Engineering manual Nr. 06

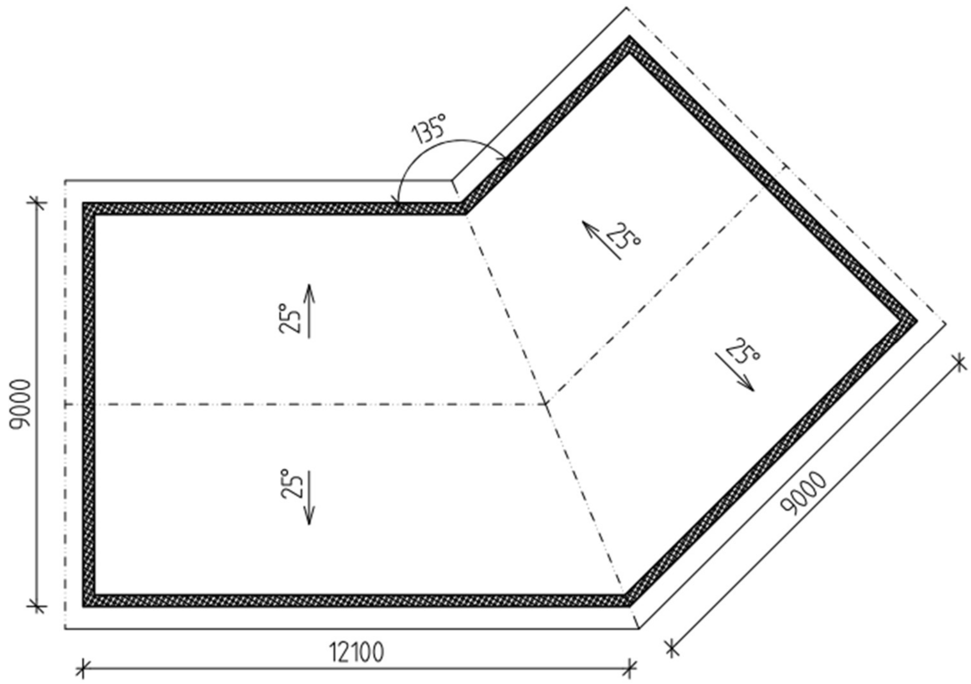
Updated 06/2025

Manual modelling in Truss 3D

Assignment

Program: Truss 3D
File: FineTrial.tr3

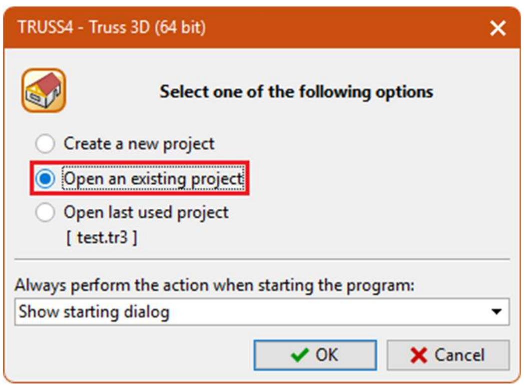
This manual will show the basic principles of manual modelling. The task is to design a truss structure for a simple dog-leg roof. The dimensions are shown in the following figure. A pitch of *25°*, a cantilever of *500mm* and an eaves height of *150mm* will be used. The structure will be modelled without the use of wizards and macros.



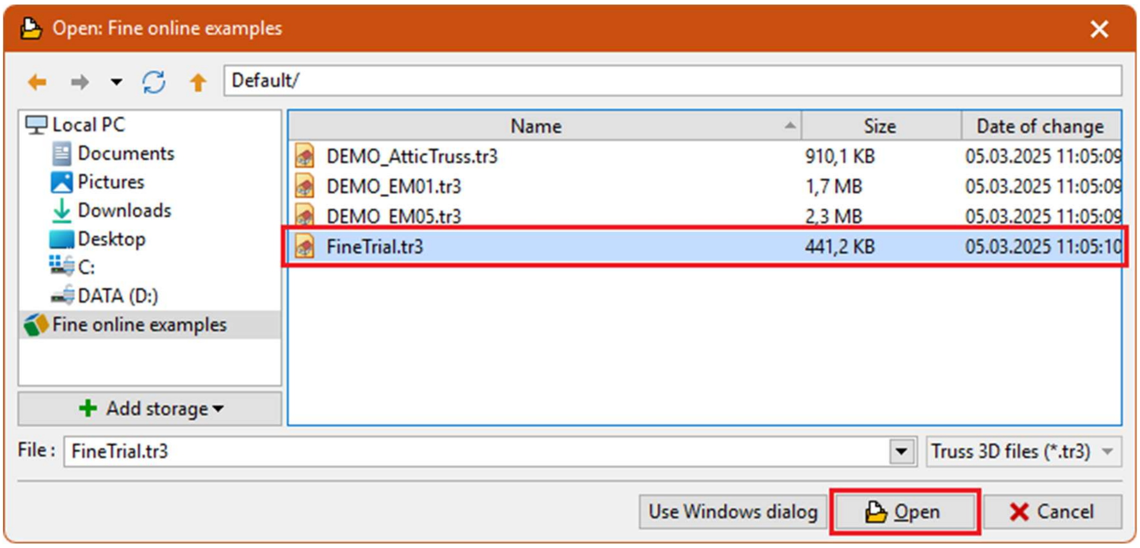
Ground plan of the structure

Starting a project and initial settings

When we start Truss 3D, we are prompted to choose how we want to start working. Select the "**Open an existing job**" option.

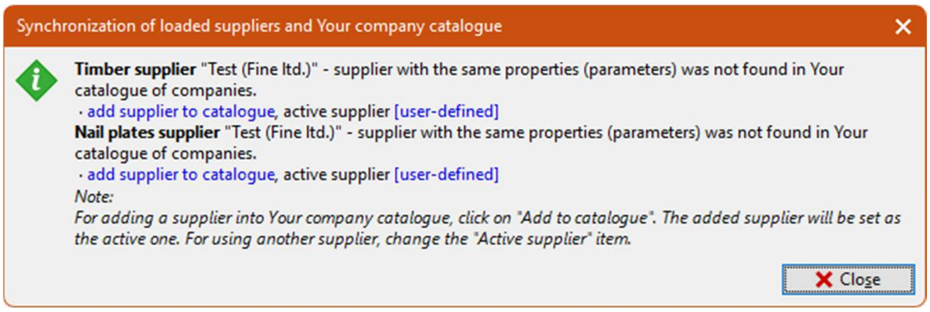


In the following window, navigate to the cloud folder "**Fine online examples**" and select the file "**FineTrial.tr3**". This file contains the default database of nail plates and the range of timber for our job.



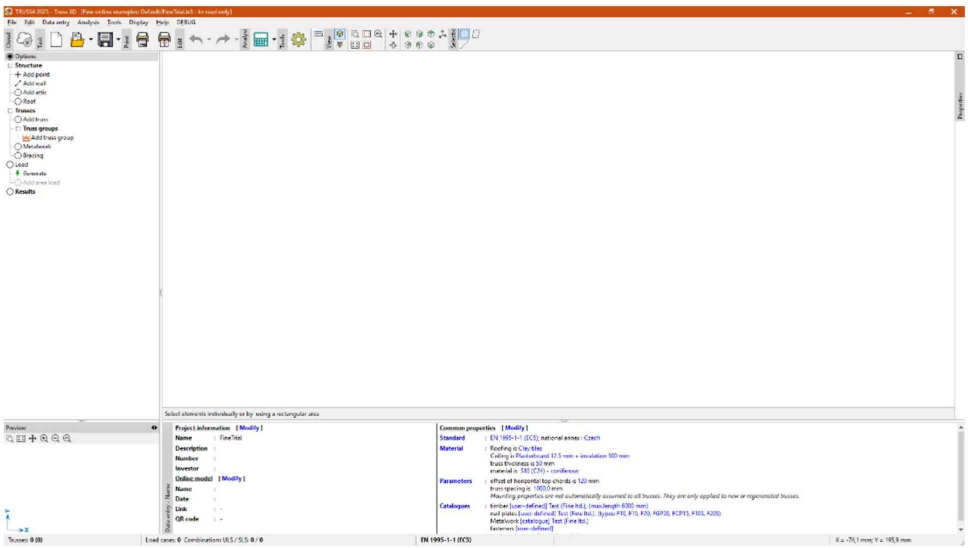
Selecting a file to be opened

When opening a file, we may get a window warning us that we will be using a range of timber or plates that we do not yet have in our catalogue. We can skip this warning by clicking the "**Close**" button.



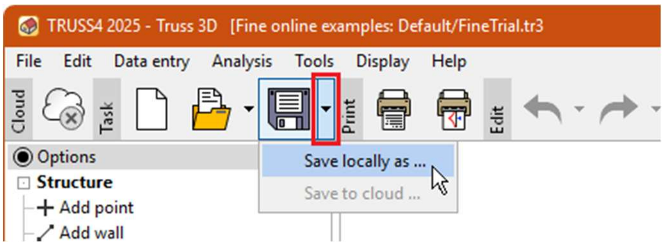
Request to add a supplier to the catalogue

After closing the warning, we will now see the basic screen of the "**Truss 3D**" program.



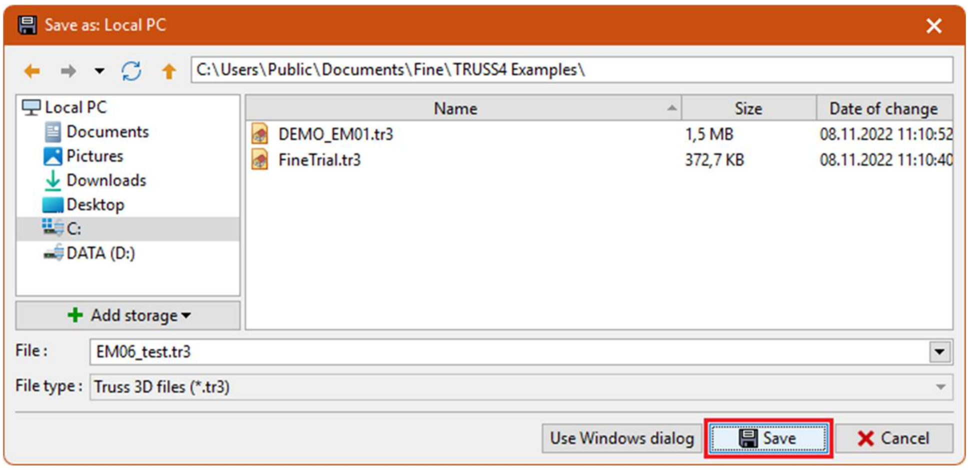
Truss 3D main window

Since we don't want to overwrite the original "**FineTrial.tr3**" file, we immediately save the project as a new project. We can use the standard path via the main menu item "**File**" - "**Save locally as ...**" or click on the arrow to the right of the "**Save**" button in the toolbar. A drop-down menu will appear, which also contains the "**Save locally as ...**" command.



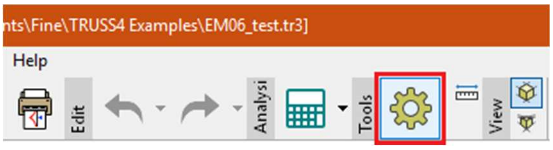
Saving the project as a new file

The folder for saving and the file name can be selected in the following window.



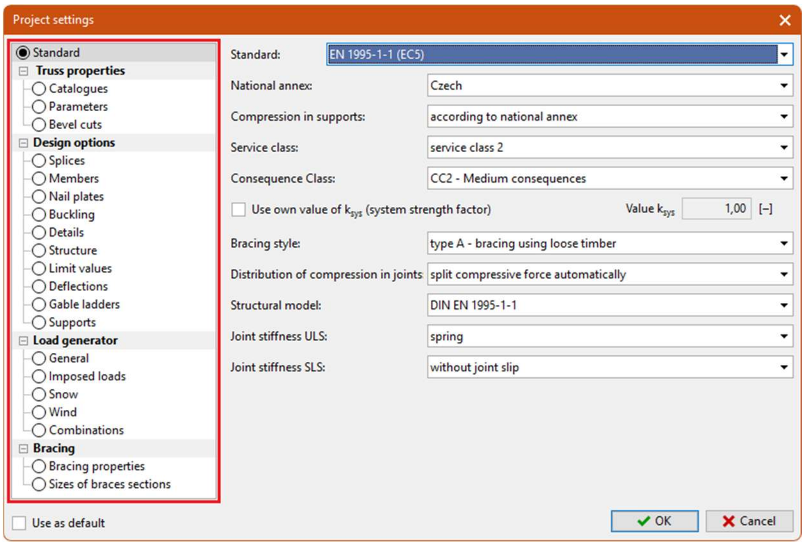
Saving the file

We can change some settings that will affect the design before we start modelling. The settings window can be launched using the corresponding button in the toolbar.



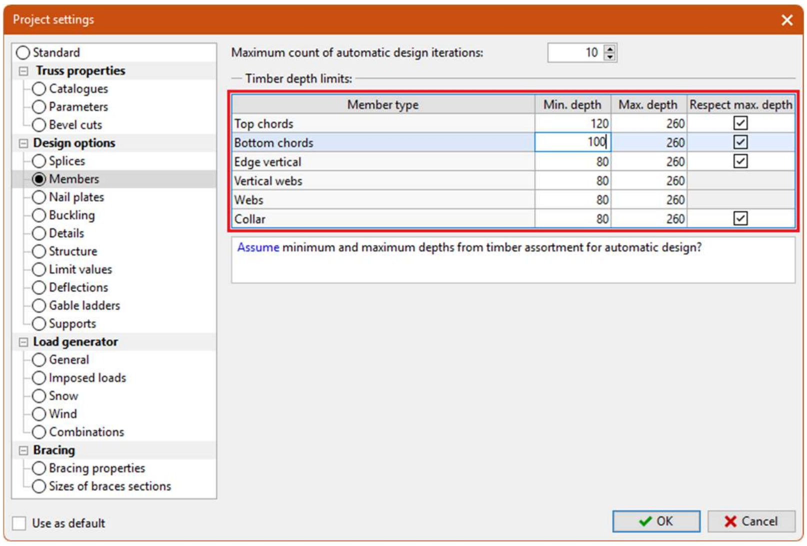
Button to edit project settings

The "**Project settings**" window will open. This window contains a tree menu in the left part, which can be used to navigate between the different pages with settings.



Window "Project settings"

We go to the "**Members**" section. This section contains the settings related to the design of the timber members. We will focus on the table "**Timber depth limits**" with the limiting dimensions of the cross-section for each type of member. Our goal is to set all top chords in the project to have a minimum depth of *120 mm* and all bottom chords to have a minimum depth of *100 mm*. We will not limit the maximum depth. After changing both values we can close the window with the "**OK**" button.

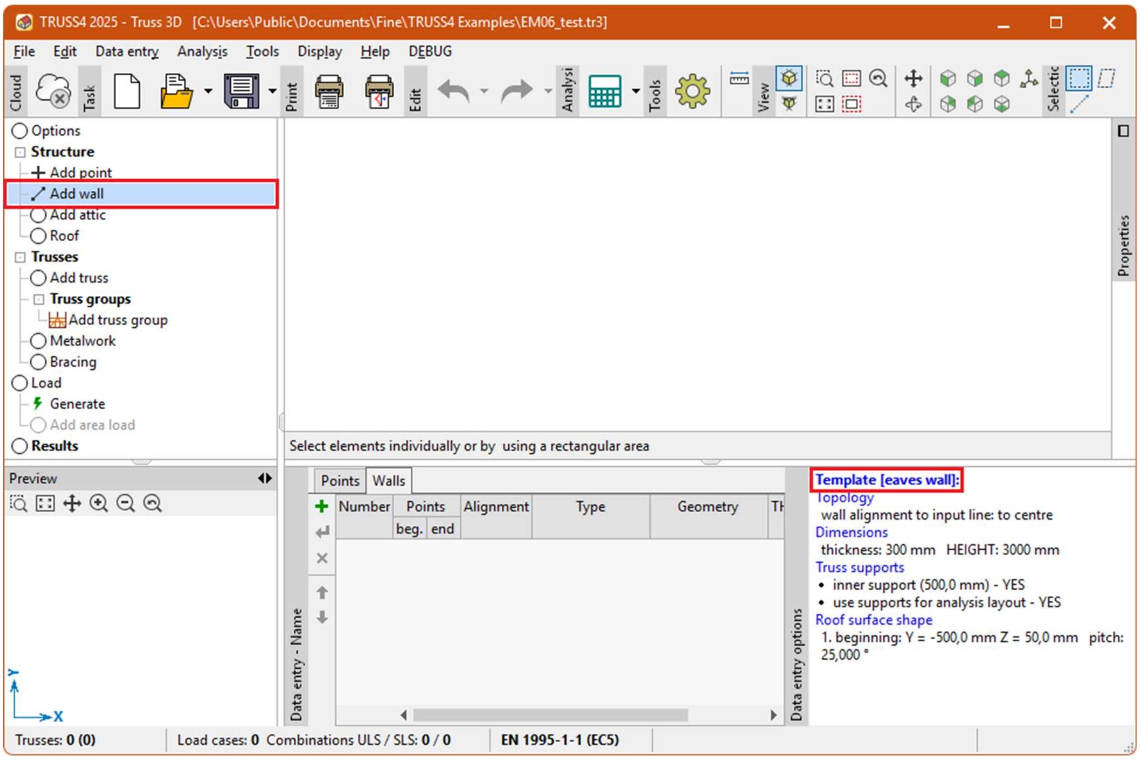


Change of the cross-section limits

Wall properties

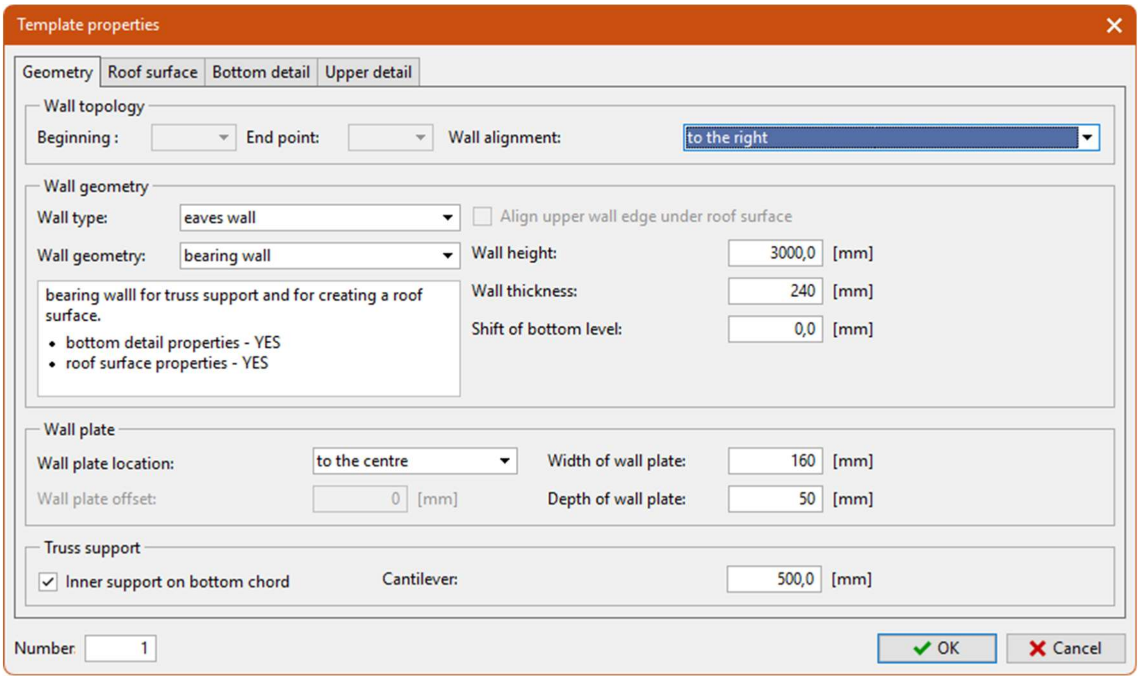
We can start to enter the building itself now. We will start with the walls. When we select "**Add Wall**" in the tree menu, a frame with the option to enter a wall template will appear in the bottom right corner. The wall template is used to specify the basic properties that will be assigned to new walls. In addition to geometry information, such as wall height and thickness or the size and position of the wall plate, the roof plane and eave detail can also be defined in this part. Clicking on the text "**Template [eaves wall]**" opens the template properties window.

TRUSS4



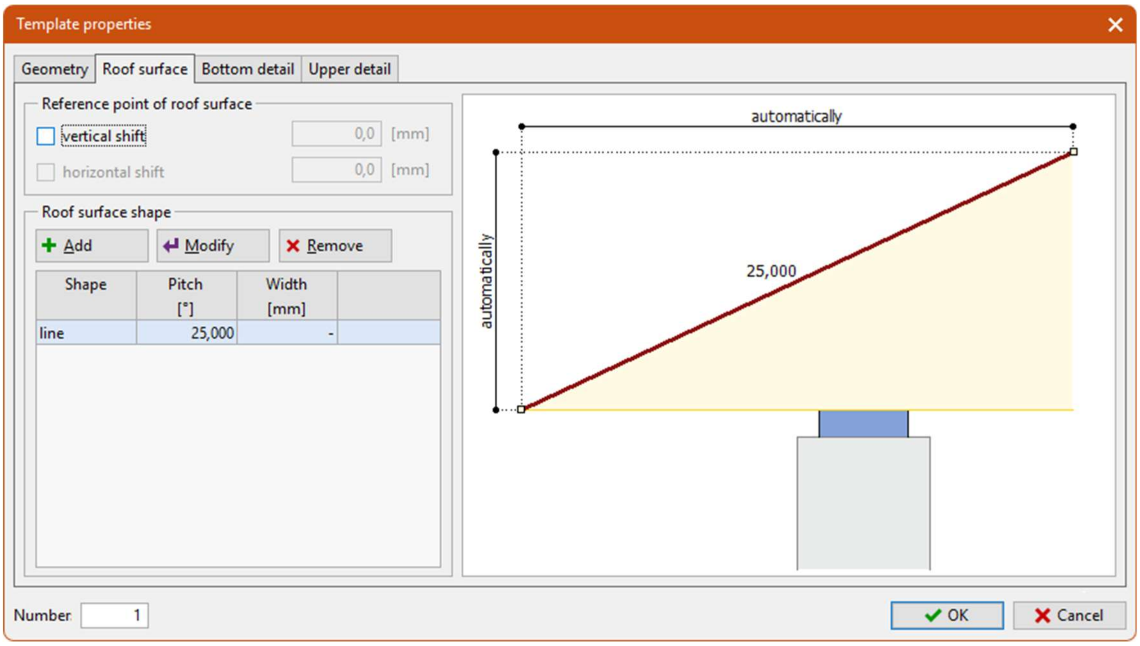
Input of wall template

First of all, we change the alignment of the wall to the input line to "**right**" on the first tab "**Geometry**". This is because we will be entering the building outline using the outer dimensions and the input line in the axis of the wall would complicate the entry. The other parameters will be adjusted according to the following figure:



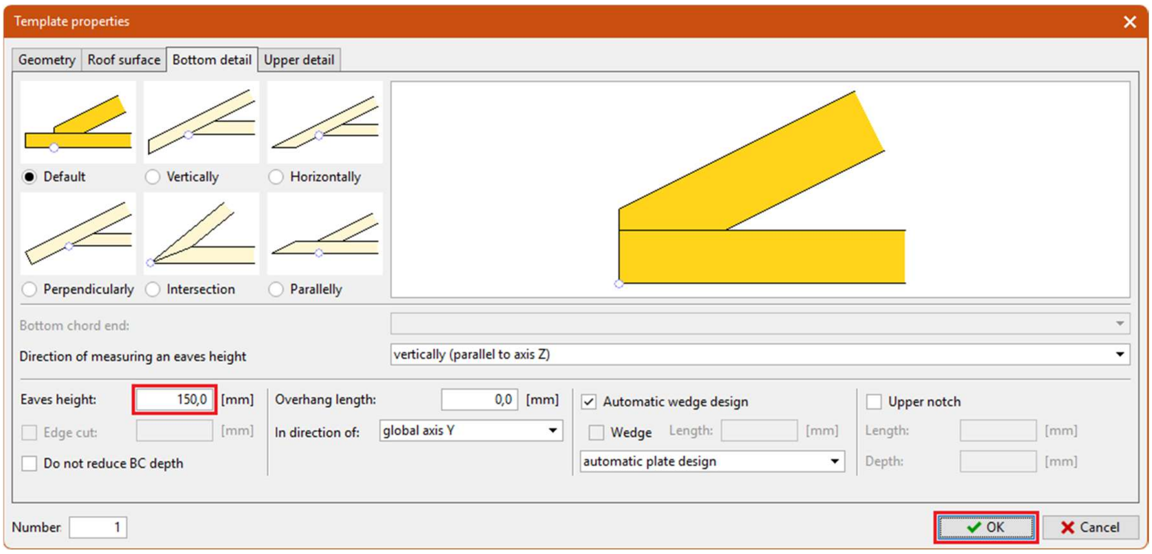
Wall template, "Geometry" tab

Go to the second tab "**Roof surface**". Here we leave the default pitch of 25°.



Wall template, "Roof surface" tab

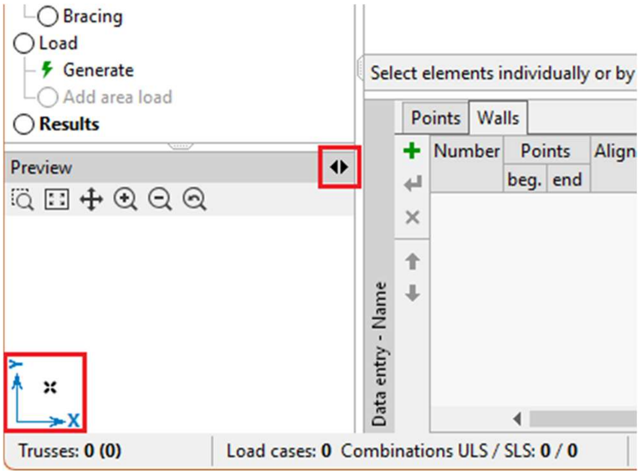
We enter the eaves height *150 mm* on the following tab "**Bottom detail**". The last tab "**Upper detail**" contains properties relevant only for eaves details with an edge vertical. However, since we do not have such a structure, we can close the template properties window with the "**OK**" button after entering the eaves height.



Input of the eaves height

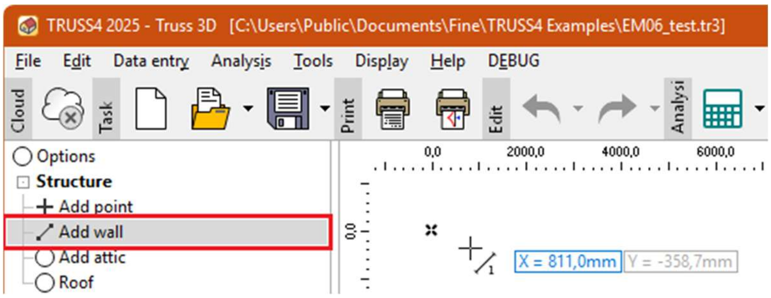
Entering the structure

We proceed to the input of the walls. The "**Truss 3D**" program has two views: a 2D view and 3D model of the structure. These views can be alternately displayed either on the main workspace, which occupies most of the program window, or on a secondary workspace, which is located in the left bottom corner. Graphical input (using the cursor on the workspace) can only be performed on the 2D workspace. So if we have a 2D view minimized in the left bottom corner (recognized by the *X* and *Y* axis cross), we move it to the main workspace using the "◀▶" button in the header of the secondary workspace.



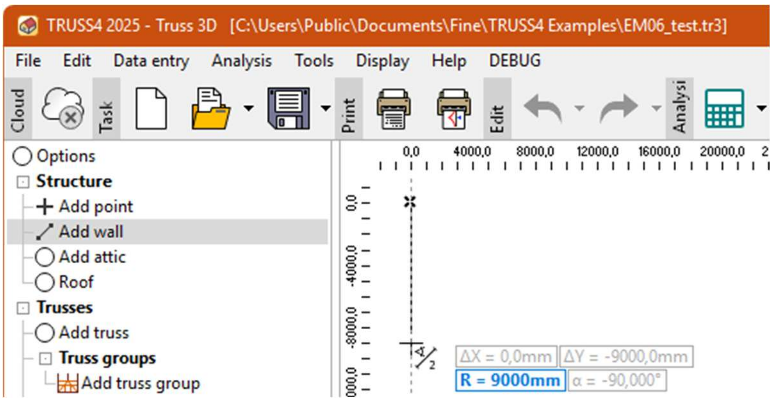
2D workspace with axis cross in the bottom left corner

We still have the "**Add Wall**" mode activated earlier in the tree menu. Therefore, the program shows the number *1* next to the cursor on the 2D workspace (indicates input of the first point of the wall) and also the coordinates with the current cursor position.



"Add wall" input mode

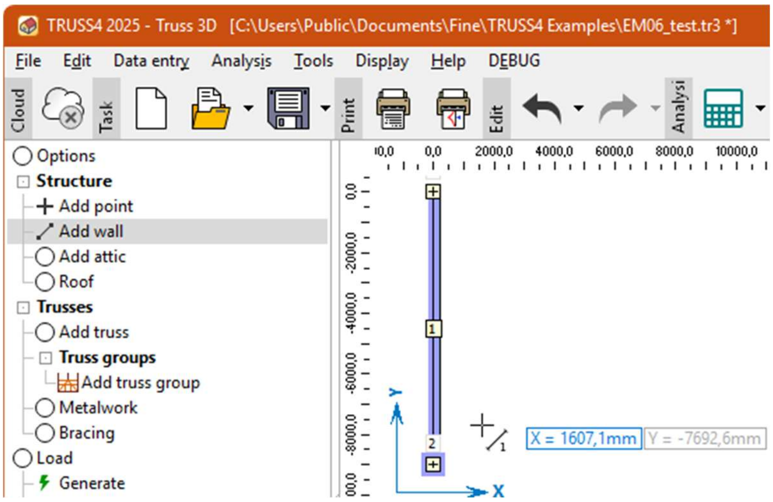
To specify the beginning of the first wall, use the "⌘" marker that represents the point *[0,0]*. To enter the second point, point the cursor downwards while typing *9000* on the keyboard.



Input of the first wall

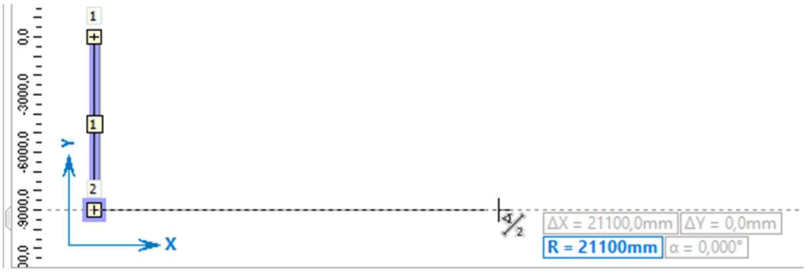
Confirm this input by pressing the "**Enter**" key. This inserts the first wall on the workspace. The wall runs downwards from the origin of the coordinate system and is *9000 mm* long. The cursor is immediately put back into the mode of entering the start point of the next wall.

TRUSS4



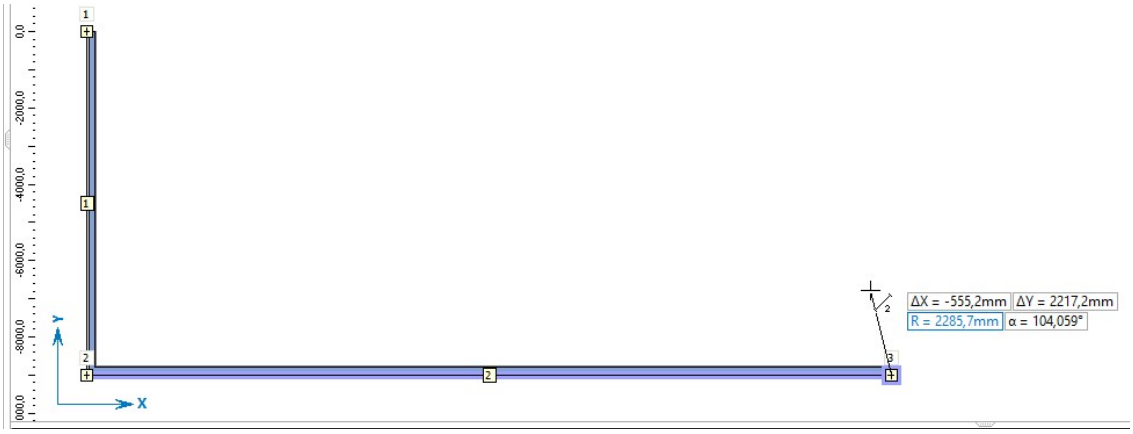
2D workspace with the first wall

As the start of the second wall we choose the point *2*, i.e. the bottom end of the first wall. We then point the cursor to the right of this point and at the same time enter the distance *21100* on the keyboard. Confirm the entry again with the "**Enter**" key.



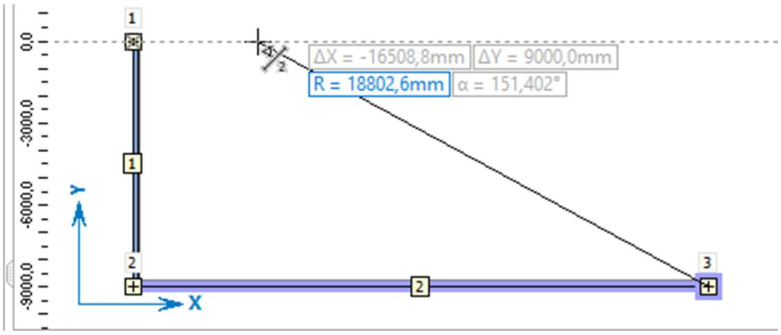
Entering the length of the second wall

The second wall was inserted into the structure. Again, we use the end point of the second wall as the start of the third wall.



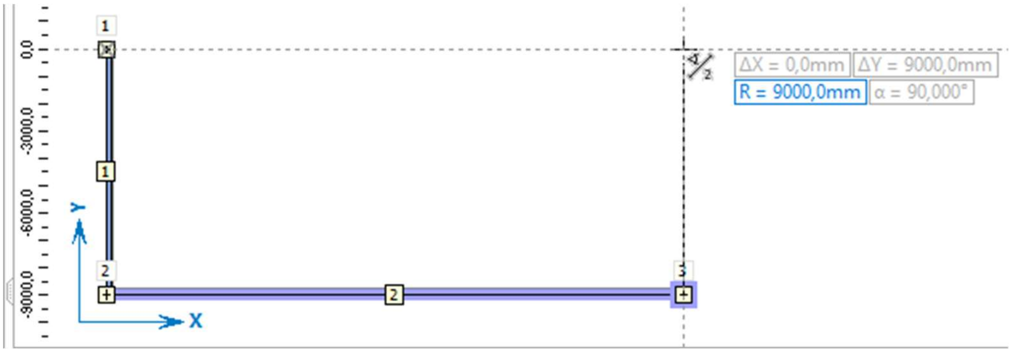
Input of the third wall

The end point of the third wall could be specified in the same way as for the previous walls. But we will also try other tools that the program offers for modelling. In this case, we will use the apparent intersection. Place the cursor over the point *1* and hold the cursor there. After a moment, the "✕" symbol will be drawn over the point marker. From that moment, you can use polar tracing from that point as well. Move the cursor to the right and you can see that a tracing ray is displayed from point *1*.



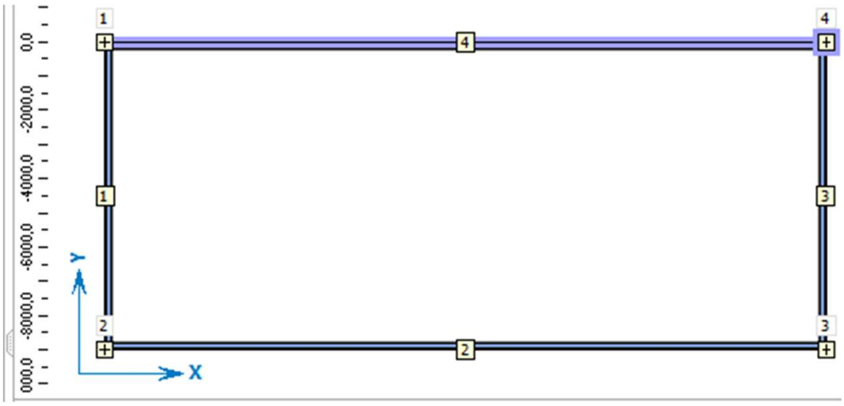
Tracing from point 1

Move the cursor to the planned fourth corner where the horizontal ray from point *1* intersects with the vertical ray from point *3*.



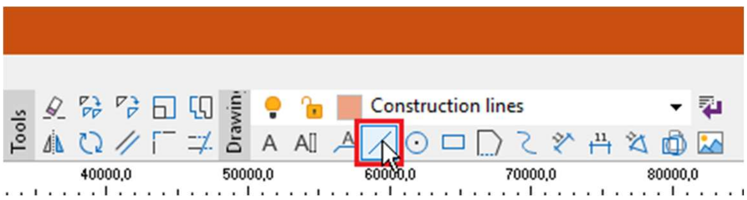
New intersection of two rays

Use this intersection as a grip point and enter the end point of the third wall at that location. The last wall will then be used to close the entire perimeter.



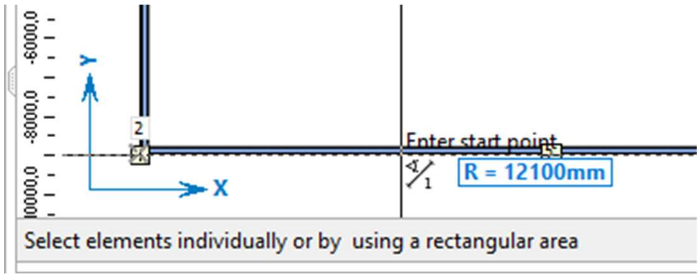
Closed perimeter of the structure

Now we modify the floor plan using several steps to match the assignment. In the top toolbar, select the command to draw construction lines.



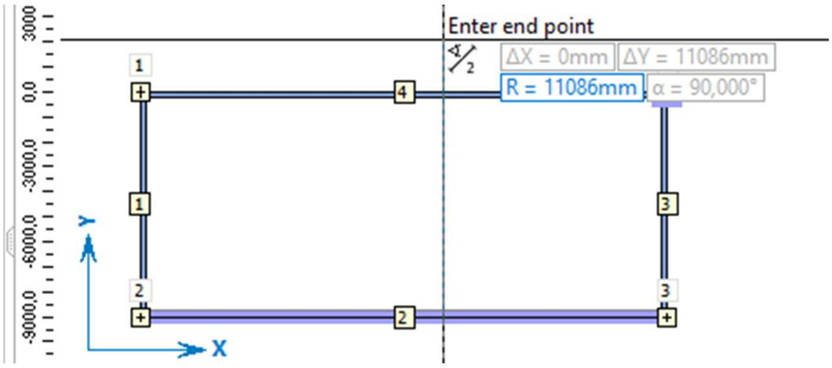
"Line" tool

Use this tool to draw a vertical construction line at a distance of 12100 mm from the left edge of the ground plan. We will use polar tracing again. This time to specify the first point of the line. Place the cursor over the point *2* and wait until the "✕" mark appears. We can then enter the origin of the line relative to this point. Point the cursor to the right and enter offset *12100* on the keyboard.



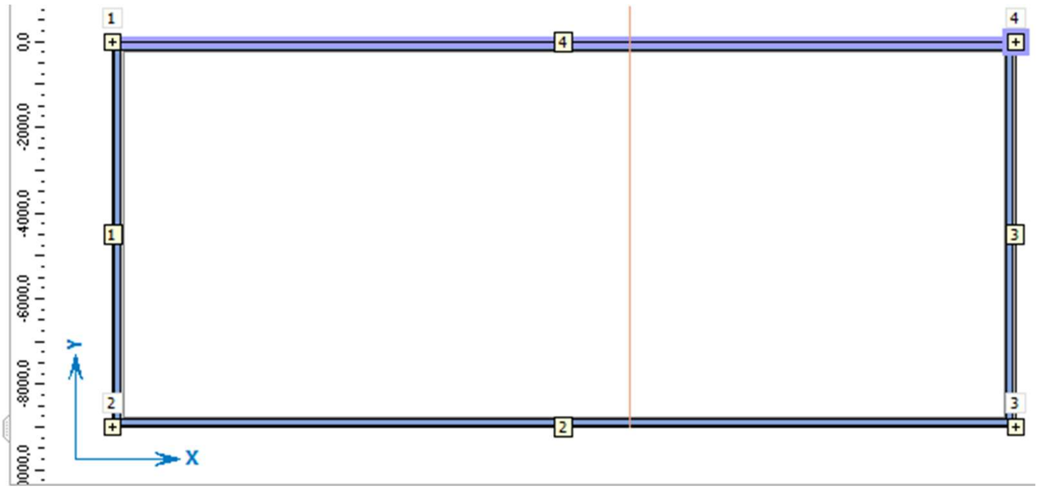
Beginning of the construction line created with the help of tracing from point 2

Confirm the entry by pressing the "**Enter**" key. This sets the start of the construction line to 12100mm from the point *2*. The end of the construction line does not need to be specified exactly; it is sufficient if the line is vertical and extends over the building.



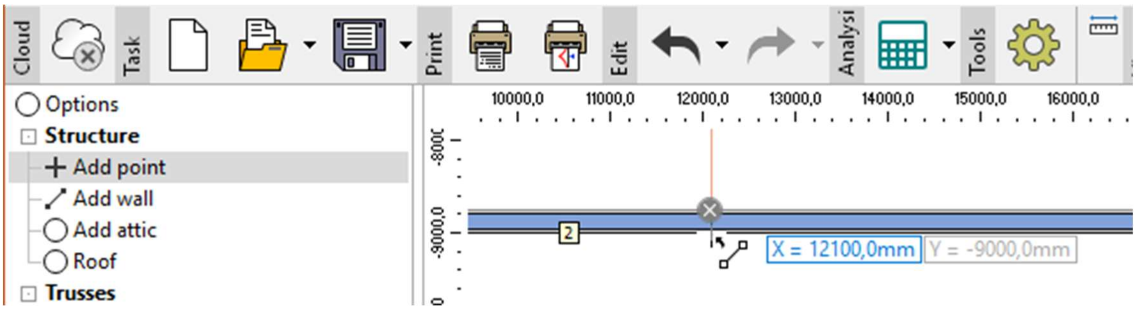
Entering the second point of the construction line

Now we have a rectangular ground plan with one construction line.



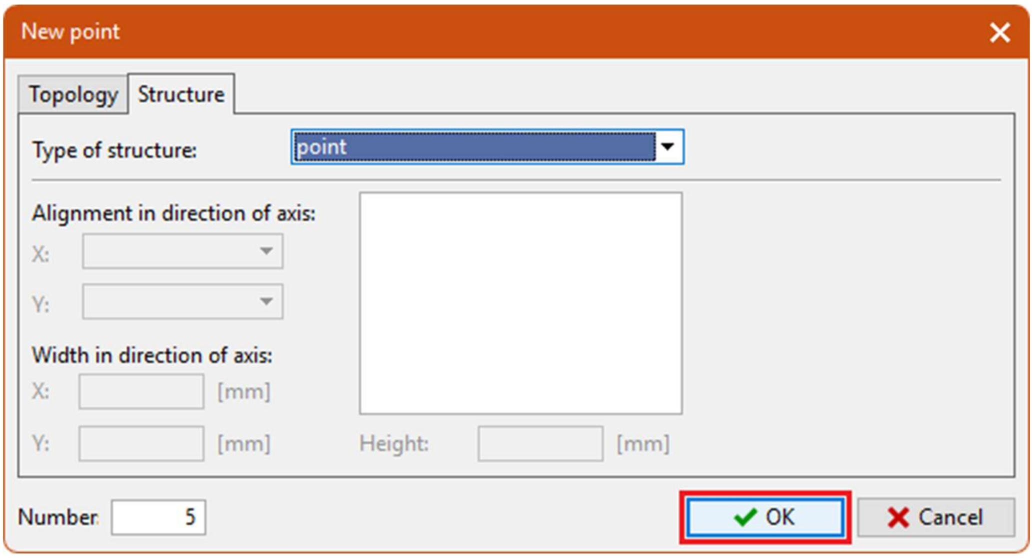
Inserted construction line

We will continue by adjusting the ground plan to the final shape. In the tree menu, select the "**Add point**" tool. Insert one point by clicking at the intersection of the construction line and the wall *2*. The program will automatically align cursor to the snap points; we can use any of them (intersection of the line with the wall edges or with the wall plate).



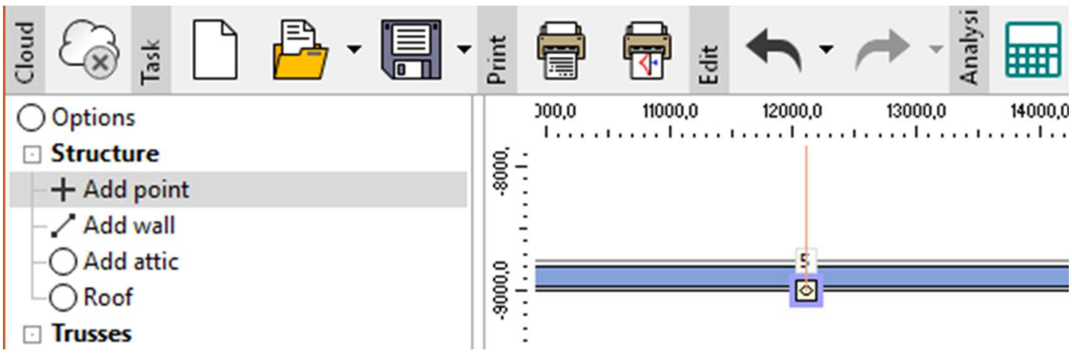
Inserting point using a snap point

The "**New Point**" window, which contains the point properties, appears after clicking. In this case, just confirm the window with the "**OK**" button.



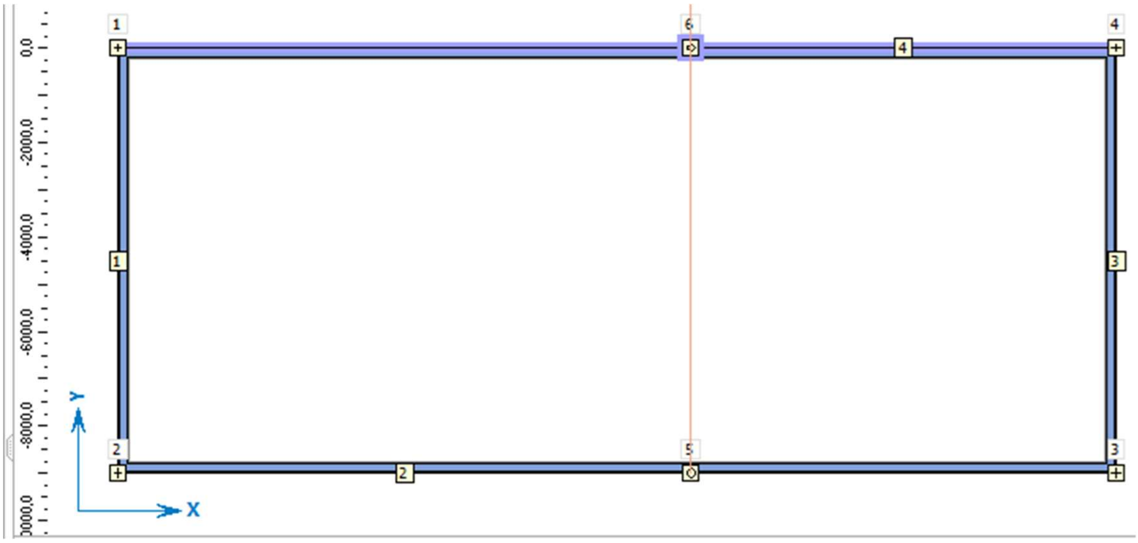
"New point" window

A new point marker will then appear on the wall at the point of intersection.



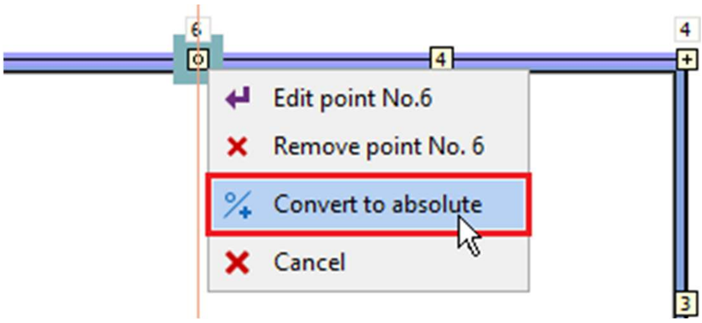
New point on the wall

Enter the second point at the intersection of the construction line and wall *4* in the same way.



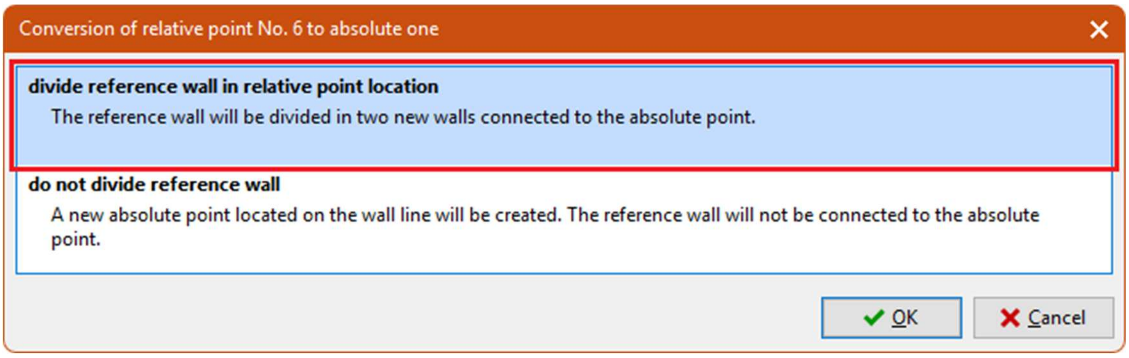
Ground plan with inserted points

Now divide the longitudinal wall at these points. Right-click on the insertion point and open the context menu. There, select the "**Convert to absolute**" tool.



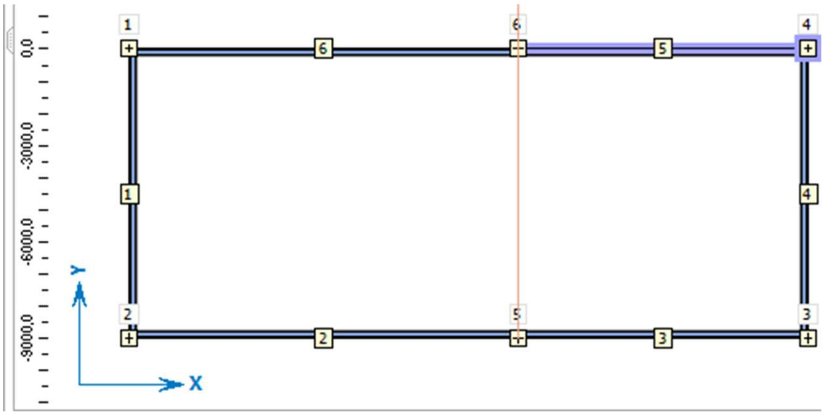
Context menu of the point

In the following window, select the option "**Divide reference wall in relative point location**" and close the window with the "OK" button.



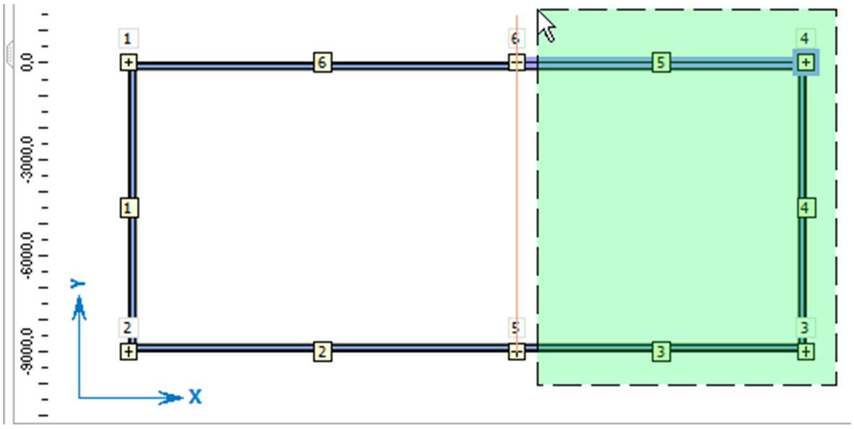
Dividing reference wall

Do the same procedure for the second inserted point. The result is a structure with six individual walls in total.



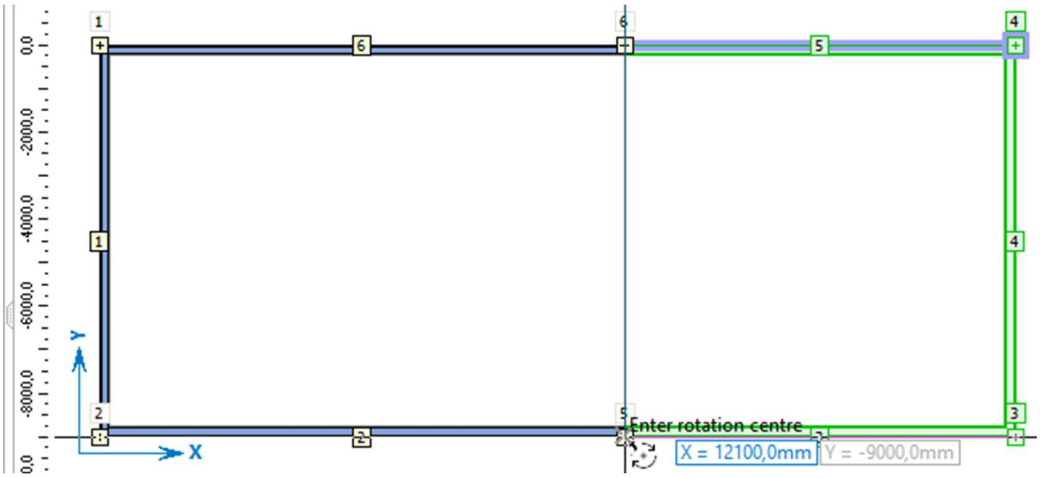
Ground plan with divided walls

The next step is to rotate the right part. The easiest way to select the three walls on the right is to make a rectangular selection by dragging from right to left. In this case, the selection area will be highlighted in green and all elements that at least partially interfere with the selection will be selected. When dragging from left to right, the selection is highlighted in blue and only objects that are completely within the rectangle are selected.



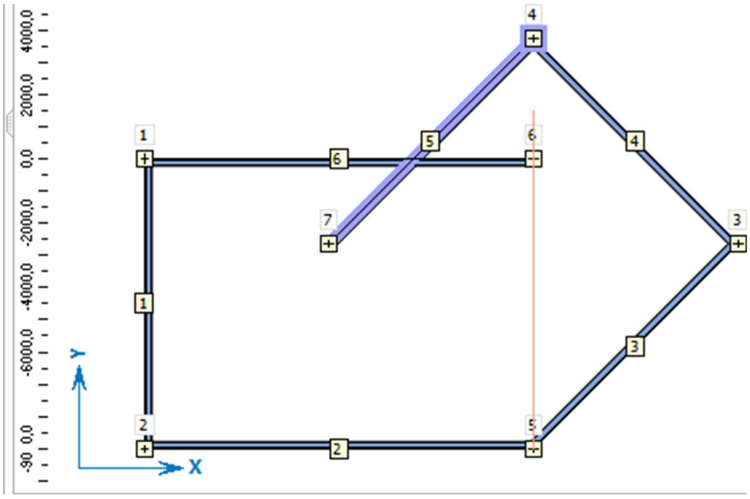
The selection of the right part of the structure

Then select the "**Rotate**" command in the "**Tools**" bar. First, we must specify the centre of rotation. Click on point *5*.



Specifying the centre of rotation

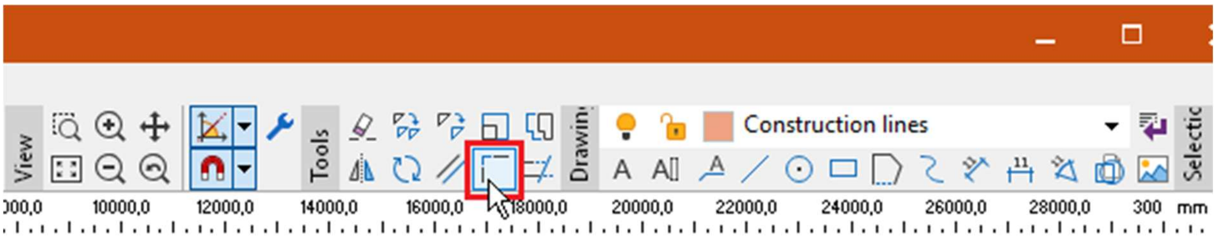
The following step is to specify the angle of rotation. We can enter the value 45 on the keyboard and confirm the entry with the "**Enter**" key. A simpler option is to just rotate the selected part of the structure with the cursor. Thanks to the fact that the program offers alignment in significant directions, it is very easy to get the rotation in the required direction. Just move the cursor in the direction of rotation and confirm the correct position by clicking when the cursor displays "**α=45°**". The design after this operation looks as follows:



Structure after rotating the right part

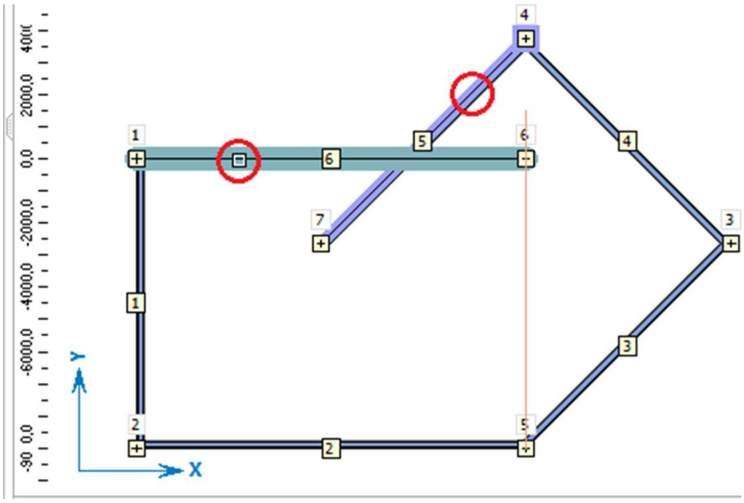
"Rotate" tool modes
When executing the "**Rotate**" command, it is possible to select also other tool modes in addition to the normal moving of selected objects by rotation. The modes are selected with the corresponding keyboard shortcut after selecting the centre of rotation. For example, if you press the "**c**" key before entering the angle, the tool will switch to the "**Copy**" mode.
The following options are available in addition to the basic rotation mode:
Copy - a mode that creates a copy of the selected elements in a new position. The original selected elements remain in place. This mode is activated with the "**c**" key. After selecting the mode, the rotation angle is entered as in the basic mode.
Reference - This mode allows you to specify a reference angle that represents the initial state of rotation. This mode is useful if the selected elements need to be aligned to existing objects or if the exact rotation angle would have to be calculated manually. It can also be used, for example, to align the scanned drawing to the orthogonal system of the workspace. The mode is accessed with the "**r**" key. After selecting the mode, the reference angle is first entered with two points. This is followed by entering the rotation angle, which already corresponds to the standard mode.

The right part of the structure is rotated, the next step is to shorten walls *5* and *6*. We will use the "**Extend**" tool for this operation.



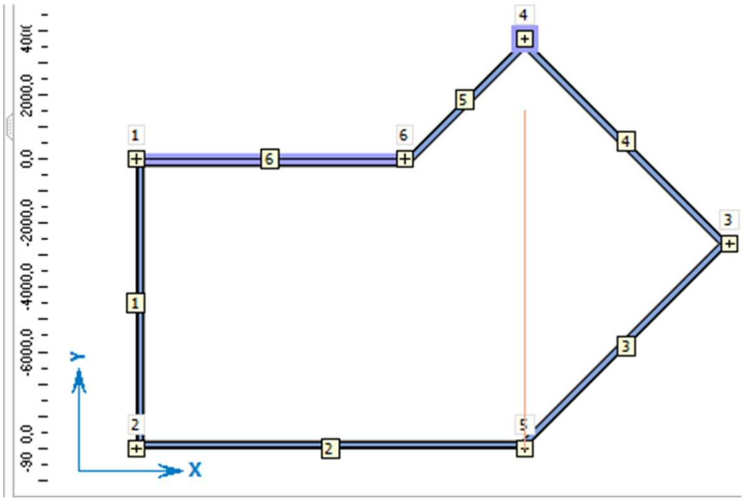
Choice of "Extend" tool

After selecting the "**Extend**" tool, click on walls *5* and *6* in the parts that you wish to keep.



Specifying walls 5 and 6 for the "Extend" tool

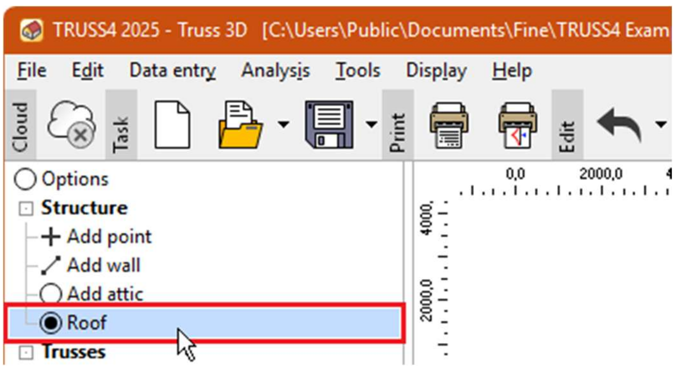
The result is a closed structure.



The final geometry of walls

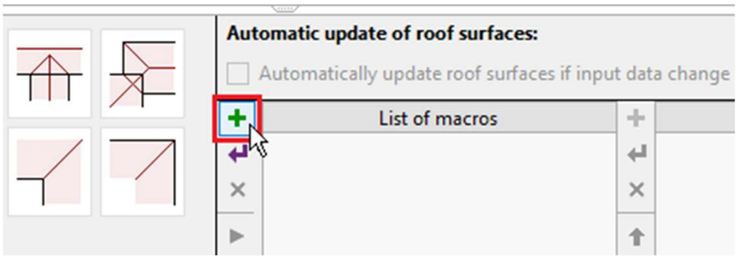
Creating roof planes

The next step is to create the roof planes. Go to the "**Roof**" section of the tree menu.



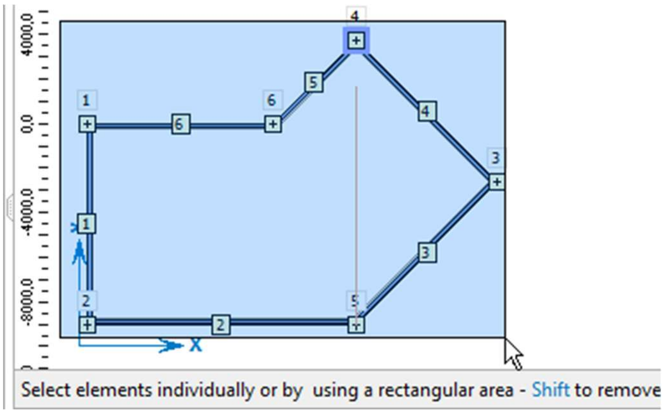
Part "Roof"

The roof planes are built using the tools in the bottom input frame. The program allows to record the individual steps that make up the roof planes in a macro that is automatically executed whenever the geometry of the structure changes. We will use this option as well. In the toolbar to the left of the "**List of macros**" table, use the "**+**" button to start recording a new macro.



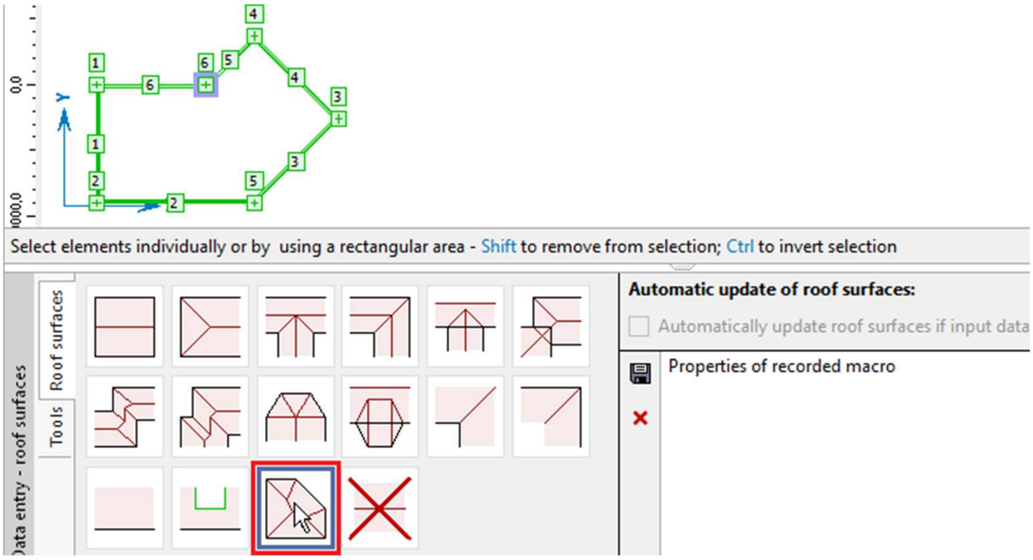
Recording a new roof macro

In this case, the construction of the roof planes will be simple. We will use only one step, namely the creation of roof planes over the general polygon. Before running the tool, all the walls forming the polygon must be selected. Since we have no internal walls in our structure, we can select the entire structure. Either by using the cursor on the workspace or the keyboard shortcut "**Ctrl**" + "**A**".



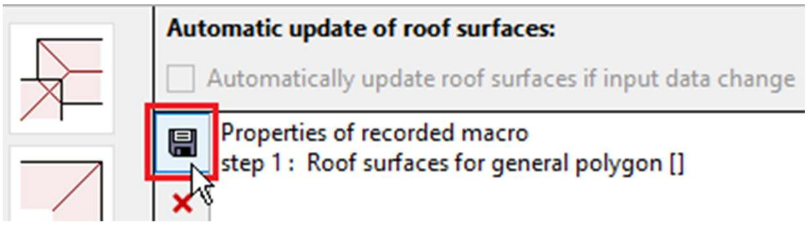
Selection of walls forming the polygon

The selected walls are then highlighted in green. Then use the "" button to select the "**Roof surfaces for general polygon**" tool, which will calculate the roof geometry.



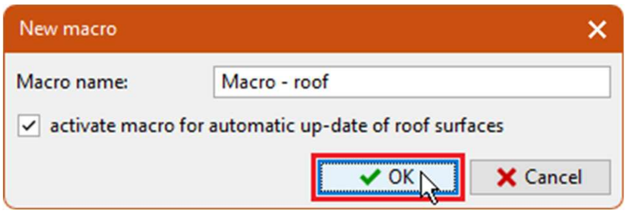
Running "Roof surfaces for general polygon" tool

No more steps are needed. Save the macro using the "" button (Finish and save record).



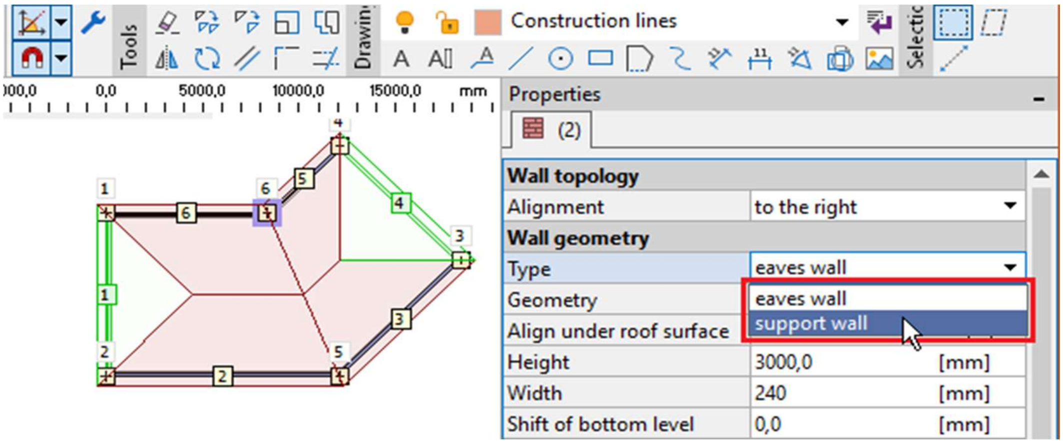
Saving the macro

The program prompts us to enter the name of the macro. After entering it, confirm the creation of a new macro by clicking "**OK**".



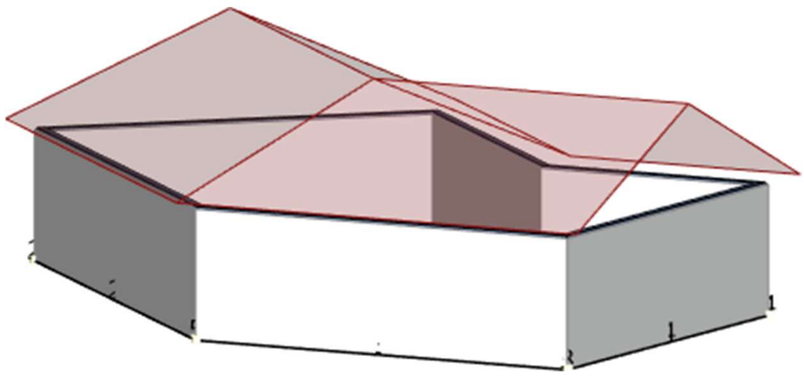
Dialog box for entering the macro name

We did not change the template properties while entering the walls, so the roof planes are created for all walls. However, according to the specification, we must create a gable roof, not a hipped one. We must modify the properties of the gable walls. Select the gable walls 1 and 6 on the workspace and change their type to "**Support wall**" in the "**Properties**" sidebar.



Changing the type of front walls

After changing the type of front walls, the roof macro is automatically run and the roof planes for the front walls disappear. Result is a gable roof.

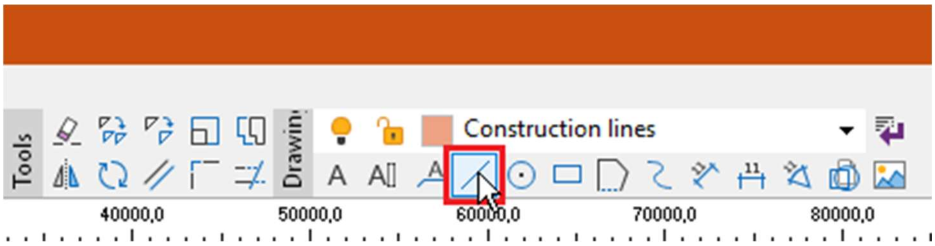


Structure with roof planes

This gives us the geometry of the structure and we go on to specifying the trusses.

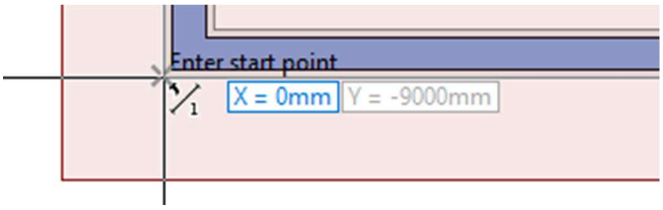
Input of trusses

Go to the "**Trusses**" section of the tree menu. We plan to place the first truss at 500mm from the outer edge of the gable wall. There are several options for specifying this, we will use the user-friendly procedure of using a construction line. We enter it at the outer edge of the gable wall and then move it to the correct position. The truss is then entered using the intersections with the eaves lines. Again, select the "**Line**" tool in the toolbar at the top of the window.



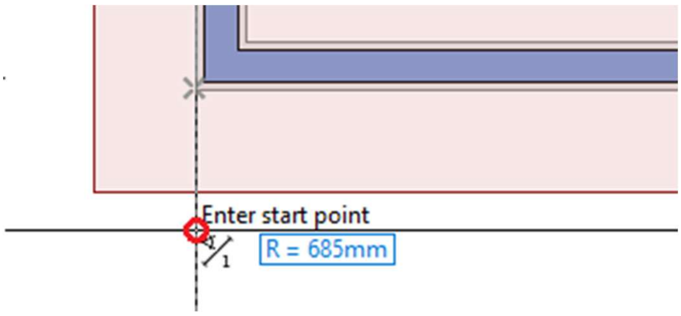
The "Line" tool

When specifying trusses, it is advisable to use clear grip points, such as intersections, and avoid ambiguous cases where a structural line is drawn to the edge of the roof plane but no actual intersection occurs at the point. When inserting trusses in such cases, there is a risk that the truss will only snap to the end of the construction line and the additional data for building the truss shape from the reference walls will be missing. For this reason, we will again use polar tracing when entering the line. Place the cursor over the corner of the building and hold the cursor in place for a moment. The "✕" symbol will appear in the corner of the building. From that moment, we can use polar tracing from that point.



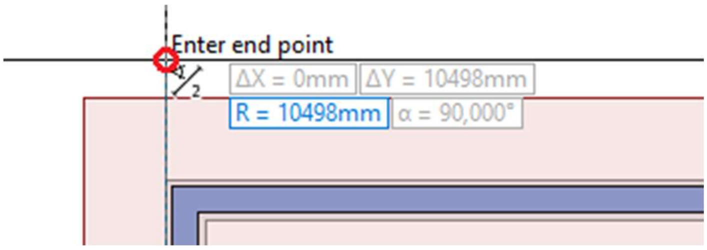
With polar tracing symbol

Now we can move the cursor lower, below the level of the eaves line of the wall *2*, and click to enter the beginning of the line.



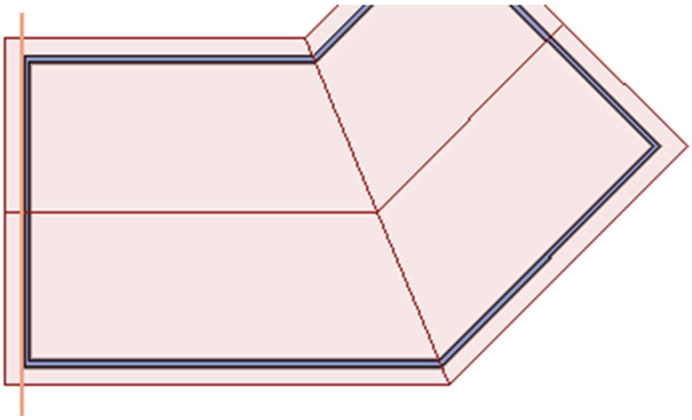
Entering the first point of the line

The construction line will run along the outer edge of the gable wall. Enter the second point behind the eaves line of the wall *6*. Use the vertical alignment of the cursor when entering.



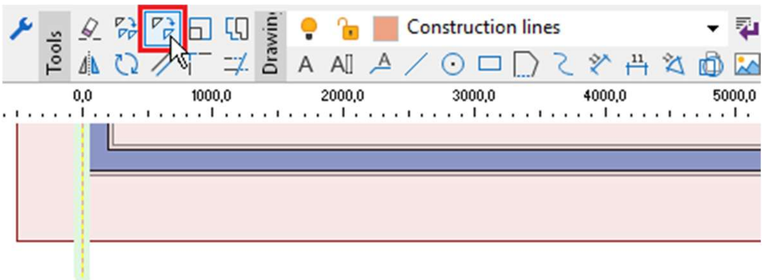
Entering the second point of the line

Now we have a construction line in the ground plan that exceeds the eaves lines.



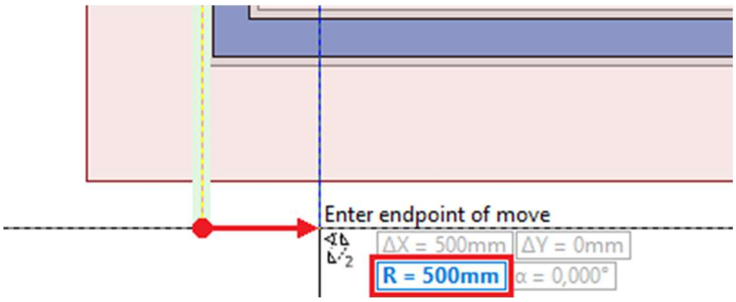
The roof with the entered line

In the next step, we will move the line to the place where we plan to insert the truss. Select the line and use the "**Move**" function from the "**Tools**" bar.



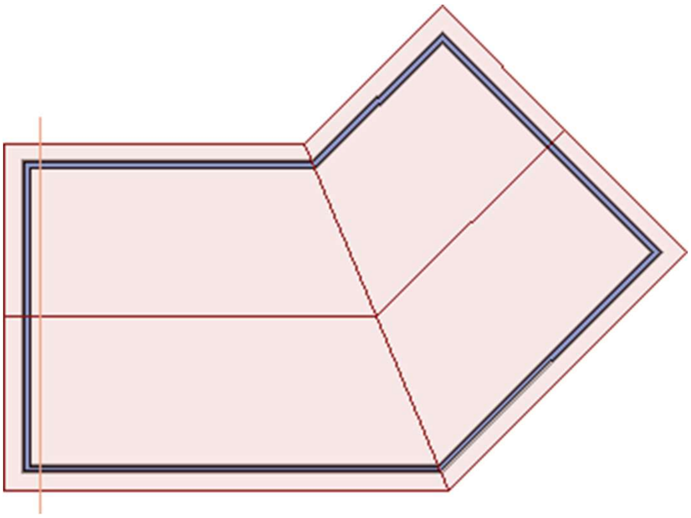
The "Move" tool

Then enter the first point of manipulation on the workspace, point the cursor in the right direction, and enter the offset "**500**" on the keyboard. The entered value is displayed next to the cursor. Confirm the entry with the "**Enter**" key.



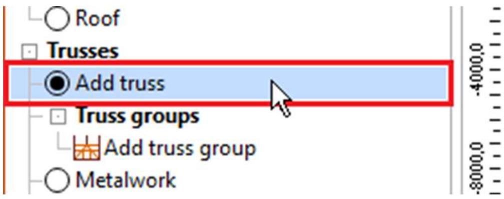
Entering the value of displacement

The line will be moved by 500mm to the right to the place where the truss will be inserted.



Structure with shifted line

The next step is the insertion of the trusses. In the tree menu, select the "**Add truss**" command.

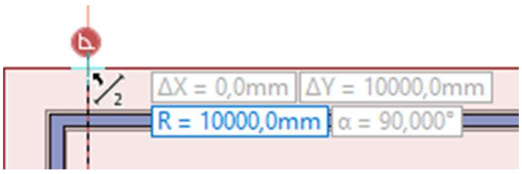


Tool "add truss"

The beginning of the truss is the intersection of the construction line with the eaves line of the wall *2*. This orientation is preferable because the stubbed trusses in the central part will have the same origin as the basic trusses. Therefore, it will not be necessary to rebuild the production line significantly when manufacturing the trusses.

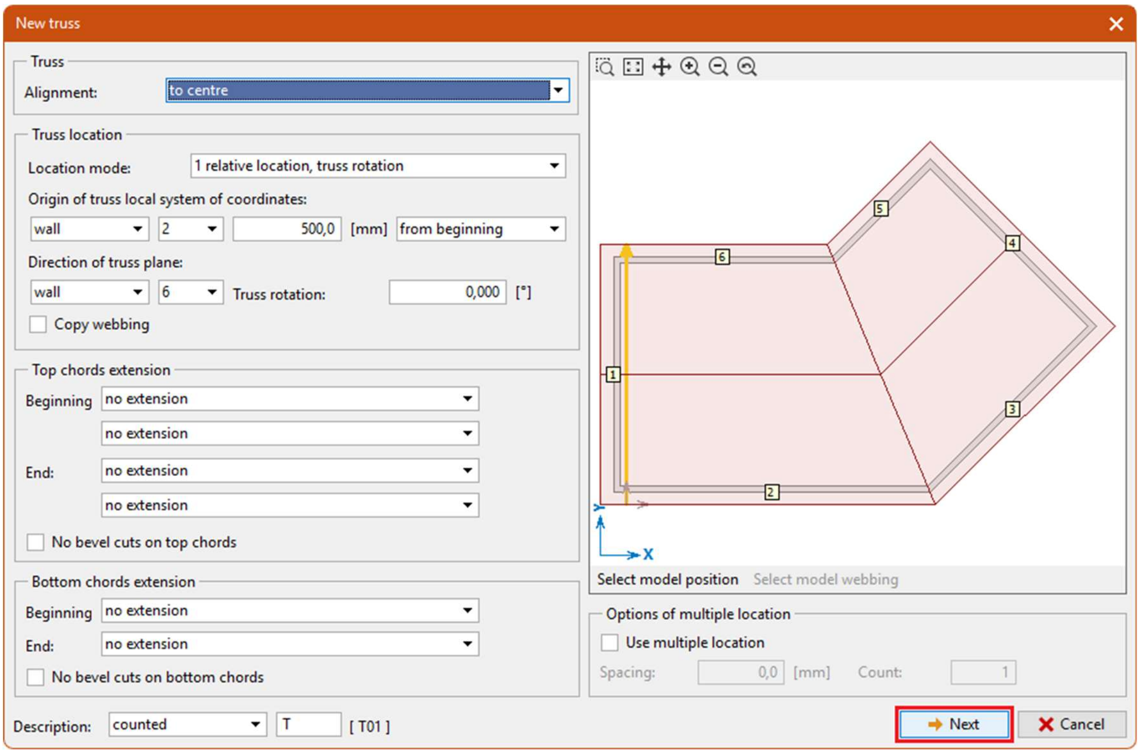
Input of truss beginning

Enter the end of the truss at the intersection of the construction line and the eave line of the wall *6*.



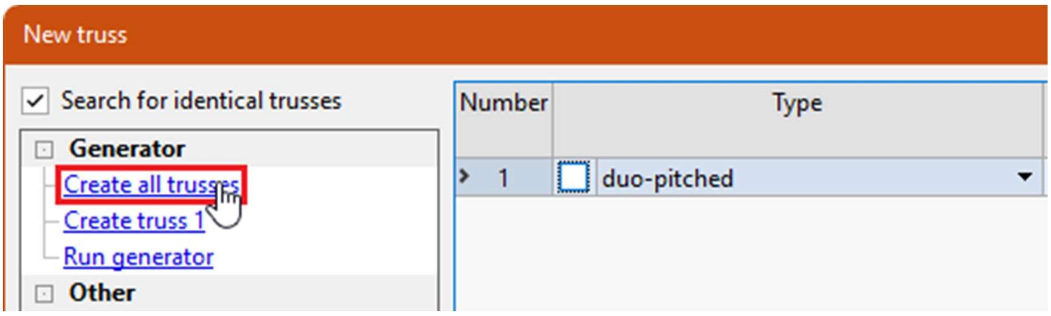
Input of truss end

The "**New truss**" window is displayed after entering the beginning and end of the truss. In the first part it is possible to modify the position of the truss in the structure. In this case it is not necessary to change any data and it is possible to go to the second part of the window using the "**Next**" button.



"New truss" window

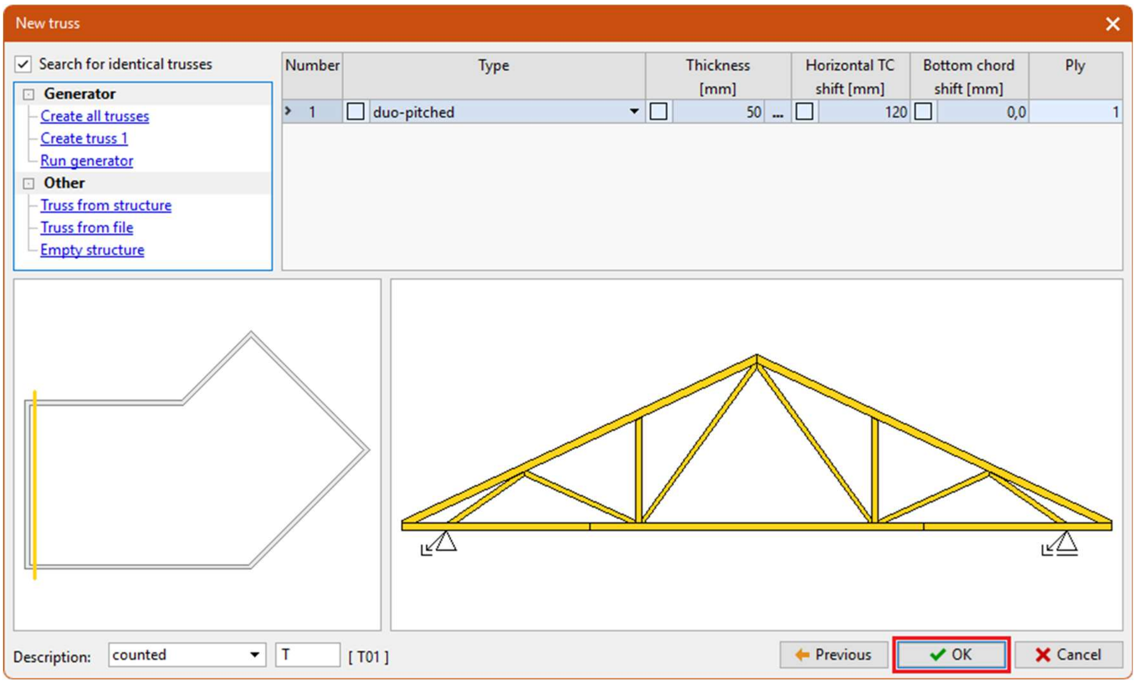
The second part of the window is used to create the geometry of the truss. Use the "**Create all trusses**" tool in the upper left corner of the window.



"Create all trusses" function

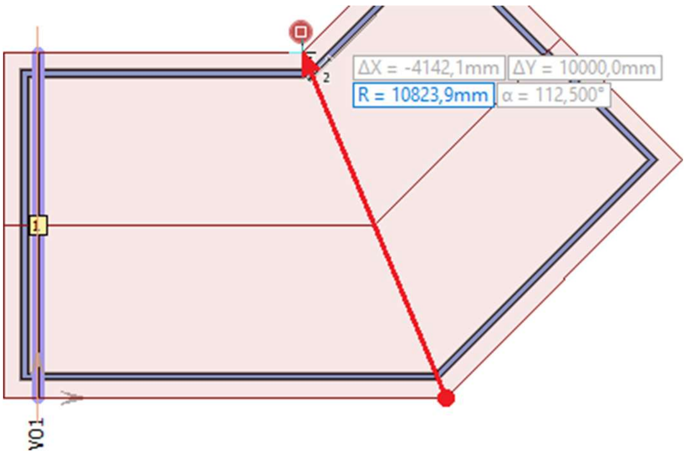
The program creates the truss shape and designs the configuration of infill members. The resulting truss is displayed in the right part of the window. The truss is inserted into the structure using the "**OK**" button.

TRUSS4



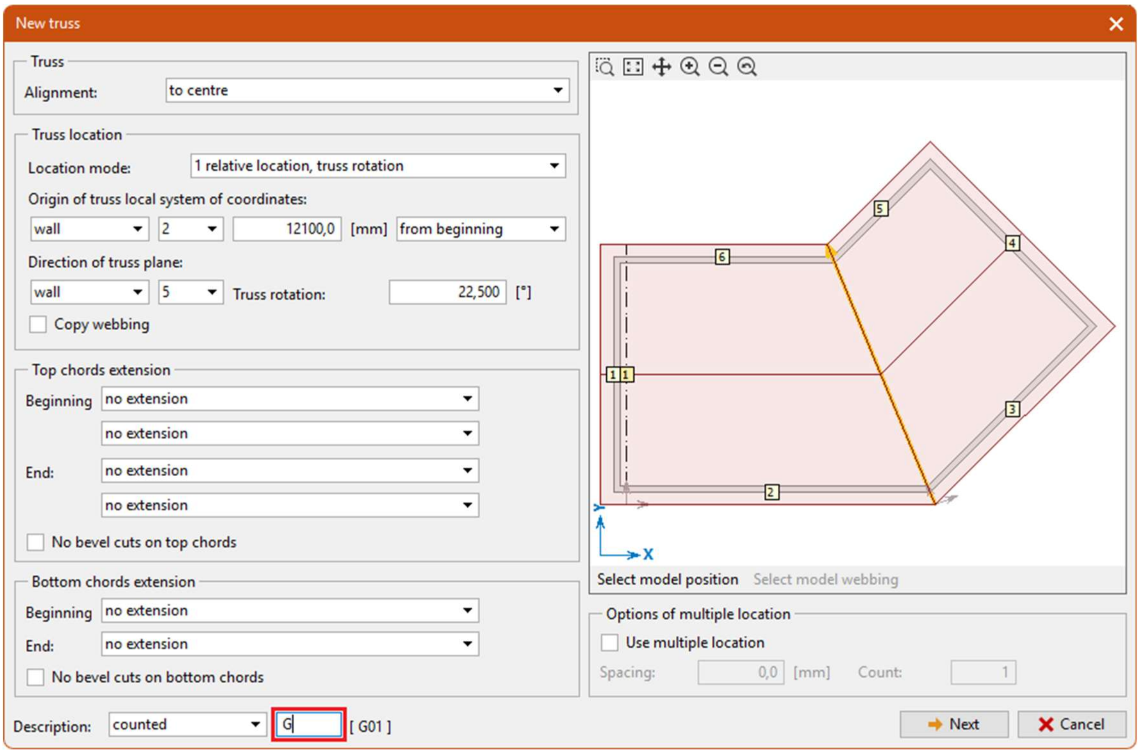
Created truss

After inserting the truss into the structure, the program remains in the truss entry mode. Insert the second truss into the roof. This time, it will be a girder running under the valley. Enter the beginning of the truss at the intersection of the eaves lines of the walls *2* and *3*, and the end at the intersection of the eaves lines of the walls *5* and *6*.



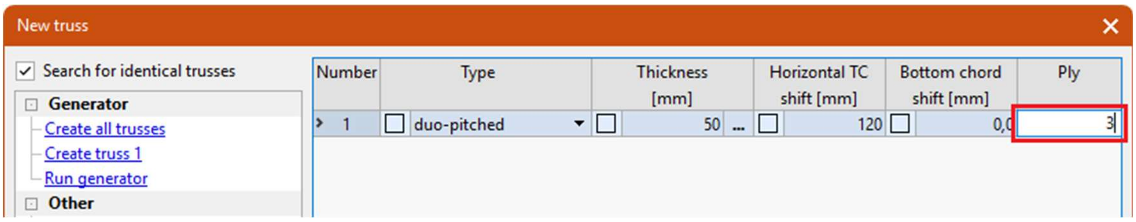
Input of the girder

In the first part of the "**New truss**" window, the position is again loaded from the graphic input. This truss will be the main load-bearing truss in the structure, so we will clearly distinguish its name for production and assembly purposes. In the left bottom part of the window, change the prefix before the number to "**G**". This modification will rename the truss to "**G01**". Then we can continue with the "**Next**" button to the second part of the window.



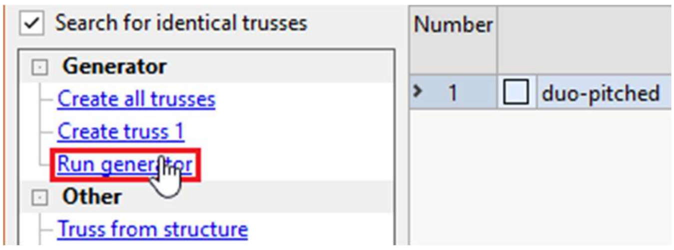
Changing the truss name

Since the girder will support more trusses, it will be defined as 3-ply. We also select a webbing that will have short bay lengths on the bottom chord. We will enter the value "**3**" in the "Ply" column of the table on the right side of the window.



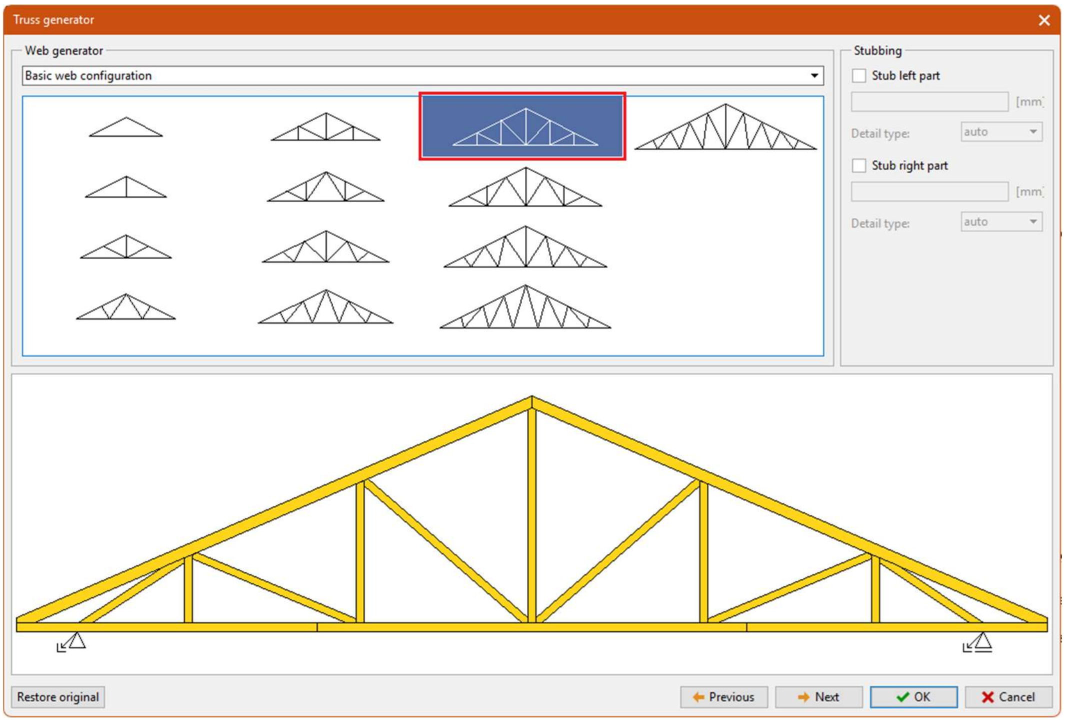
Changing the ply option

Then we can create the truss. Since we want to select the more suitable web configuration, we will use the "**Run Generator**" tool. This gives us the possibility to affect the truss generation.



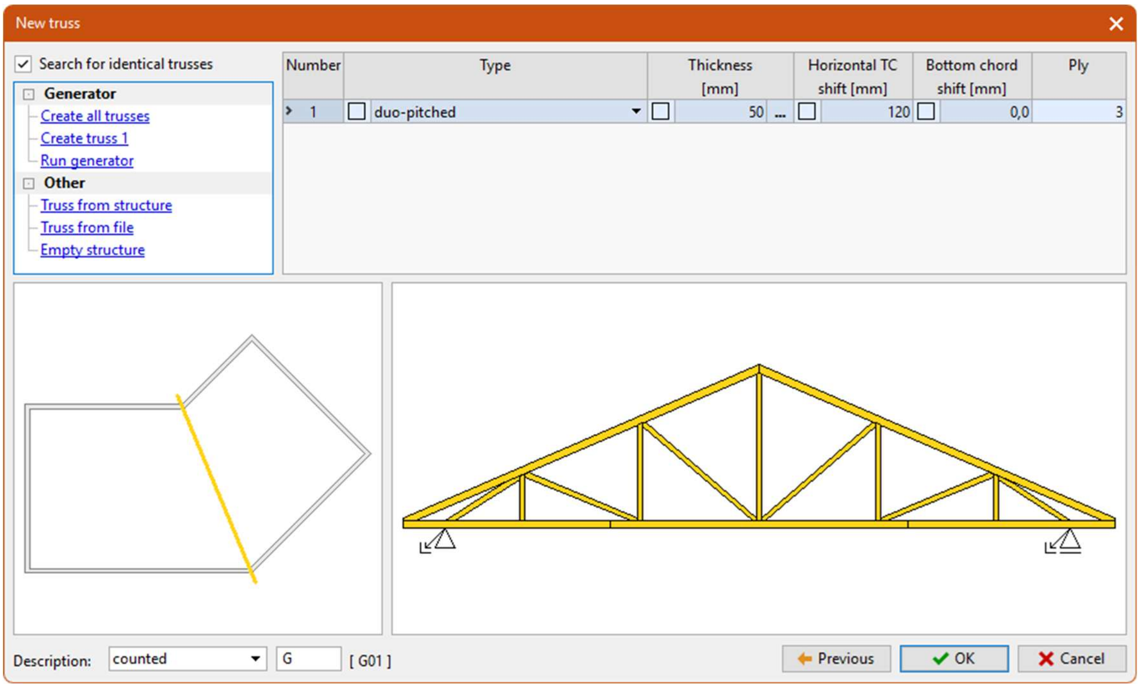
"Run generator" tool

The "**Truss Generator**" window acts as a tool for easy editing of the generated truss. In individual steps it is possible to affect the geometry of the truss, the configuration of infill members and the positioning or orientation of the individual webs. In our case, it is sufficient to change only the web configuration. In the window, we select a configuration with five verticals and four diagonals. Close the window with the "**OK**" button.



Changing the web configuration

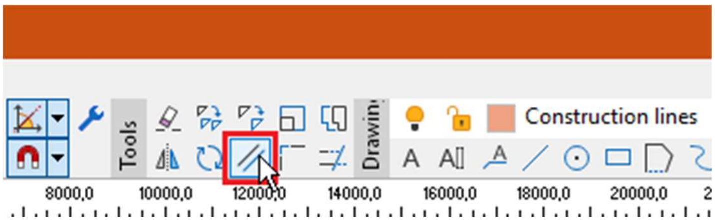
The truss has been created, so we can insert it into the roof by clicking "**OK**".



Created girder

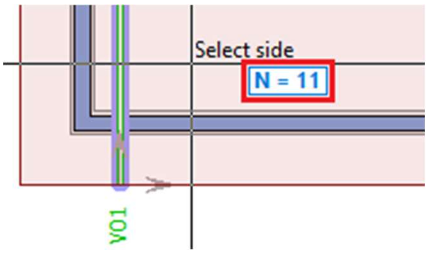
We have inserted the basic trusses and now we will finish the roof structure using the manipulation tools. Since we no longer need the construction line, we can select and delete it, for example by using the "**Del**" key.

We will fill the left part of the roof with copies of truss *T01*. To create them, we use the "**Equidistant**" command, which is used to copy line objects multiple times with the specified axis distance.



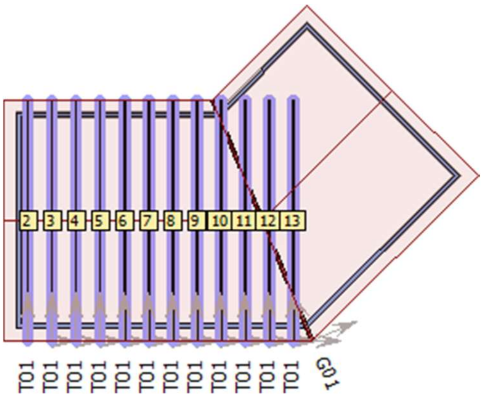
"Equidistant" tool

After selecting this command, we need to specify the distance that will be used for making copies. In our case, we will enter a value of *1000*, i.e. the planned axial distance between the trusses. Confirm the input with the "**Enter**" key. Then we must select the object to be copied. In our case it will be truss *T01*. Then we have the possibility to enter the number of copies on the keyboard. In our case we will enter the value *11*, which will be displayed at the cursor.



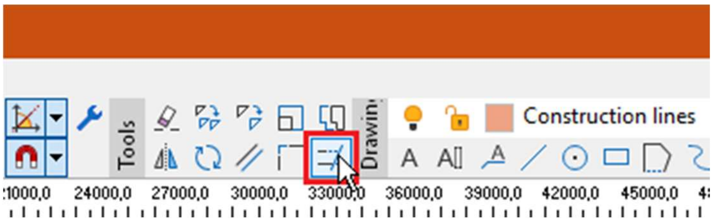
Specified number of copies

Finally, just specify the semi-plane where the copies should be created. Click anywhere on the workspace to the right of the *T01* truss. Eleven copies of the truss will then be created to the right of the original position, with an axial distance of *1000mm*.



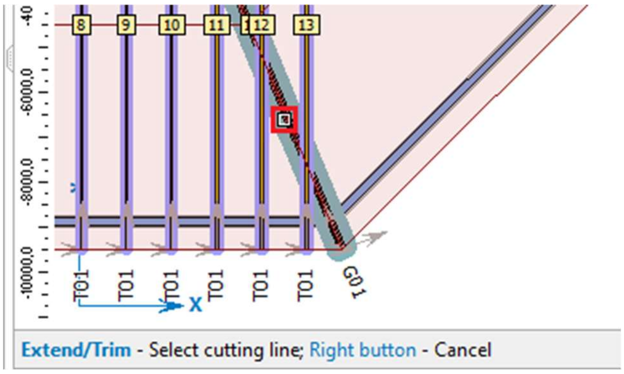
Structure with new trusses

The trusses fill the left side of the building, but partially intersect with the girder *G01*. The next step is to trim these trusses. Use the "**Extend/Trim**" tool to do this.



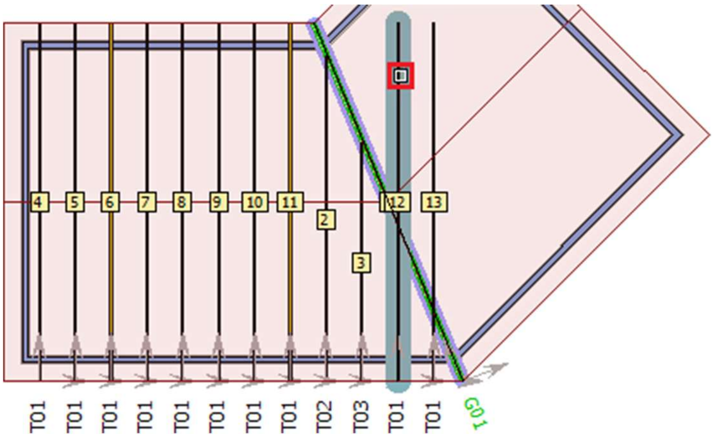
"Extend/Trim" tool

In the first step, we need to select a trimming object. In our case it will be the truss *G01*.



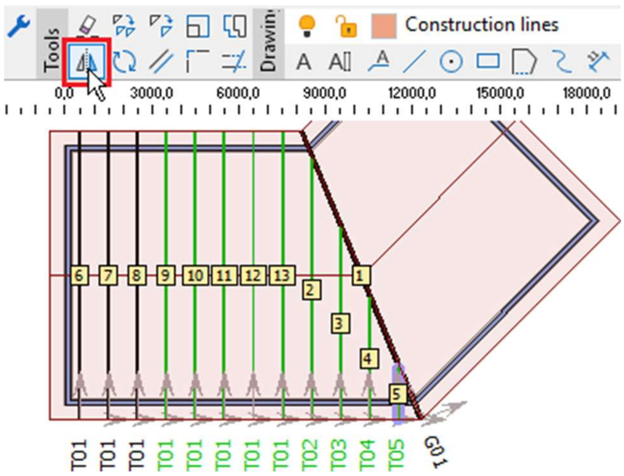
Selection of trimming object

Then just click on the ends of the *T01* trusses that should be cut off. The trusses will be trimmed one by one against the girder *G01*.



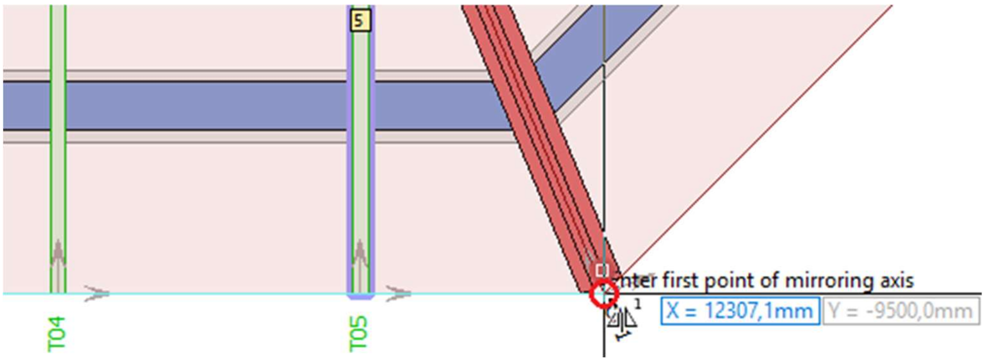
Trimming trusses

The selection of trusses to be trimmed can also be done by crossing them out. In this case, however, there would be a risk that we would insert also a wall into the selection, which would complicate our work. When trimming step by step, we also have control over the selection order, and therefore over the resulting numbering of trusses. After trimming all the trusses, we will need to fill the right part of the roof with identical trusses. The easiest way to do this is by mirroring the selected objects. Select all stubbed trusses and also the five *T01* trusses and then choose the "**Mirror**" command.



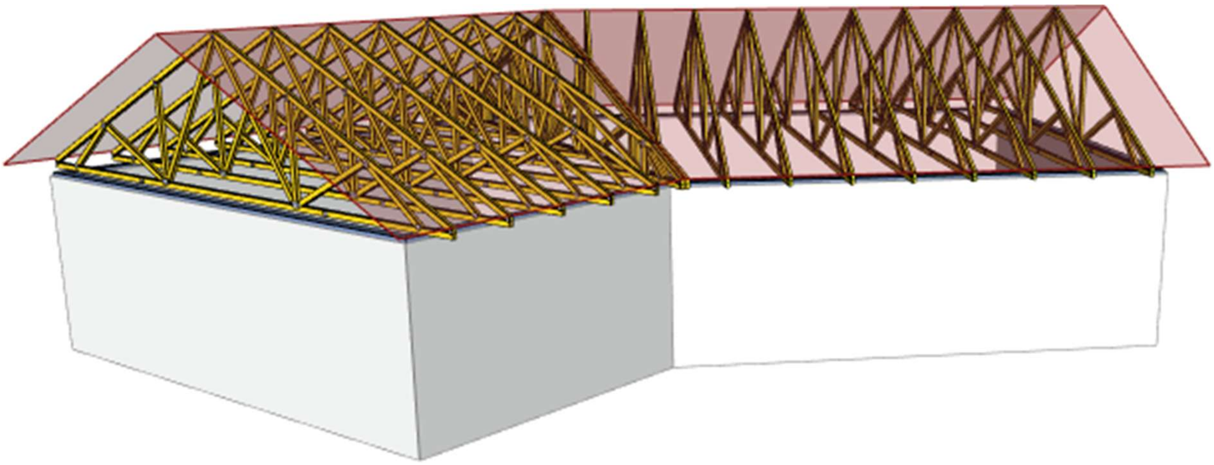
Selecting the "Mirror" tool

Since we have already selected the objects to be mirrored before pressing the "**Mirror**" button, the tool guides us to enter two points defining the axis of mirroring. We select the intersections of eaves lines on both sides. In this operation, we must be precise and zoom in sufficiently on the structure. There are many grip points (for example on the girder) which could cause inaccurate mirroring.



Specifying the beginning of the mirroring axis

After specifying the second point of the mirroring axis, the program prompts whether to perform the mirroring in copy or move mode. Press the "**Yes**" button to confirm the copy mode. The trusses will then fill the remaining part of the roof structure.

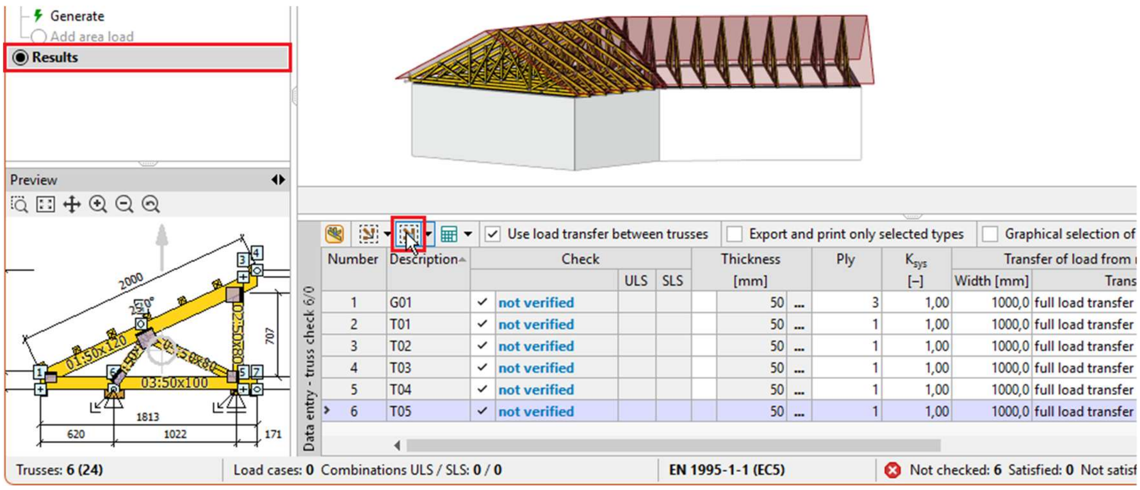


The resulting structure

The structure is now complete. The last step is the automatic design of the structure.

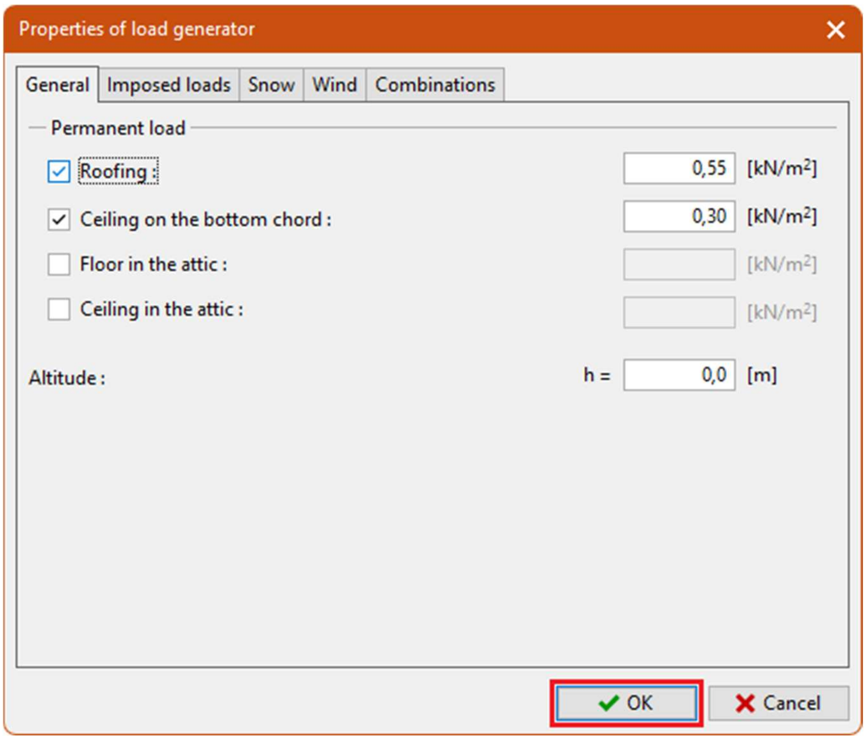
Automatic design

In the tree menu, go to the "**Results**" section and start the automatic design. This can be started, for example, with the "**F8**" key or with the "" button in the toolbar above the truss table.



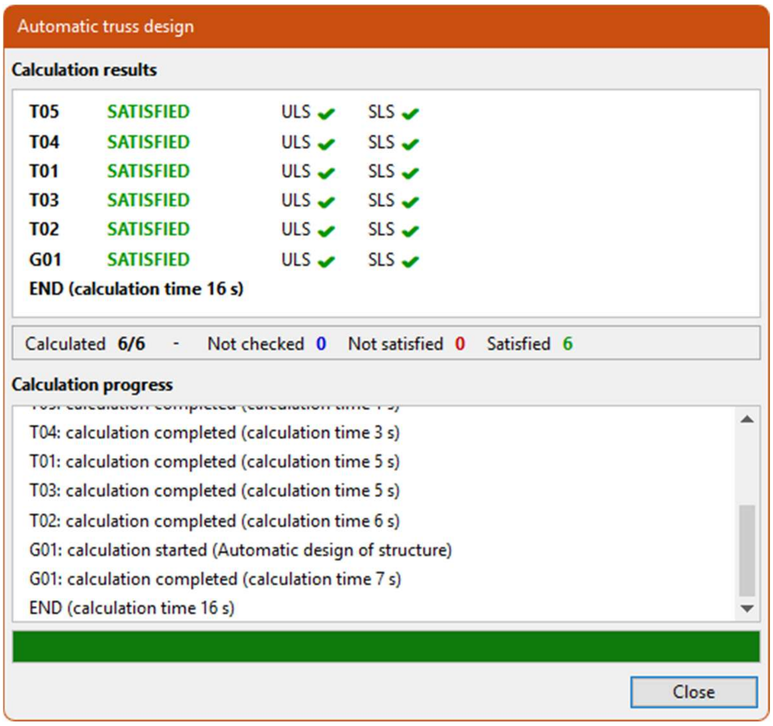
Running the automatic design

Before the first analysis it is necessary to enter the load parameters. Therefore, the "**Properties of load generator**" window is displayed first. For this project, just confirm the default values and close the window with the "**OK**" button.



Load generator

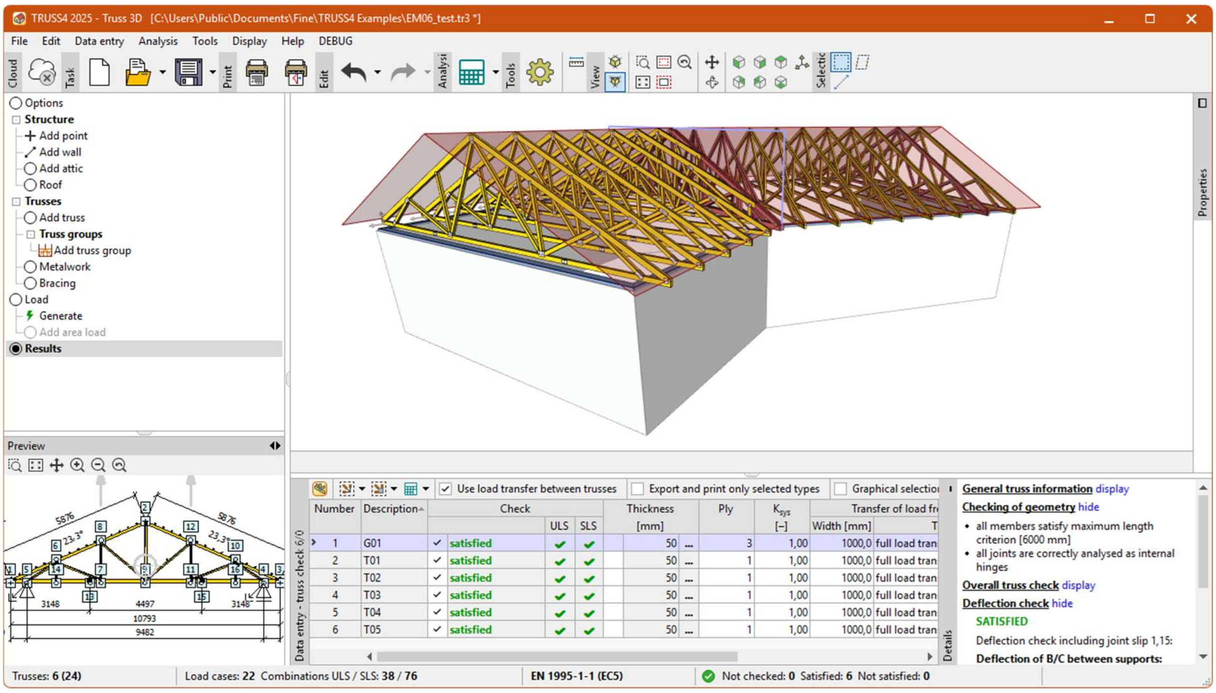
After confirming the loading data, a window that shows the progress of the calculation is launched. When the calculation is complete, it can be closed by clicking the "**Close**" button.



Analysis report

TRUSS4

This completes the design of the roof structure.



Finished structure

The created structure is stored in the file "**DEMO_EM06.tr3**" in the "**Fine online examples**" folder.

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