Engineering manual Nr. 05

Updated 11/2024

Dividing the truss into production parts

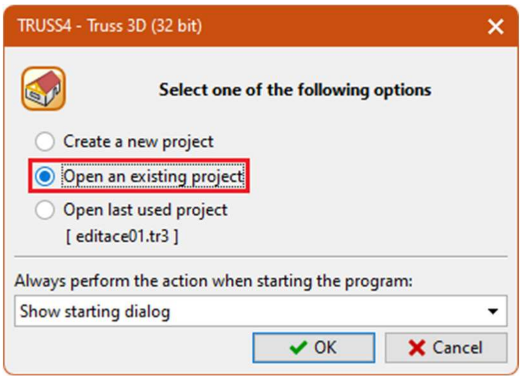
Assignment

Program: Truss 3D
File: FineTrial.tr3

This manual deals with the division of large trusses into production parts. The manual process of dividing a truss will be explained, including the work with structural documentation outputs. The basis for the work will be a 22m span duo-pitched truss with a 7° slope and a edge vertical height of 800mm.

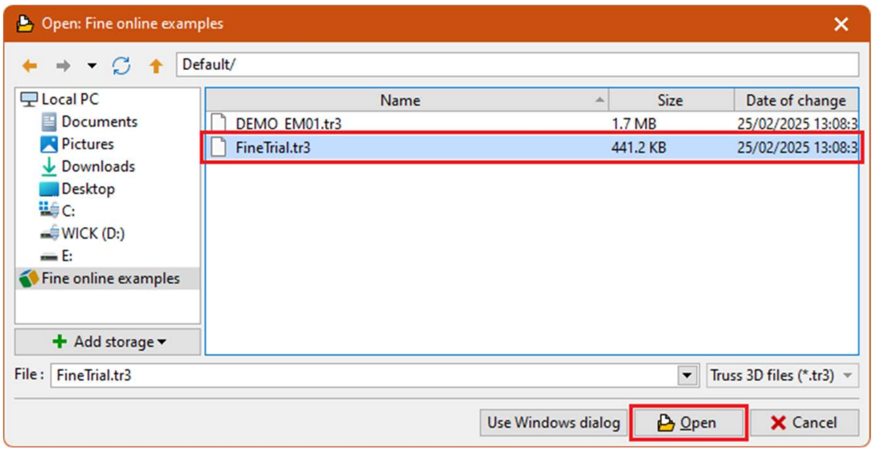
Creating a truss

Start the "**Truss 3D**" program. To work with identical settings, we load a predefined empty file as a template at the beginning. This will ensure consistent results even for installations containing customized default settings for a specific market. In the initial window, select "**Open an existing project**" and confirm with "**OK**".



Initial program offer

This will launch the "**Open**" window for selecting a project file. We select the existing *FineTrial.tr3* file from the *Fine Online Examples* folder. Confirm the selection with the "**Open**" button.

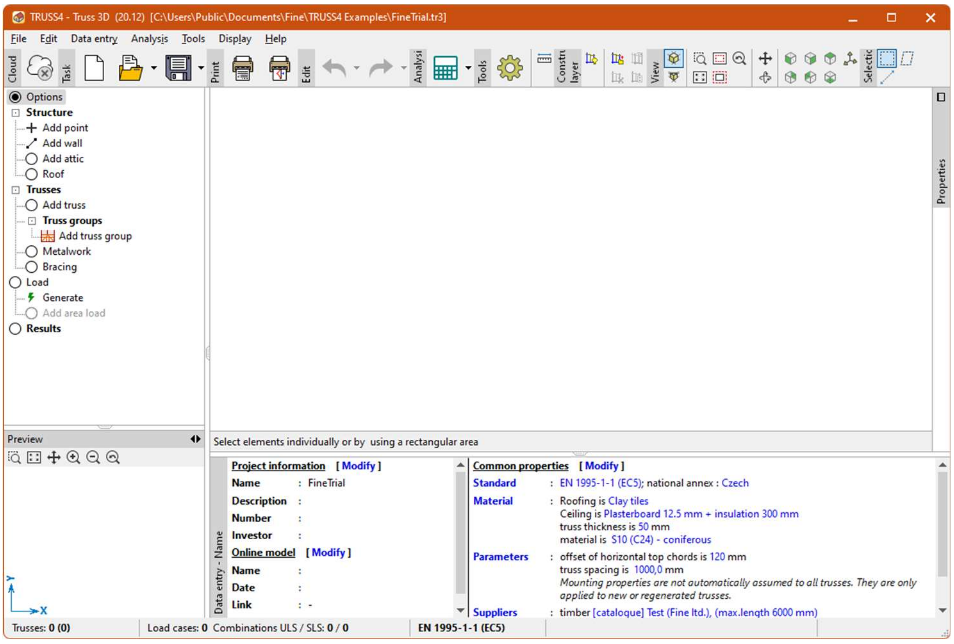


Opening a file

When you open the file, a window may pop up warning you that we will be using a timber or nail plate database that we do not yet have in our catalogue. We can skip this warning by clicking the "**Close**" button.

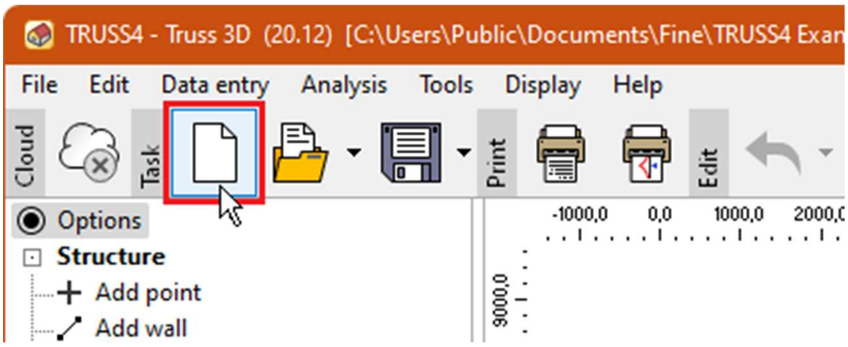
Option to add a supplier to the catalogue

This step will load the settings that we will use during the work. The main "**Truss 3D**" window will appear.



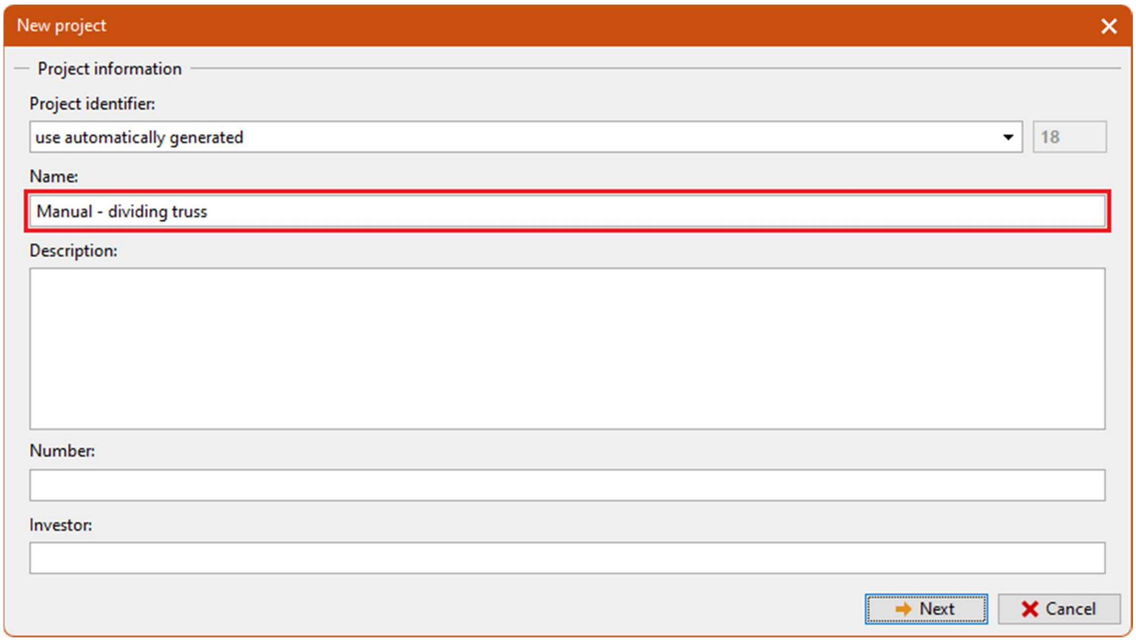
Main window of "Truss 3D"

Now we can create a new project. We will use the "**New**" button in the main toolbar for this. Alternative way is to use the keyboard shortcut *Ctrl+N*.



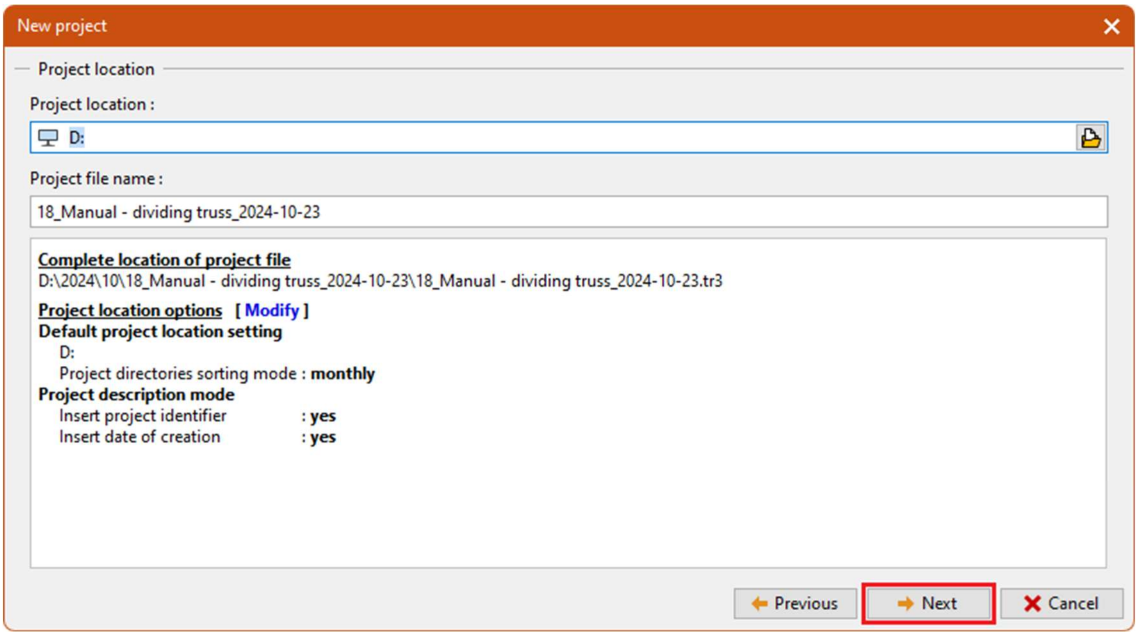
The "New" button

The wizard for setting up a new project will start. On the first page, just fill in the project name. Then you can continue with the "**Next**" button to the next page.



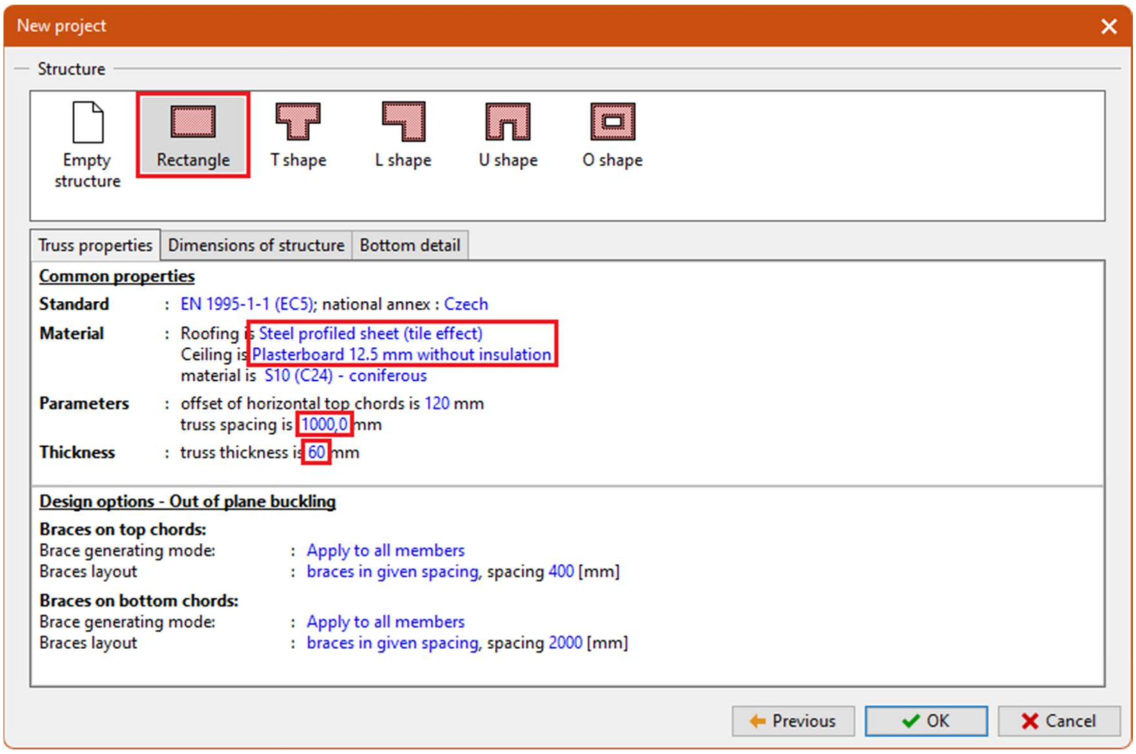
Input of the project name

We can change the directory for saving the project in the next window and, if necessary, modify the rules for sorting projects. We skip this window by clicking "**Next**".



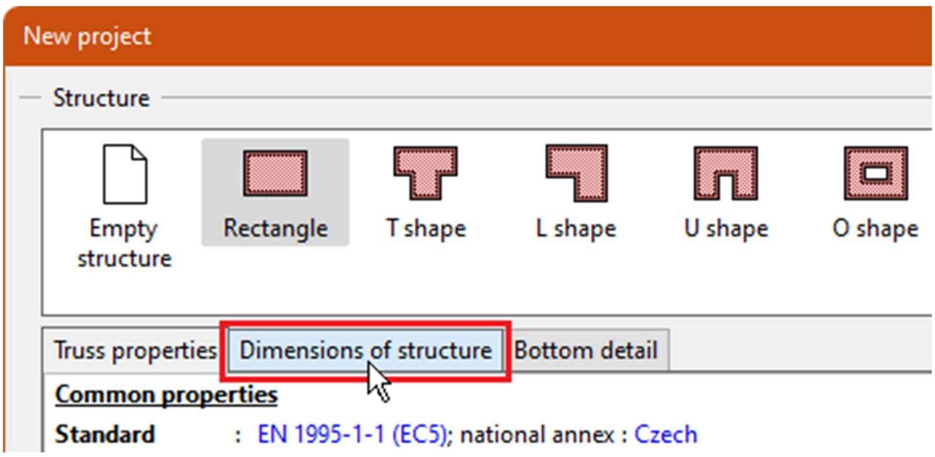
Project location and sorting

The following window allows you to select the basic shape of the building (select "**Rectangle**") and the basic parameters of the project. Enter the typical truss spacing of *1000mm*, the truss thickness of *60mm* and select the roofing and ceiling material. Specifically, the items "**steel profiled sheet (tile effect)**" for the roofing and "**Plasterboard 12.5mm without insulation**" for the ceiling. The default database of roofing and ceiling may vary in installations for specific markets. If the material database does not contain the above items, we will select others with similar weights. This is *0.15kN/m²* in both cases.



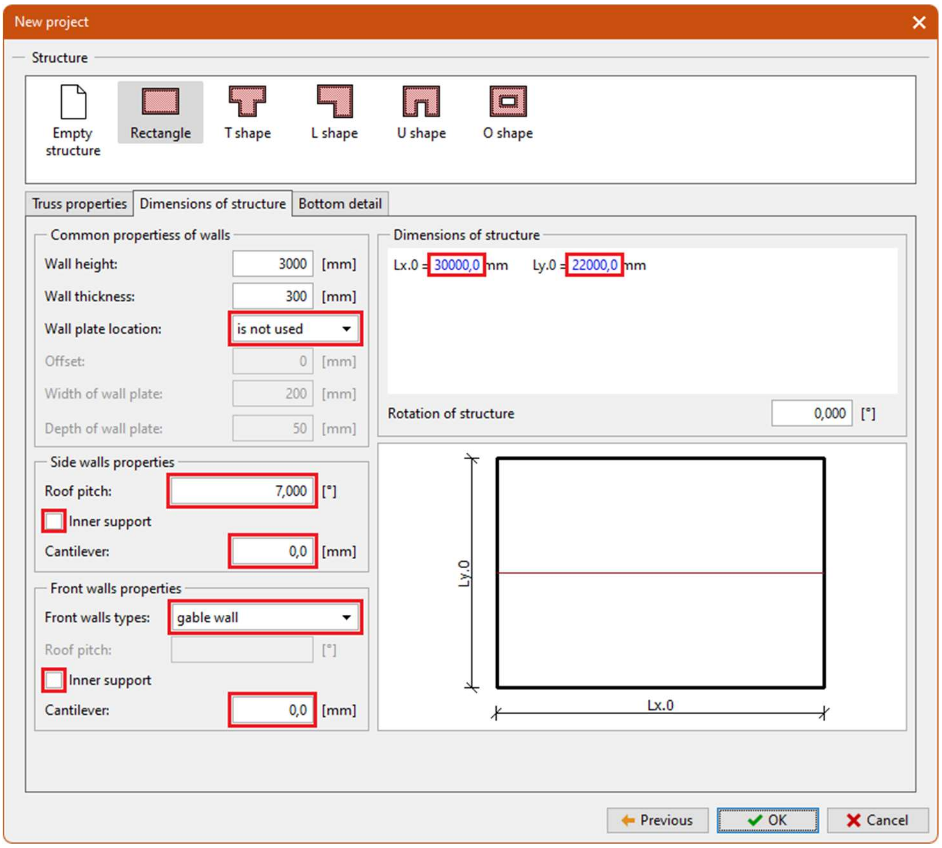
Basic project parameters

Then switch to the second tab of the dialog box, which is called "**Dimensions of structure**".



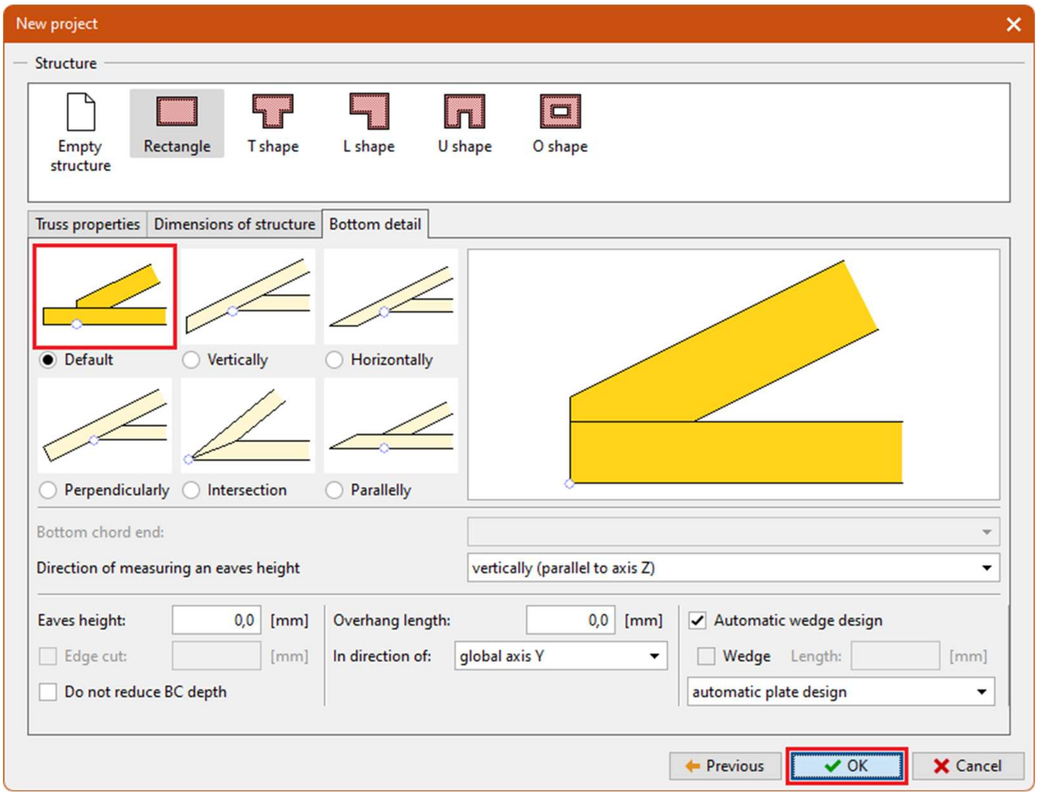
Switching to the "dimensions of structure" tab

On the "**Dimensions of structure**" tab, enter the parameters according to the following figure. The length of the building will be *30000mm*, width *22000mm*. The structure will not contain wall plates. For the side walls we will enter a slope of *7°*, for the front walls we will choose the "**gable wall**" option. For both types of walls, disable the "**Inner support**" switch and specify a truss edge distance of *0mm*.



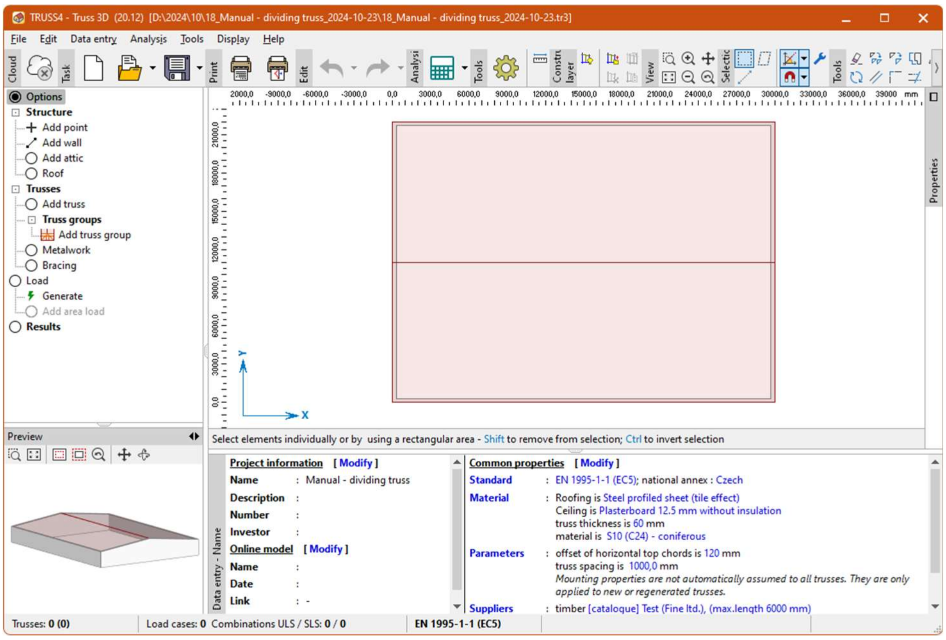
Changes in the "Dimensions of structure" tab

On the last tab "**Bottom detail**", we just make sure that we have selected the detail type "**Standard**".



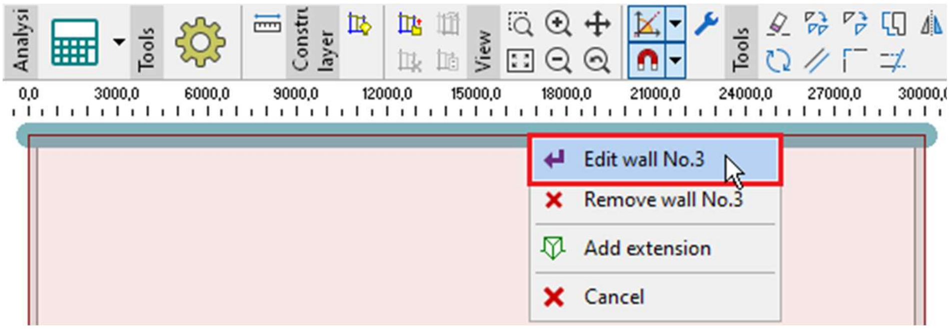
The "Bottom detail" tab

Then we can exit the wizard with the "**OK**" button and return to the main program window.



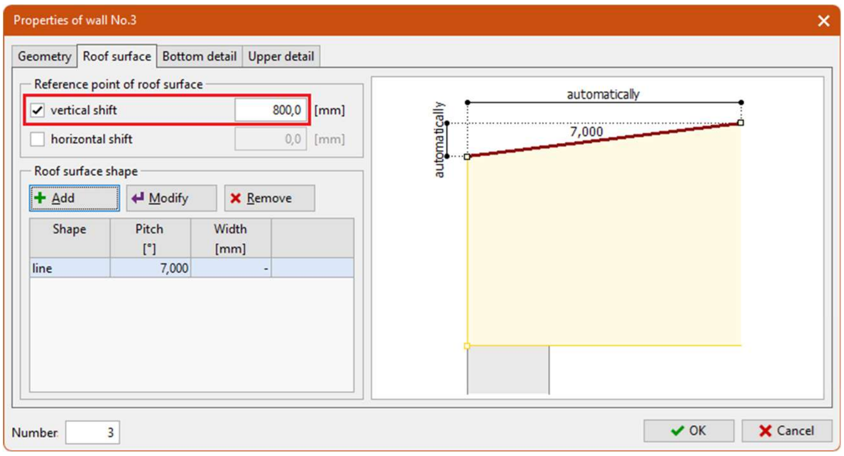
Main program window with the created structure

The initial wizard did not allow us to enter an eaves detail with an edge vertical. Therefore, we will now modify the eaves walls that carry the roof plane information and change the detail. Right-click on the wall *3* and select "**Edit wall No.3**". Alternatively, we can also double-click on the wall with the left mouse button.



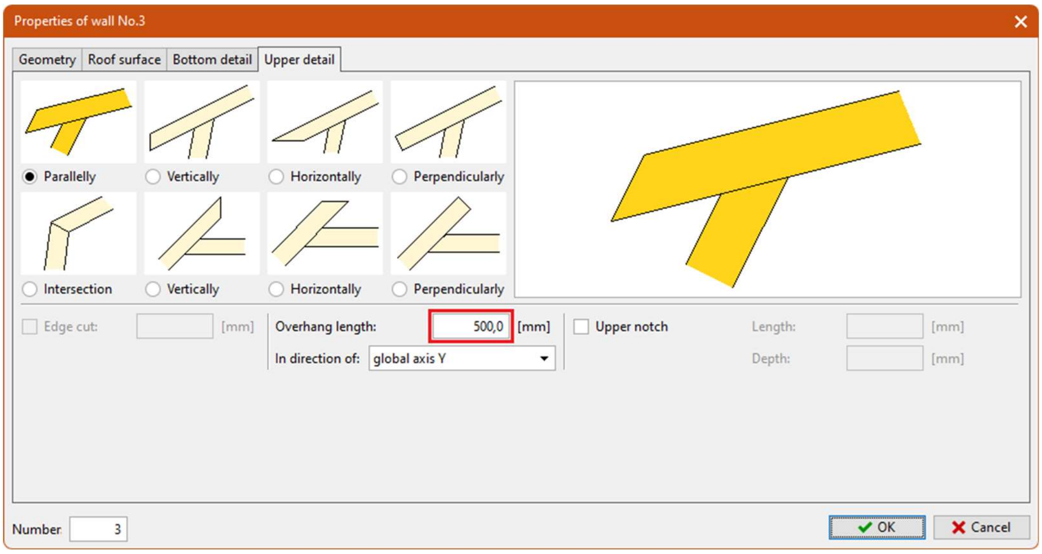
Edit wall properties No.3

Go to the "**Roof surface**" tab and check "**Vertical shift**". Then enter the value 800mm in the input field. Then switch to the "**Upper detail**" tab.



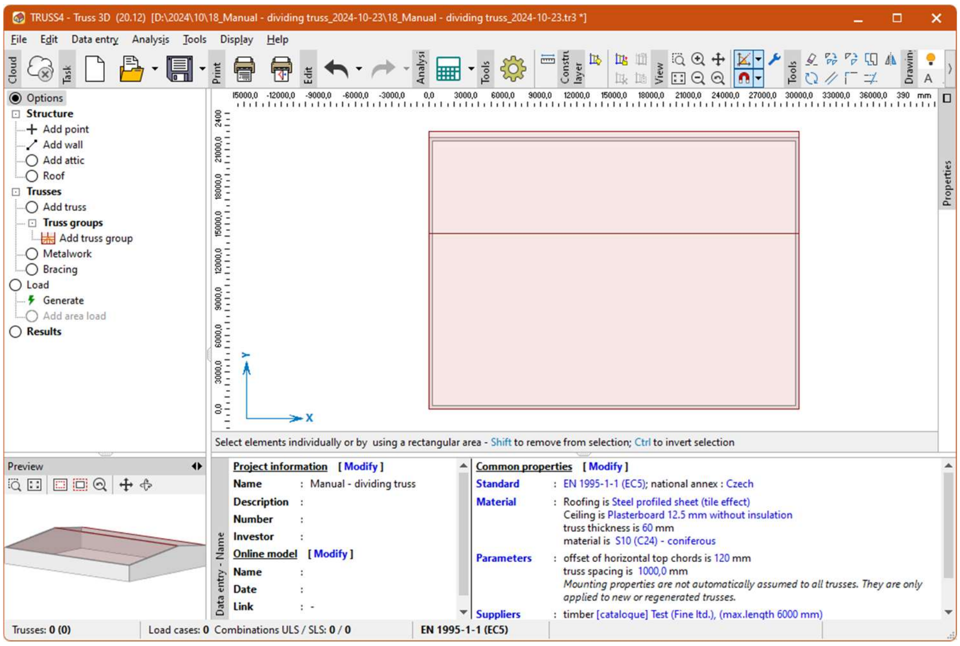
Input of vertical shift of the roof plane

In the "**Upper detail**" tab, enter the value *500mm* in the "**Overhang length**" item. Then you can close the wall properties window with the "**OK**" button.



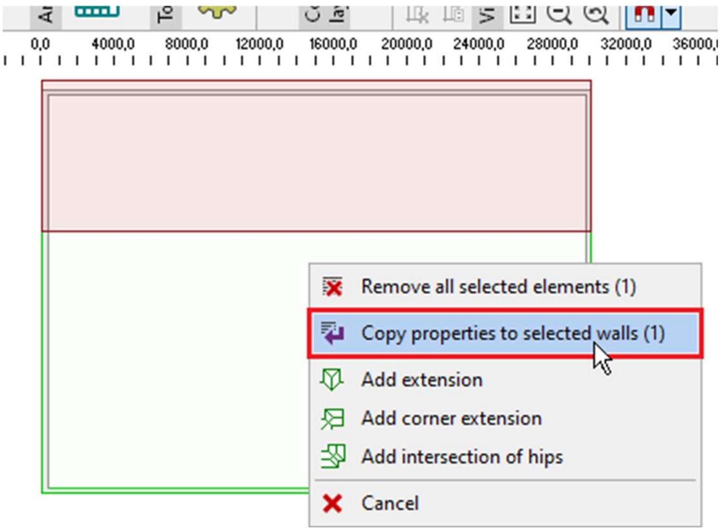
Input of overhang length

When we return to the main window, the roof planes are redrawn and the structure is now asymmetrical.



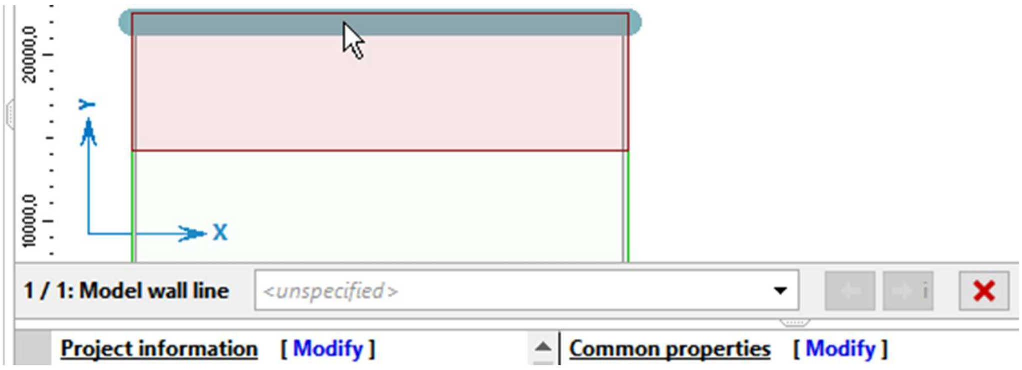
Structure with one modified wall

The final step in specifying the shape of the roof is to copy the properties of the roof plane to the other side of the building. First, select the bottom longitudinal wall by left-clicking. The selected wall will be highlighted in green, including the associated roof plane. Next, right-click above the workspace and select "**Copy properties to selected walls**" from the context menu.



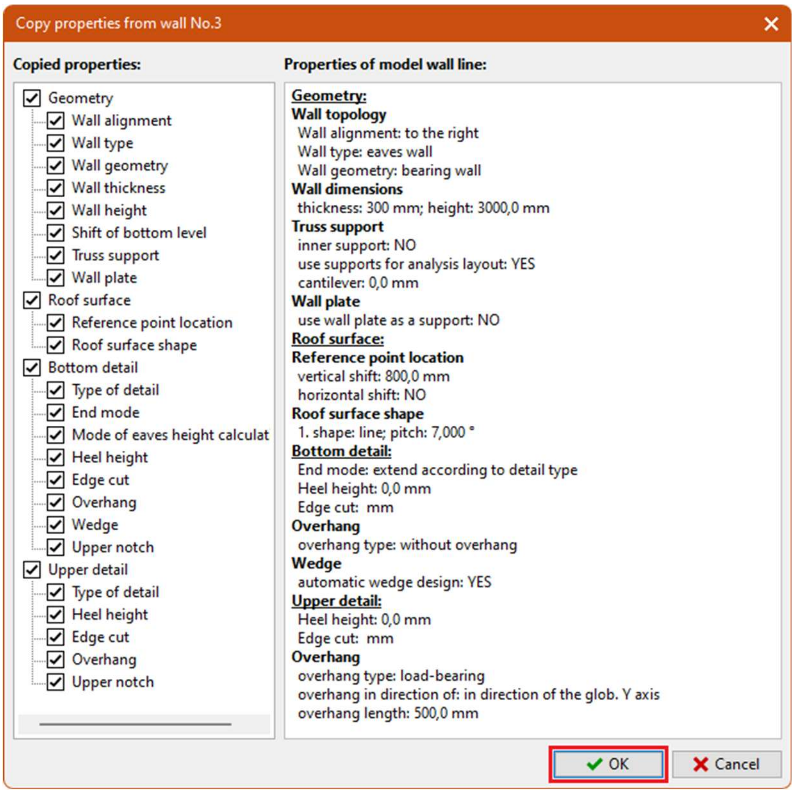
Tool for copying wall properties

The "**Copy properties to selected walls**" tool prompts us to select the walls to which the properties are to be transferred. Select the top longitudinal wall and confirm the selection with the right mouse button.



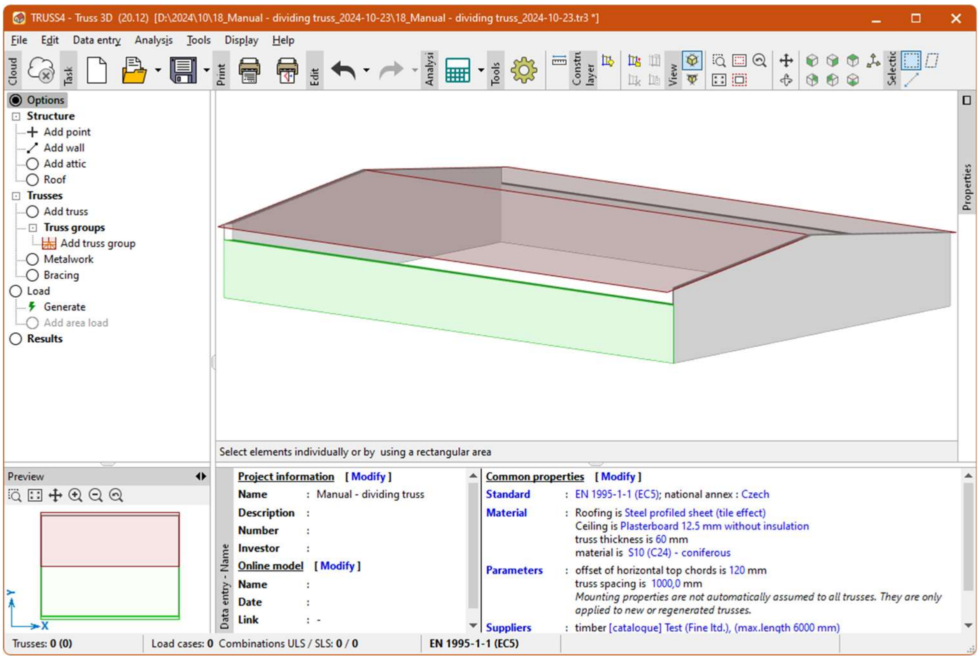
Selecting the wall where properties are copied

After confirming the selection of the walls, a window appears in which we can select the properties to be copied. We can copy all properties, so we just confirm the window with the "**OK**" button.



Window with copied wall properties

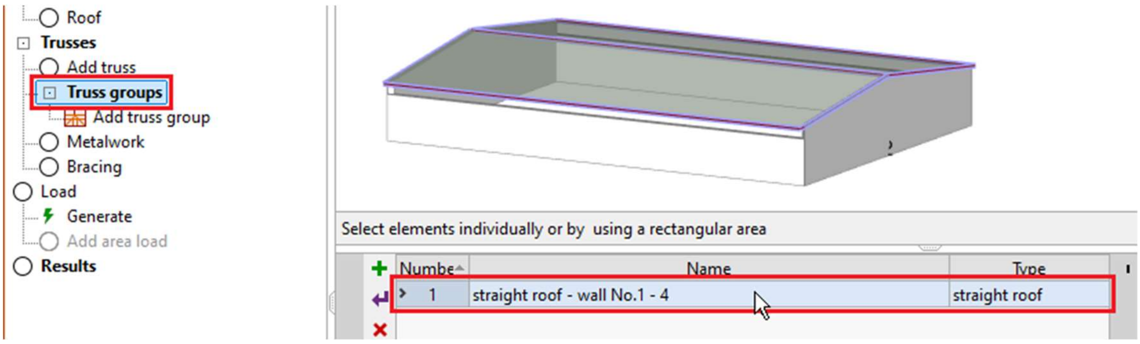
After copying the properties, the structure is symmetrical again. We have now finished specifying the external shape of the structure and can move on to creating the trusses and specifying the loads.



Final shape of the roof

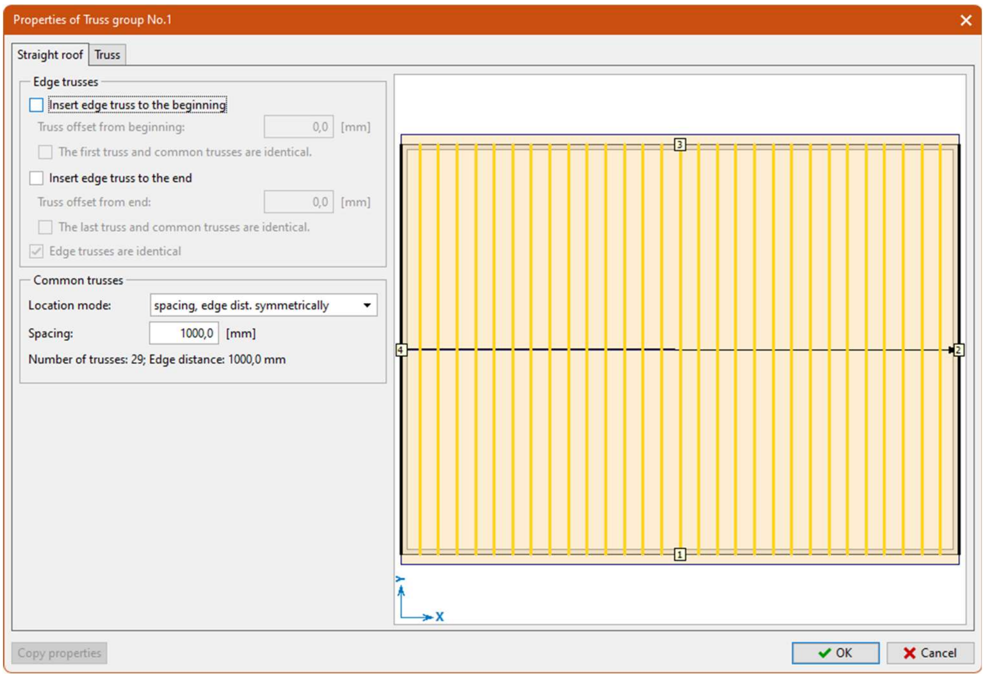
Trusses and loads

Trusses can be inserted into the structure either individually or using predefined groups. We will use the group option, as it is a faster way for such a simple roof. Switch the tree menu to the "**Truss groups**" mode. In the table in the bottom frame, the "**Straight roof**" group is already specified, but it does not have trusses created yet. Open the properties of this group by double-clicking on a row in the table.



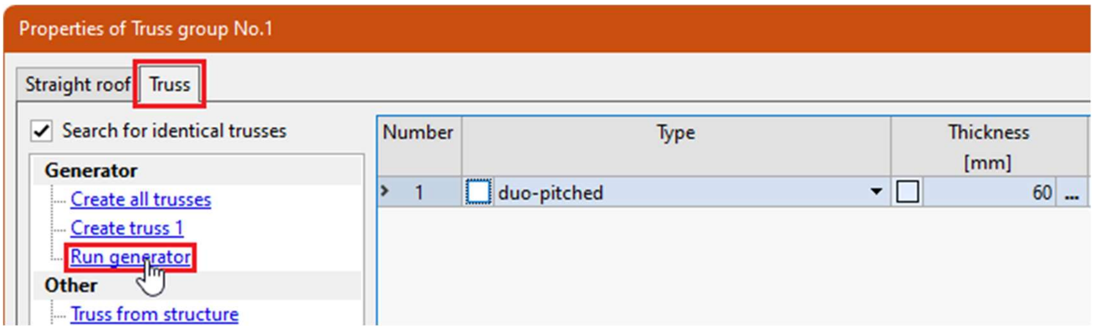
"Straight roof" truss group

The window with the properties of the truss group will open. Here it is possible to define the arrangement of trusses in the group by distance or number and the possibility to insert gable trusses. These can be created as a separate truss type and the distance from the outer edge of the wall can be also defined. In our case we will leave the default arrangement. The trusses will be spaced regularly at 1000mm, the remaining distance is distributed symmetrically to both ends.



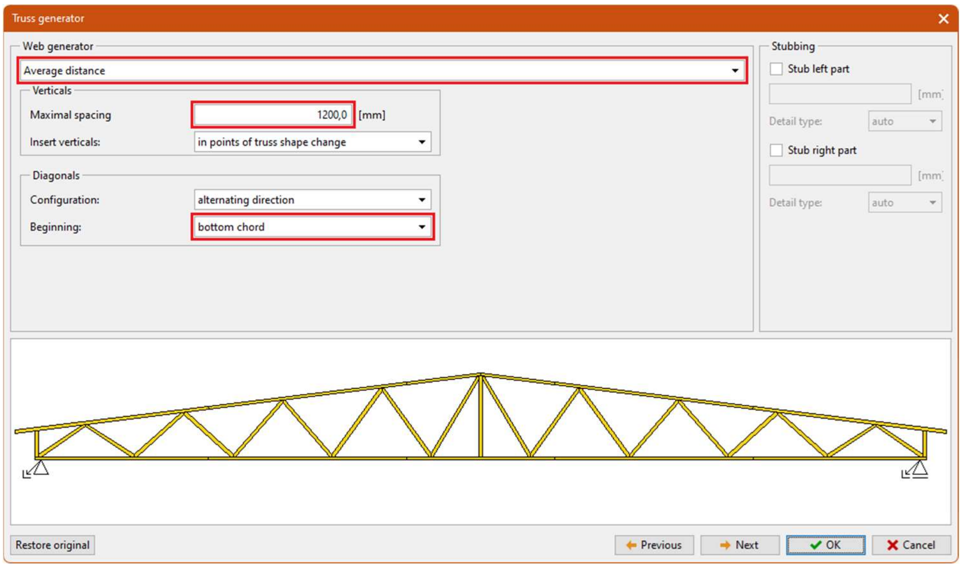
Properties of the truss group

We can now move to the second tab "**Truss**", which is used to create the trusses themselves. There will be only one type of truss in our group, we can recognize it by one row in the table that fills the upper right part of the window. Since we want to choose a custom infill for this truss, we will use the "**Run Generator**" function from the tree menu.



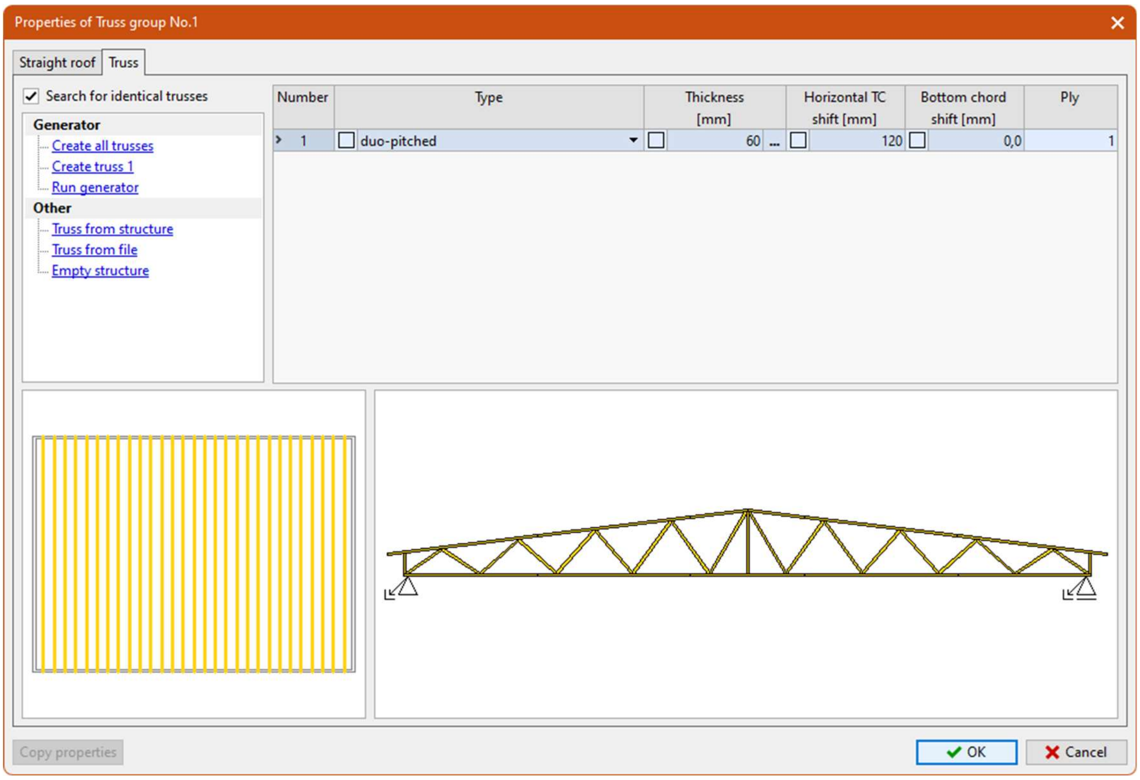
The "Run Generator" command

The "**Truss Generator**" window is launched, which offers multiple options for creating truss infill members. We choose the "**Average distance**" method. Next, we specify a maximum distance between the joints of *1200mm* and select "**bottom chord**" as the start of the generation. Confirm the input with the "**OK**" button.



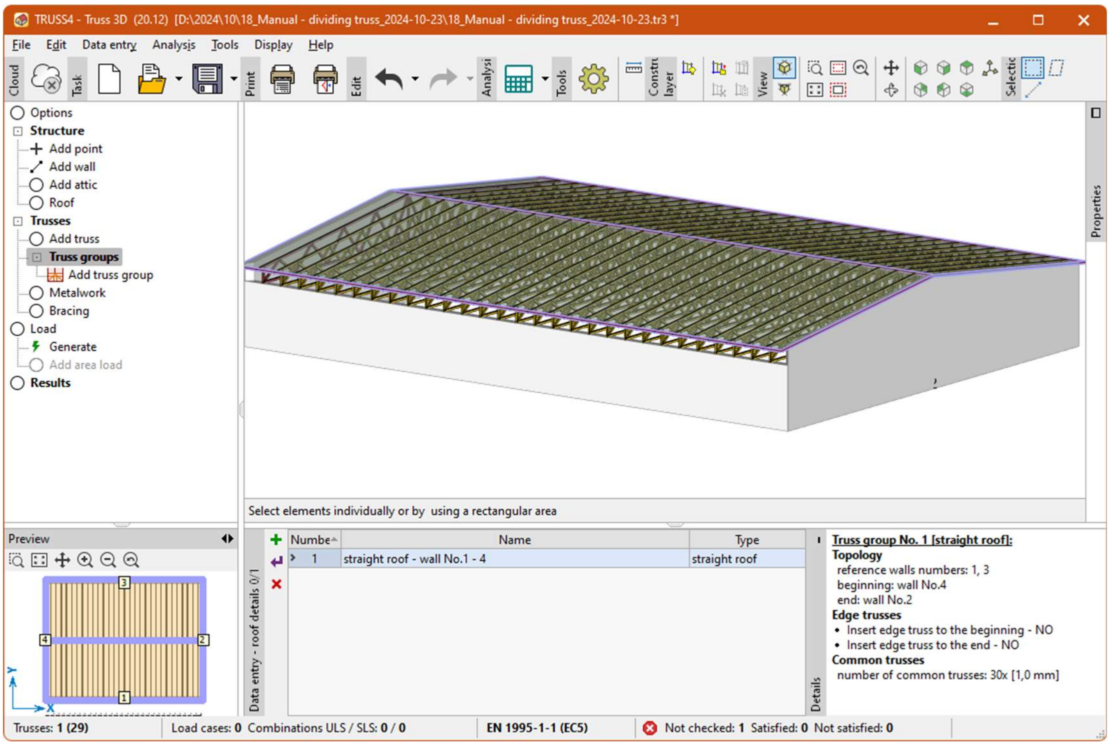
Selected parameters of the infill generator

After the generator is finished, the truss will be displayed in the bottom part of the window. Close the truss group properties window with the "**OK**" button.



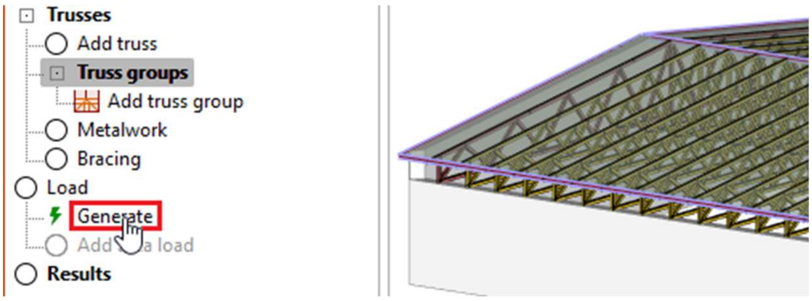
Properties of a truss group with a created truss

After closing the window, the trusses are drawn on the workspace.



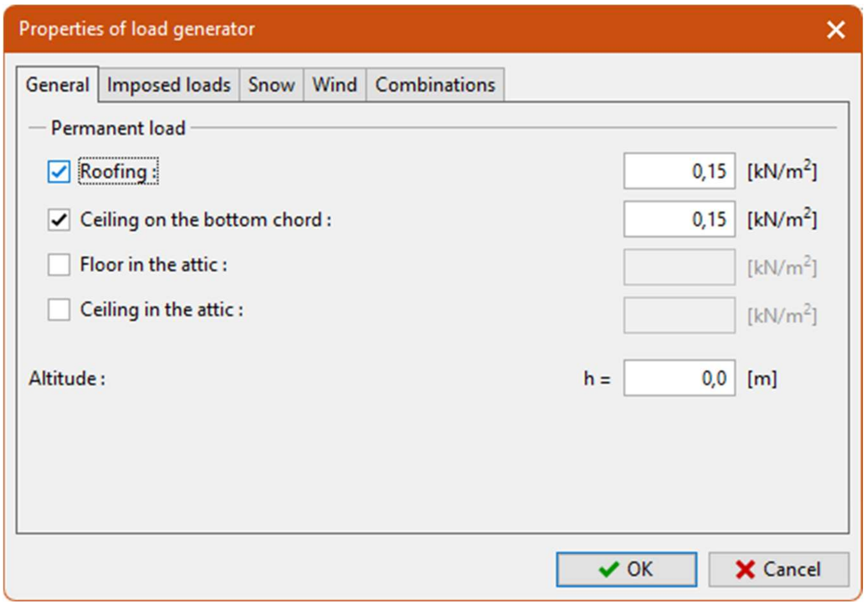
Structure with trusses

Since we have created the trusses, we can move on to entering the loads. Use the "**Generate**" function in the "**Load**" section of the tree menu.



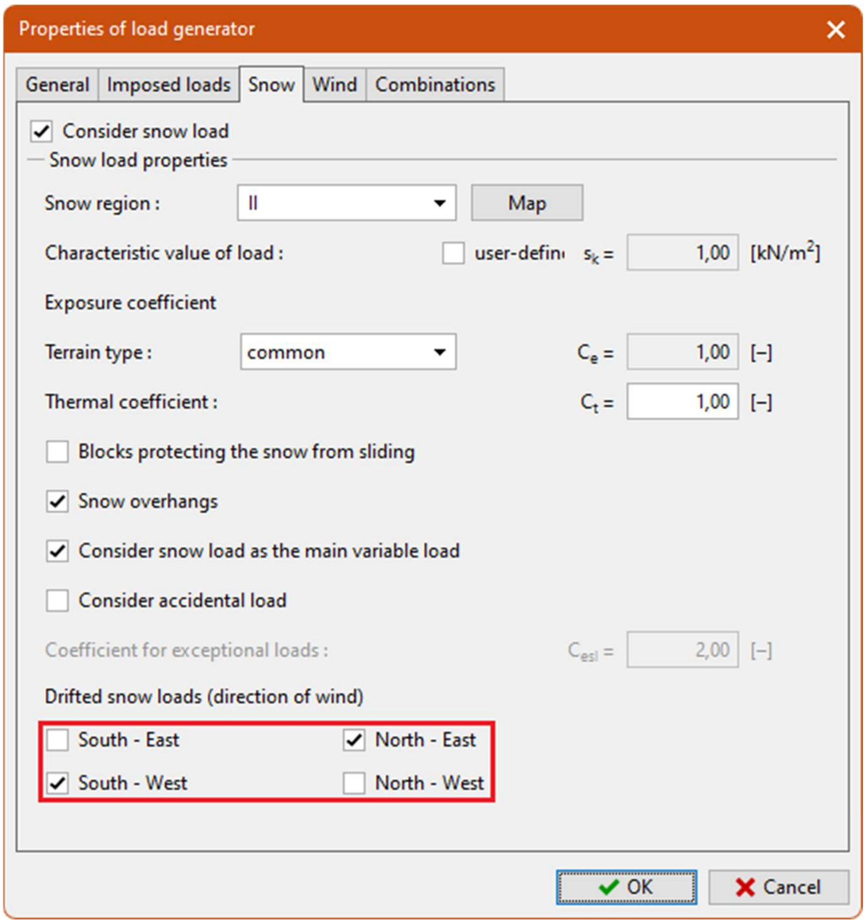
Load generation

A window that allows you to enter the load properties is opened. It is possible to enter permanent loads and other additional parameters affecting the load determination, such as altitude, in the first tab "**General**". We have initially selected the material of the roofing and ceiling. The loads for the roofing and ceiling were therefore pre-loaded according to this selection. The following "**Imposed loads**" tab contains the parameters affecting the variable loads associated with the use of the structure. We leave the default values here and go to the "**Snow**" tab.



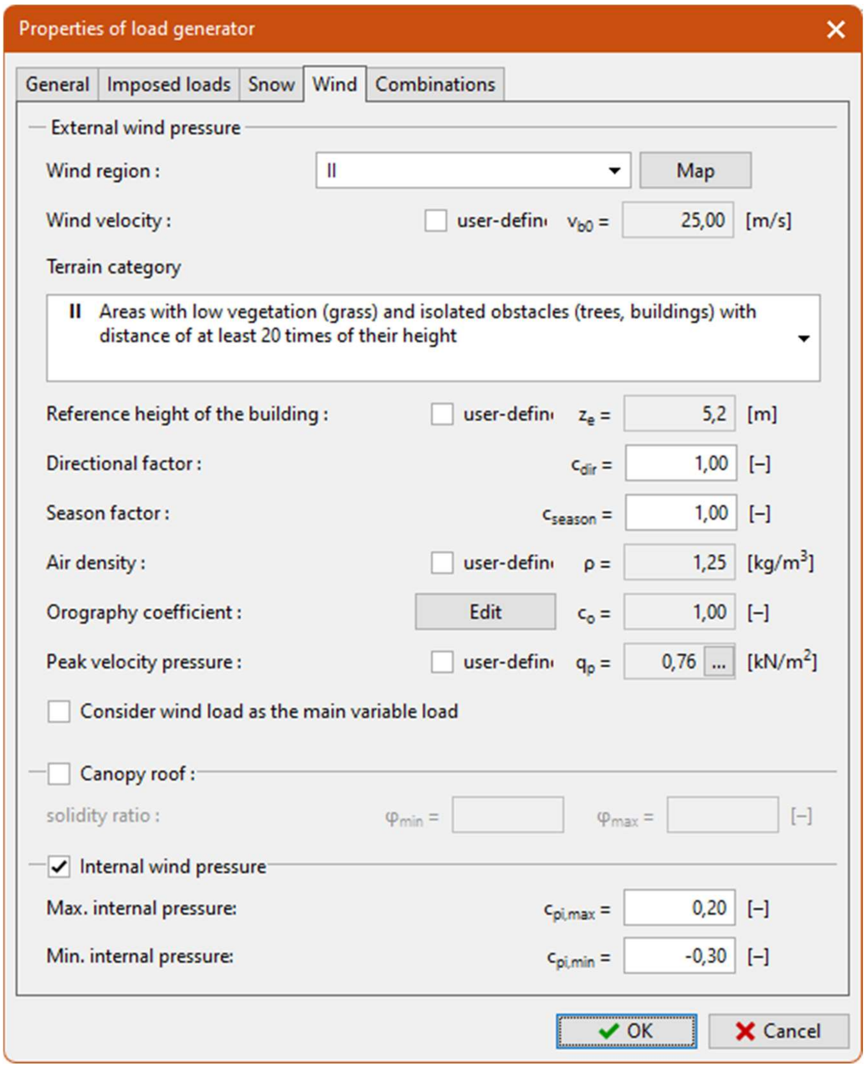
"General" part of the load generator

Leave the default settings on the "**Snow**" tab as shown below, but reduce the number of snow loading directions considered. Four directions are suitable for more complex roofs of the hipped type. For a simple gable roof, two mutually perpendicular directions are sufficient. Therefore, only the directions "**South-west**" and "**North-east**" are ticked. This modification can reduce the number of load combinations and thus the length of the analysis. Translated with DeepL.com (free version)



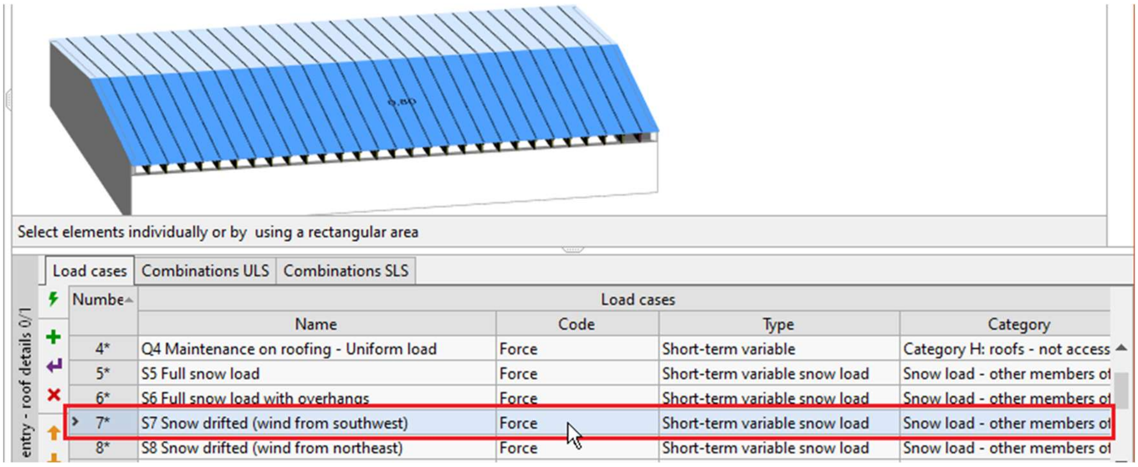
Reduction of directions for snow accumulation

Parameters affecting the wind load calculation (wind area, internal wind pressure, etc.) can be adjusted in the following section "**Wind**". We will not make any changes from the default.



Wind load properties

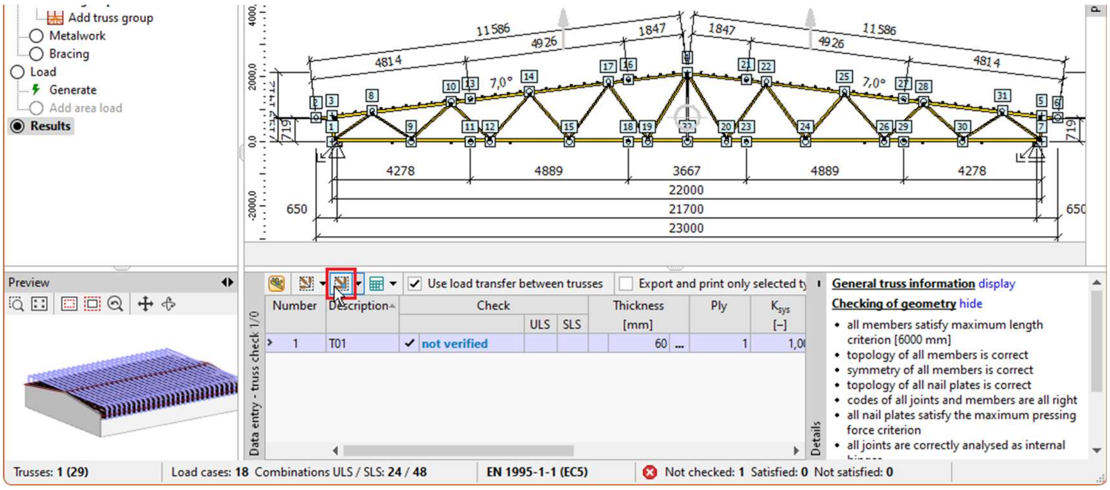
Similarly, the last tab "**Combinations**", which contains options to influence the generation of load combinations, will remain unchanged. Therefore, we can close the window with the "**OK**" button. In the bottom table we will then see a list of load cases and in the next tabs also the combinations for the ultimate and serviceability limit states. As you move through the table, the load arrangement corresponding to the load case is displayed on the workspace



Active load case displayed on the workspace

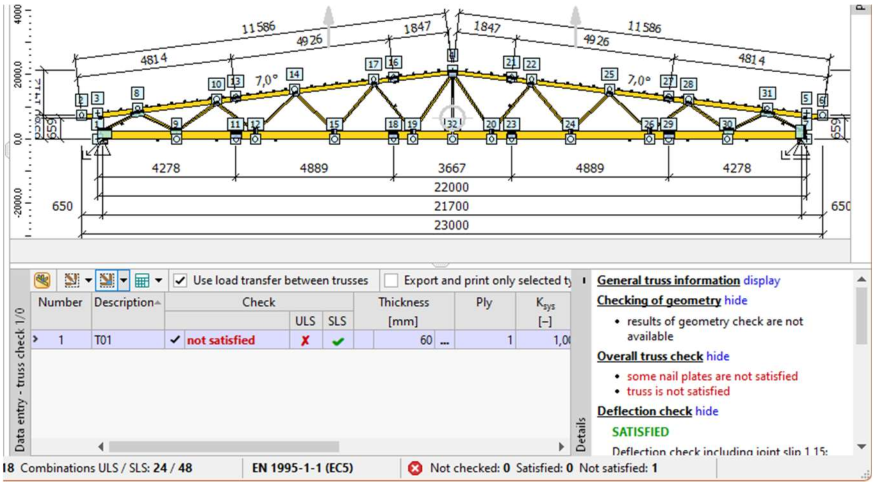
Basic analysis

After entering the load, we can go to the "Results" section and start the automatic design. The automatic design is started using the "" button in the toolbar above the truss table. Alternatively, the *F8* keyboard shortcut can also be used.



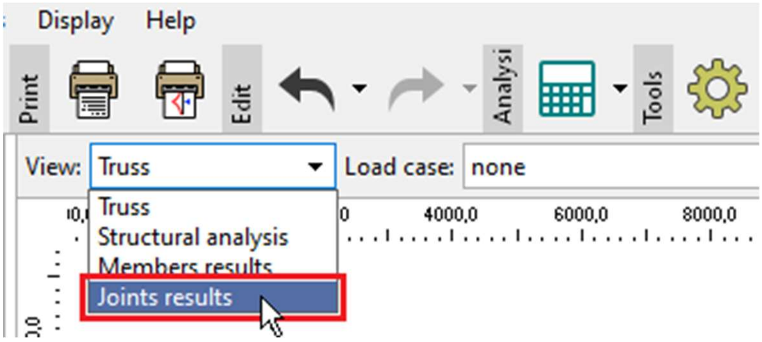
Running automatic design

After the design was completed, the program marked the truss as not satisfied. A more detailed listing to the right of the truss table indicates that some of the nail plates are not satisfactory.



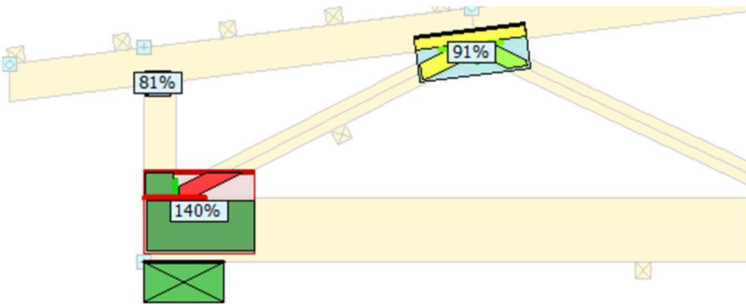
Result of automatic design

We display the detailed results for the nail plates. In the "**View**" drop-down list located in the toolbar above the 2D workspace, select "**Joints results**".



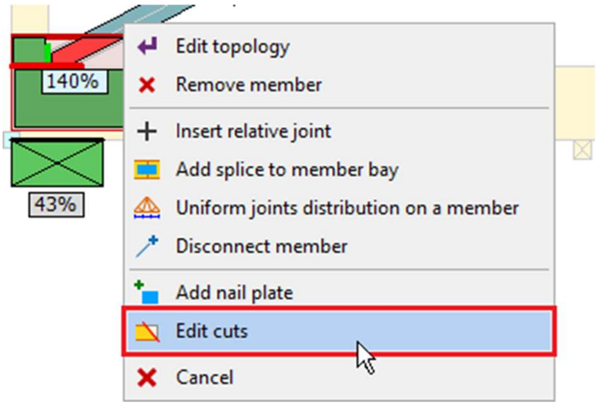
Switch to result display mode for nail plates

Subsequently, we can see that the nail plates in the supports do not pass. On closer inspection, we can see that the joint line between the bottom chord on one side and the webs on the other side fails.



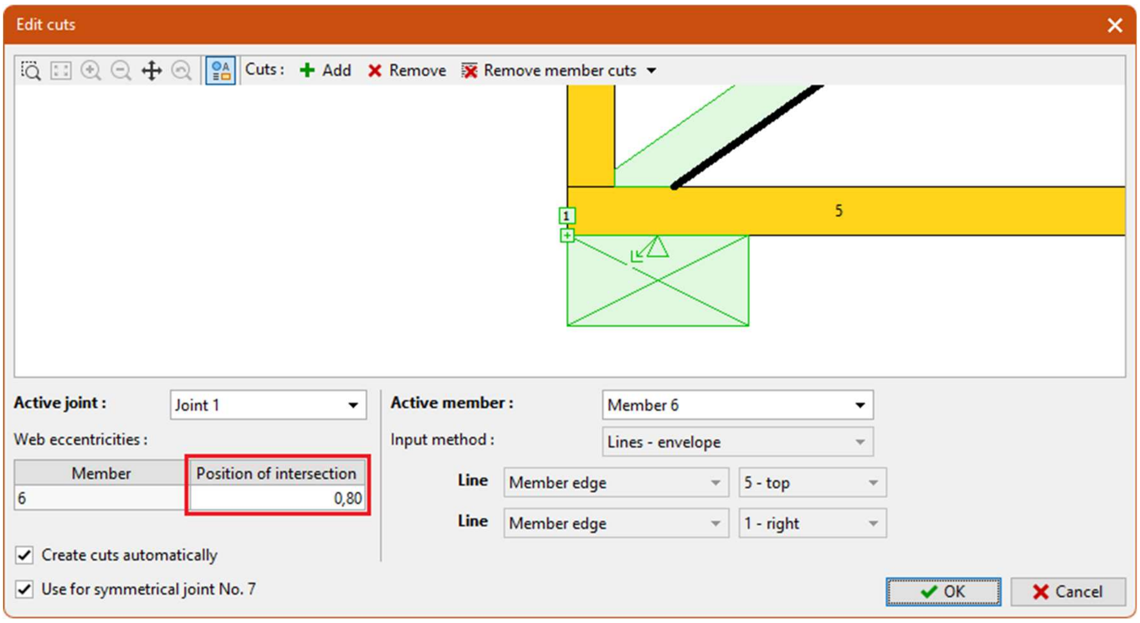
Nail plate with noncompliant joint line

This situation may occur if the joint line is too short at a given place and is not able to carry the vertical load. Increasing the size of the nail plate will not help in this case, it is necessary to increase the length of the joint line. This can be done either by increasing the depth of the edge vertical or diagonal web or by adjusting the eccentricity of the web. We will use the second method as this will not increase the volume of timber in the structure. Right-click on the web to bring up the local menu and select the "**Edit cuts**" tool.



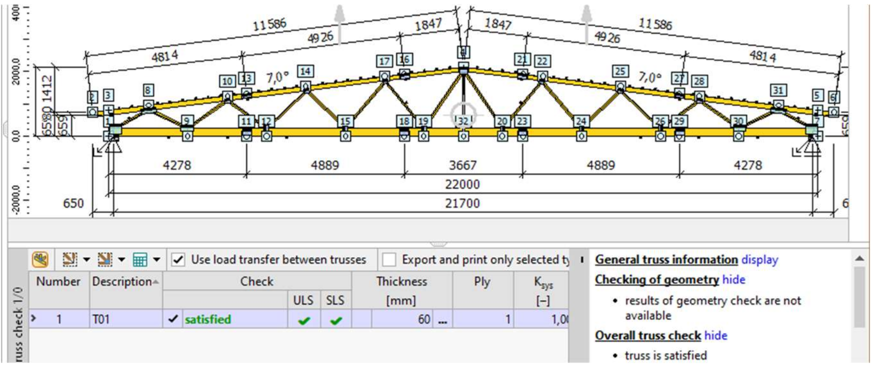
Starting the window for modifying cuts in the joint

We will see a window that is mainly intended for manual editing of cuts in the joint. However, we will only use it to adjust the eccentricity of the connected web. This can be changed in the "**Web eccentricities**" table. The eccentricity is entered as a dimensionless number in the range *<0;1>*. This number expresses the relative distance of the intersection of the end cuts from the bolded web edge. The default value of *0.5* means that the intersection is exactly on the member axis. We need to move the intersection further away from the highlighted edge. So we enter a value of *0.80*. Close the window with the "**OK**" button.



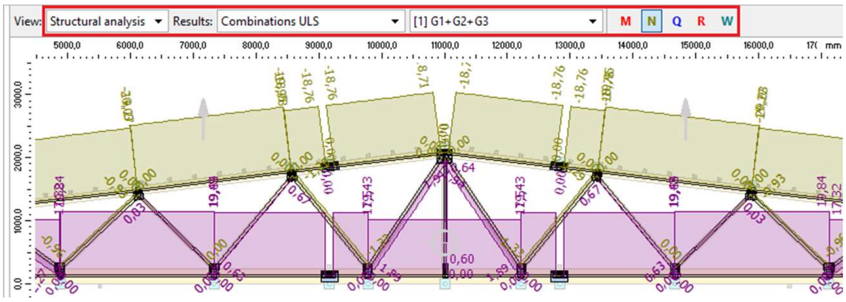
Change of web eccentricity

After the repeated automatic design, the truss already passes.



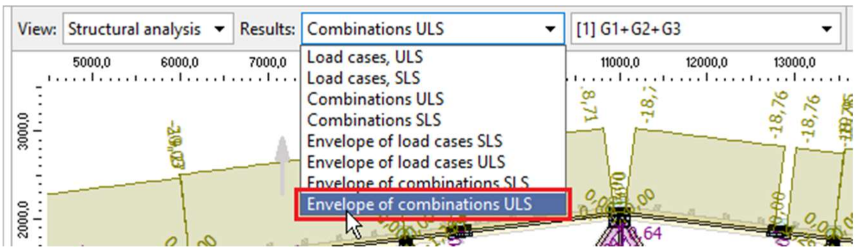
Satisfactory truss after the automatic design

The "**Results**" section offers not only the design results, but also, for example, the internal forces and deformations. Now we will see how to easily obtain the value of the normal force in the bottom chord at the point of the planned split. We will be interested in the value for the combination with permanent load cases as well as the maximum value across all combinations. These values can then be used to design the connection. The results display is set in the toolbar in the 2D workspace header. First, select the "**Structural analysis**" display mode. Then we can select which results we are interested in from the "**Results**" list. We can choose from results for load cases or combinations. Values for both ultimate and serviceability limit states are available. We will select "**Combinations, ULS**" as the forces will be used for the design of the load bearing connection. A list of combinations for the ultimate limit states will appear in the following drop-down menu. We will leave combination *1* displayed, which contains all permanent loads in the structure. The last part of the toolbar contains buttons for selecting the displayed quantities. We leave only the "**N**" button on, which represents the normal (axial) force. We can then see on the workspace that all the permanent loads produce a force of about *18kN* in the middle part of the bottom chord.



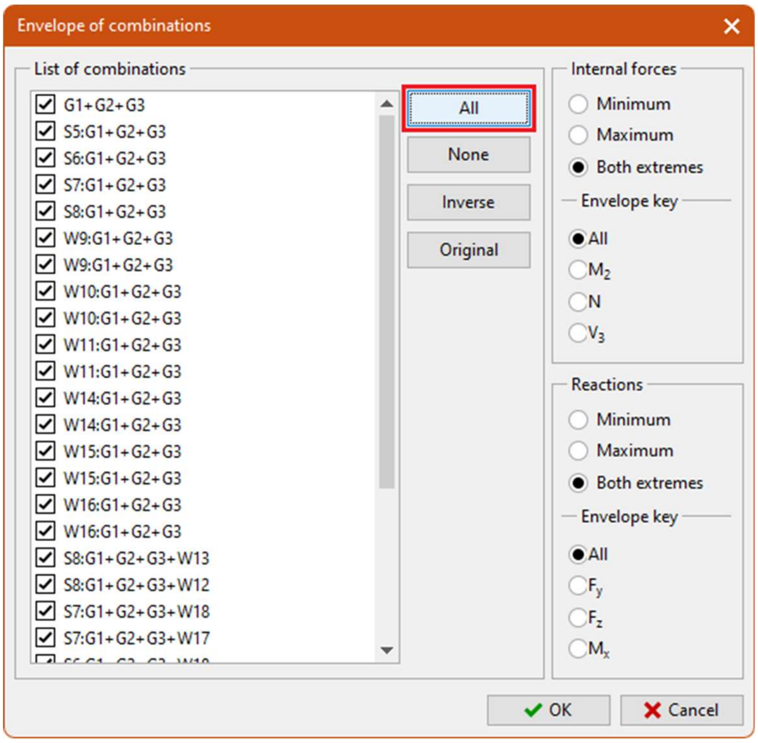
Display of normal forces in the central part of the truss for all permanent loads

If we want to see the maximum values of forces from all combinations, we can use the envelope of combinations. The envelope always plots the worst value of the selected quantity from the pre-defined combinations at each point of the structure. Thus, in the "**Results**" drop-down list, we select the "**Envelope of combinations, ULS**".



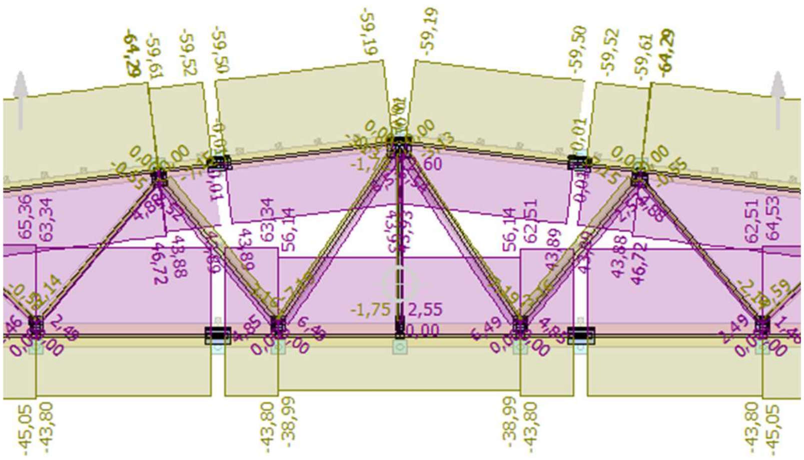
envelope of combinations for the ultimate limit states

When this option is selected, the envelope properties window is displayed. This is primarily a list of the combinations contained in the envelope. We include all combinations from the design using the "**All**" button. We do not need to change any other parameters, we close the window with the "**OK**" button.



Properties of the envelope

The maximum values of the normal force are then displayed on the workspace. Since minima and maxima are detected within the envelope, two force values are plotted at each point. The negative value indicates the maximum compressive force, and the positive value represents the maximum tension.

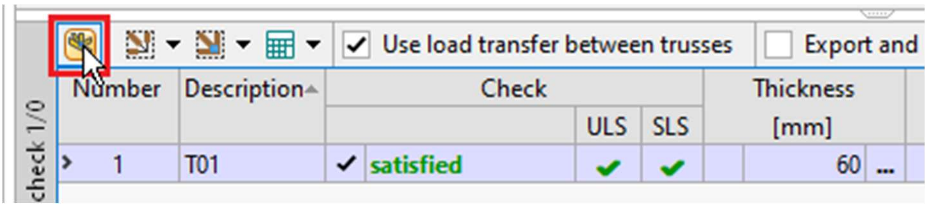


Axial forces for an envelope of ULS combinations

Using these forces, we can already make a preliminary design of the connection. We now move on to the dividing of the truss itself into two separate production parts.

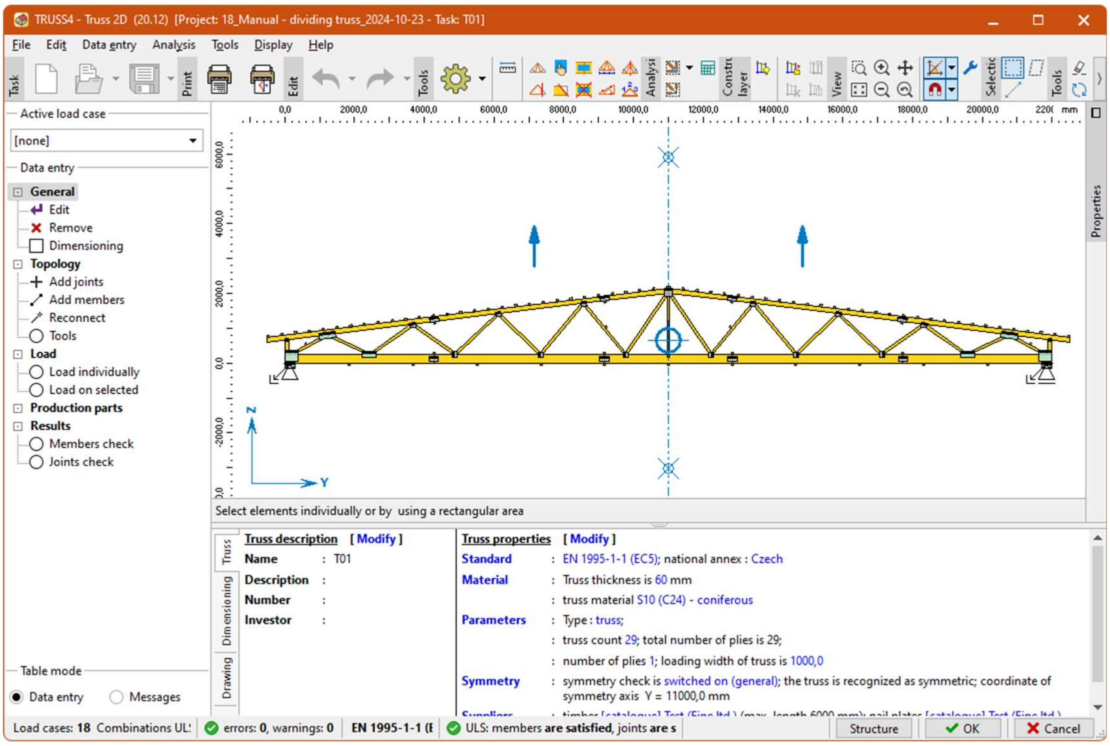
Modifying the topology

The division of the truss into production parts will be done in "**Truss 2D**", as there are more topological tools available. Start the program with the "" button in the header of the truss table.



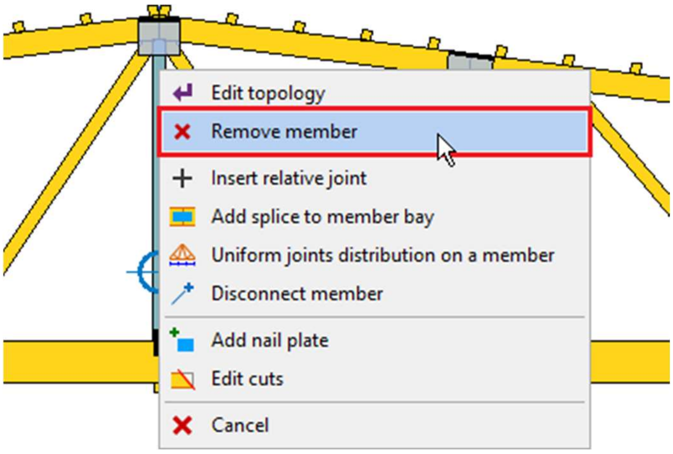
Starting the "Truss 2D" program for the active truss

The program "**Truss 2D**" is launched. We can see our truss on the workspace. We will make our modifications here.



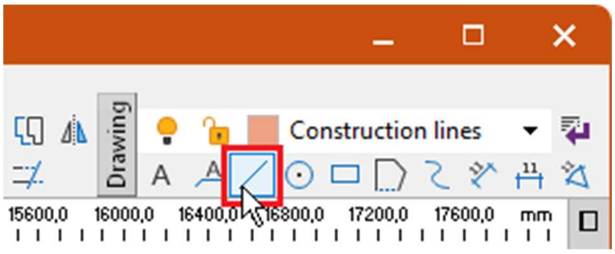
"Truss 2D" program window

The first modification will be to remove the vertical web in the apex. Click on the web with the right mouse button and select "**Remove member**" in the context menu.



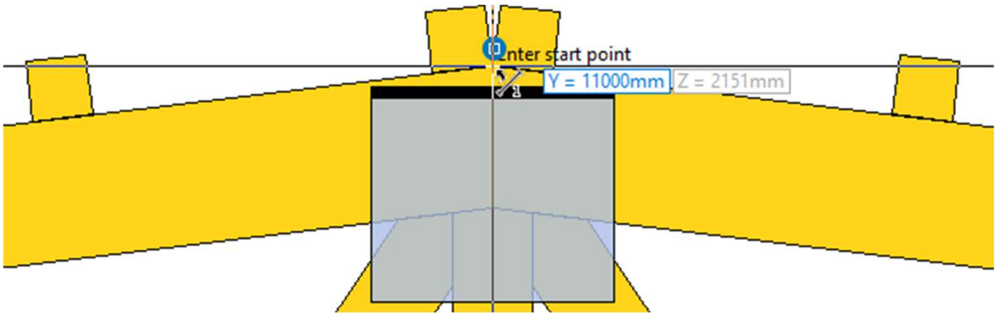
"Remove member" tool in the context menu of the member

Then we will draw a construction vertical line that will go through the apex of the truss. Select the "**Line**" command in the "**Drawing**" toolbar.



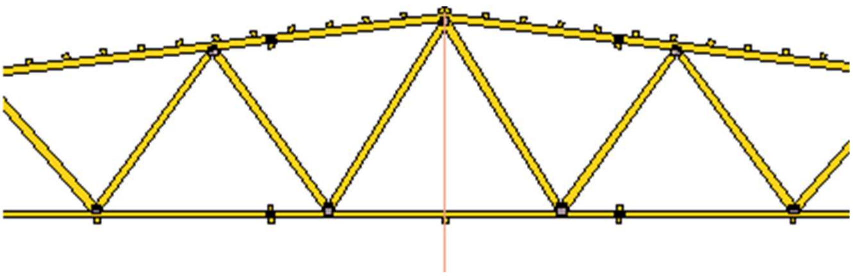
"Line" tool

Specify the truss apex as the first point of the line.



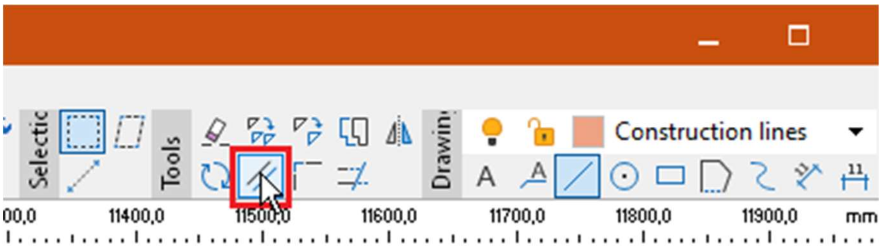
Specifying the beginning of the line

Now we must enter the second point of the line. If we move the cursor downwards, it will automatically align itself vertically to the first point. Enter the second point so that the line extends over the bottom chord of the truss.



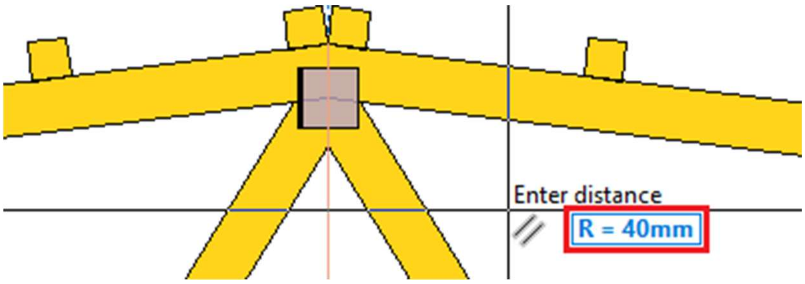
Truss with drawn construction line

Use the line when specifying two verticals. One will be placed in the left part of the truss, the other in the right part. They will be *80mm* apart. In the "**Tools**" bar, select the "**Equidistant**" tool, which is used to create a parallel line at a predefined distance. We want to create two lines parallel to an already defined line: One will be located to the left of the line at a distance of *40mm*, and the other to the right at the same distance.



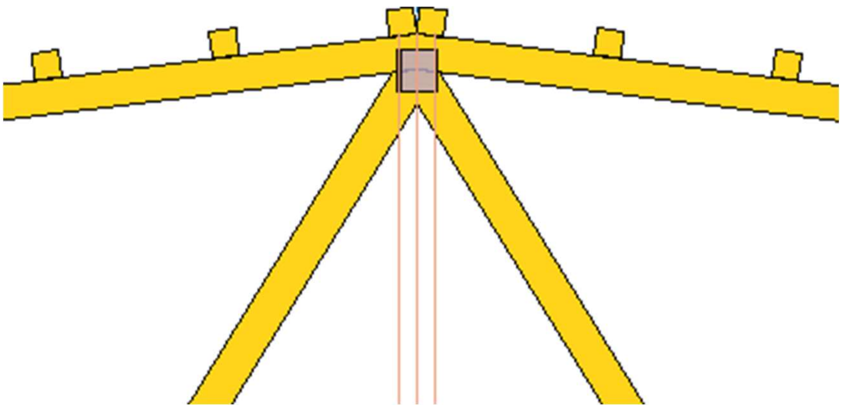
"Equidistant" tool

The program first prompts us to enter the distance. You can enter either numerically on the keyboard or with two points on the workspace. We will use the numeric input, enter the value *40* (the value is displayed at the cursor) and confirm with the *Enter* key.



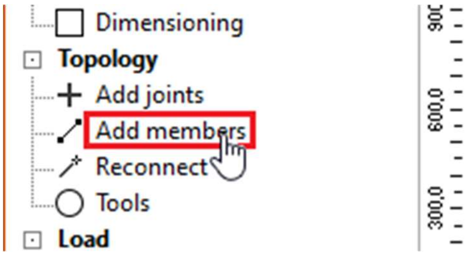
Specifying the distance of a parallel line

Now, click on the line to select the object to which we will create a parallel and then click anywhere in the left part of the truss. This click selects the side where the parallel is to be inserted. Then we click on the line again, but the next click will be on the right side of the truss. So now we have three vertical lines around the apex of the truss.



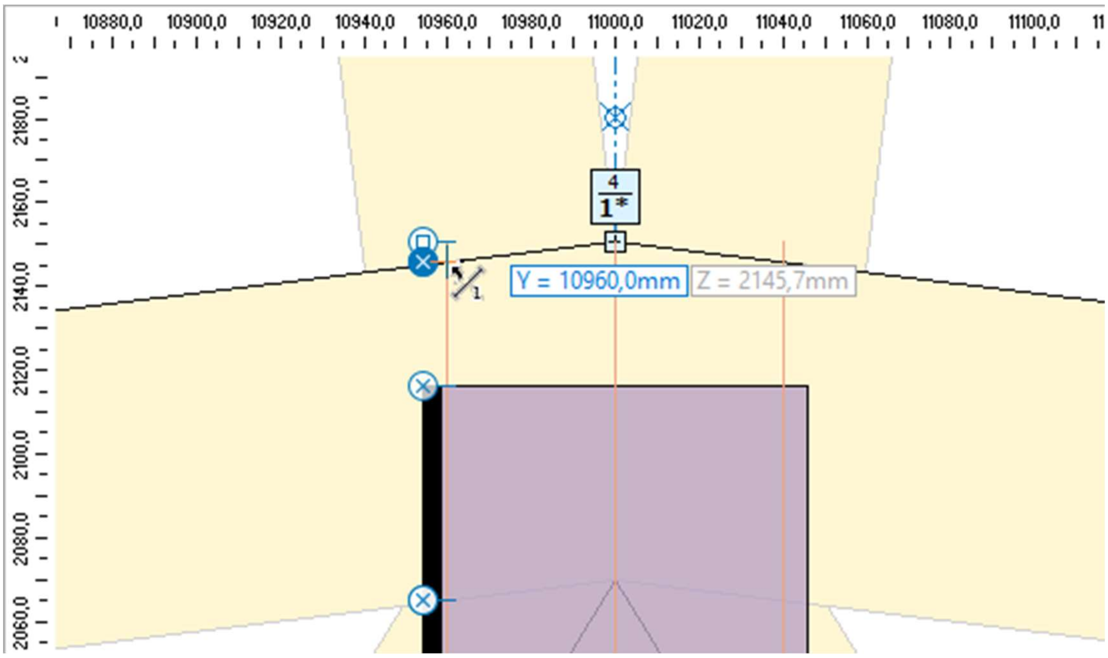
Truss apex with three lines

The new parallel lines have created snap points on the top and bottom chords, which we will use to specify the new vertical webs. In the tree menu, select the "**Add members**" command.



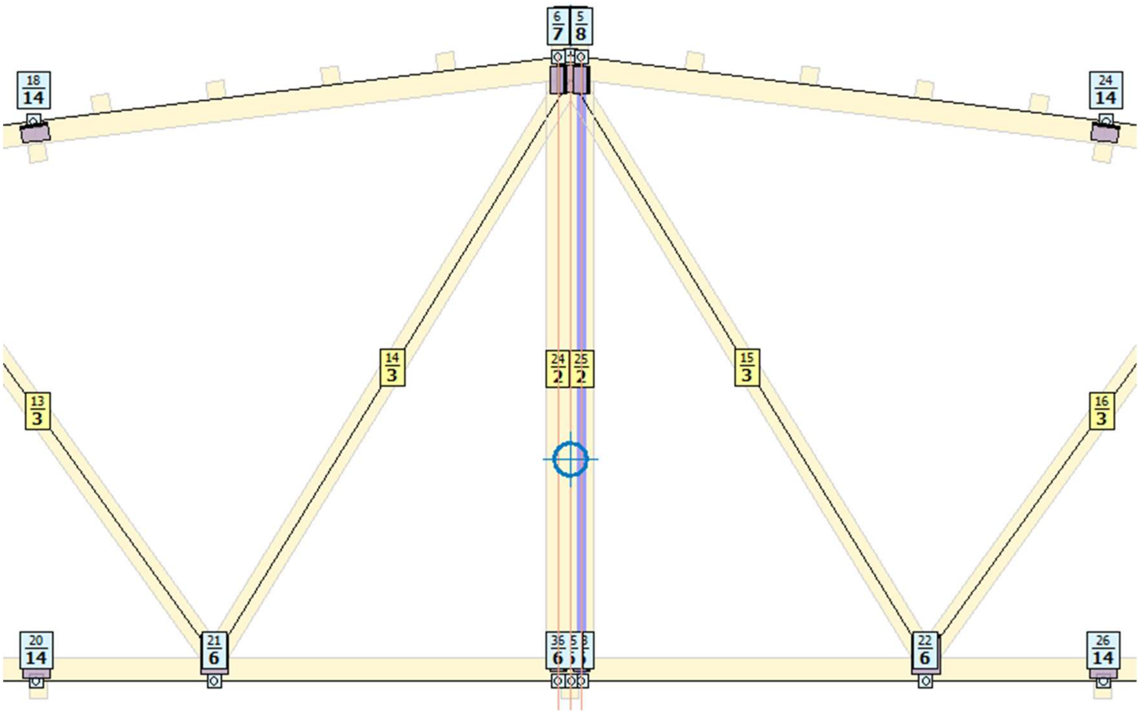
Tool "Add members"

Now you just need to click on the intersection of the construction line with the edge of the top or bottom chord one by one. This will insert a vertical web into the structure and create two new joints with nail plates. When selecting the top point, it is necessary to get close enough to the structure, as the intersection of the segment with the chord edge is very close to the end of the construction line. A small enlargement runs the risk of selecting the wrong snap point and not attaching the vertical web to the top chord.



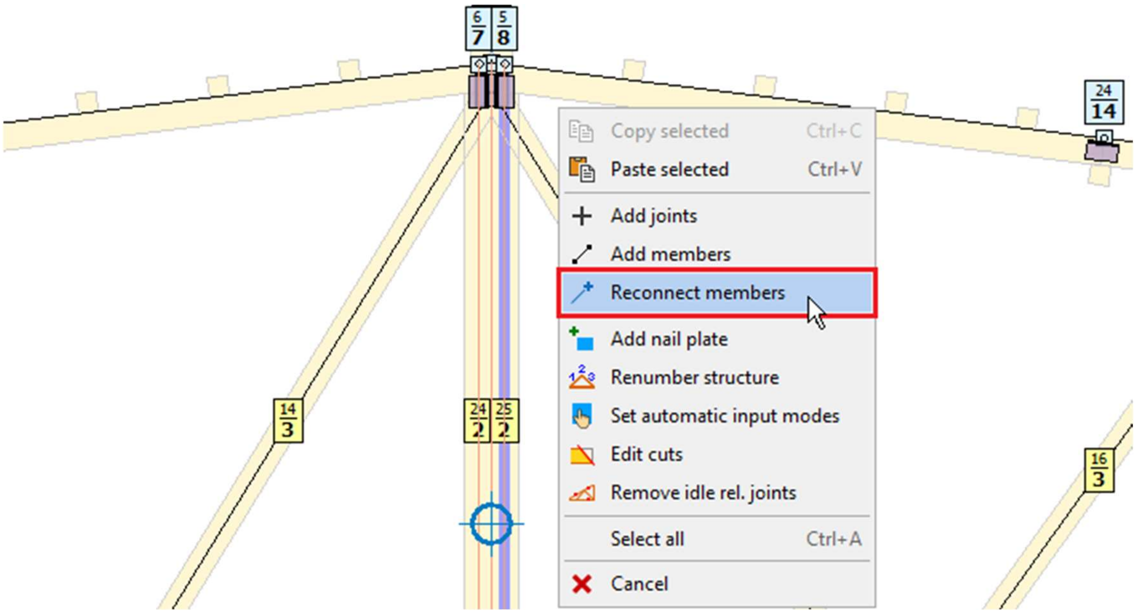
Specifying the first point of the vertical web at the intersection of the line and the edge of the top chord

Repeat the same procedure for the vertical on the other side. The result is then a truss that has two vertical webs that intersect the two diagonal webs at the apex.



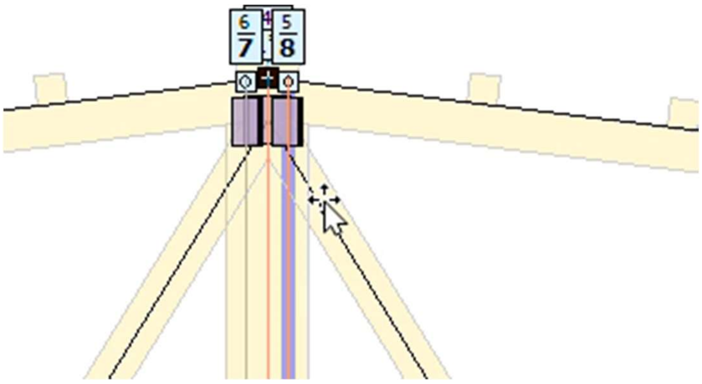
Truss with two new vertical webs

The collision of the new vertical webs with the diagonal webs can be solved easily using the "Reconnect members" command. This can be found in the workspace context menu.



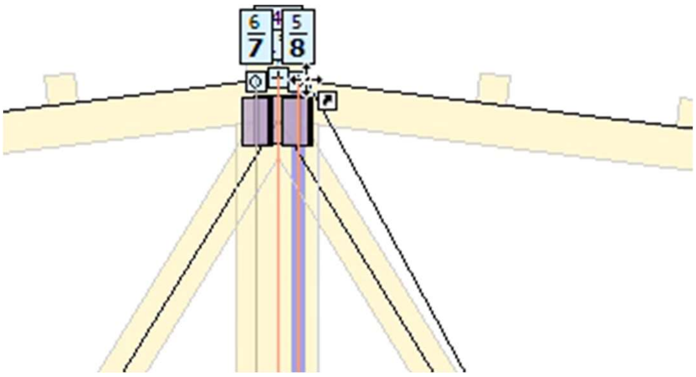
""Reconnect members" tool

Reconnecting members works in the style of *Drag and Drop* operations, i.e. dragging while the left mouse button is pressed. Move the cursor over the web and press the left mouse button. Hold the button down while moving the end of the web. In this mode, the appearance of the mouse cursor changes.



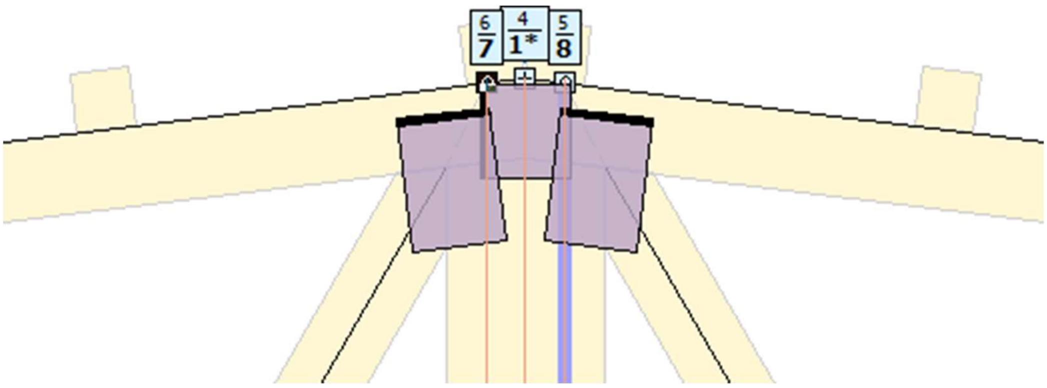
Appearance of the cursor over the member that can be reconnected

Place the cursor over the joint mark. The cursor will change again. It indicates that it has snapped to the joint.



Appearance of the cursor after dragging the member end over the joint

Appearance of the cursor after dragging the member over the jointSo release the mouse button and the web will be switched to the end of the vertical member. Repeat the procedure in the same way on the other side. The reconnect mode allows you to reconnect any number of members sequentially, so there is no need to run the command again.



Modified position of webs in the truss apex

Exit the "**Reconnect members**" mode by right-clicking. The next modification will be to split the bottom chord into two separate parts. Since we already have an empty relative joint in the middle of the bottom chord, we just need to turn it into an absolute one.

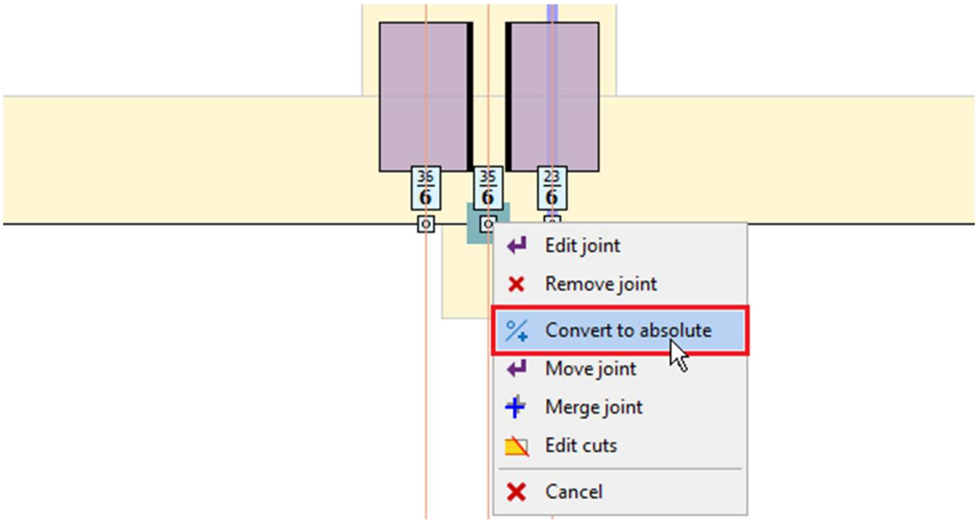
Relative and absolute joints

*The program distinguishes between "**Absolute**" and "**Relative**" joints. They differ in the way their position is defined. The other properties (e.g. support options) are identical.*

***Absolute joints** are defined by the Y and Z coordinates in the global coordinate system and are mainly used in places where the outer shape of the truss changes. If the absolute joint lies on a continuous member, there is no link between the joint and the member.*

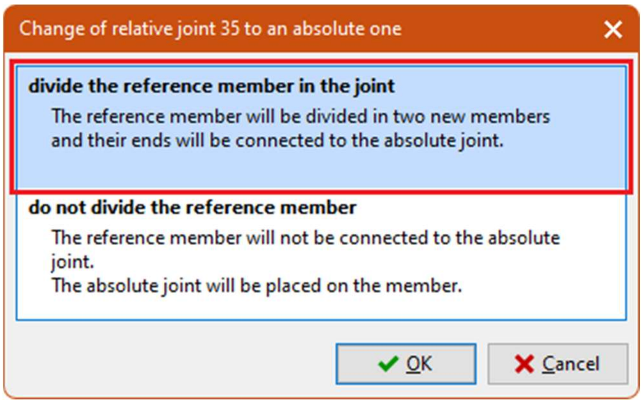
***Relative joints** have their position defined by their location on the reference member. Thus, when the position of the reference member changes, the position of the relative joint also changes. This type is used for intermediate joints on outer chords.*

By changing the type of the joint, the bottom chord will be split. In the context menu of the jointer we use the command "**Convert to absolute**".



The "Convert to absolute" command in the context menu of the joint

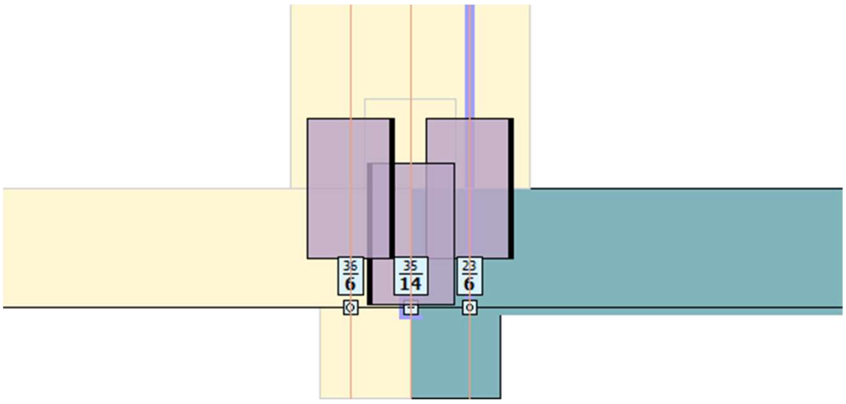
A window appears in which we select the first option "**Divide the reference member in the joint**". Confirm the selection with the "**OK**" button.



Choice of dividing the bottom chord into two parts

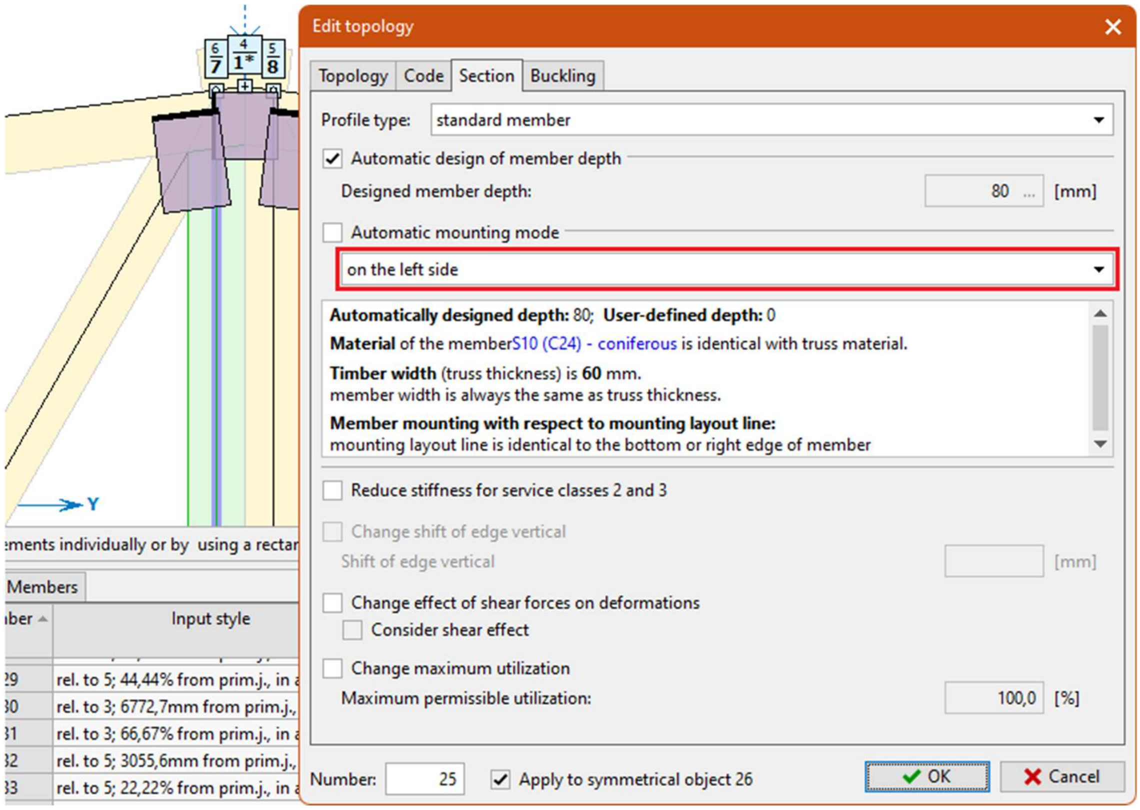
Conversion of relative joint to absolute
The relative joint can be converted to absolute in two ways:
Divide the reference member in the joint - *the program splits the reference member into two separate parts that will be connected to the absolute joint. The position of these new members will be therefore changed with any subsequent displacement of the joint. This modification can be used when the outer shape of the truss is to be changed (for example, to create a camber of the bottom chord).*
Do not divide the reference member - *in this case the reference member remains unchanged. The relative joint will be changed to an absolute joint which will no longer have any connection to the member. The joint can then be arbitrarily moved out of the original reference member. This option can be used in cases where we want to delete the chord but keep the infill members going into it.*

After this modification, the bottom chord was divided into two parts and the mark of the joint was changed. The absolute joint has a mark with a cross, the relative one with a ring.



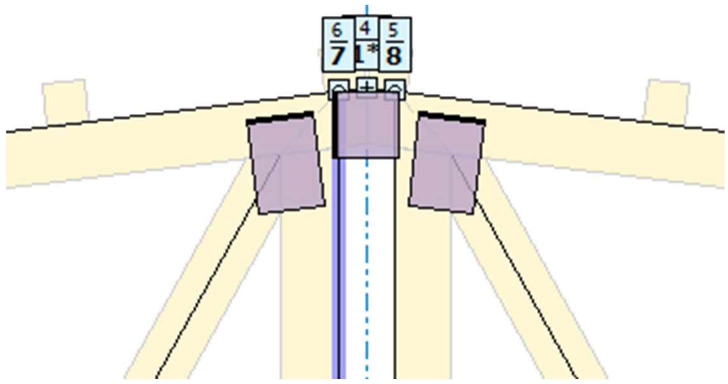
Bottom joint with divided member

The last operation we do is to change the horizontal alignment of the verticals webs. Double-click on the vertical web in the left part to open the "**Edit topology**" window and uncheck the "Automatic mounting mode" setting in the "**Section**" tab. This will open a drop-down list in which we can select the alignment method "**on the left side**". Close the window with the "**OK**" button.



Changing the alignment

After closing the window, the alignment of the second vertical web will change as well, because it is a symmetrical element.

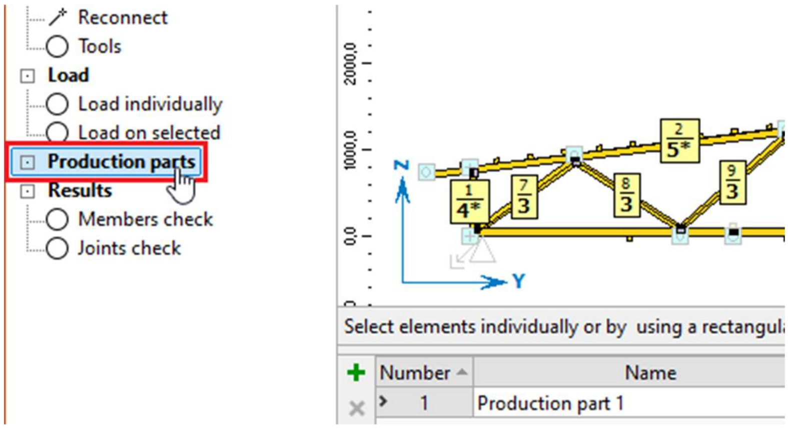


Truss apex with aligned vertical webs

This concludes the topological modifications and we proceed to split the truss into two production parts.

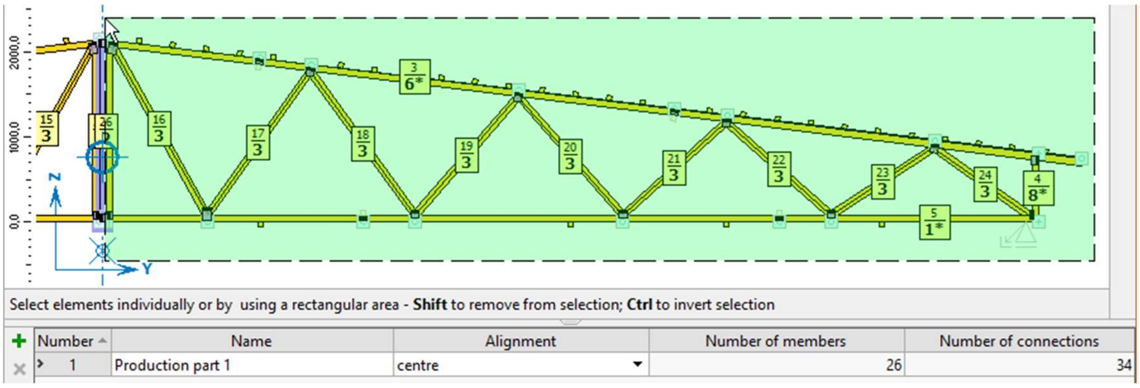
Splitting the structure

If we want to produce the truss in parts, we have to divide it into production parts. So we go to the "**Production parts**" node of the tree menu.



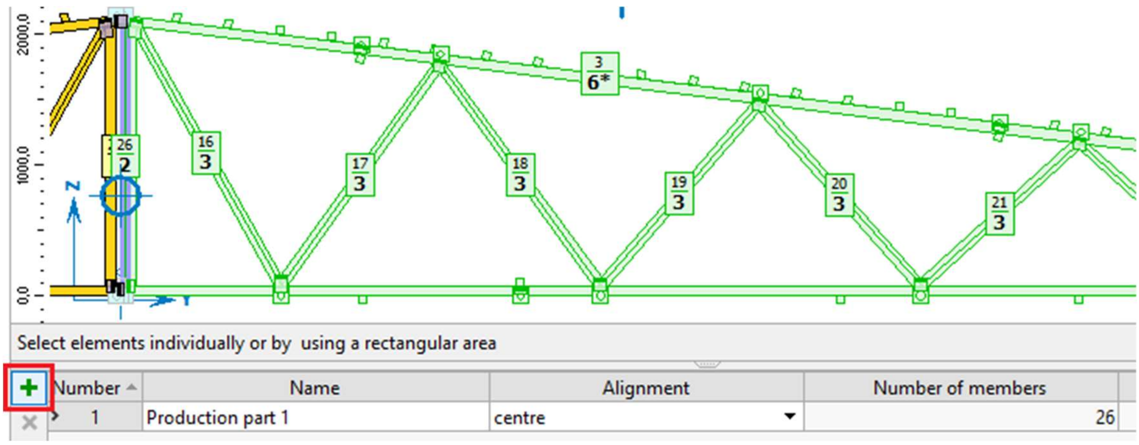
"Production parts" in the tree menu

Select all the members and joints in the right half. The easiest way to select elements is by dragging on the workspace from right to left.



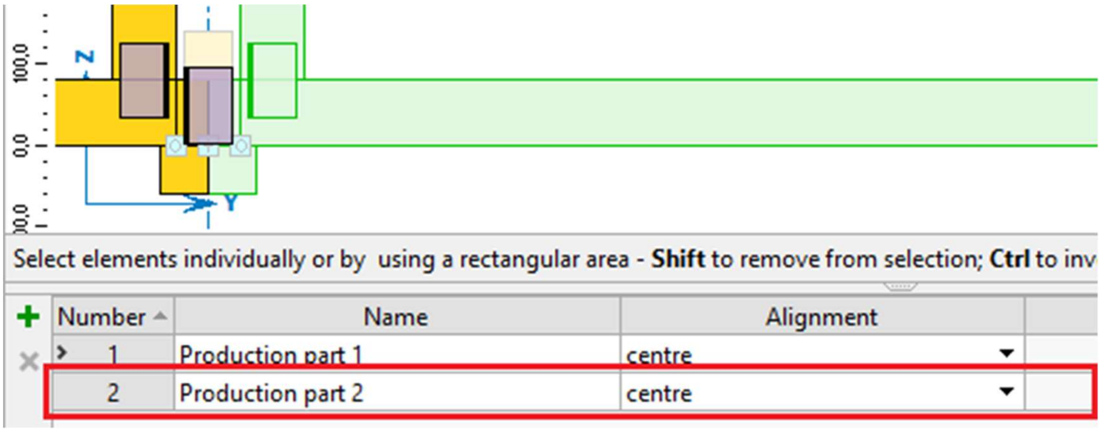
Graphic selection of members and joints in the right part of the truss

When making the selection, we must pay particular attention to the area around the apex, where the two production parts will interact. Should we not be able to select the vertical web near the top or the plates that connect them at first, for example, it is possible to add these elements to the selection by simply clicking the left mouse button. On the other hand, if we accidentally selected some elements from the left part, we can remove them from the selection by clicking them while holding down the *Shift* key on the keyboard. If we have the selection correctly defined, we can create a new production part by clicking on the "✚" button to the right of the production part table.



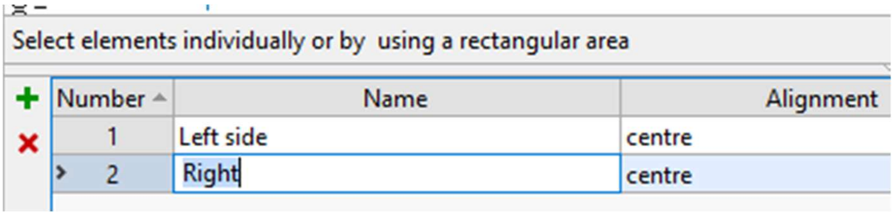
Creation of a new production part

A new entry "**Production part 2**" will appear in the table of production parts.



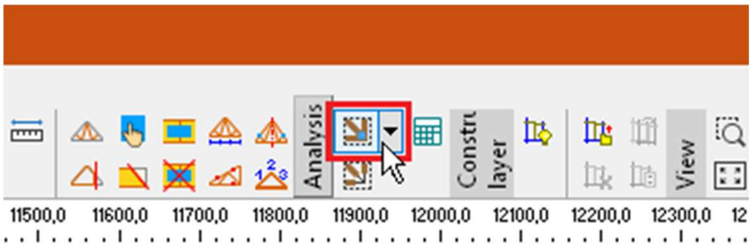
Newly created production part in the table

The table allows custom naming of production parts by simply clicking in the "**Name**" column. Rename the production parts to "**Left side**" and "**Right side**".



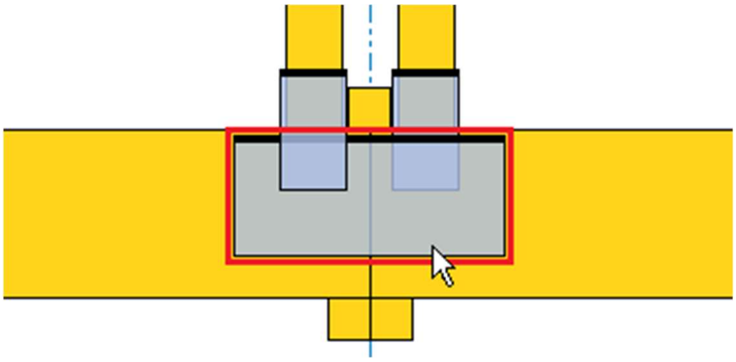
Renaming of the production part

In the next step, we will specify the properties of the joints that connect the two parts. But before that, it is advisable to run the automatic design of the structure to have an idea of the real dimension of the members, especially the chords. The automatic design can be started with the corresponding button in the toolbar or with the *F8* key.



Starting automatic design

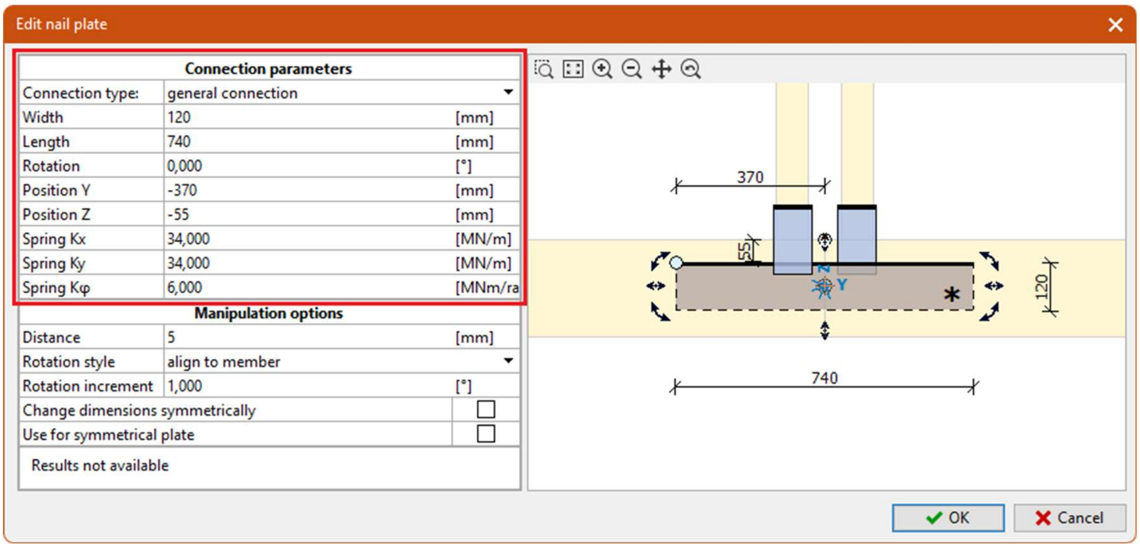
Once the automatic design is complete, we have obtained the real depths of all the members. This gives us an idea of the actual geometry of the joints in which the structure will be divided. So far, we have regular nail plates in these joints (apex and middle of the bottom chord). We will want to replace them with a structural joint made on site before the final installation of the truss into the structure. This will be a bolted joint with steel plates. We will first look at the tension joint on the bottom chords. Double click on the nail plate to open the plate properties window.



Nail plate in the bottom chord to be modified

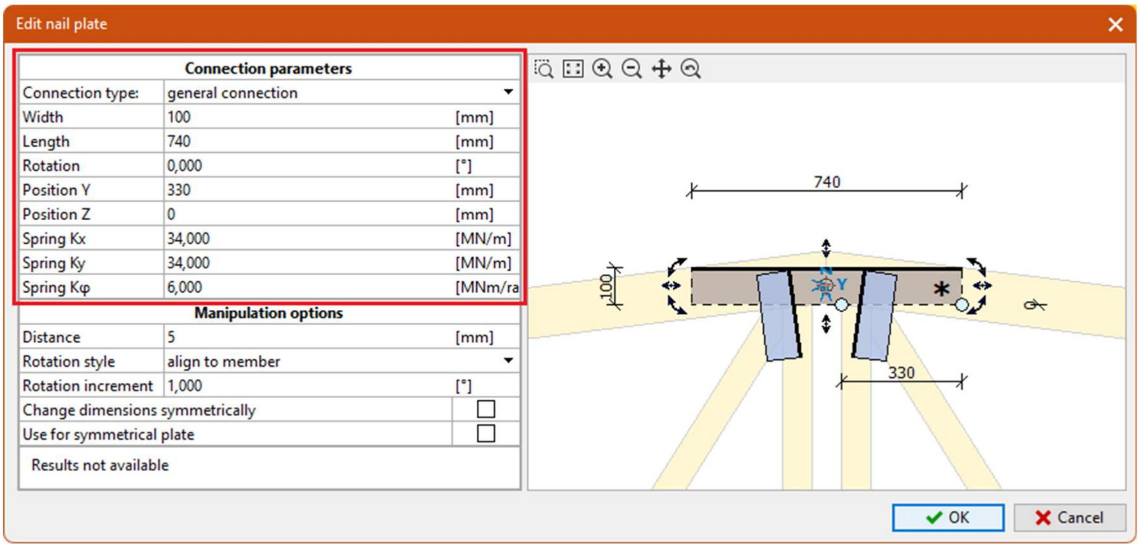
After double-clicking on the nail plate, the "**Edit nail plate**" window opens, which is used to modify the position and geometry of the plate. The basic operation we will perform is to change the type of connection to "General connection". This will turn the nail plate that is being analyzed and goes into the lists of material into a theoretical connection that is only used to connect the production parts and transfer forces between them. The basic properties are the geometry (width, length, position) and the stiffness of the connection. The geometry of the structural connection affects the creation of the structural scheme. Therefore, care should be taken to ensure that it corresponds as closely as possible to the real connection. In particular, it is essential to ensure that only those components that will be connected by the actual joint are overlapped. In our case, the bolts will only be arranged in the two bottom chords. Therefore, care should be taken to ensure that the rectangle of general connection does not interfere with the vertical webs. The entered stiffness should correspond to the actual stiffness of the connection, obtained during the design and verification of the fasteners. It has an influence on the resulting deflection of the structure and may partly affect the redistribution of internal forces in the structure. The properties of the connection are given according to the following figure. I.e. a centrally located connection will have a dimension of 740x120mm, the stiffness in displacement will be *34.00MN/m*, the stiffness in rotation will be *6.00MN/m/rad*.

Orientation of the general connection
*Even though the rectangle of the structural connection serves only as a geometric envelope to create the basic geometry of the structural model, it is advisable to select as the "**Length**" input the dimension that is in the direction of the main stress of the connection. In our case, this is the horizontal dimension in the direction of the tensile force in the chord. When listing the internal forces in the joint, this direction is used as the main direction for tension/pressure differentiation in the joint. If we were to specify the joint in such a way that "**Length**" represents the vertical dimension, we would get the correct force values in the static documentation, but the sign convention for tension/pressure would not be consistent with common practice. Also for this reason, the main (length) dimension is highlighted with a bold border in the structural connection.*



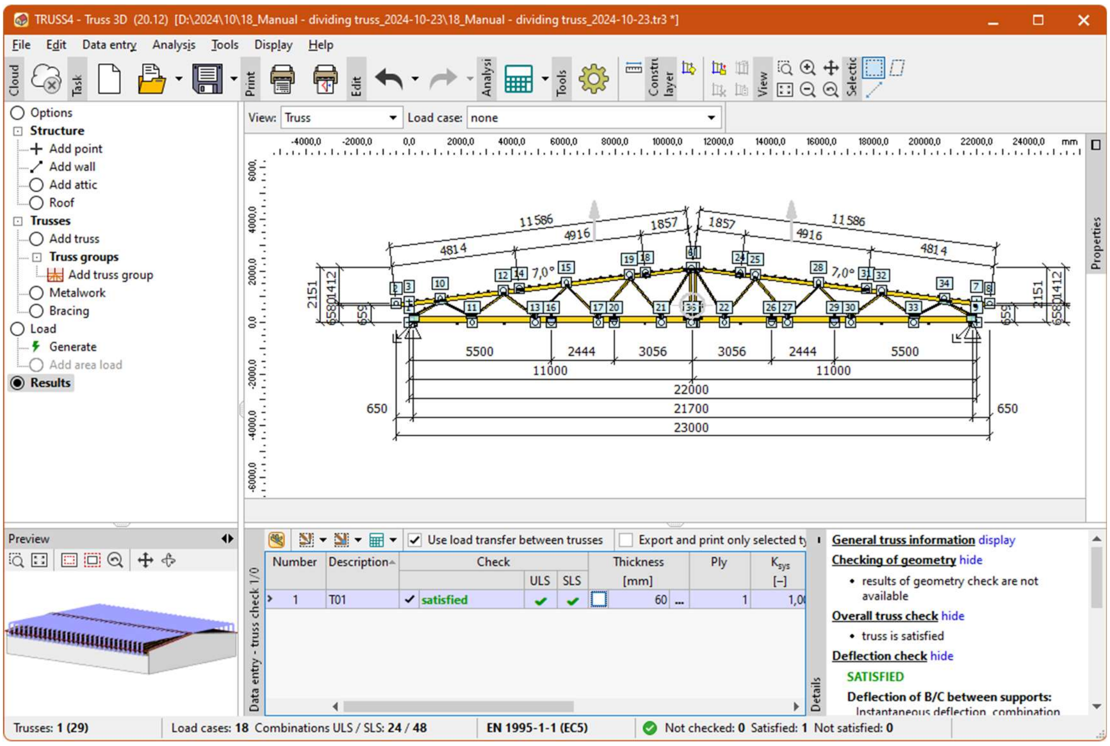
Modified properties of the joint on the bottom chord

Follow the same procedure for the joint in the apex of the structure. The values entered can be seen in the following figure.



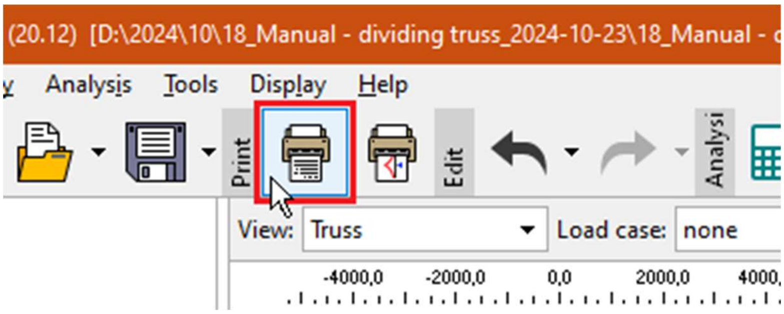
Properties of the general connection in the apex

After entering the general connections, we can start the automatic design again. The results are OK, so we can exit the "**Truss 2D**" program with the "**OK**" button in the bottom right corner of the window. The results will be transferred to "**Truss 3D**"



Designed truss in "Truss 3D"

If the design of the general connections has been carried out on the basis of the preliminary values of the internal forces, it is now advisable to check whether the designed connections are also suitable for the final truss. The program provides an easy way to obtain the forces in these connections. The table of these forces is included in the documentation of the structural analysis. Let's start the printing window for the text documents. We can use the item "**Print...**" in the "**File**" section of the main menu or the corresponding button in the main toolbar.



Button for printing text documents

Select that you want to print the document "**Structural analysis**" in the main toolbar of the print window. Then select "**Forces in general connections**". Since we have simple structural connections that connect only one member in each production part, it is more convenient to use the "**For production parts**" output method. The table for each joint and production part lists the maximum values of the normal force (positive value represents tension, negative represents compression), bending moment and shear force, and the corresponding forces from the combination.

Methods of listing forces in structural connections
The maximum value of the given component (normal, shear force, bending moment) including other components for the given load combination is always displayed in the table within the listing of internal forces in the connections. The maximum value is highlighted in bold. A negative sign for the normal force indicates the pressure in the connection or in the component. The structural analysis document offers two ways of listing the forces in general connections:
For production parts *- for each general connection, the results for each connected production part are listed. This is the sum of moments and the vector sum of forces from all members that the general connection overlaps in the given production part. The resultant of the forces is then converted into the coordinate system of the connection, the normal force represents the force in the longitudinal direction (the main direction, which is indicated by the highlighted edge of the rectangle), the shear force is perpendicular to this direction. This method of listing is particularly suitable for simple connections where only one member is overlapped in each production part. In more complex connections, these values can then be used to design the bearing capacity of the connection plate.*
For members *- The force and moment values are listed separately for each member being overlapped by the general connection. The member coordinate system is used. Thus, the direction of the normal force corresponds to the axis of the member. In addition to the normal and shear forces, the vector sum "**N+V$_3$**" is also listed, including the angle "**α**", which represents the deviation of the resultant of the forces from the axis of the member (timber grain). These values correspond to the design principle for connections according to EN 1995-1-1 and can therefore be used directly for the design of fasteners for the member.*

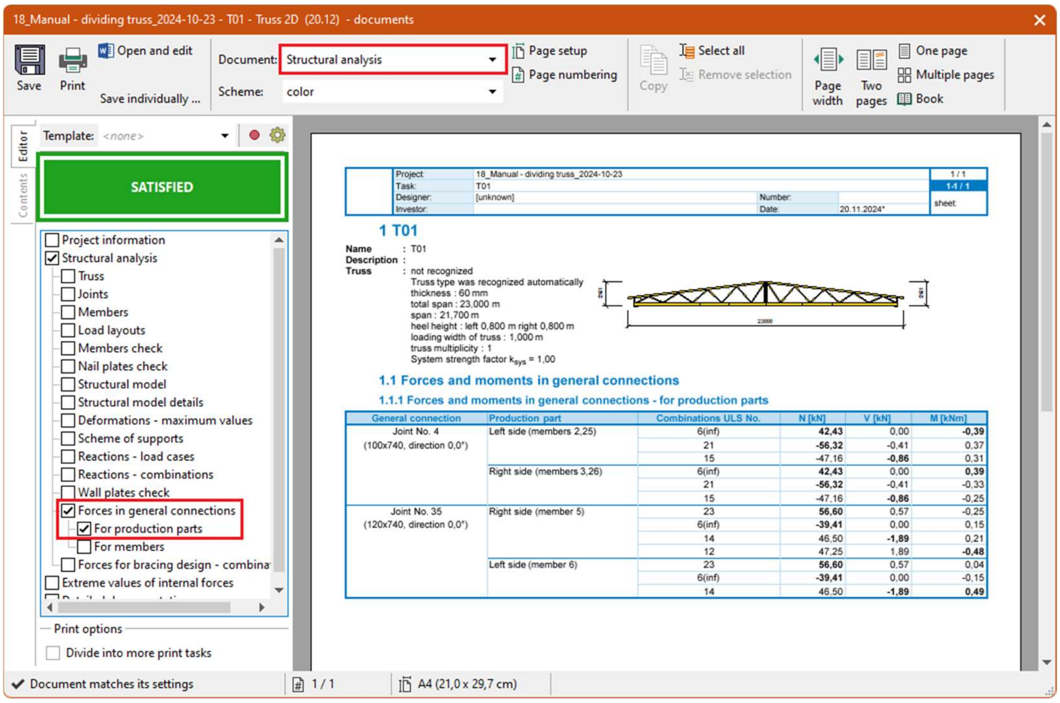
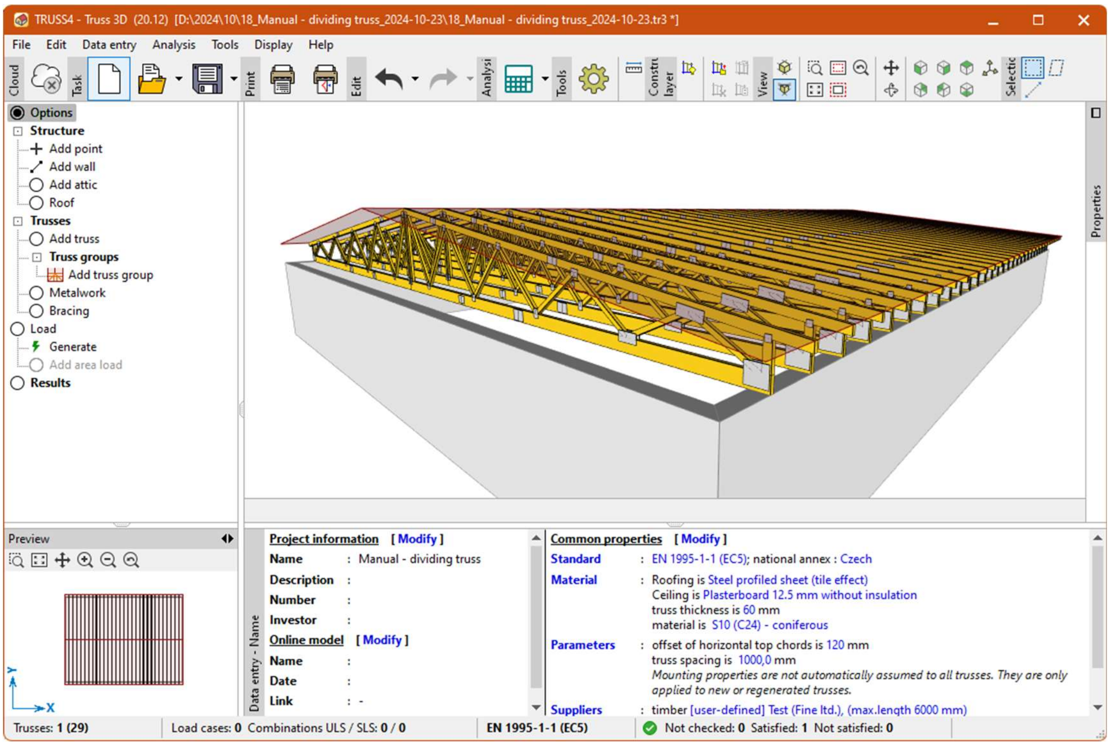


Table of forces in general connections

This completes our design of truss divided into production parts.



Structure with divided truss

Final project file *DEMO_EM05.tr3* can be found in the cloud folder "**Fine online examples**".

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