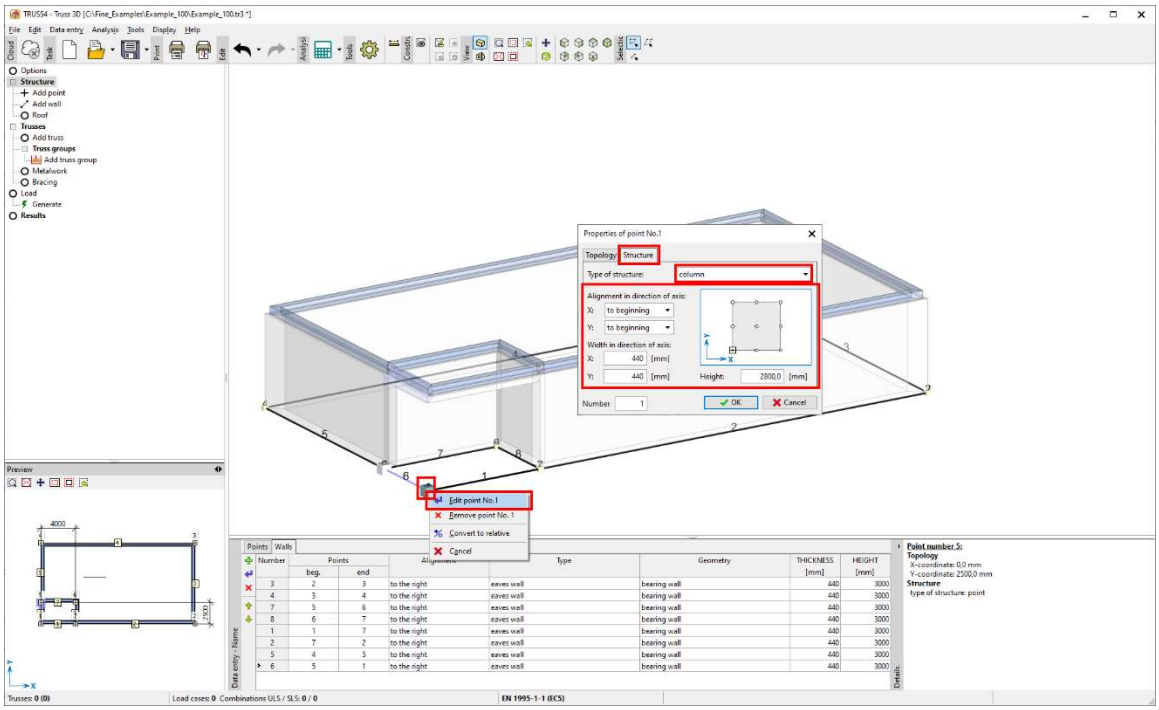
The adjustment of the wall is performed by right mouse click on the wall and selecting “Edit wall No. X” from the context menu.

NOTE: Generally, the right mouse click opens the context menu. Its content differs according to section and object.



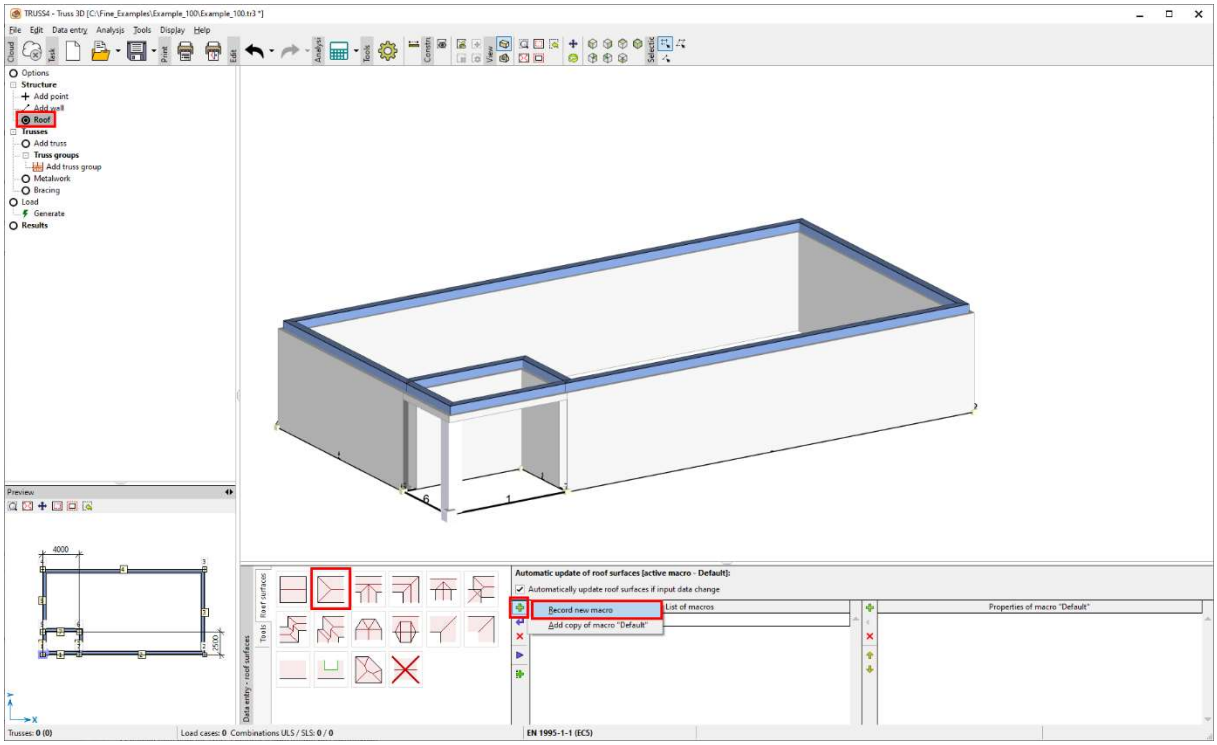
In the same way we change the properties of the edge point to create a column (again this has no impact on design and analysis).

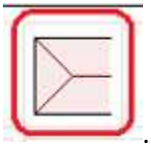
NOTE: The editing of an object (point, wall, truss, group etc.) can either be done via the context menu or by double clicking on the object itself.



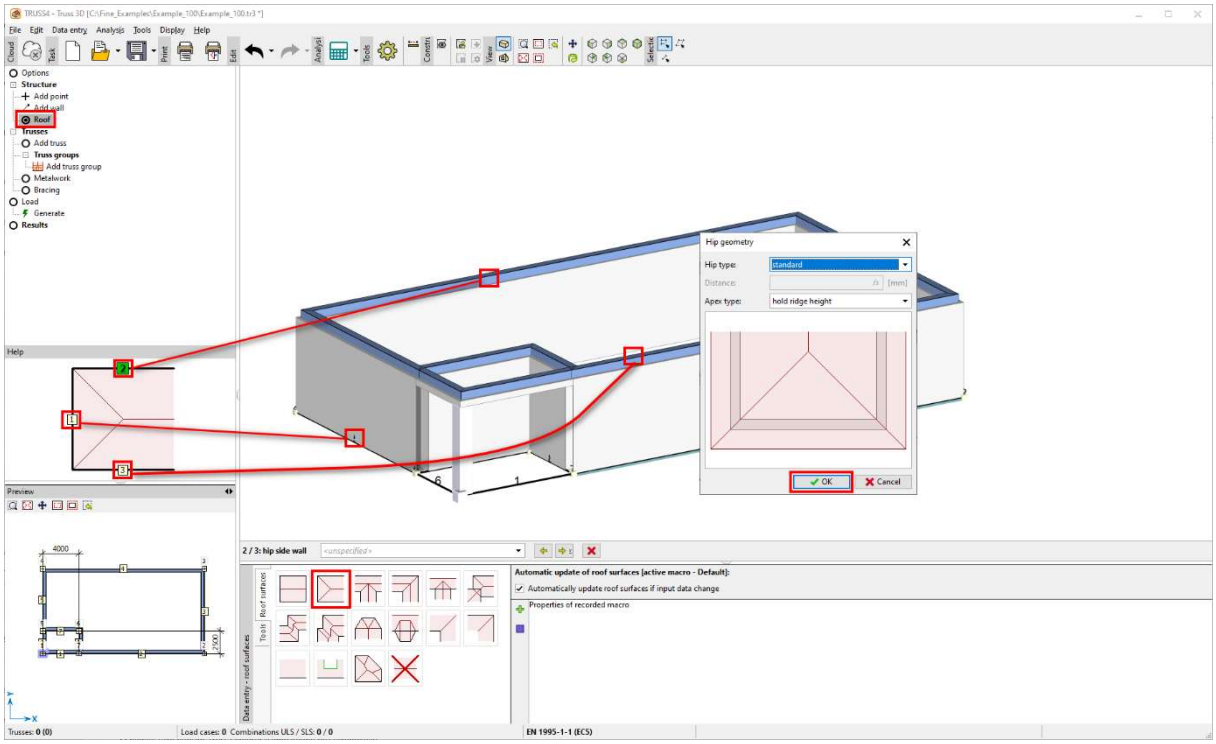
The next step happens in the options window in section “Structure” and the item “Roof”. Here you create roof planes. By clicking on “+” you run the roof plane editor.

NOTE: It is helpful to save the roof planes in macros, mainly in cases where the need to be edited later. In some cases of more complex roof geometries, it is necessary to work with partial macros which correspond to the truss shapes in the appropriate roof areas.



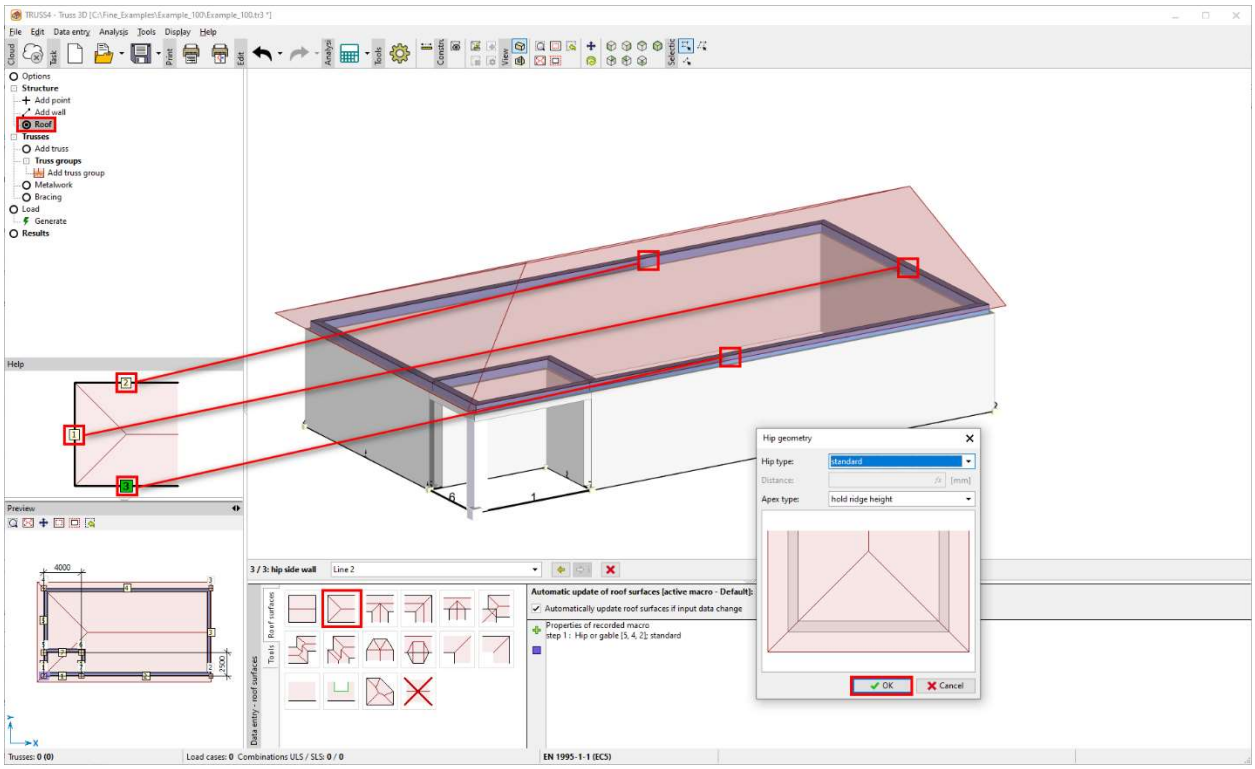
We create the roof planes step by step “hip or gable”  .

Underneath the options windows a helping guide window appears. We click on the individual walls in the following sequence hip gable wall (1), hip side wall (2) and hip side wall (3). We close the “Hip geometry” dialog with “OK”.

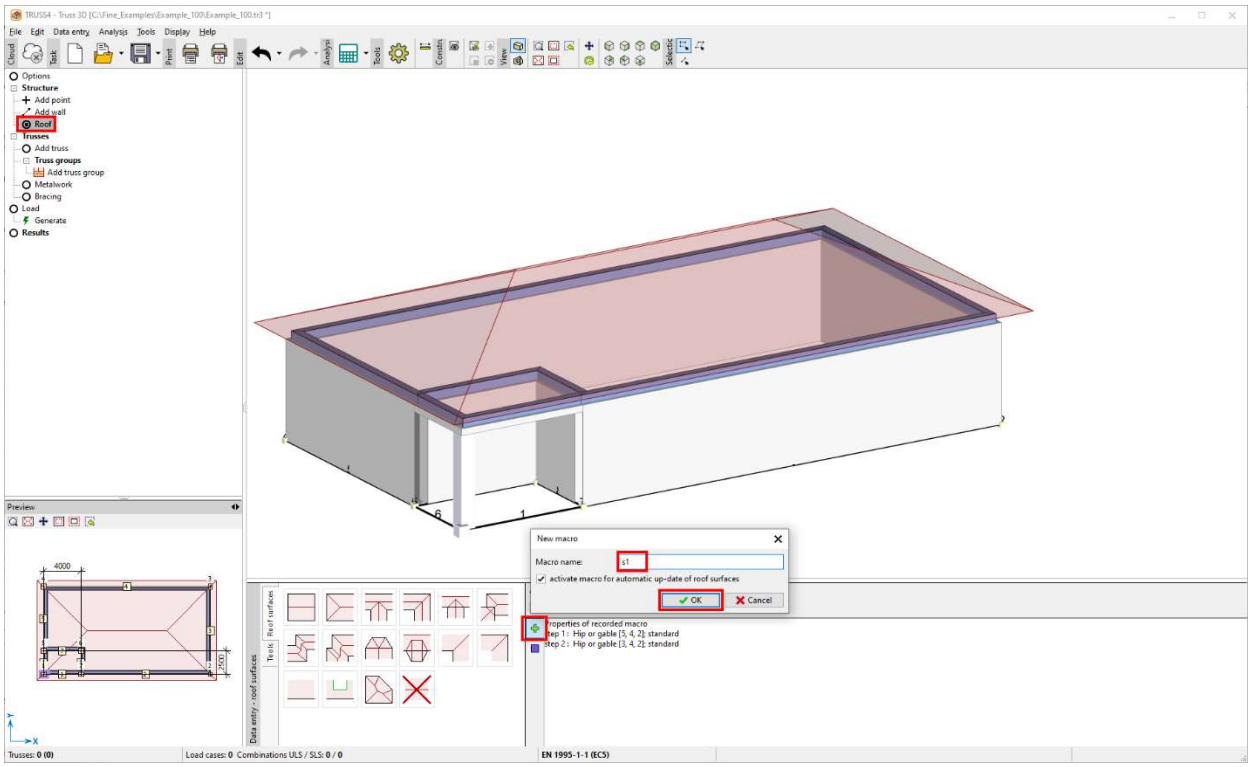


Repeat this for the other hip. Select “hip or gable”  and click on the wall on the opposite side of the building again in the sequence hip gable wall (1), hip side wall (2) and hip side wall (3). .

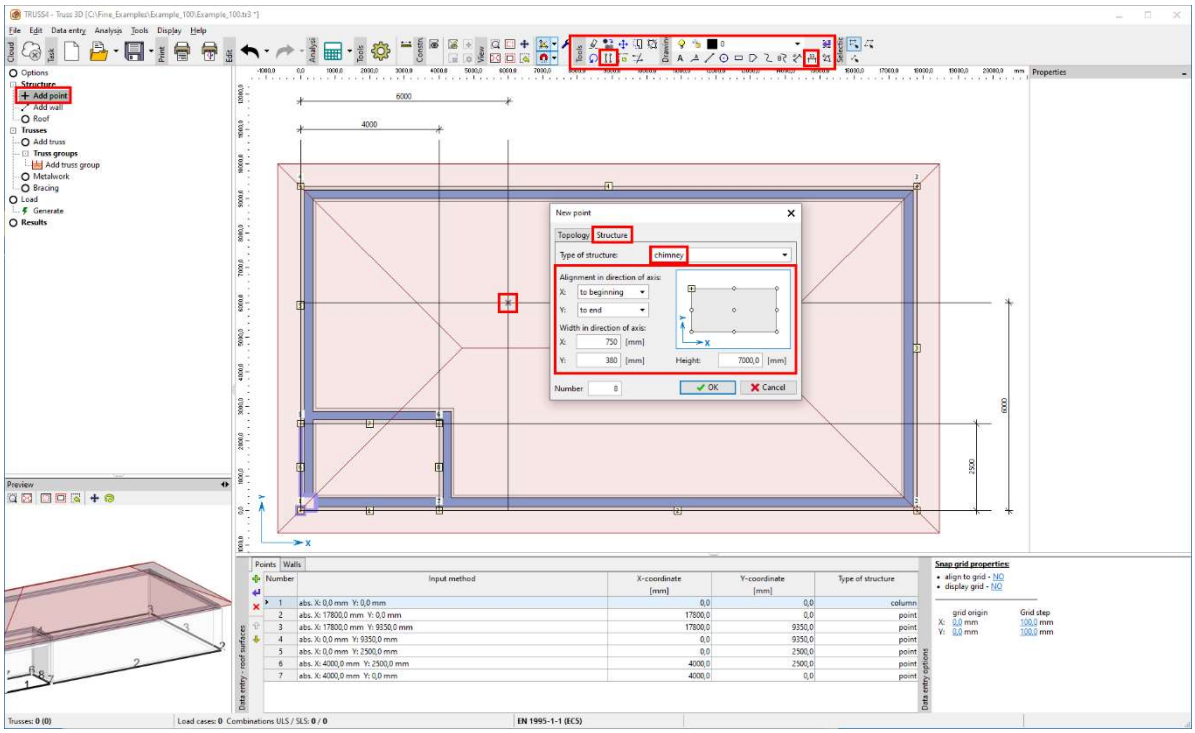
NOTE: The creation of roof planes often depends on the order of the selected walls. In some complex cases it may be necessary to use so called auxiliary walls that do not represent a support (so they are not physically present) but define the roof plane.



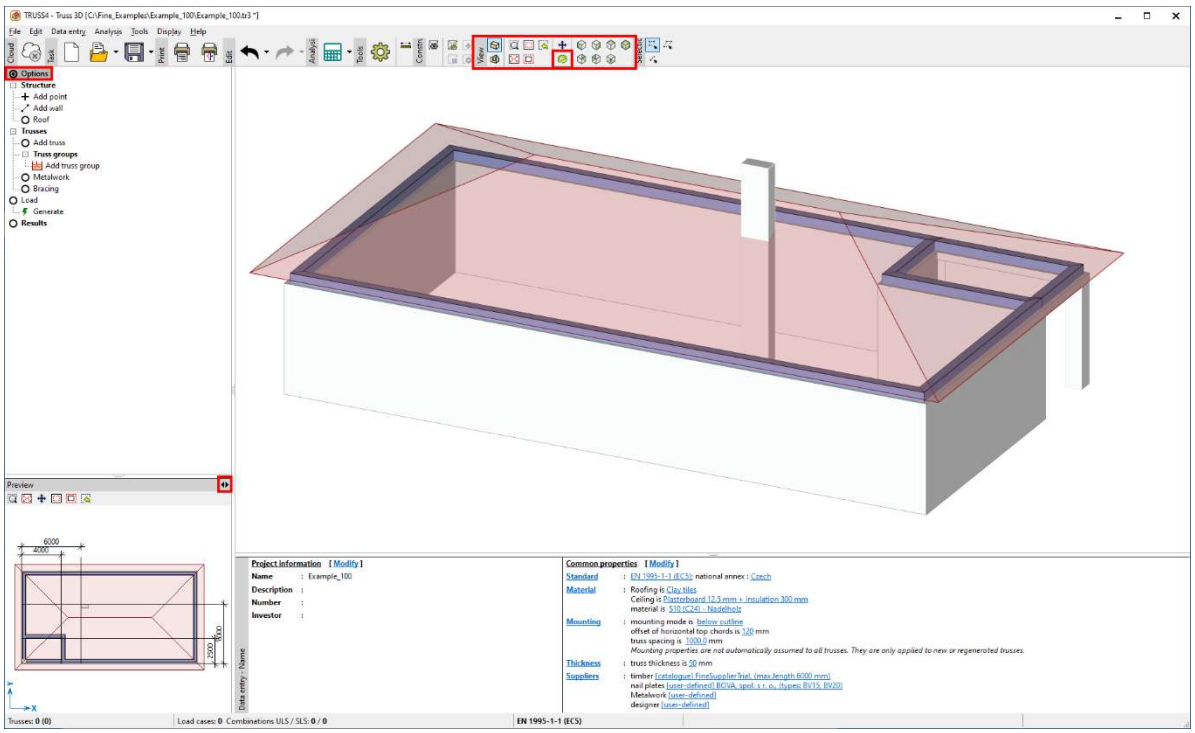
We save the recorded macro by clicking the “+” symbol and name it “s1”.



Further we add a chimney using construction lines and parallels. We add a point where those lines intersect and assign it the properties of a chimney.

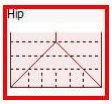


Before we proceed to the next section, we check the model thoroughly if it meets all specifications.



We delete the construction lines as we won't need them anymore. First, we switch back to plan view, then hit the <ESC> key in order to deselect eventually previously selected objects, then left click on the drawn lines and dimension lines so they get highlighted with green colour and finally hit the <Del> key to delete the selected elements.

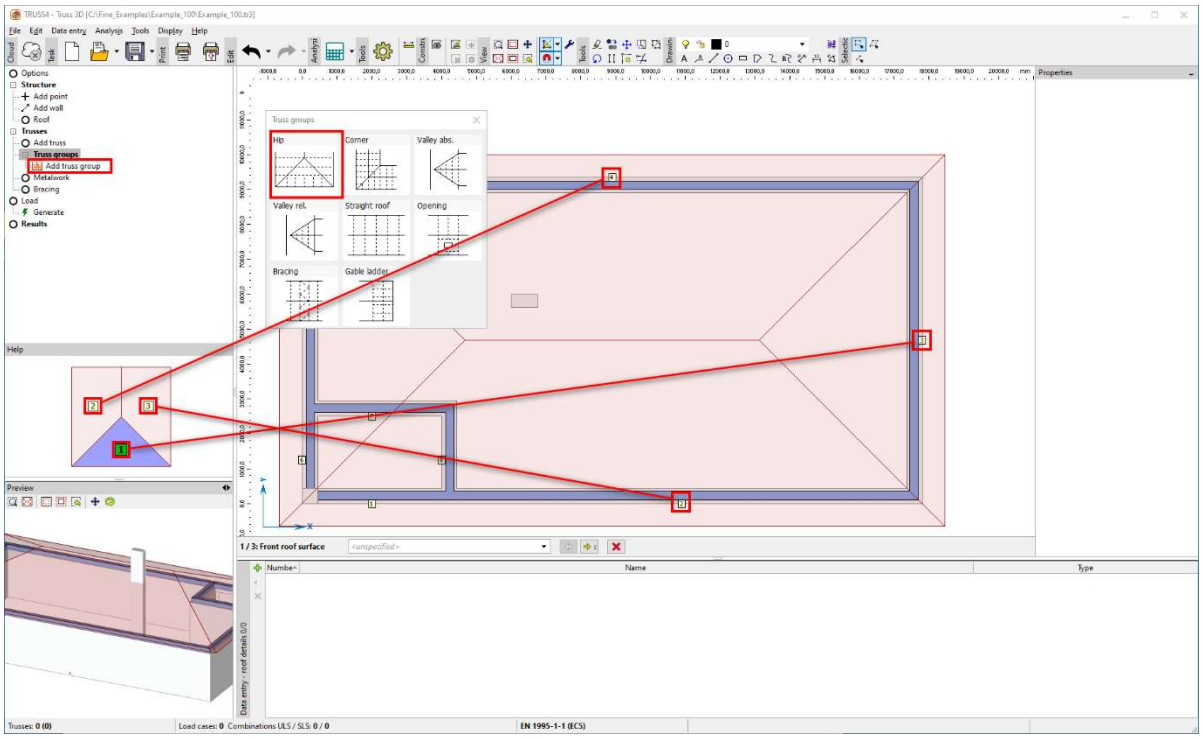
Trusses – Input of truss groups



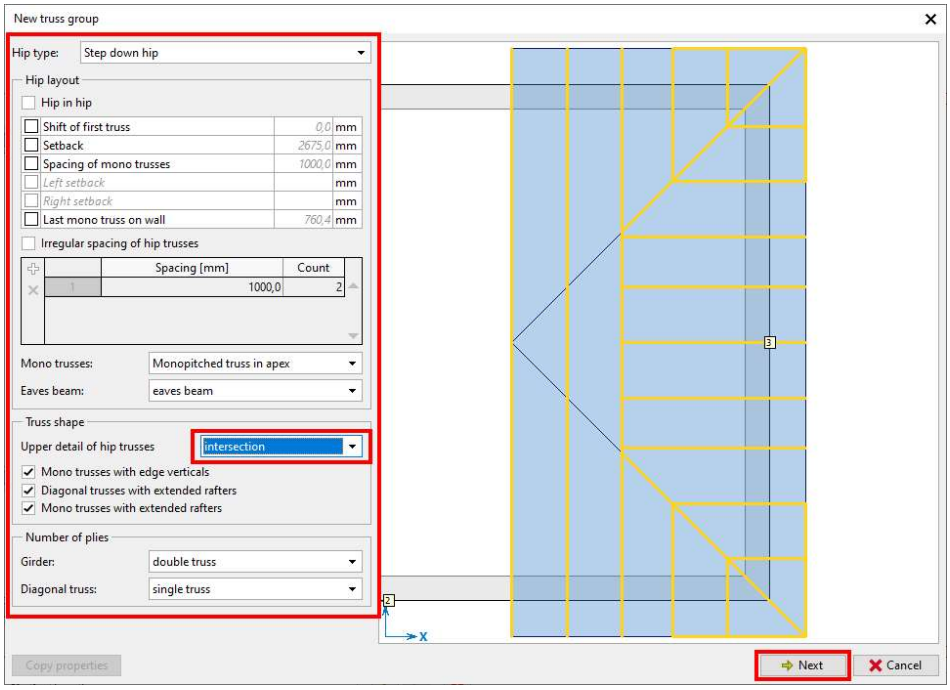
In the section “Trusses” in the options windows we select “Add truss group” and then “Hip”

Following the graphical help, we click on the respective roof planes (or walls) in the following sequence: first the front roof surface (right -1), then the side roof surface (upper -2) and the side roof surface (lower -3).

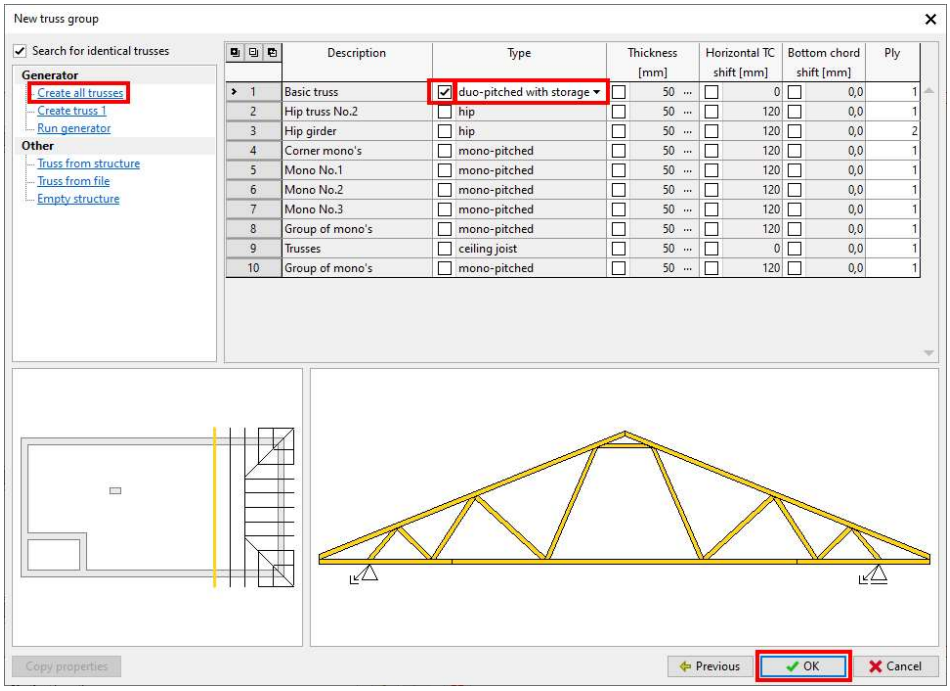
NOTE: The order of the selected walls defines the origin of the trusses which is important for production and assembly.



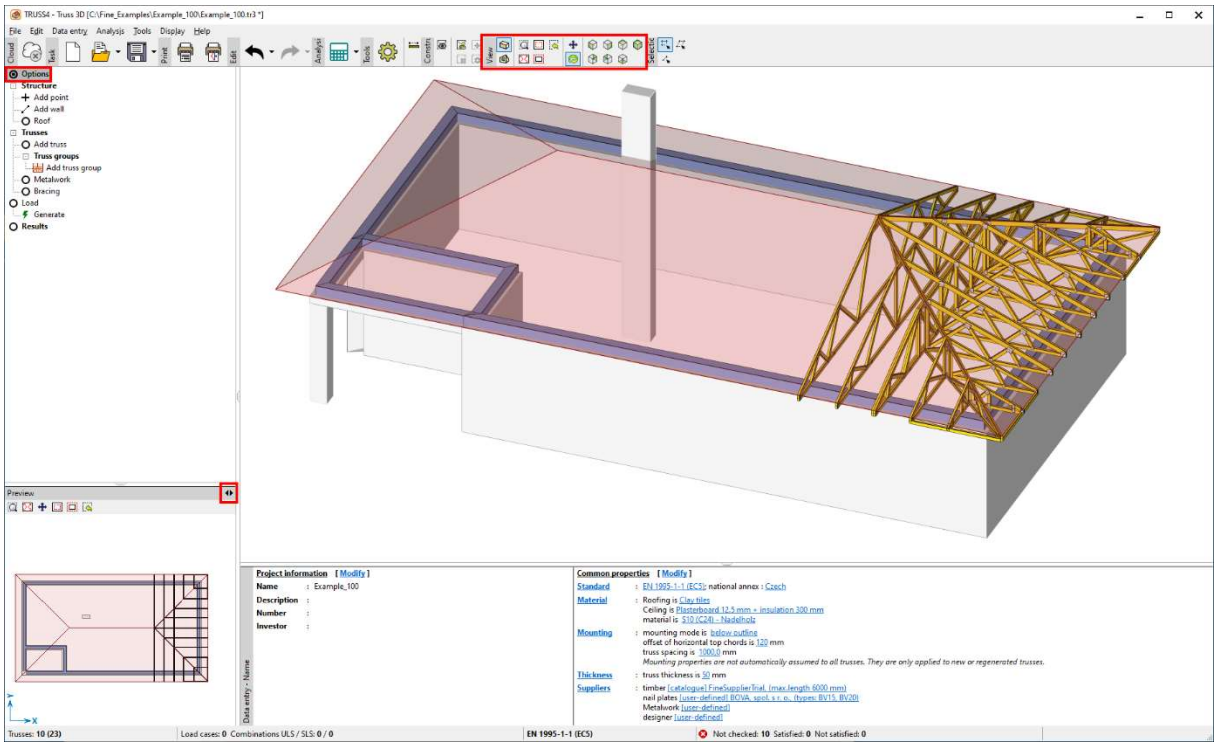
After the appropriate selection of the roof planes a dialog appears with various input parameters for the truss group. Here we are dealing with a hip and accept all given settings except the upper detail of hip trusses which we set to “intersection”. A click on “Next” brings us to the second phase of the truss input.



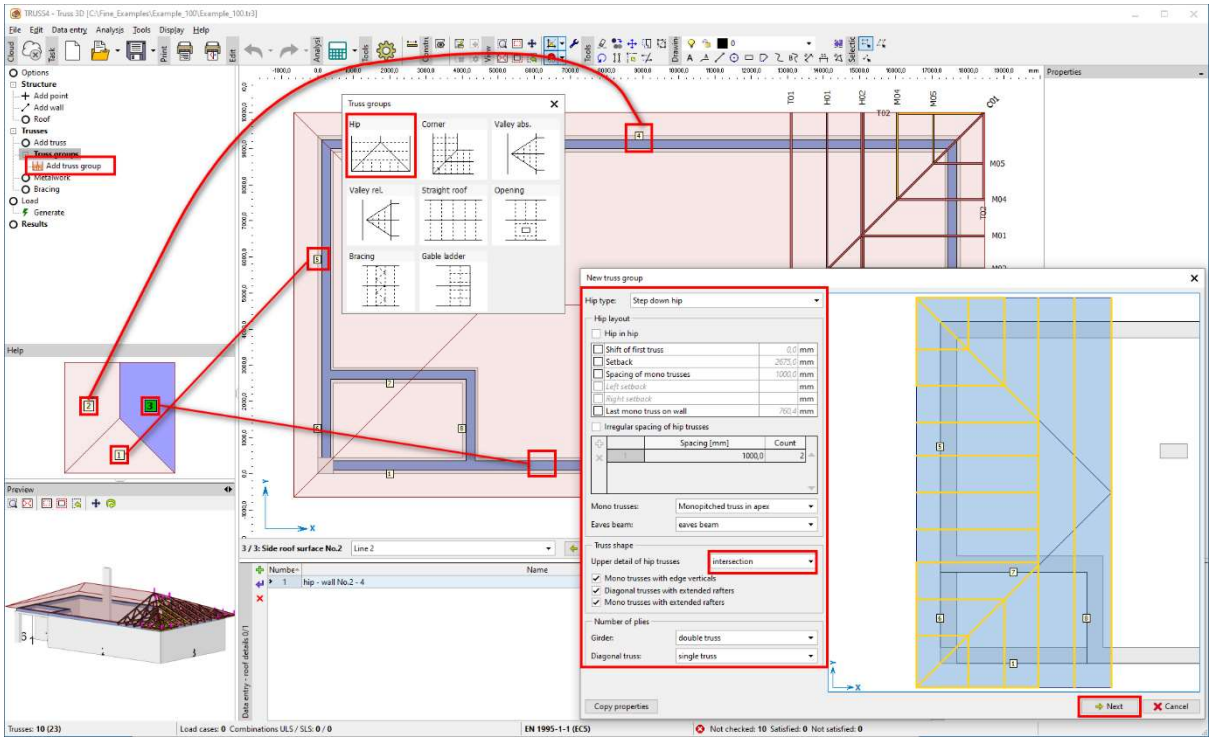
In the second phase we see all automatically identified truss types. We leave most of them as they are, we only change the basic truss into “duo-pitched with storage”. Clicking on the hyperlink “Create all trusses” the program creates them all. By clicking on “OK” the program inserts the trusses into the model.



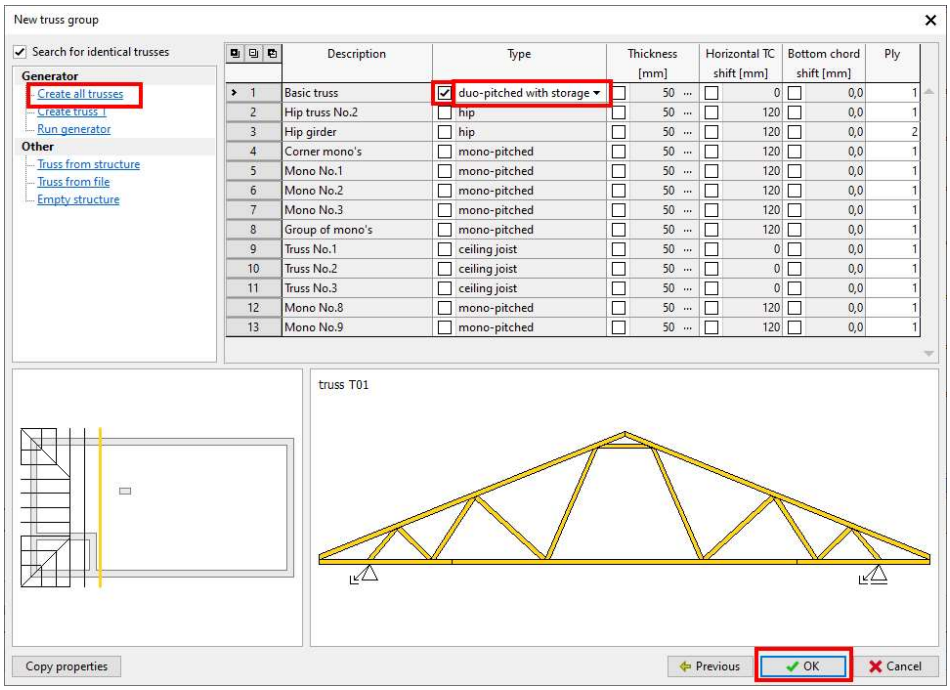
Before we continue with editing, we check the model thoroughly if it meets all specifications.



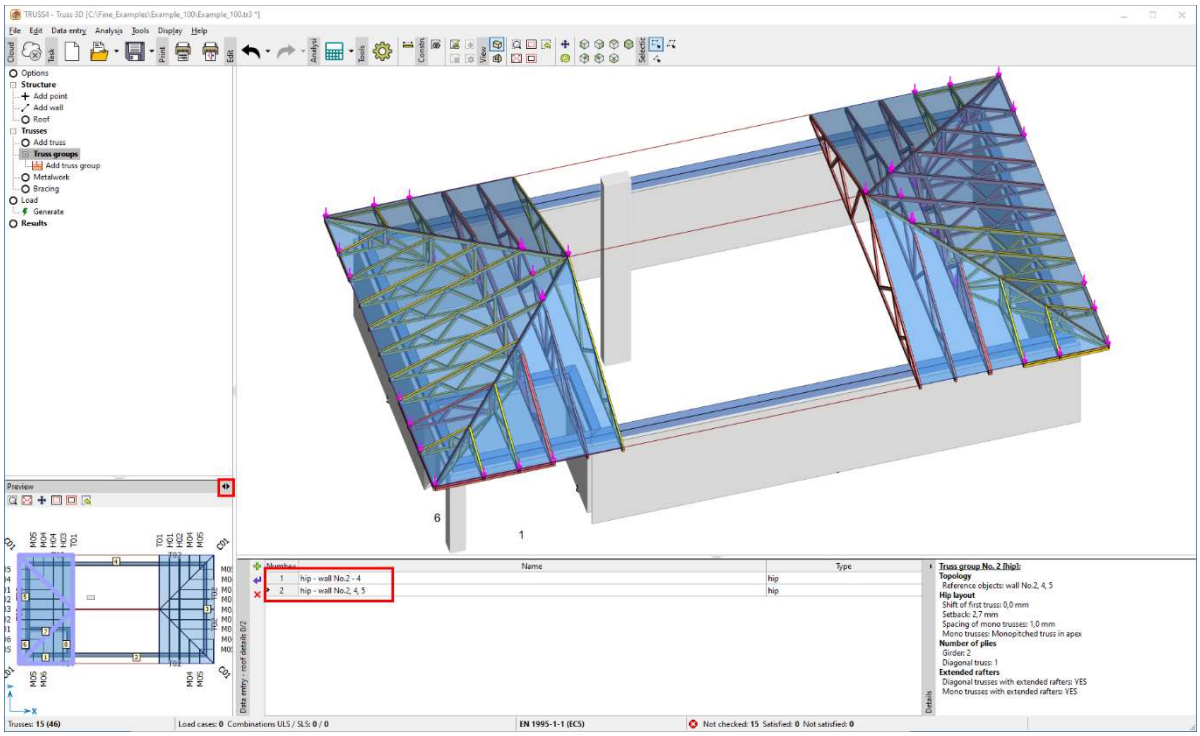
We create the other hip group in a similar way. We click on the walls in this sequence: front roof surface (right -1), side roof surface (upper -2) and side roof surface (lower -3). We end the input of the first phase with a click on “Next”.



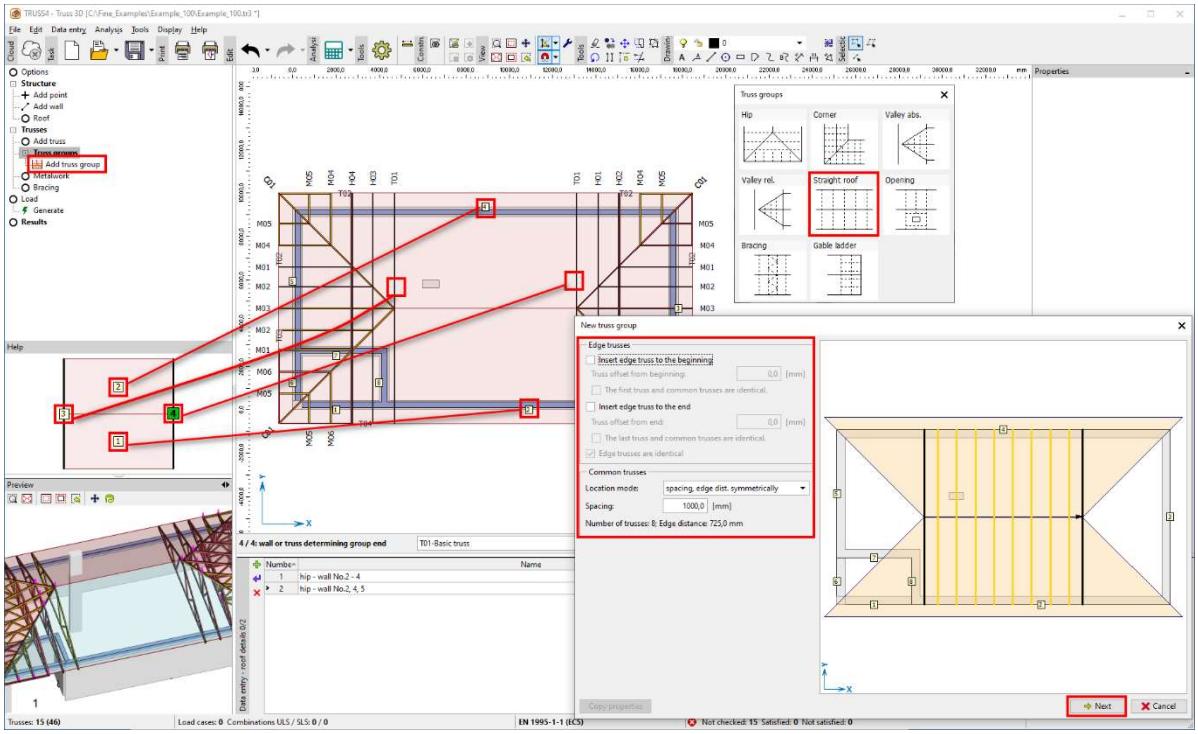
We create the truss as before and insert them by clicking on “OK”.



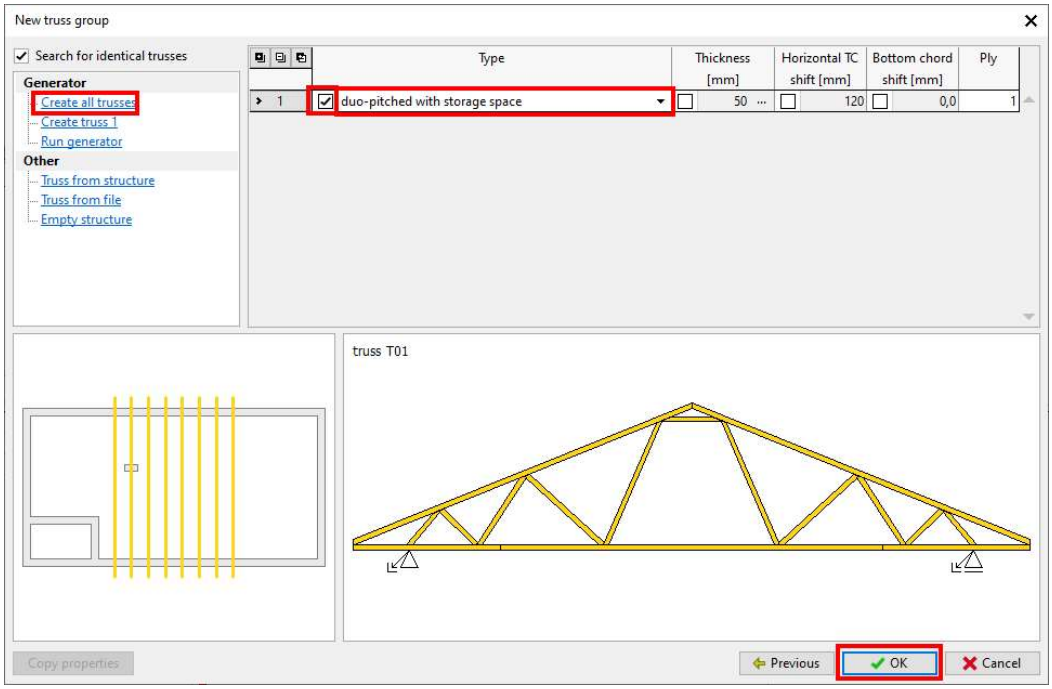
In the section “trusses”, “Truss groups” of the options window we see the truss groups graphically highlighted and the lower table they are listed. Any subsequent editing or deleting of existing truss groups can be either handled within the table, by double clicking on the object or right click or selection from the context menu.



In an analogue way we insert the truss group “Straight roof”.



Again, we select “duo-pitched with storage”, create all trusses and close the dialog with “OK”.



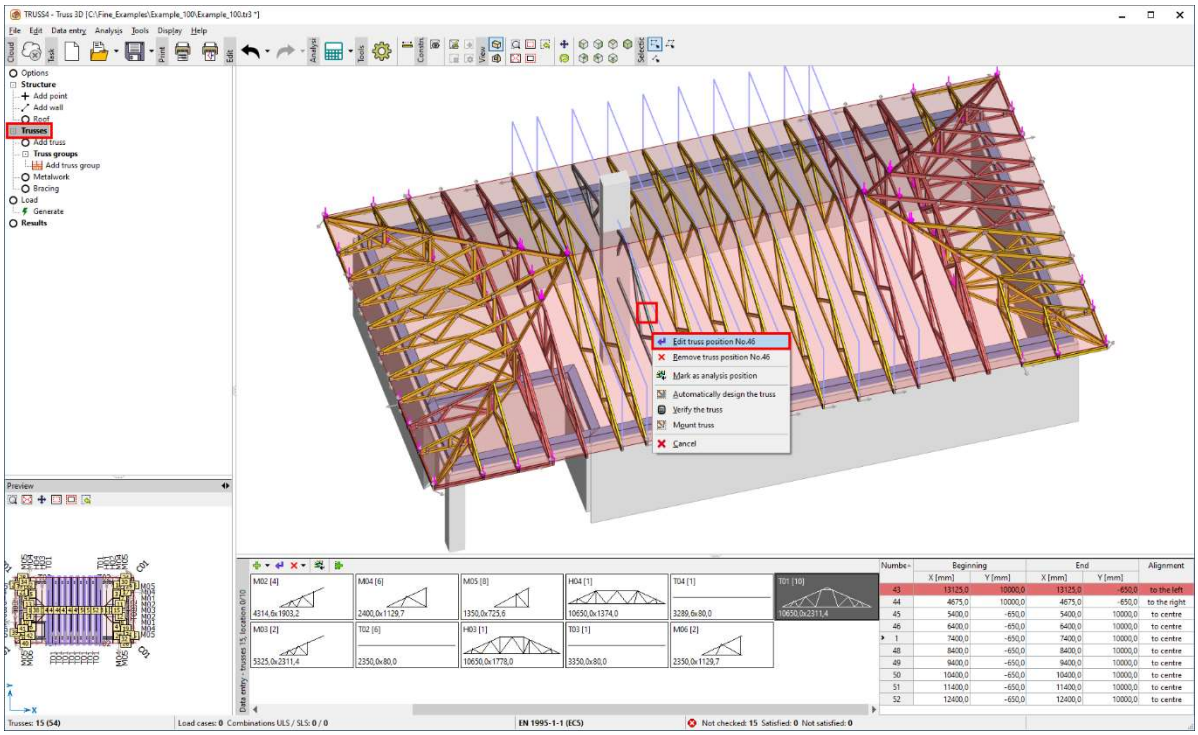
Trusses – Edit location and manual inserting

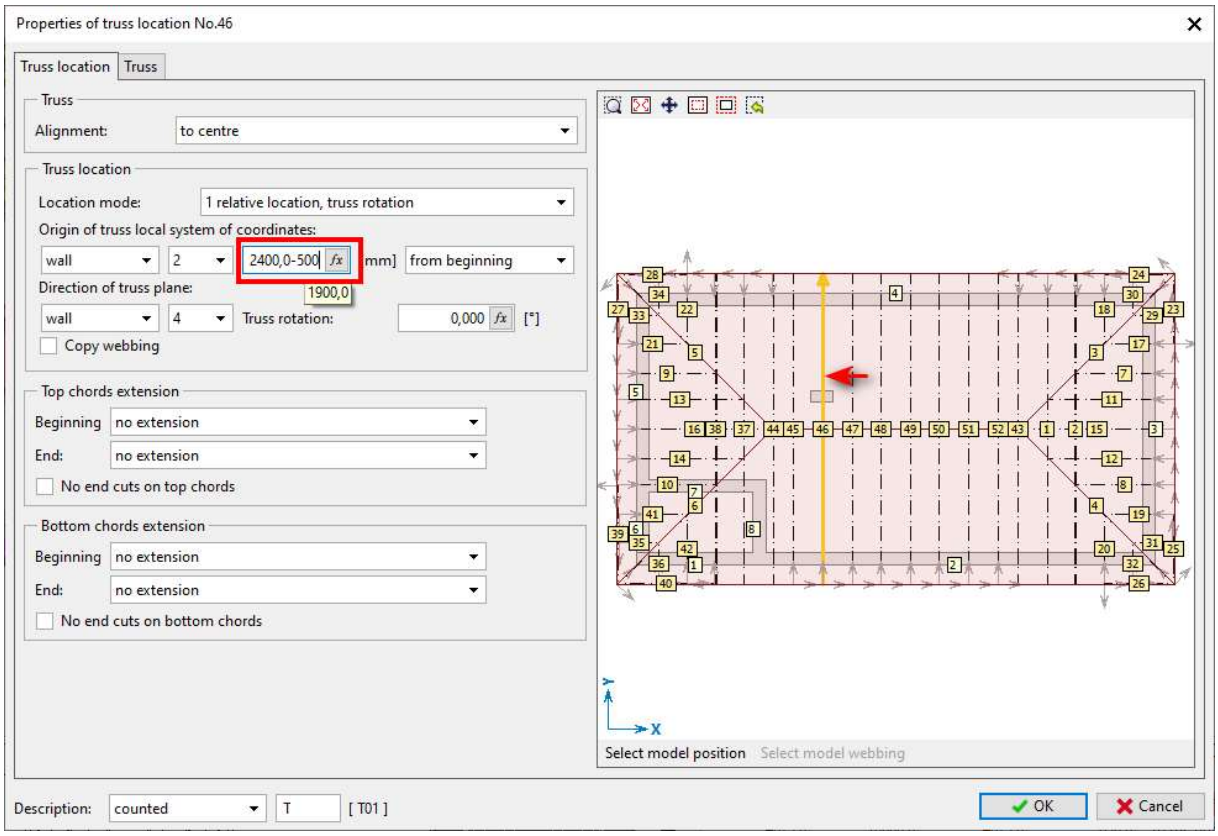
We shift the truss that collides with the chimney by changing its location.

NOTE: Locations of trusses are defined as either both sided (wall/wall, wall/truss, truss/truss), one sided (wall, truss) or general location (global XYZ coordinates). Besides these basic forms, there are special types such as bracing frames (truss/truss) and gable ladders (roof edge).

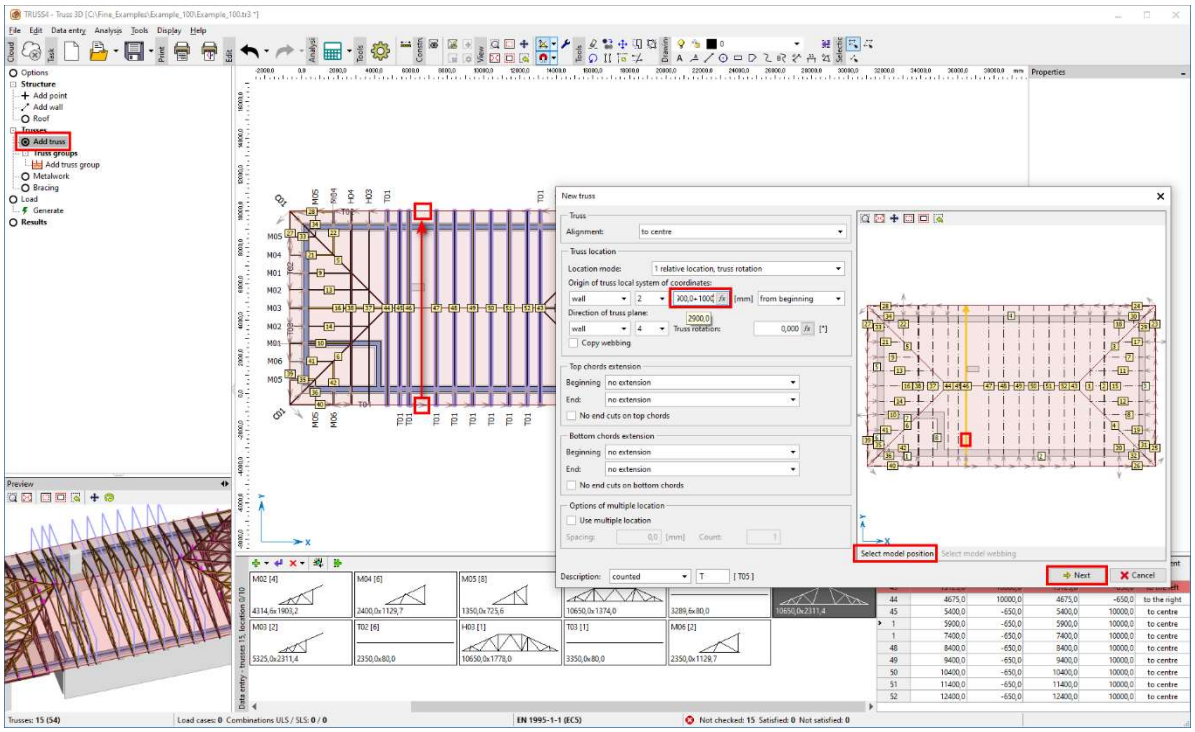
We shift the colliding truss by 500 mm to the left by inserting "-500" to the actual position value of the truss.

NOTE: All input fields in the program marked with "fx" can process mathematical operations.

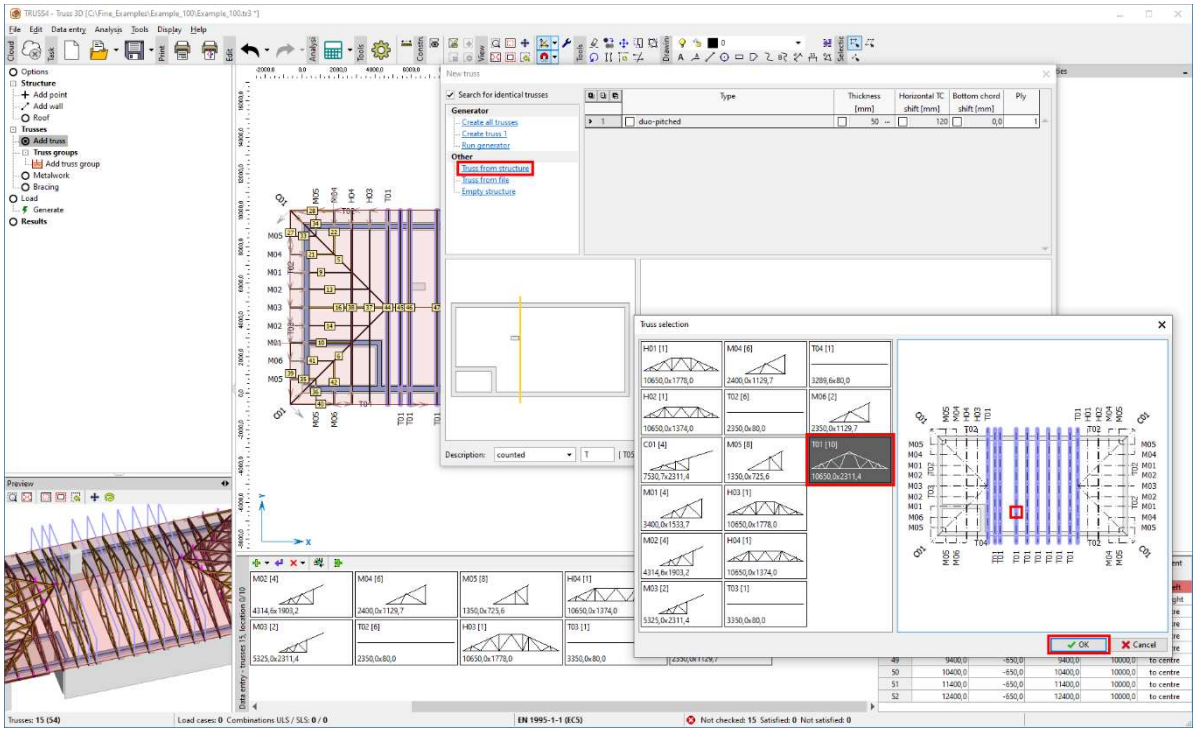




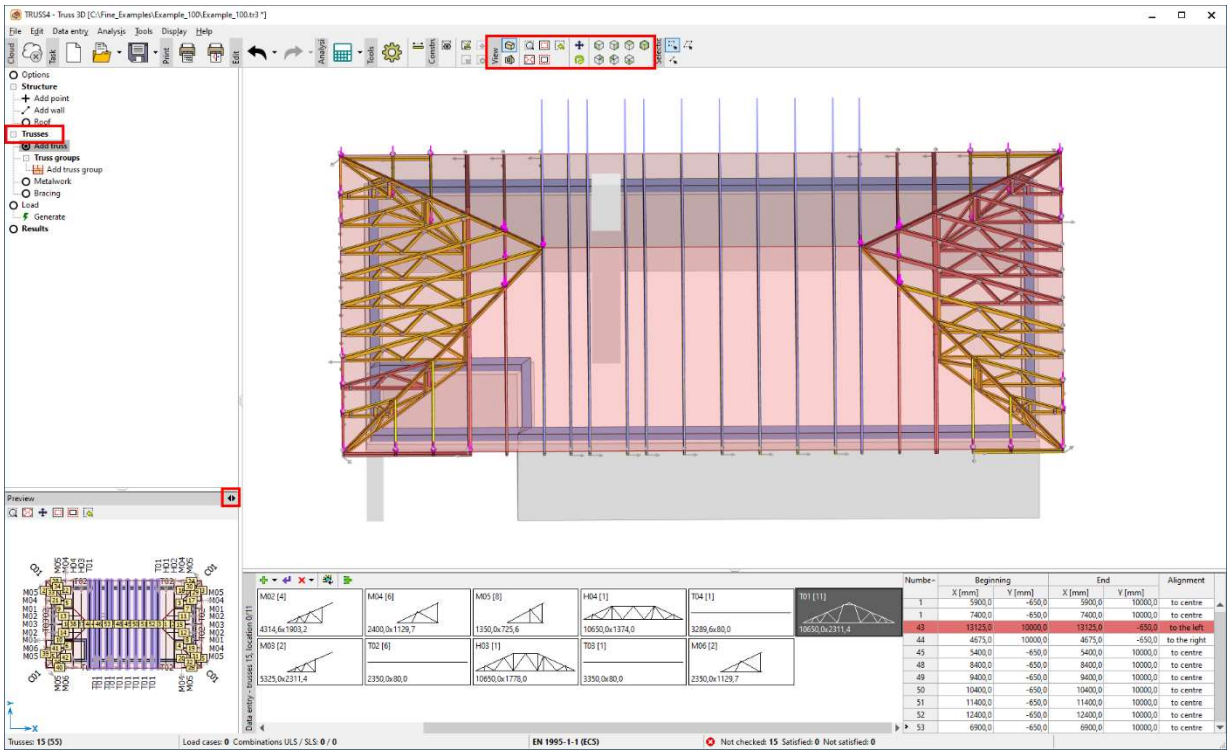
We insert a new truss by using “Add truss” from the options windows. Click on the two roof planes as shown in the picture so the location gets defined by our random click. We use the function “select model position” which assigns an existing definition of a truss to our new truss. As done before, in this case we shift the truss by 1000 mm to the right by adding “+1000” to the current location value. After the successful definition of the location click on “Next” to proceed to the second phase of the truss input.



In this second input phase we choose “Truss from structure” and click on the existing truss T01 (select either from table or plan preview)

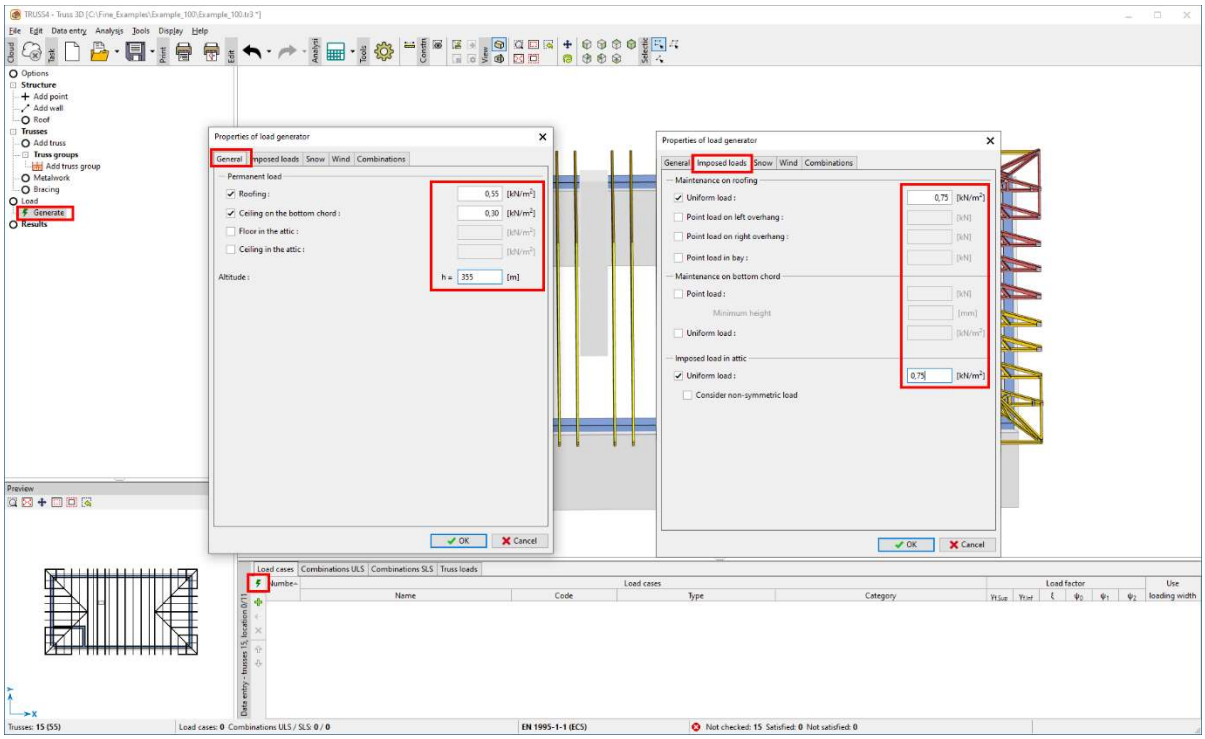


Before we go on, we thoroughly check the model if it satisfies all input and other criteria, if no trusses are missing, no collisions occur, etc.

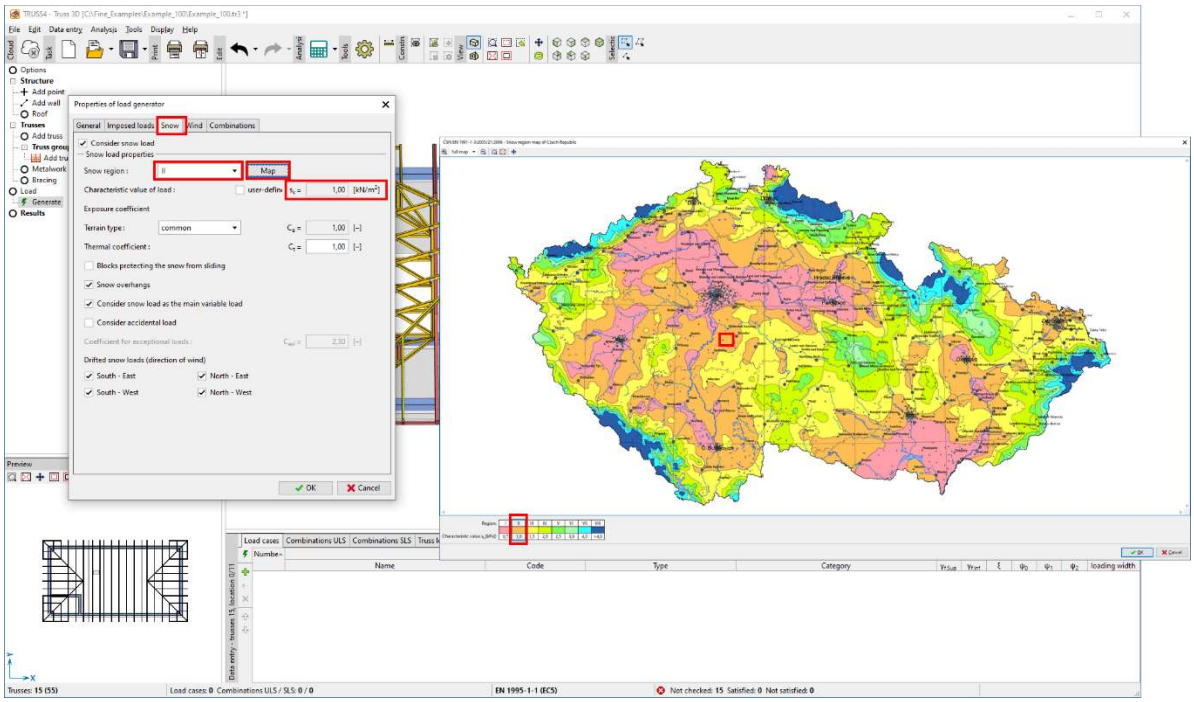


Loading

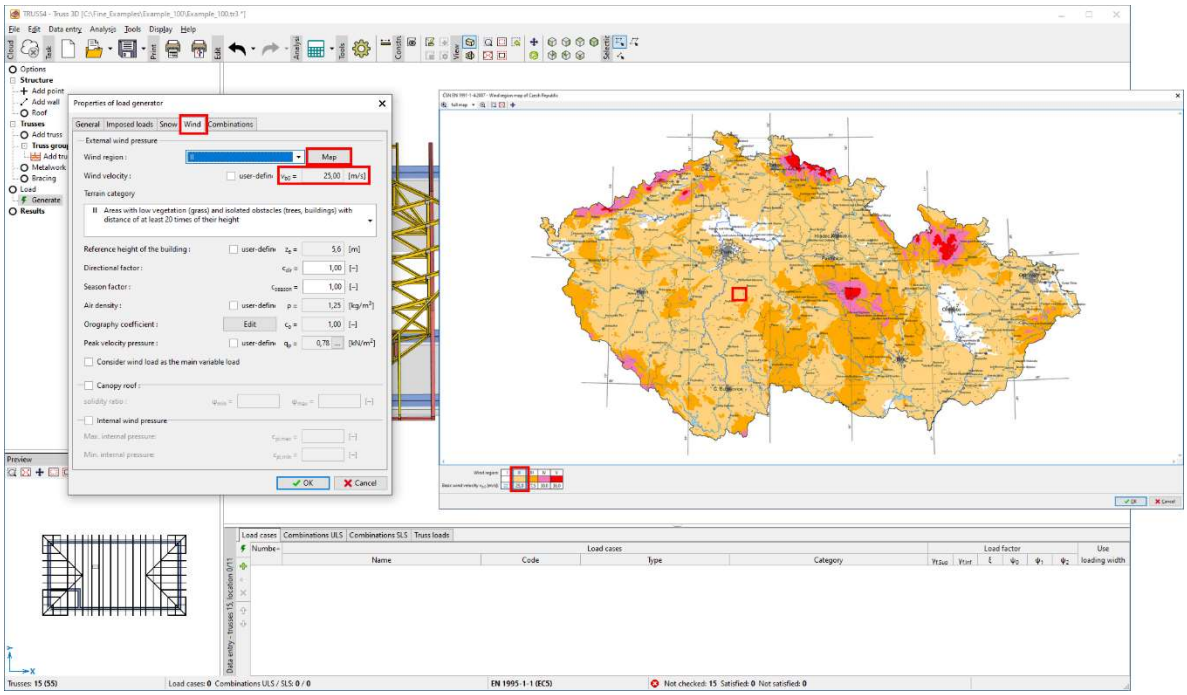
We continue with the section “Load” from the options window and its embedded option “Generate” which opens the dialog for creating load cases, load values and load combinations. The dialog holds five tabs.



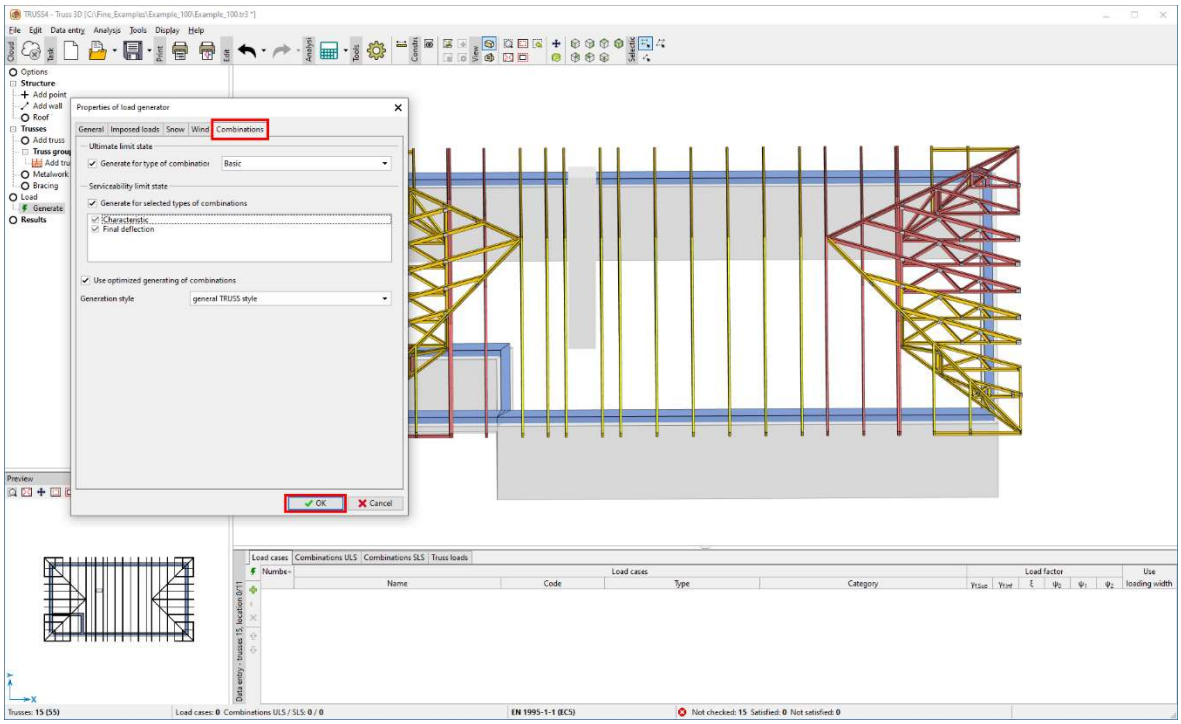
On the tab “snow” you can access an interactive map with snow zones as well as further settings that correspond to the particular design code.



The tab “Wind” allows the use of an interactive map with wind zones as well as setting further values and definitions, which respect the selected design code.

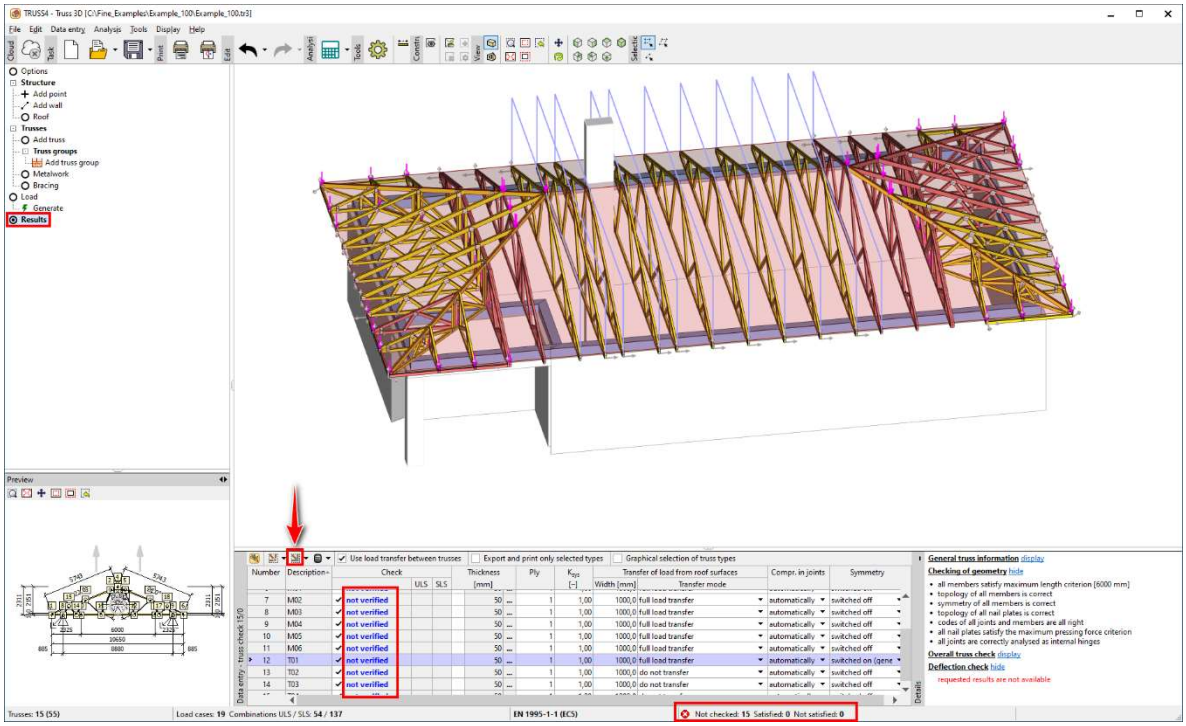


On the tab “Combinations” settings can be done in respect of the chosen design code.

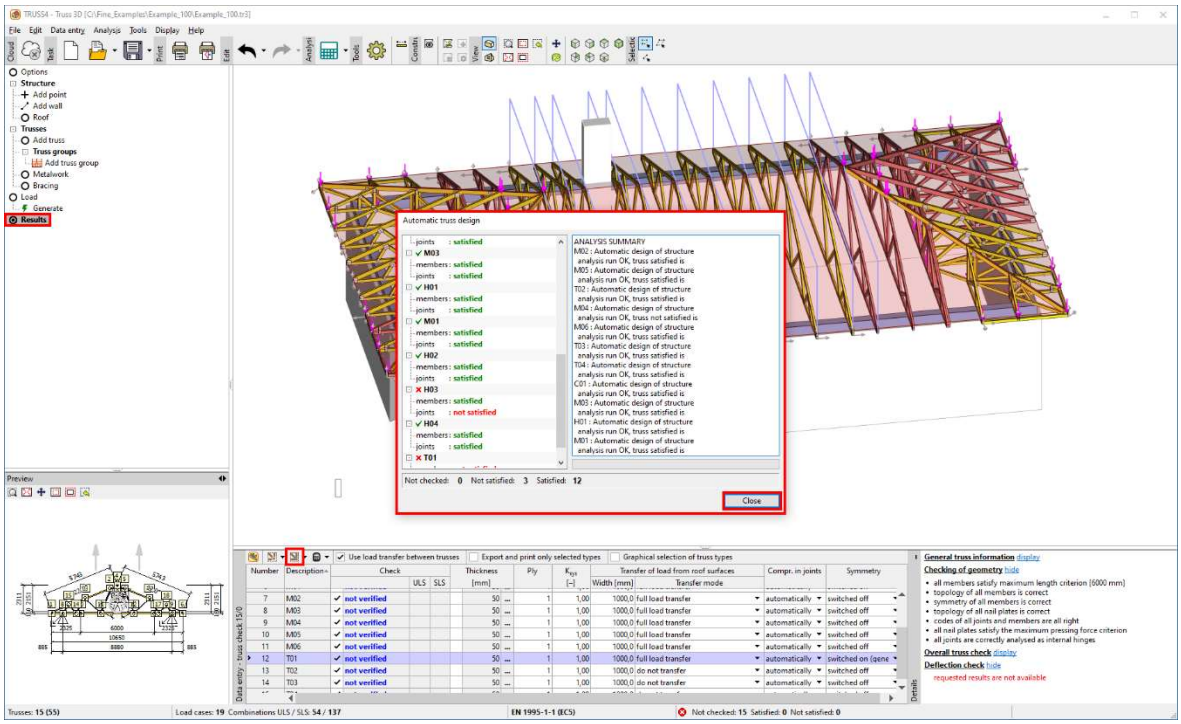


Analysis – Design and checking

In the following section “Results” and the icon “Automatic truss design” we arrive at the actual design and statical analysis. The program runs an iterative analysis process, during which supported and supporting trusses are being considered and designed gradually.

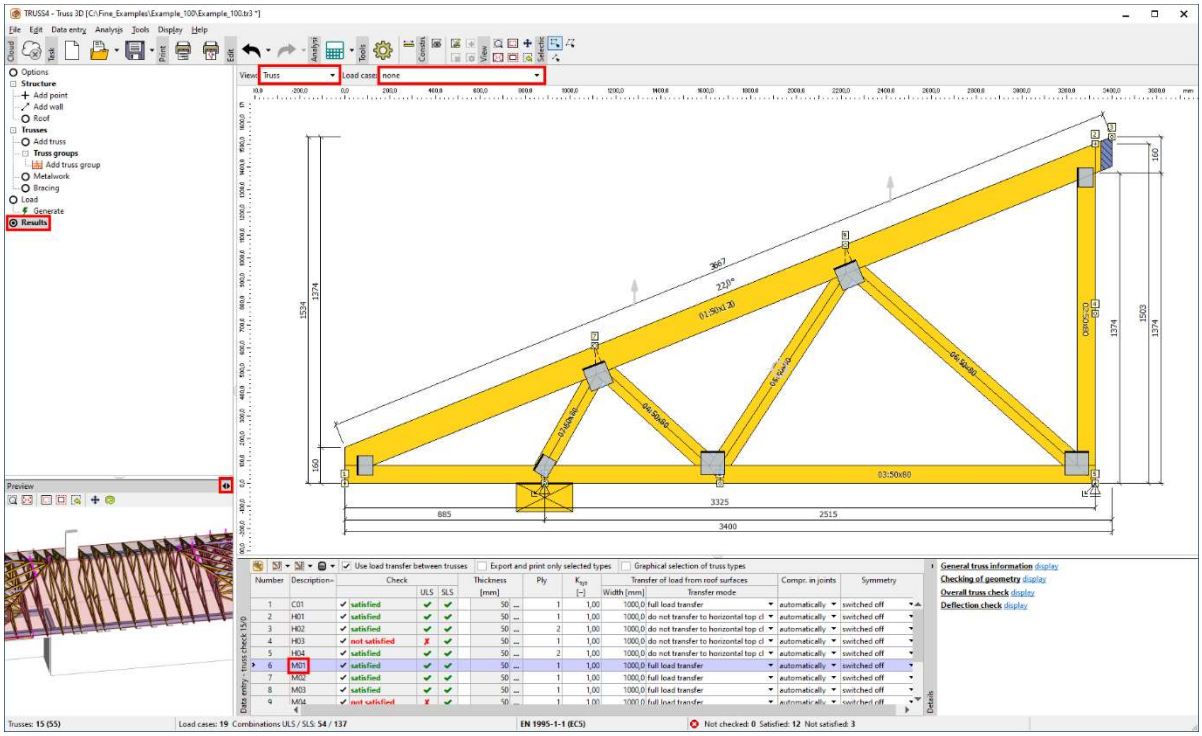


During the design the program shows the design status in a window. When the design is completed, this window needs to be closed via the button “Close”.

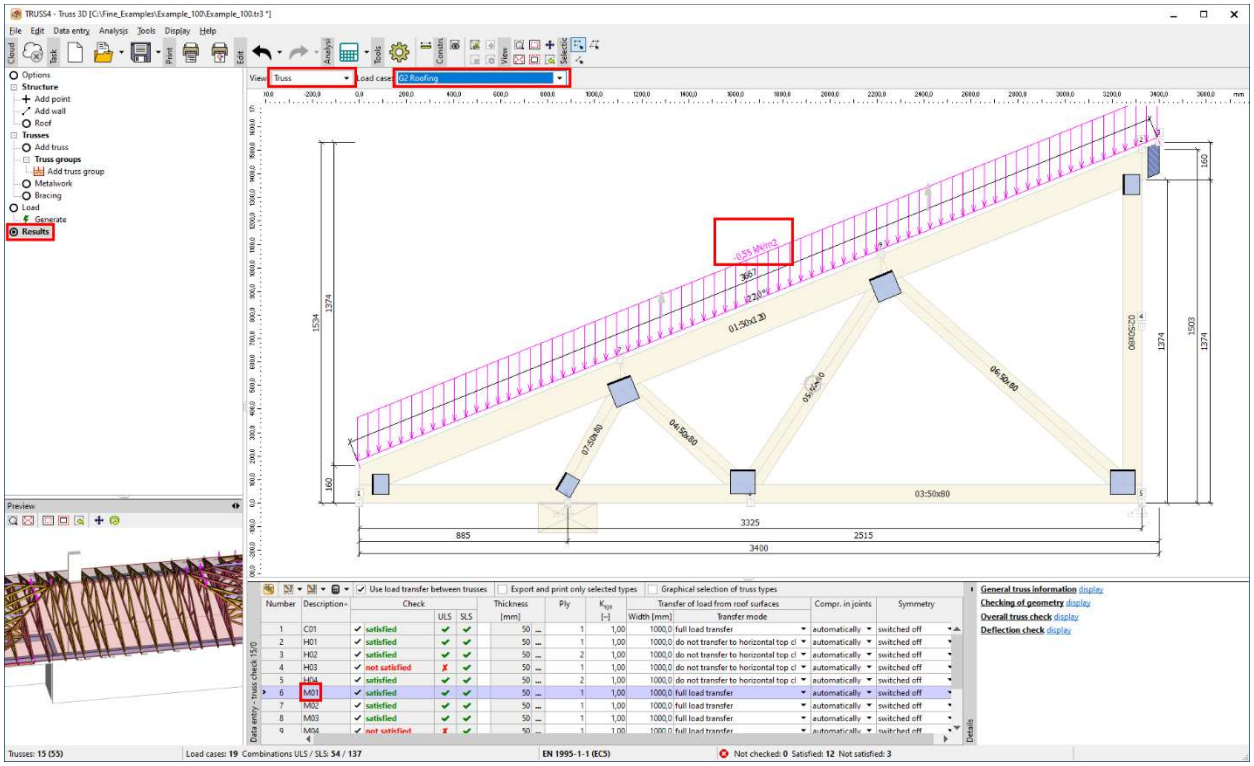


Analysis – Display of the results in the program

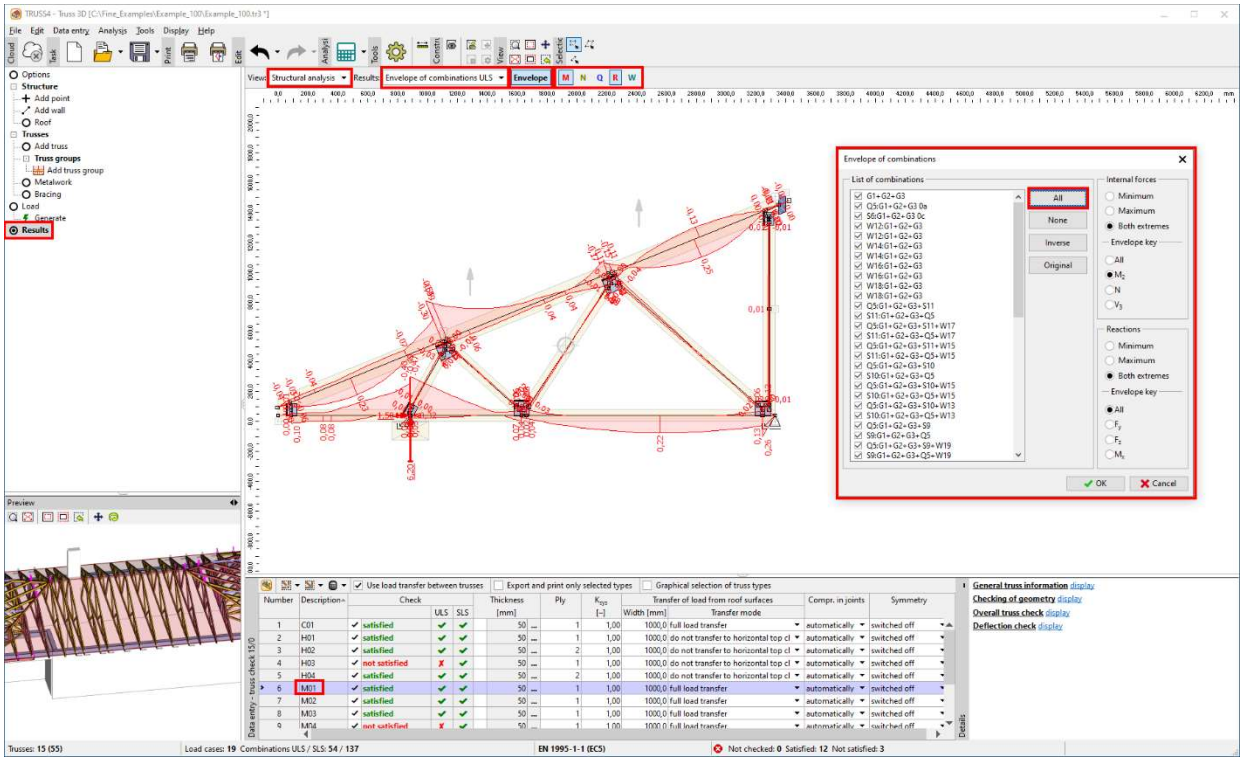
The results of the analysis and design are displayed in a clear way in the section “Results”. Individual trusses can be selected in the lower table window or graphically and various details of the results can be shown.



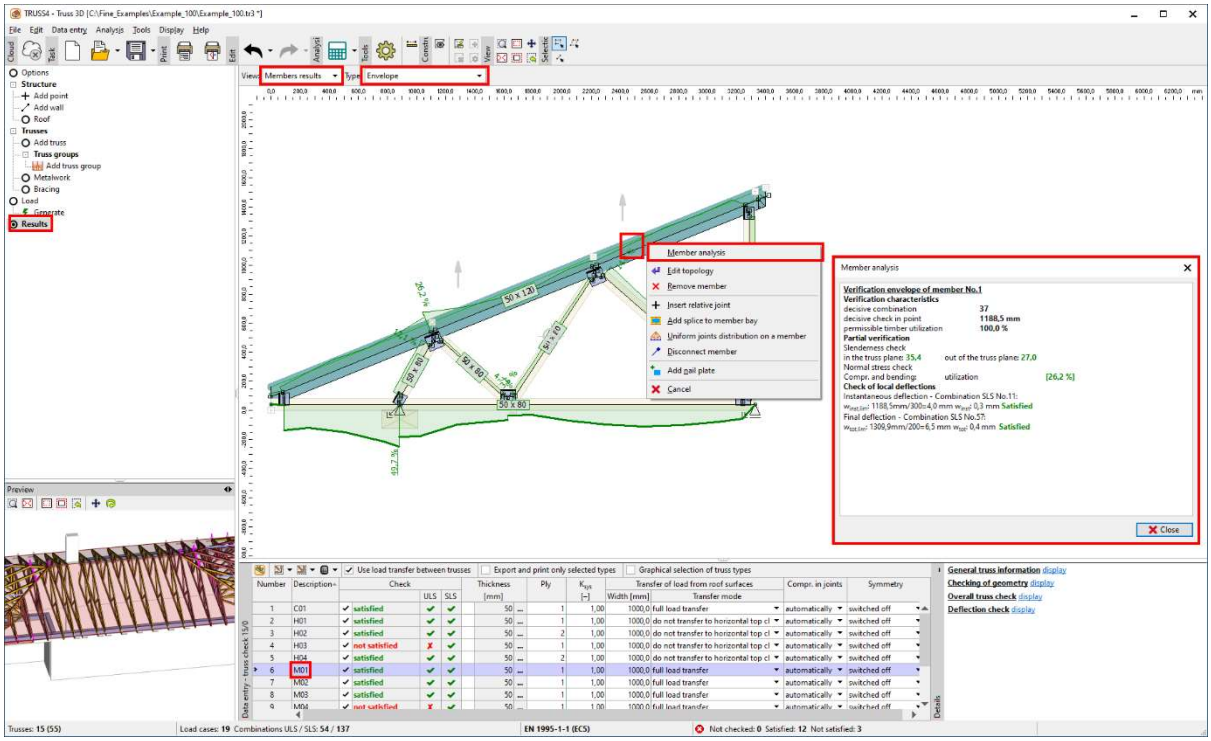
The selection “Truss” enables the display of the loads acting on the truss in the selected load case.



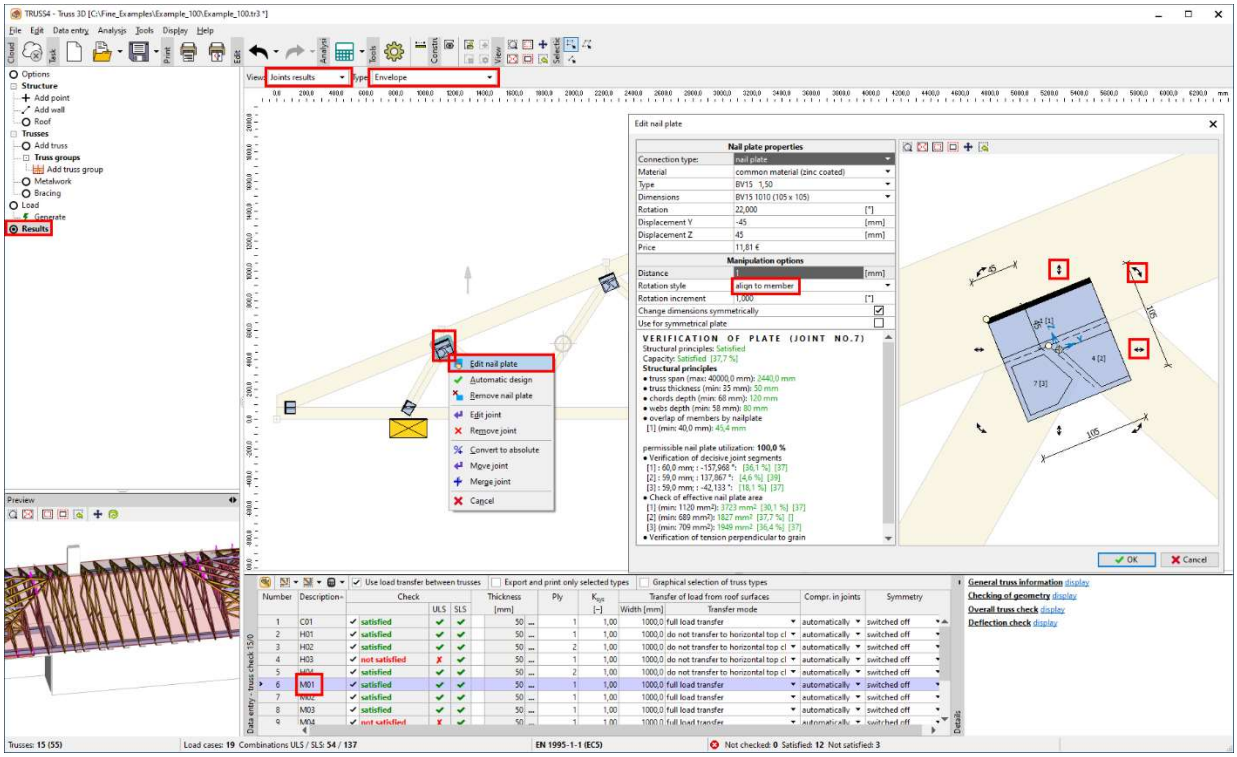
The selection “Structural analysis” provides the display of the calculated inner forces, deformations and reactions in individual load cases, combinations and envelopes of combinations for each truss separately.



The view “Members results” shows the utilisation of the members for the envelope of combinations and for load cases. A double click on a member or the selection via the context menu opens a window with detailed results for the selected member.

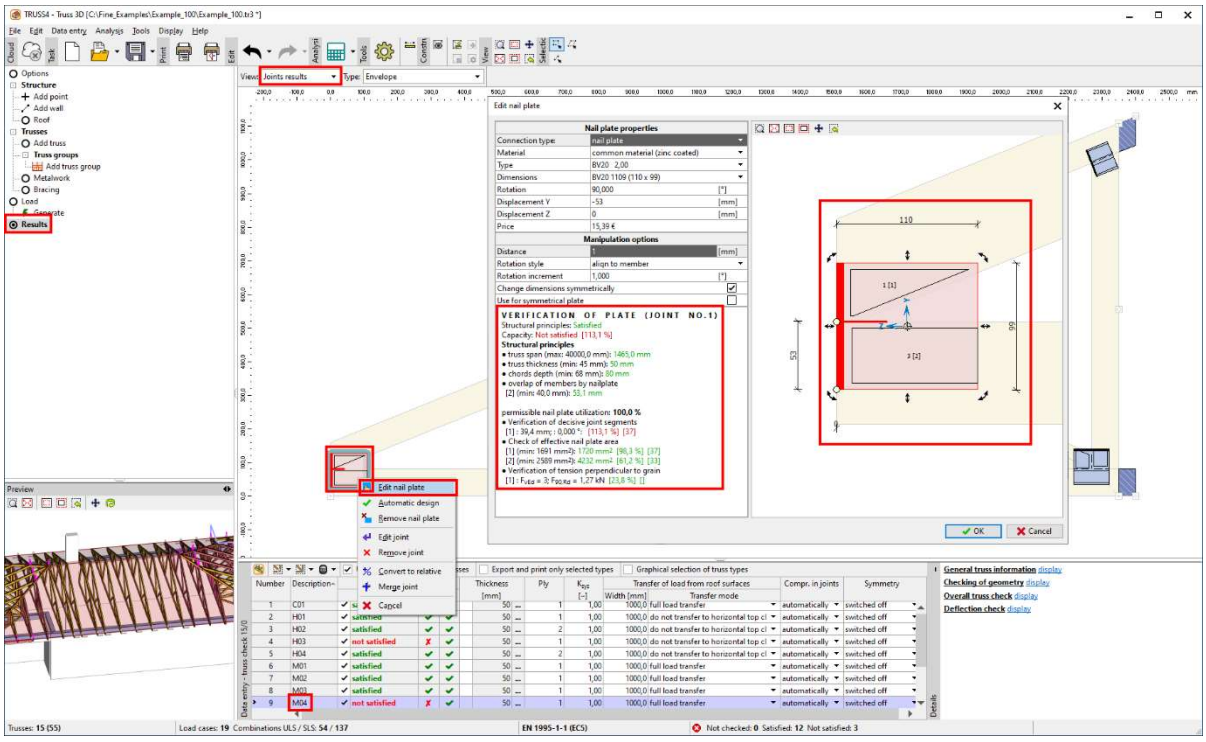


The view “Joint results” displays the utilisation of the joints for the envelope or individual load combinations. Double clicking on the nail plate or the selection via the context menu opens the plate editor in which the plate can be moved, rotated and changed in size and its orientation adjusted. Above that it contains the detailed analysis results.

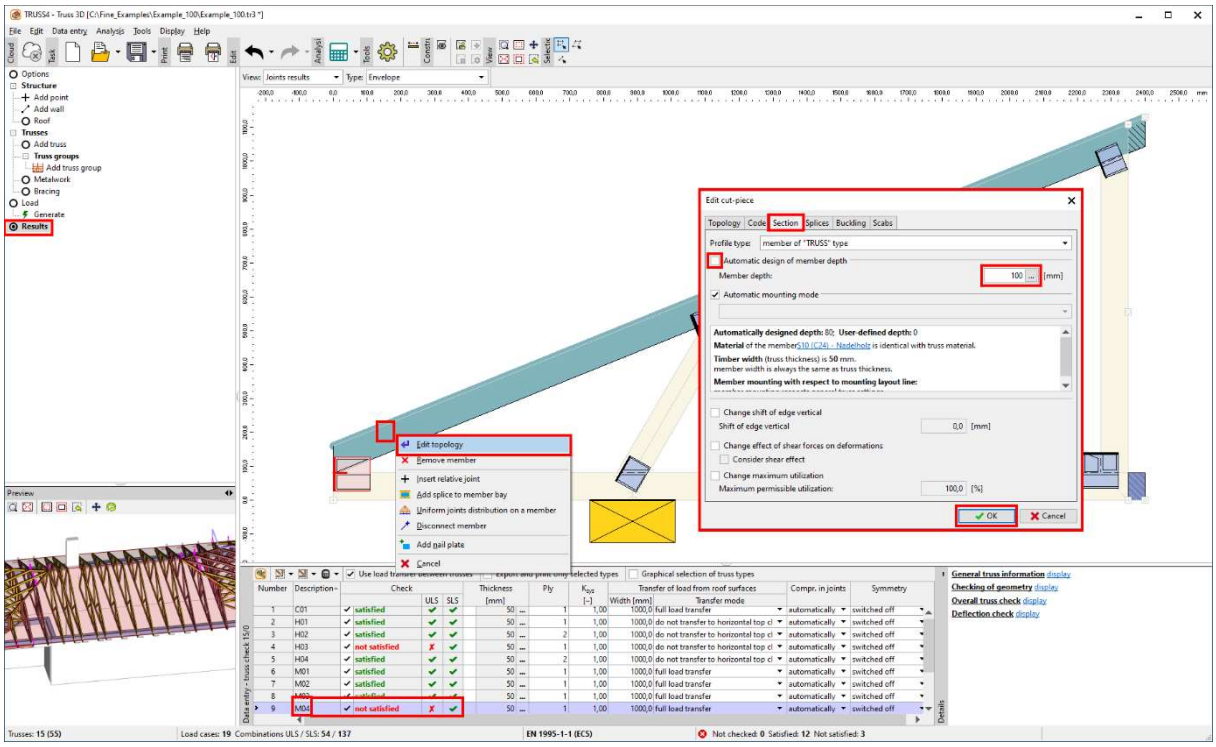


Analysis – Member editing and adjusting trusses

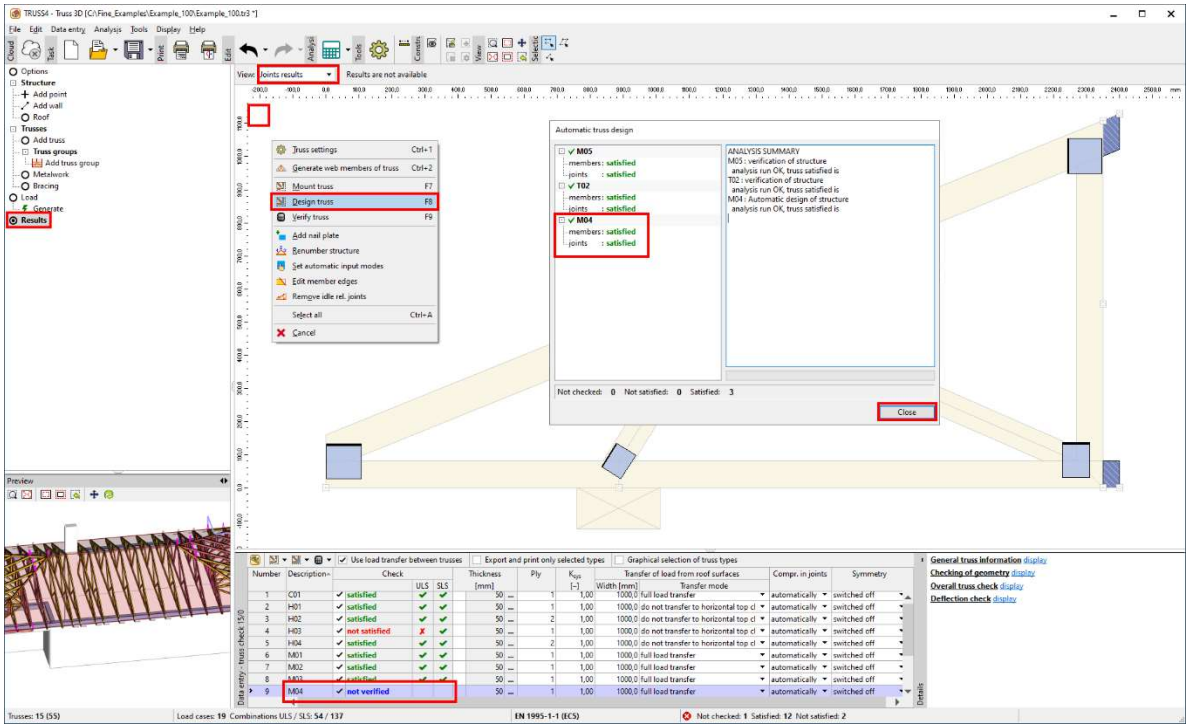
Members and joints that do not satisfy the design criteria need to be checked for the reason of the negative results and manual adjustment must happen (e.g., moving, rotating, changing the size of a plate, increasing the member's cross section, changing the bracing etc.).



In the given case the compressive stress in the joint line is decisive, which we solve by increasing the cross section of the top chord. We perform this by editing the member's topology on the tab "Section", see picture below.

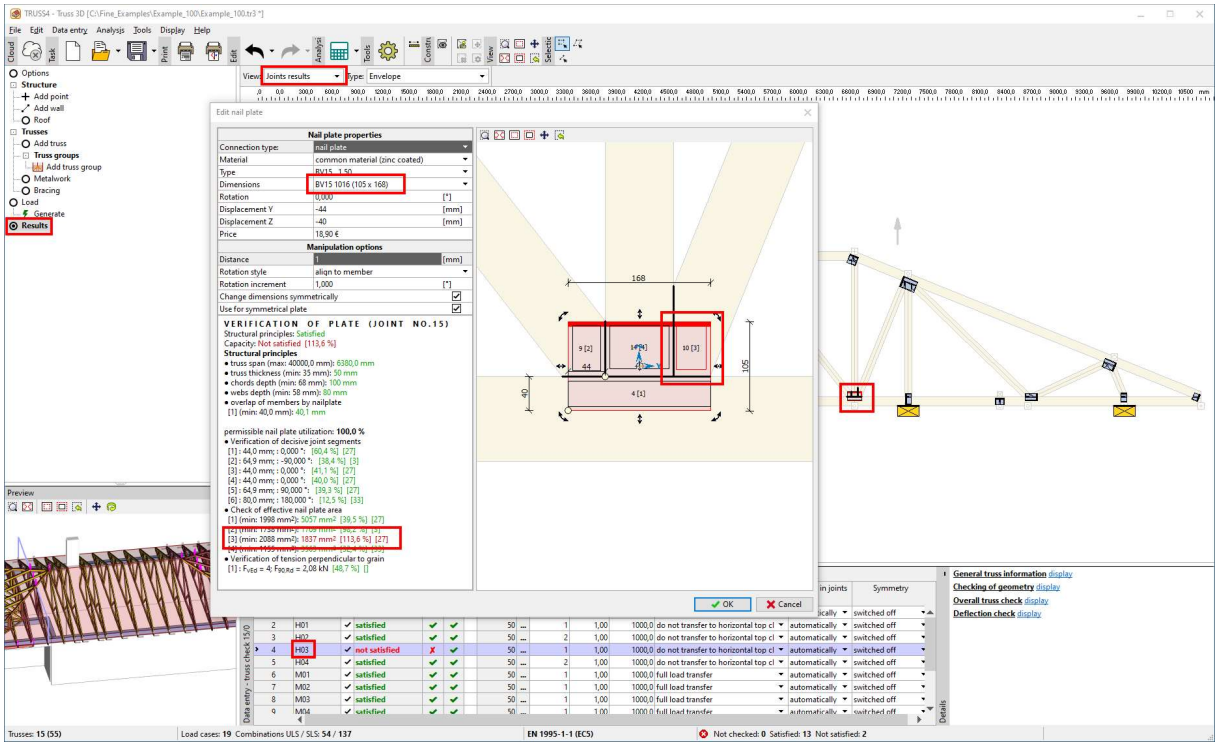


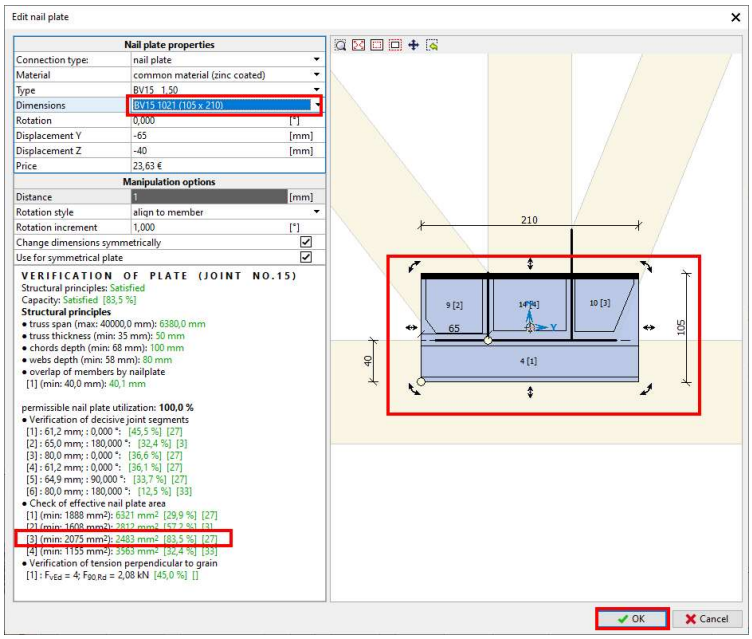
Having performed any changes upon a truss it is necessary to re-design the entire roof structure. It is enough to re-design only the affected trusses, either via the context menu of the workspace or by hitting the key <F8>.



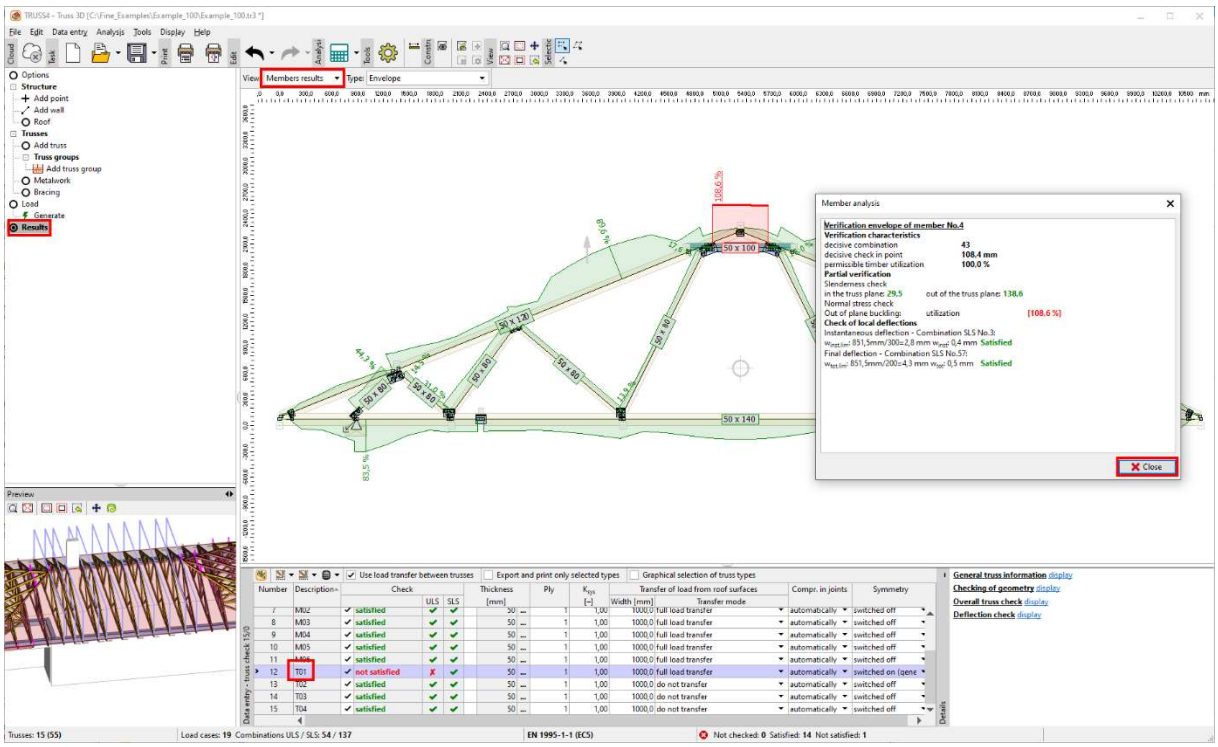
Apparently, the truss design is now satisfied.

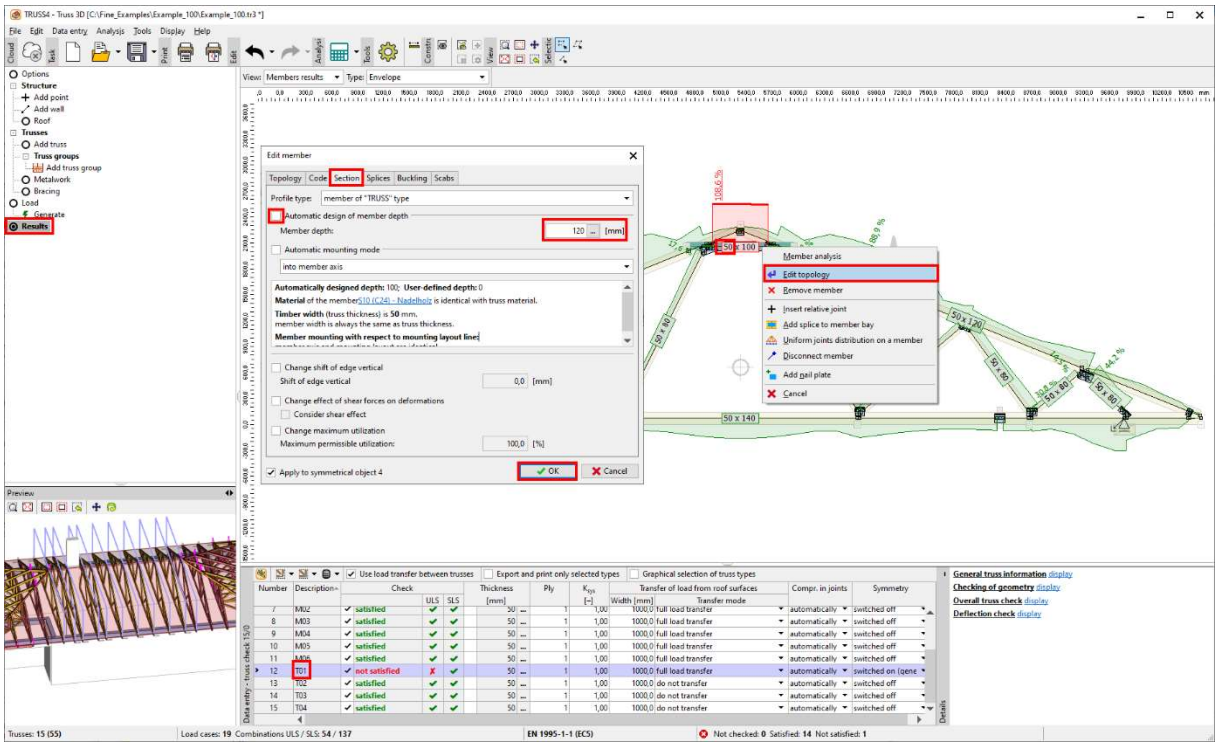
Truss H03 fails in the nail plate design which we solve by increasing the nail plate size.





Truss T01 requires a bigger cross section of the horizontal member below the apex.

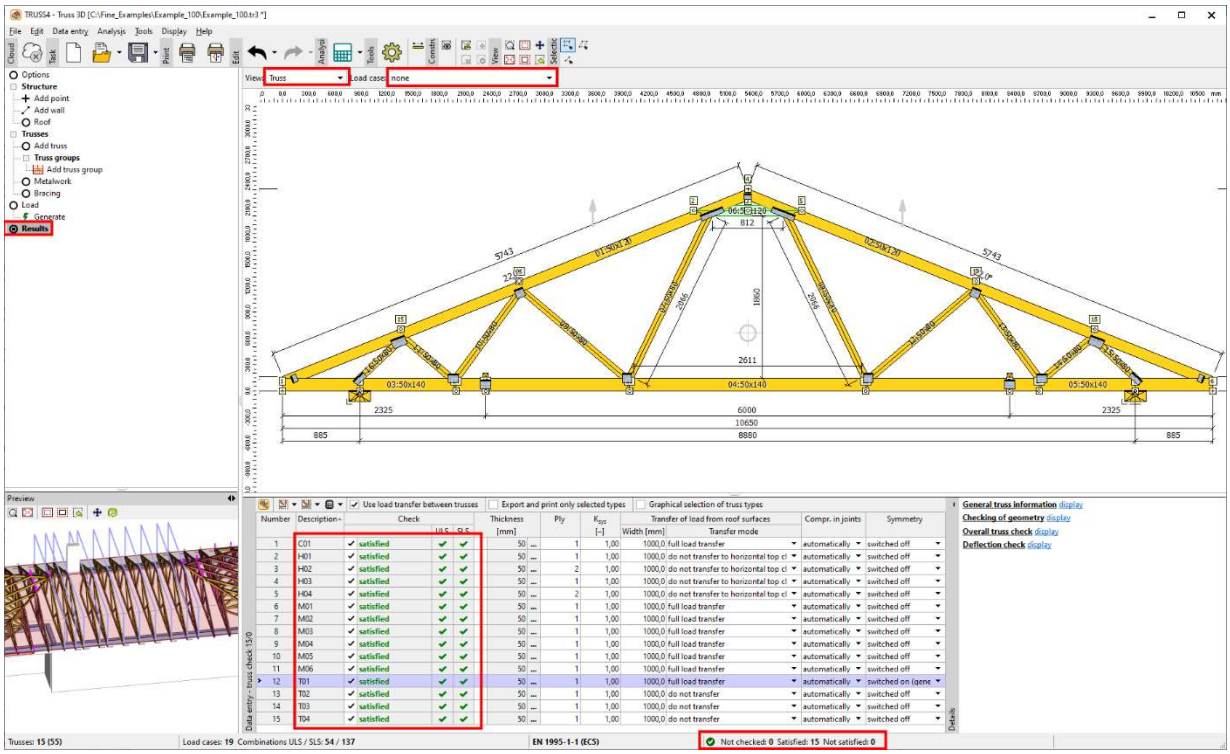




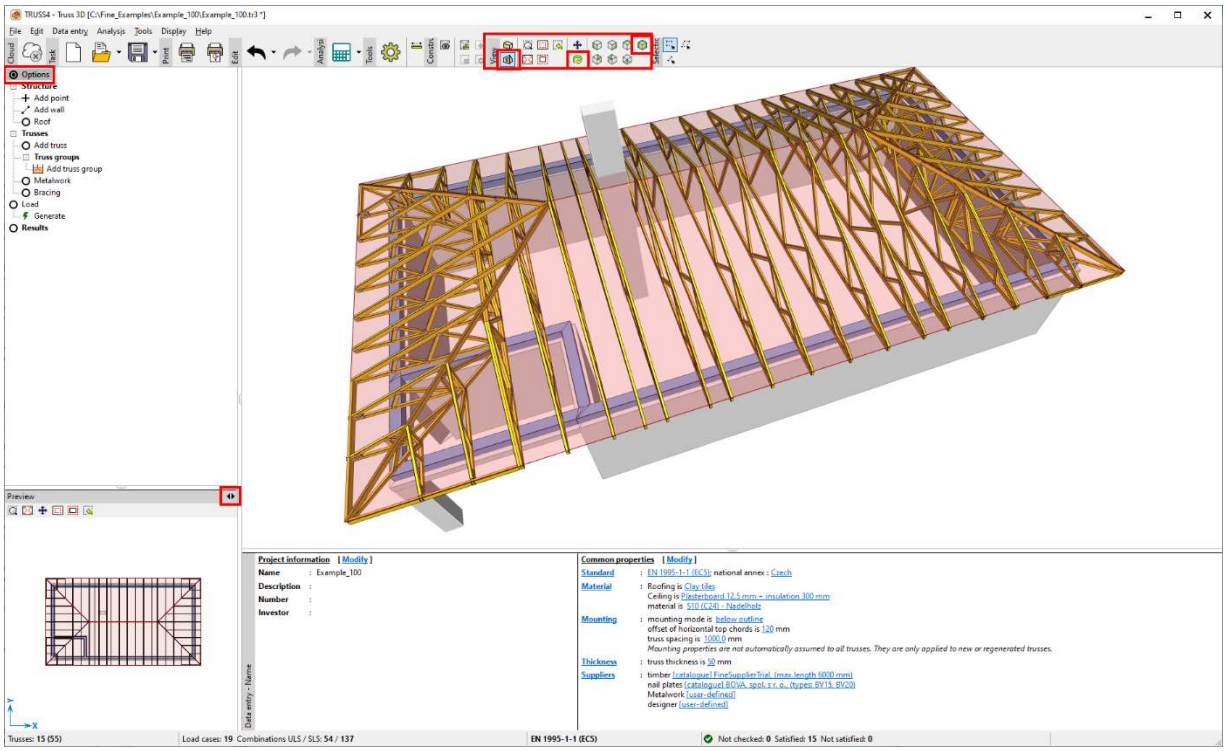
We re-design the truss with <F8>.

Analysis – successful design

Having performed all the required changes all the trusses have been analysed and successfully designed.



Before we proceed to printing and export, we check thoroughly the model for the input requirements and design criteria, if any trusses are missing or if there are any unwanted collisions etc.

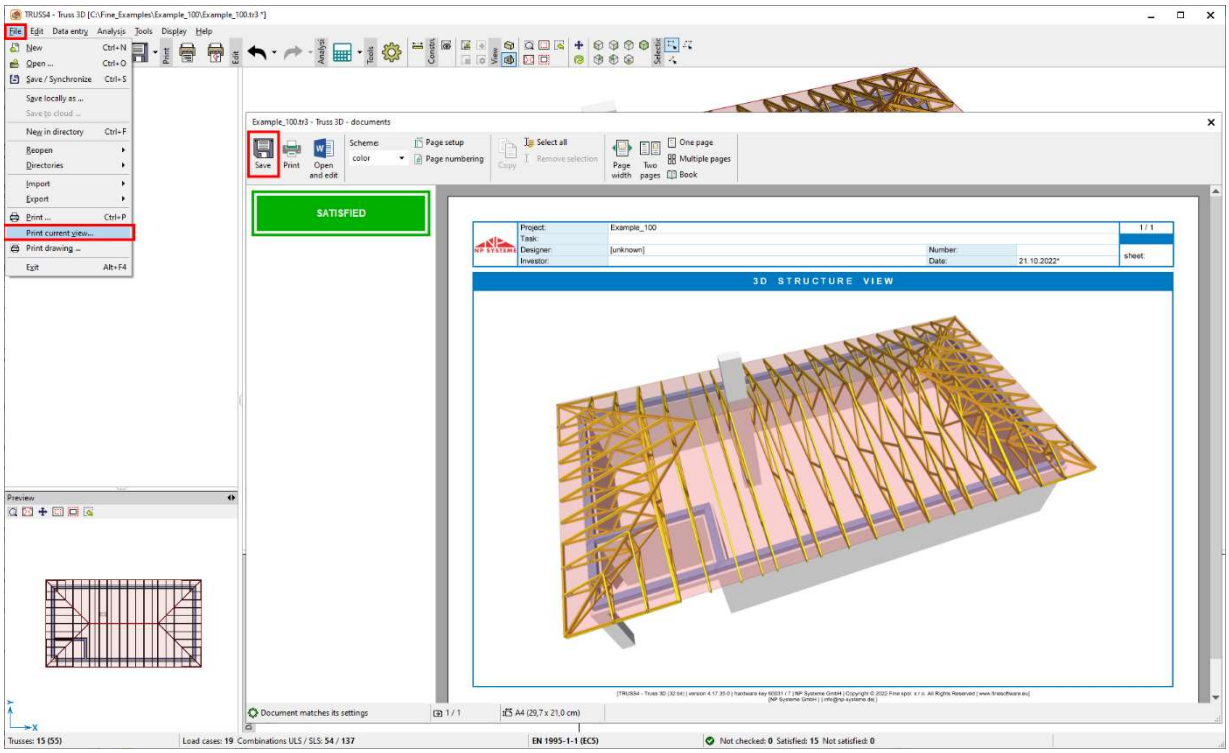


Print – Creating print previews

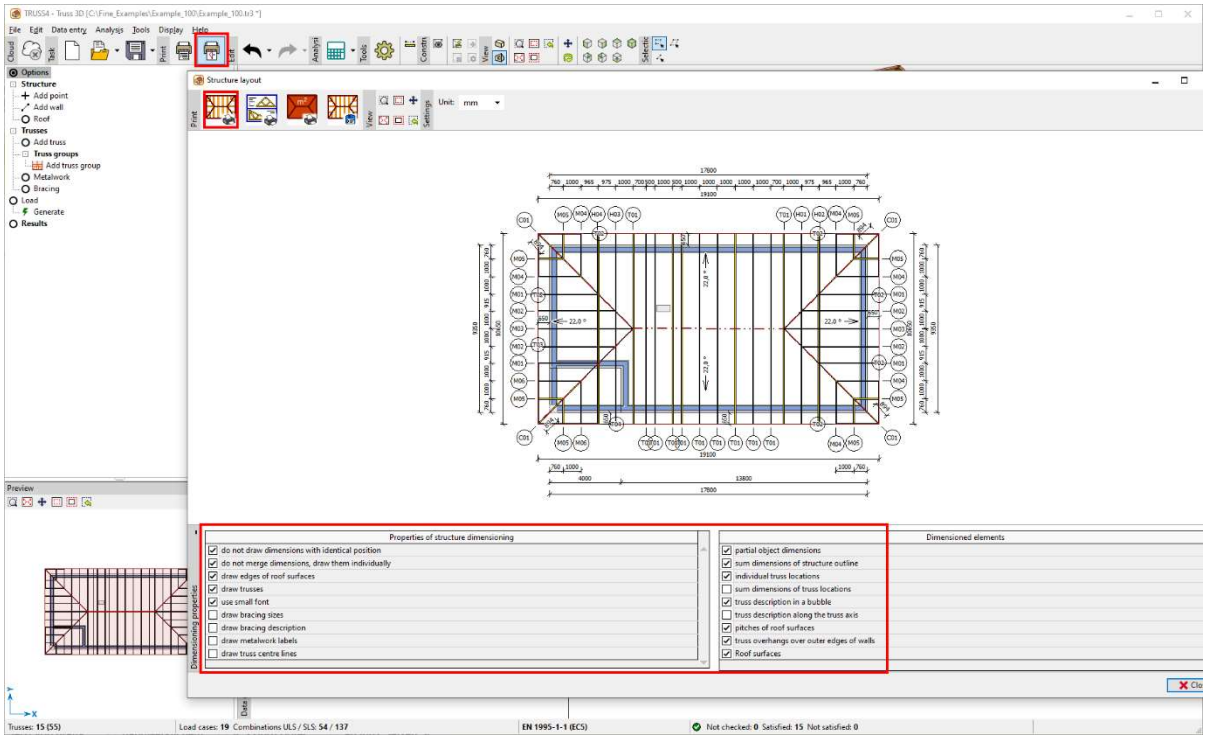
You can print the current view of the workspace via “File/Print current view ...”. This is mainly used for showing the 3d view. For this purpose, the user rotates and zooms into the model according to his own needs. In the options window “Options” should be selected.

NOTE: The 2D and 3D display differs according to the selected section in the options window, where the edited elements are considered. The most suitable section for a model display without text, highlighting or suppressing is “Options”

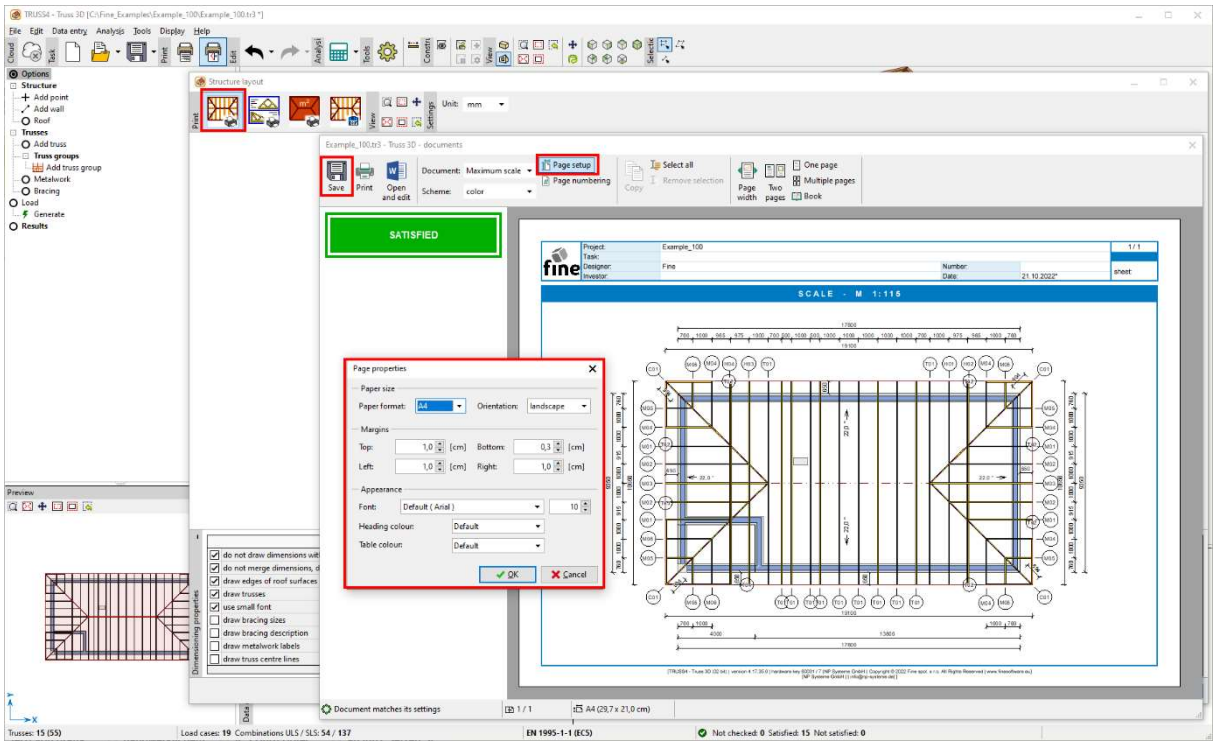
"



Printing the layout drawing is done via “Drawings...”, the user can control the display of elements and dimension lines.



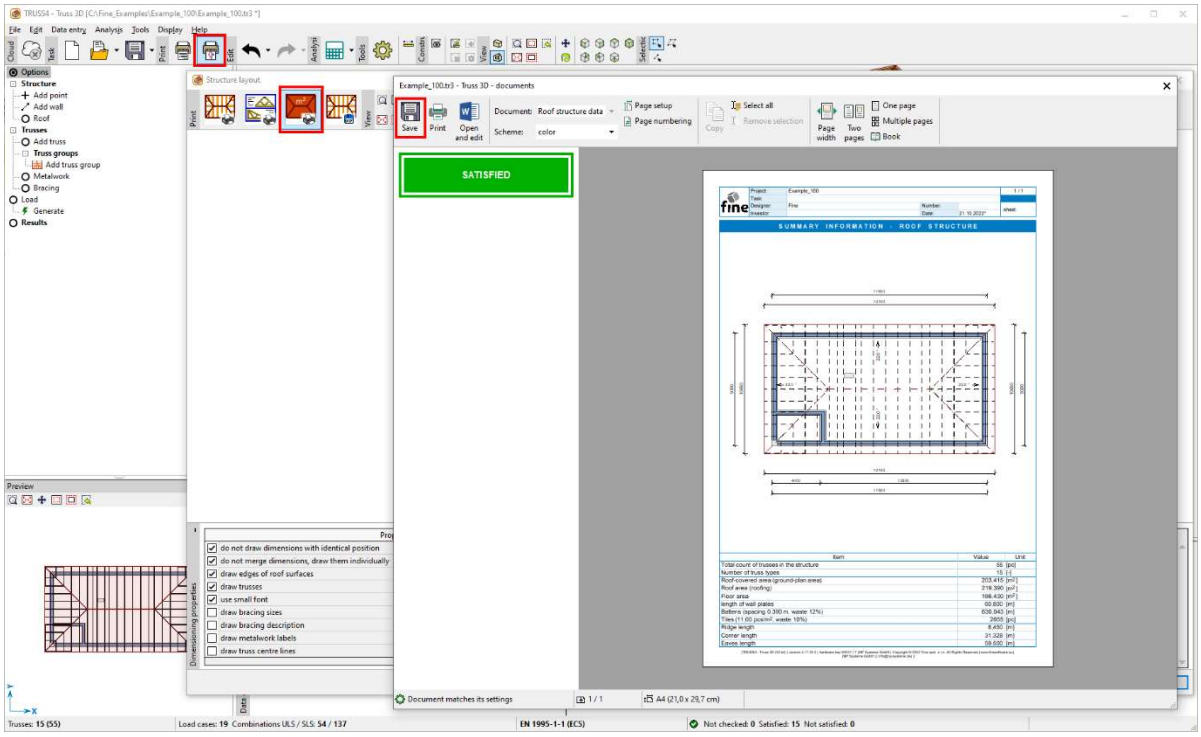
The user can define various paper sizes, margins etc. for the prints.



More print layouts can be created.

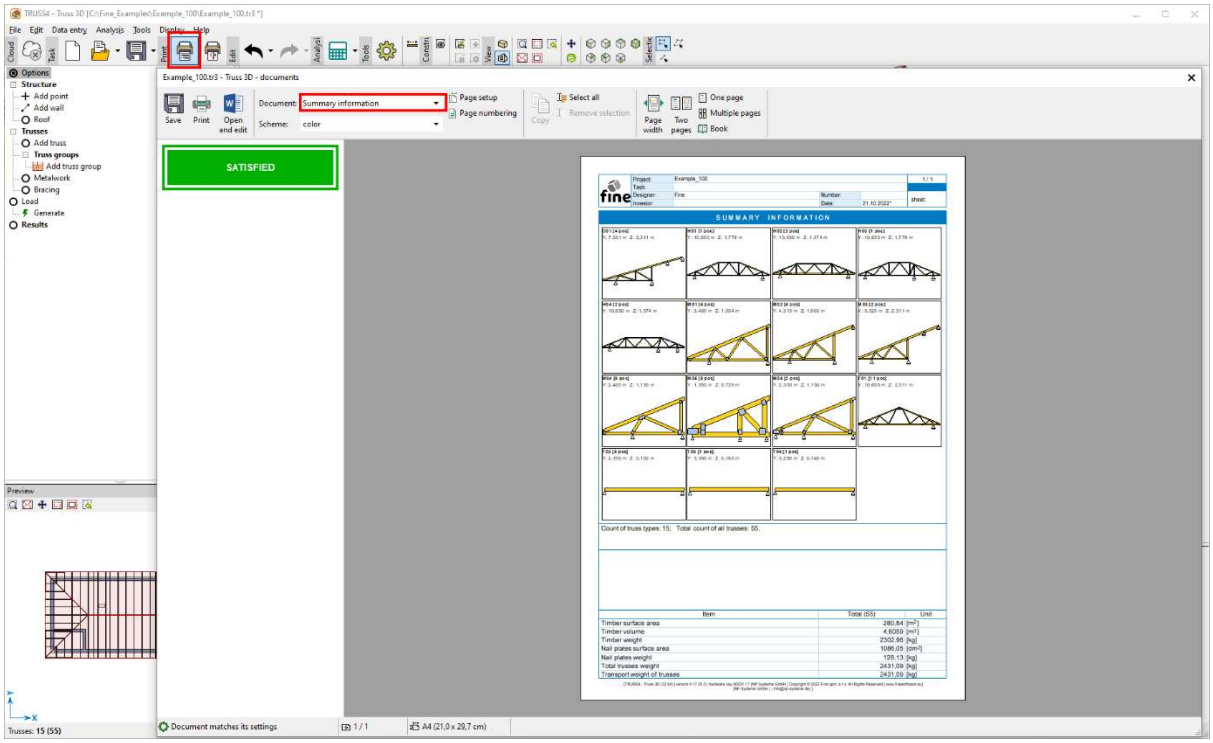
NOTE: You can save all print layouts in PDF format and also in editable formats such as DOCX, RTF, TXT et al.

You can also create 2D DXF views of the layout drawing and of truss views.

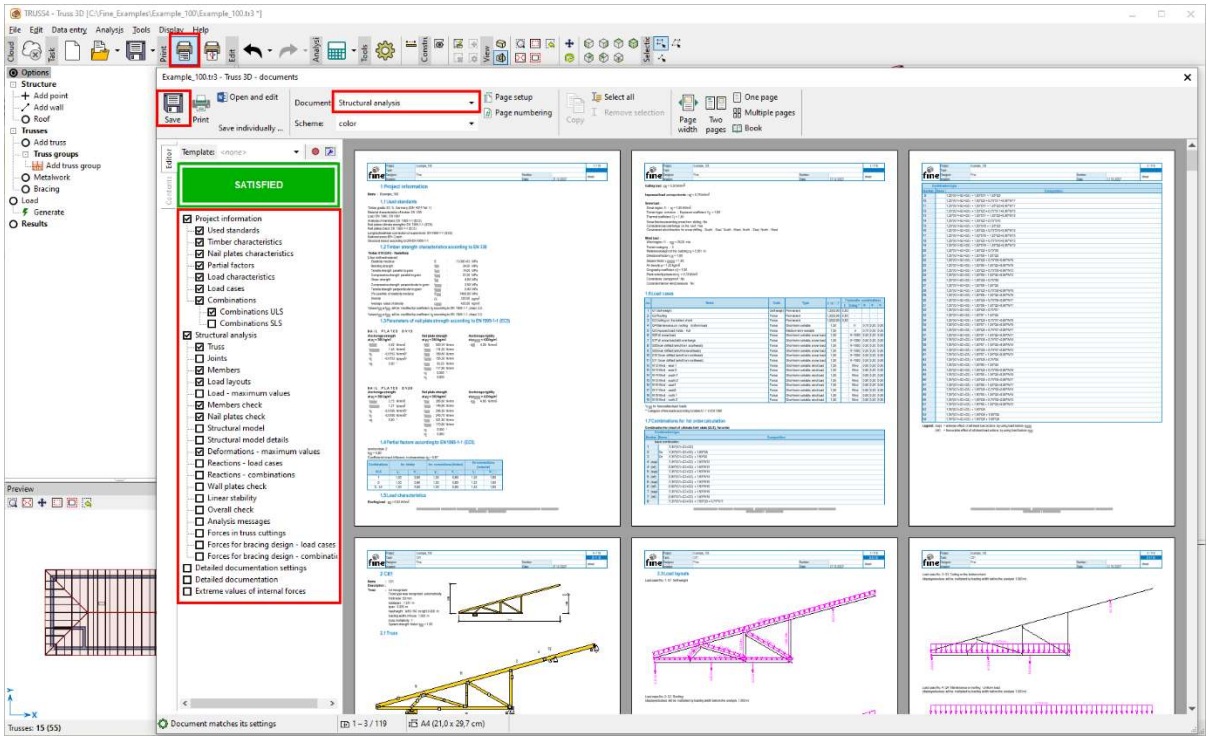


Further print layouts can be created by clicking on the icon “Print” and then selecting from various templates on the drop-down menu.

Summary information.

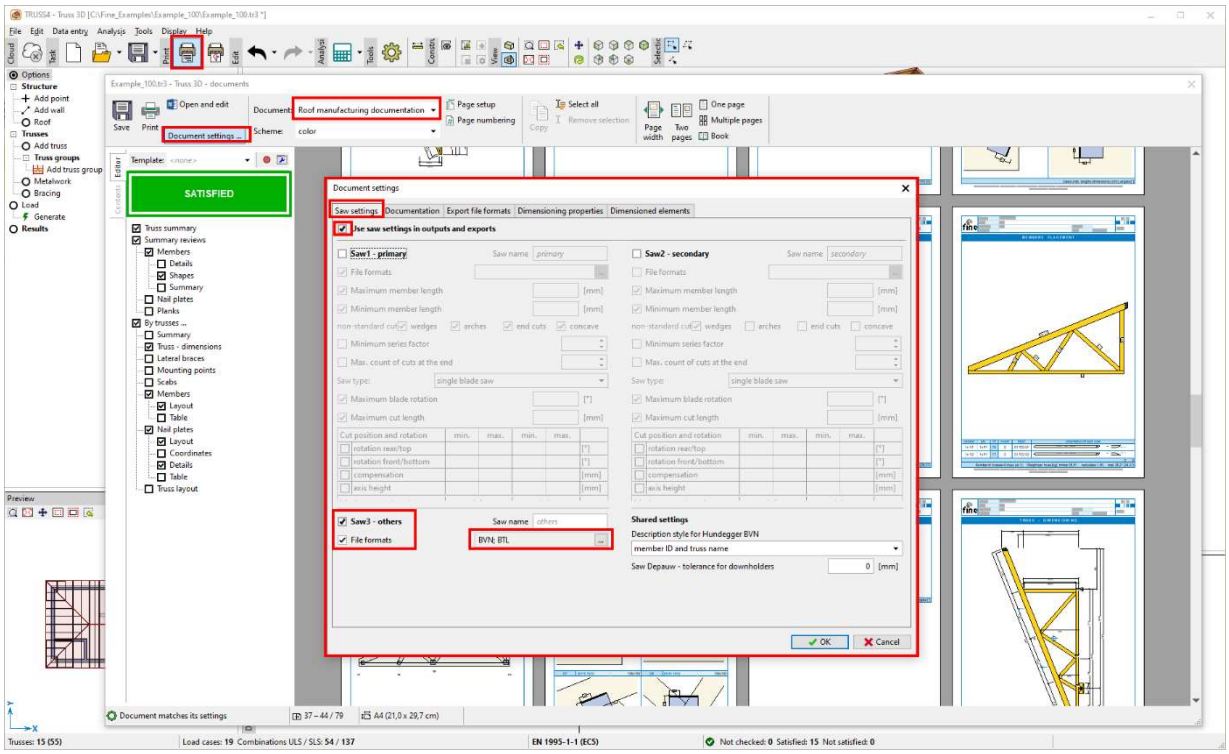


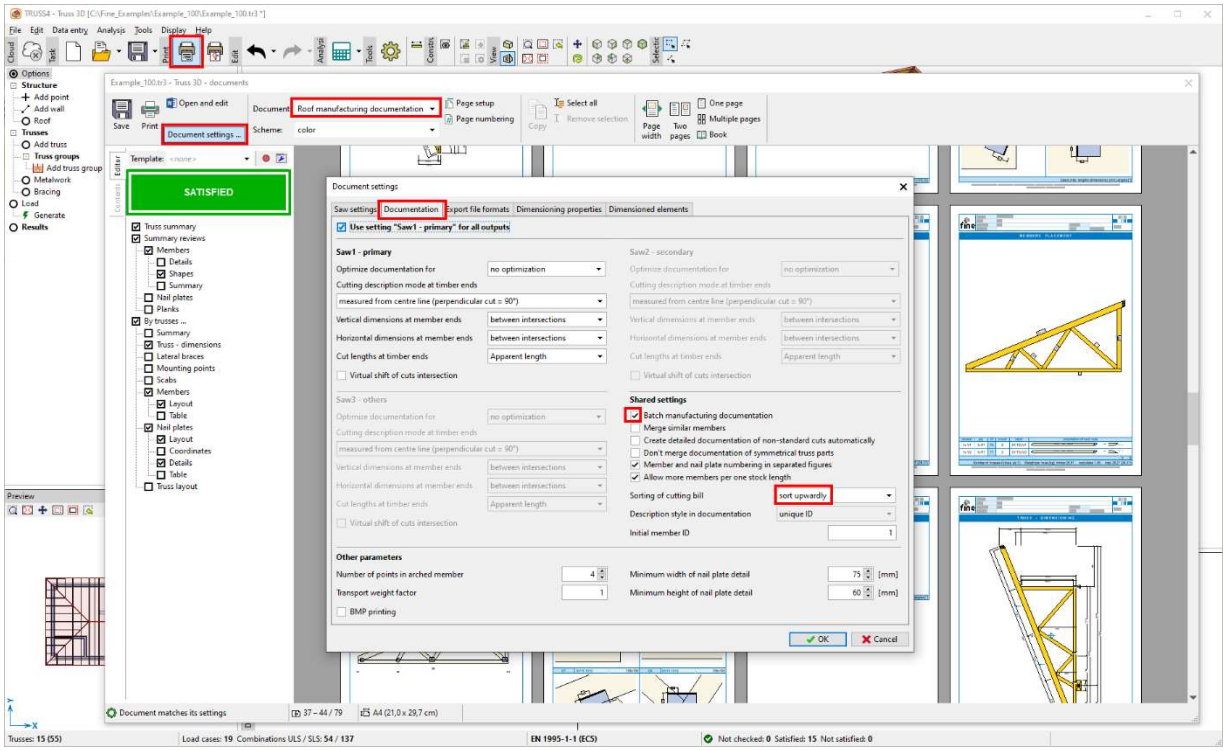
Structural analysis. In the left control window, you can select from the various content components.

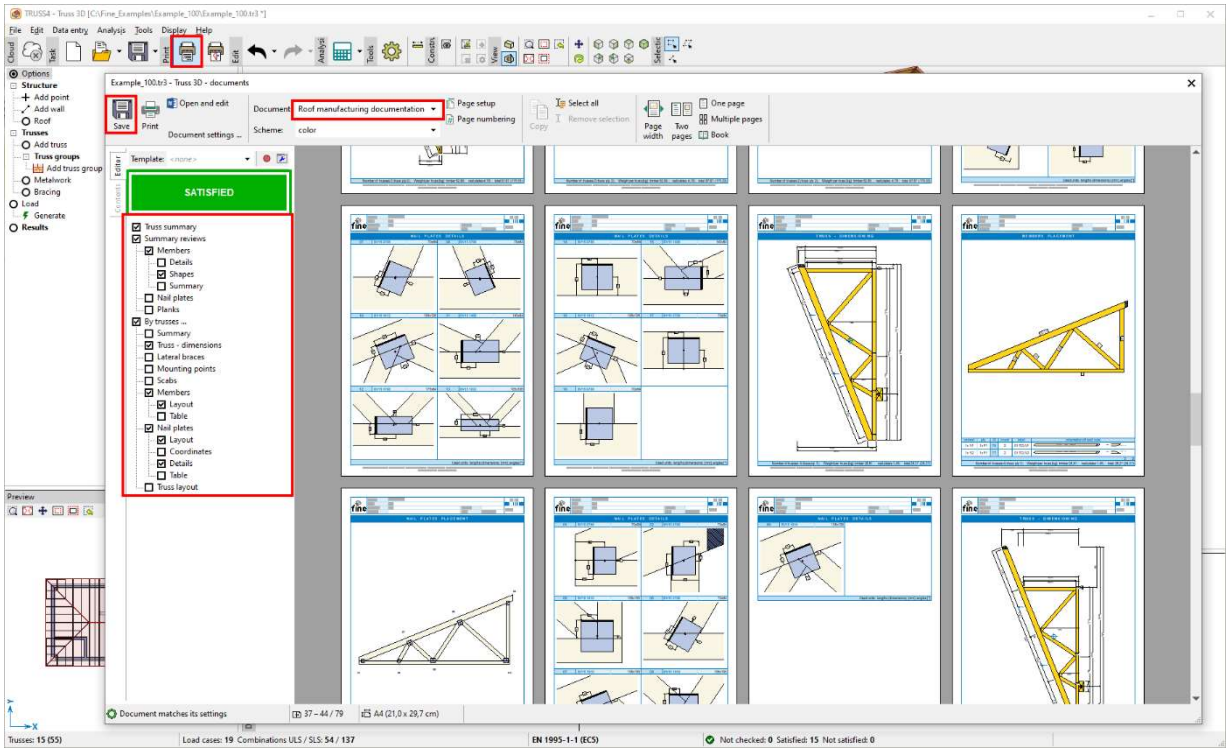


Roof manufacturing documentation. In the left control window, you can select from the various content components.

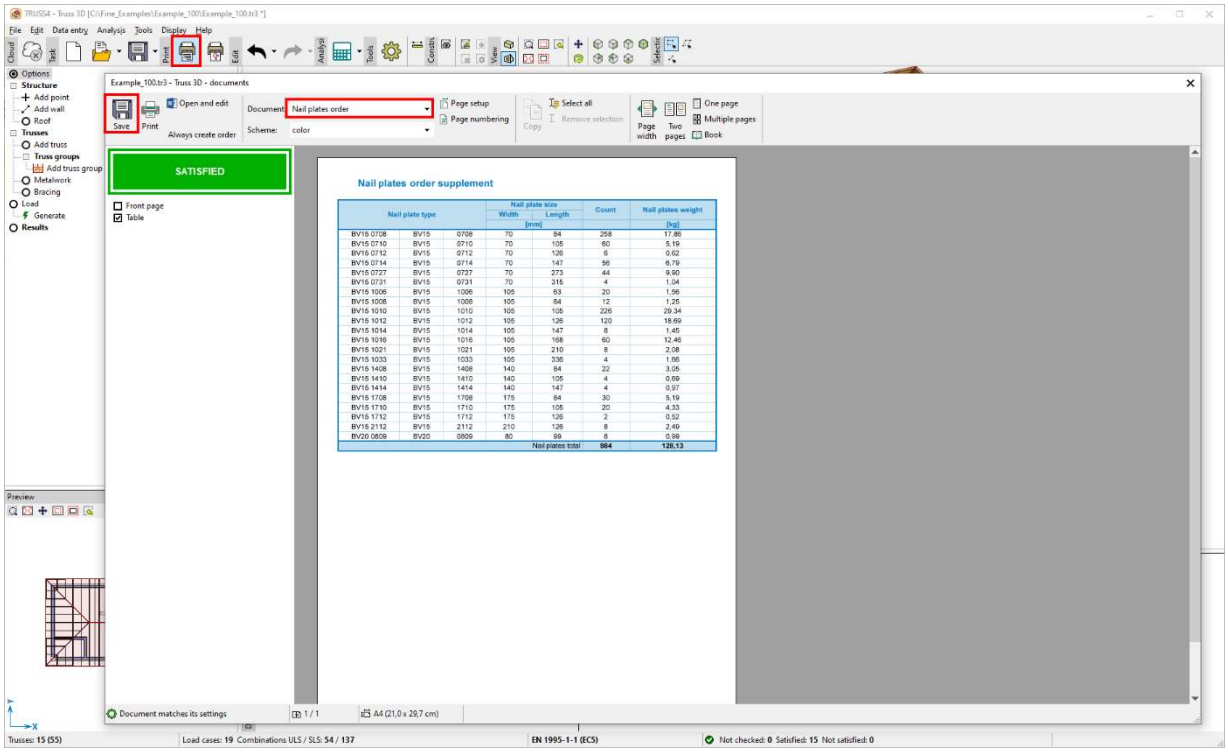
The manufacturing documentation can have many different shapes – “Document settings ...”



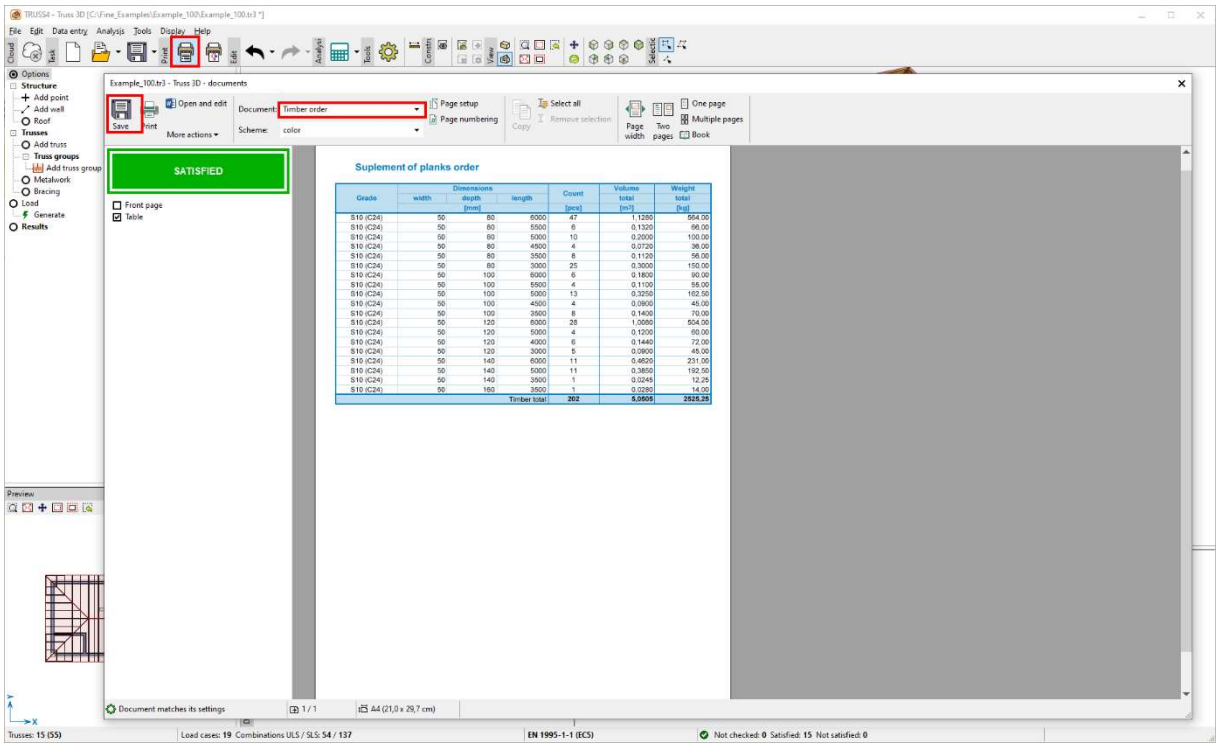




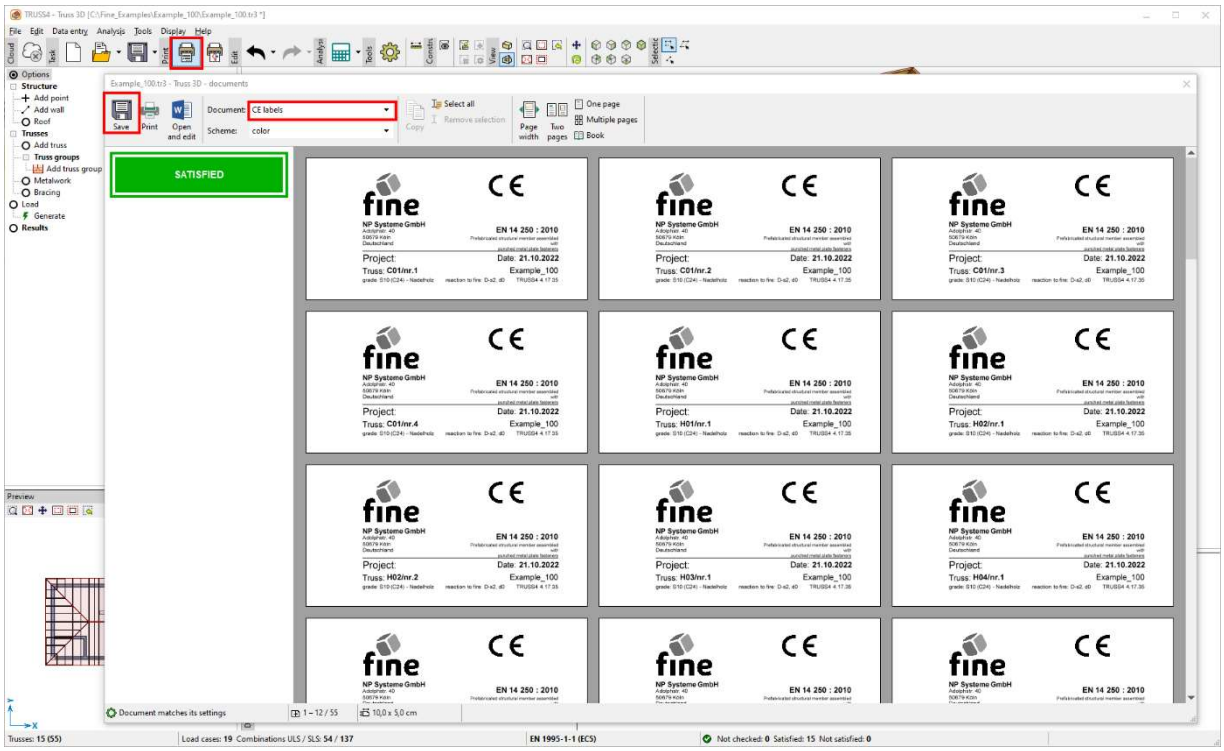
Nail plate orders.



Timber order.



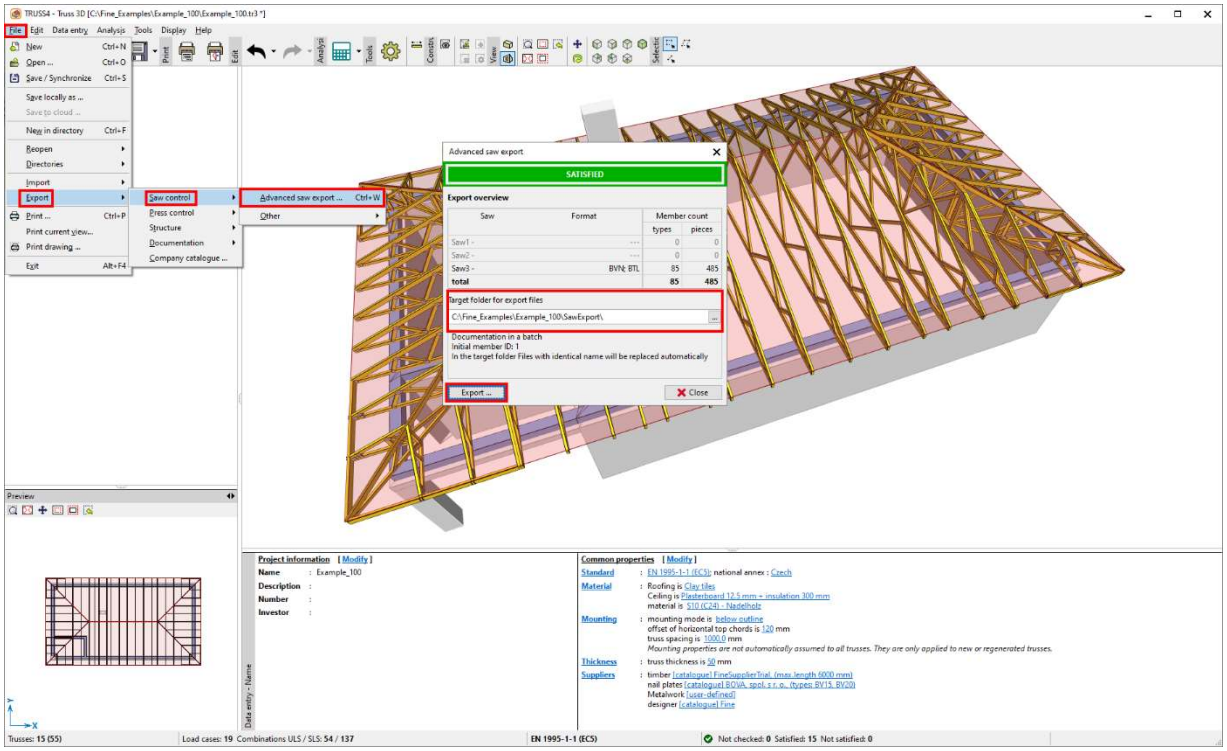
CE labels.



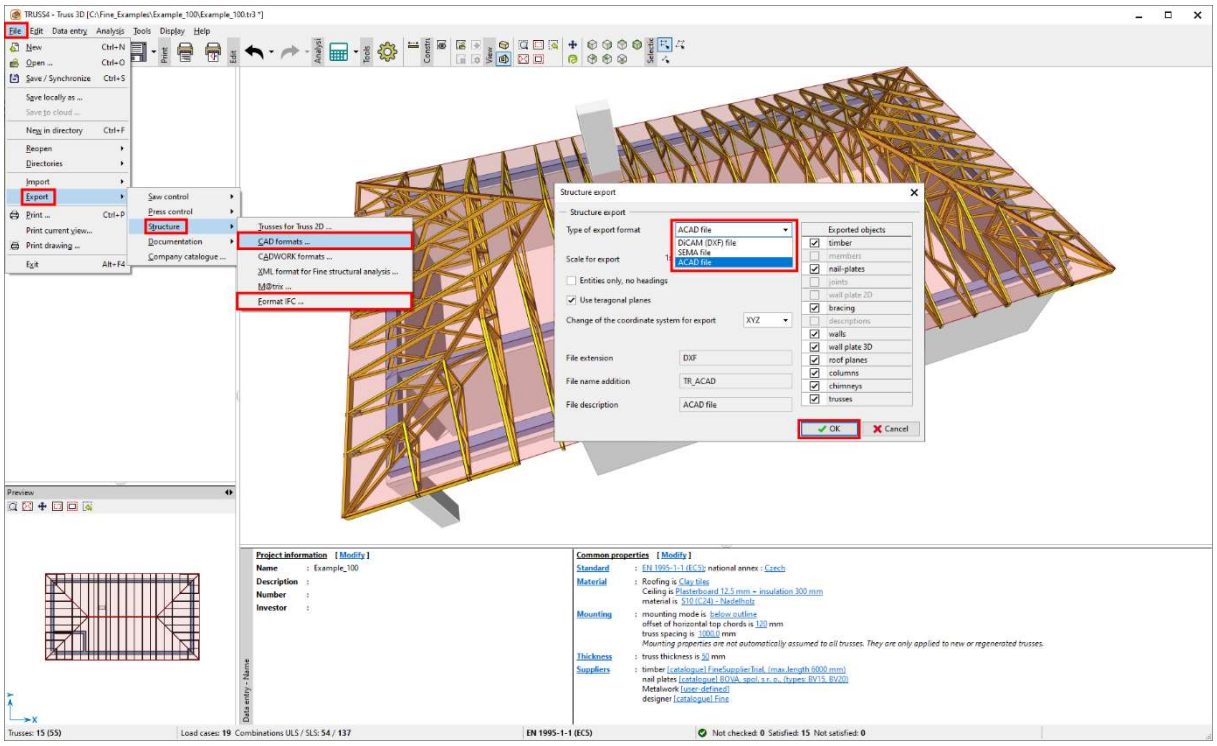
Export – CNC machines, 3D-model

There is a wide range of export files for the variety of machinery equipment (automatic CNC saws, presses, lasers).

Export files for saws



Furthermore, export types are available to other 3rd party programs, such as Dietrichs, Sema, AutoCAD, CADWORK, BIM in the formats of FTD, 3DS, DXF, IFC.



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