Allplan
Steel design modeller
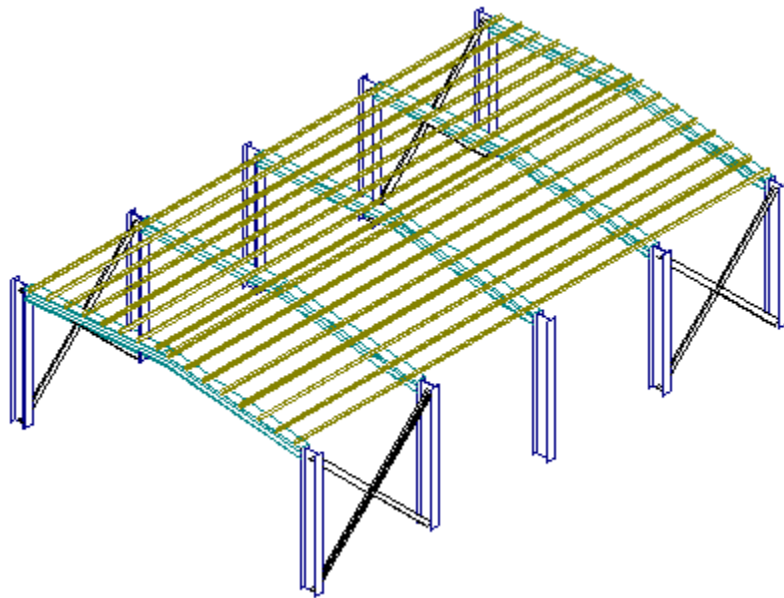
Table of contents

Welcome to Steel design modeller

Steel design modeller is a software tool that enables to model steel structures in Allplan environment combining functions of an external program with internal features of Allplan. The inserted parts of the structure can be further modified through both Allpan functions and special functions of the external module.

Utilisation of this specialised module makes modelling of steel structures including welded or bolted connections straightforward and fast.

Sample project

This book will guide you through the few steps that are necessary to create a 3D model of a simple steel hall.

The whole process of the creation of the model will be divided – for the purposes of this tutorial – into several stages:

1. explanation of template philosophy

2. input of one transverse section of the hall

3. copying of this planar frame into 3D space in order to define other "sections"

4. connection of individual planar frames in longitudinal direction

5. definition of bracing elements

6. explanation of principles of editing

At the end the structure should look like:
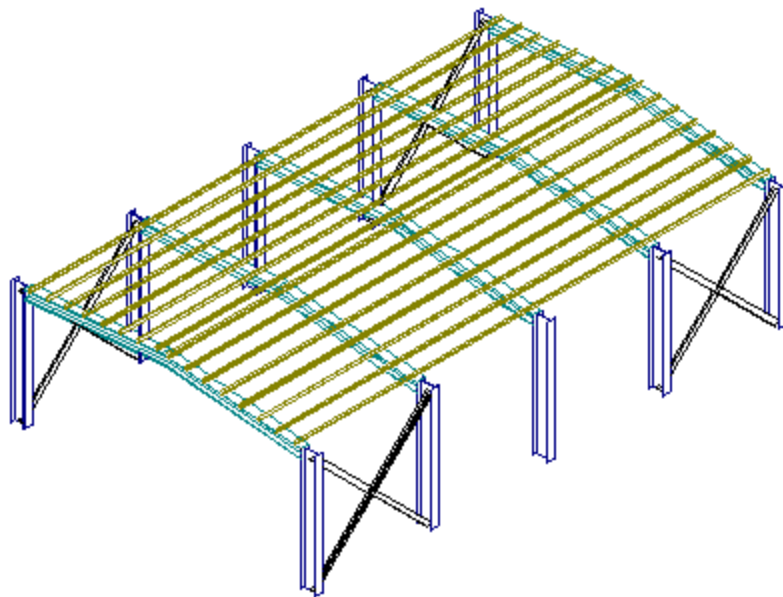
Accessing the modeller functions

Functions of **Steel design modeller** are integrated into the standard Allplan Environment. You can find them in **Add-on Module** tree.

If you select tree item **Add-on Module > SCIA Stahlbau**, modeller toolbars become accessible.

|	
Vorlage import	Imports a structure template and inputs it into the project.
Knotenverbindung	Imports a template of steel frame connection and inputs it into the project.

SCIA ESA PT Modellierer	Starts SCIA·ESA PT modeller.
SCIA Homepage	Opens the SCIA website in your browser.

Eigenschaften modifizieren	Edits the properties of a single member of the model.
Vorlage modifizieren	Edits the properties of a whole template from the model.
Bauteil schneiden	Cuts members.
Bauteildarstellung modifizieren	Edits view parameters of selected member/s.
Modifikation in ESA PT	Opens SCIA·ESA PT Modeller and takes the selected member/members into it for direct editing in SCIA·ESA PT Modeller.
Aktualisieren	Regenerates the data.
Bauteile zusammenfassen	Groups selected members together.
Bauteile auflösen	Breaks the group into original elements.

One of powerful features of **Steel design modeller** is that it enables you to model quite easily in 3D view (not just in plan view). In order to demonstrate this capability, you will create the whole sample project in 3D view.
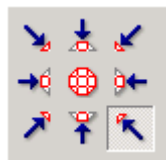
A template

Steel design modeller is based on what is called "a template".

A template is a "small, typical structure" or a "typical part of a structure". A simple planar frame is one of the predefined templates.

Note: In fact, a template may be even a large structure, but in general it is mainly used to represent a smaller part of a structure (e.g. a beam, a simple frame, a typical corner, etc.) that is used repeatedly in your design practice.

Example

It may be convenient to present a couple of examples of such templates, one simple and the other one more complex.

The first example is a simple planar frame with tapering "roof" beam. Such a structure can be input as a composition of four separate beams. On the other hand, if there are no abnormalities in the frame (such as absence of symmetry, different columns, etc.) and if a template for such a frame has been prepared in advance, it is possible to type just 3 numbers – i.e. define the dimensions of the structure – and select two cross-section types for columns and beams.

Name	BL
Geometry	
L [m]	10,000
H1 [m]	5,000
H2 [m]	1,500
Column	CS1 - IPE450V
Beam	CS2 - IPE160
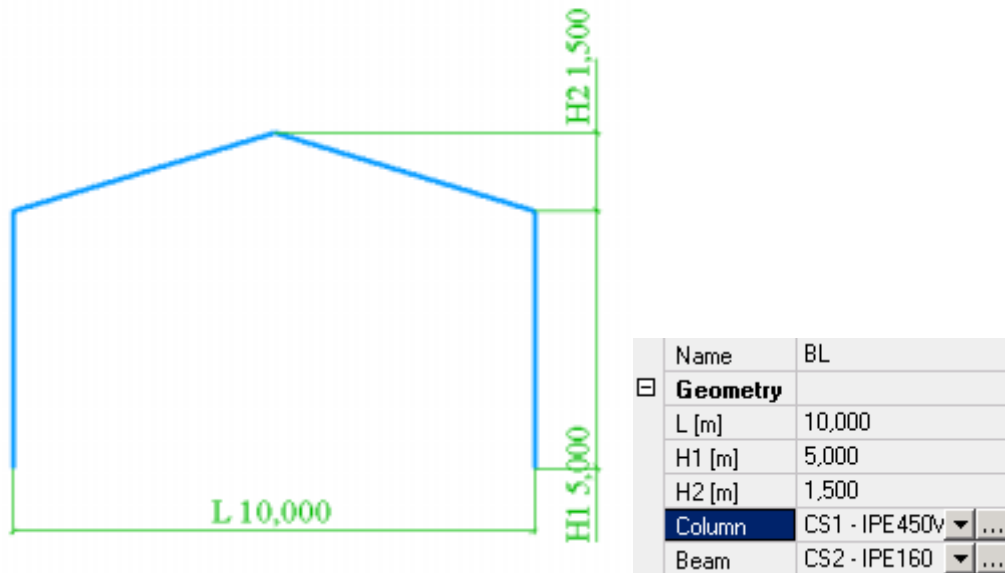
The second example is a bit more complex three-dimensional curved truss girder. In this case, the step-by-step (or we can say beam-by-beam) definition would represent the input of 75 individual beams. When compared with the necessity to input 10 parameters in the template input dialogue, the advantage of template-oriented-approach can be clearly seen.

Name	BL
Geometry	
n	4
m	4
ru [m]	10,000
rd [m]	8,500
fi [deg]	90,00
b [m]	1,000
Bb	CS2 - IPE160
Bu	CS2 - IPE160
D	CS2 - IPE160
V	CS2 - IPE160

- side view

- plan view

End of the example

A set of common templates is delivered with the program. In addition, you can create other templates in SCIA·ESA PT program. However, the creation of a new template goes beyond the scope of this book. If you are interested in this feature, please contact your local technical-support engineer.

The templates that are integrated in the program are divided into several groups, e.g. grid, beams, 2D structures, plates, etc.

Input of auxiliary 3D line grid

The modelling can be facilitated through a line grid. A line grid is a kind of a three dimensional grid. Individual vertices of the grid can be used to define points of the modelled structure.

One can imagine the line grid as a set of wire cubes placed one next to another to create a larger wire cube. The vertices of individual small wire cubes are the vertices of the line grid. What's more, the cubes may be not only regular cubes, but also other solids like a tetrahedron, irregular hexahedron, etc. The grid may be of either regular or irregular (variable) dimensions in any direction.
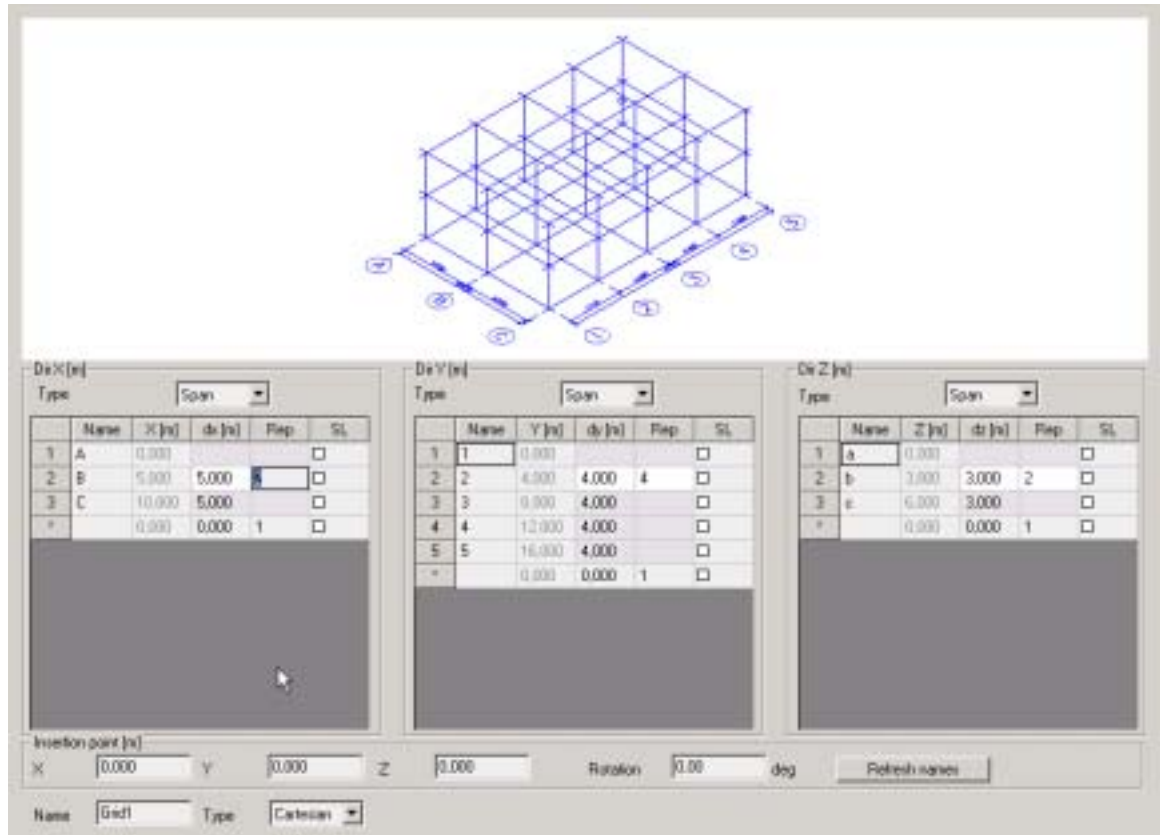
The tool is extremely useful for the definition of complex 3D structures on condition that at least some parts of the structure are regular (i.e. of the same spans or of the same height).
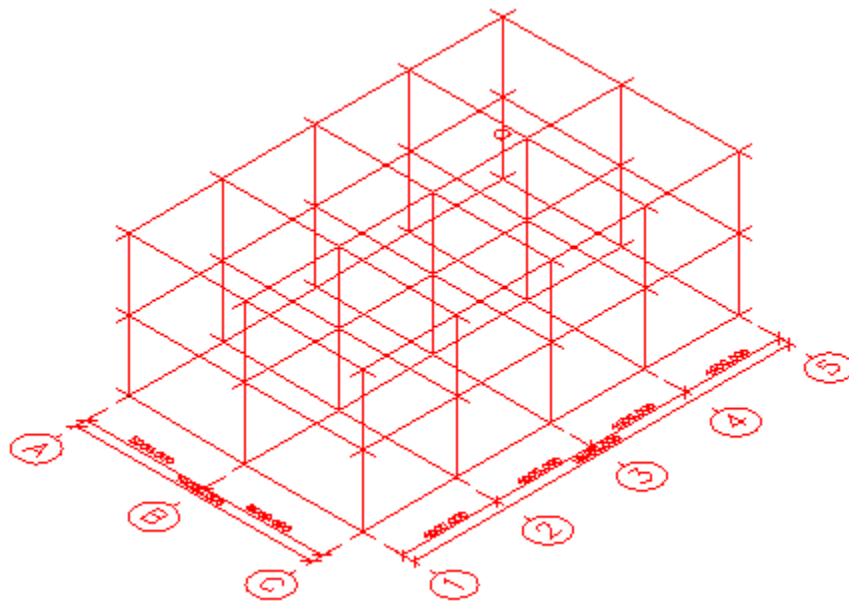
☑ Procedure

1. Call function **Vorlage import**.

2. In the **Template** dialogue, select group **grid**.
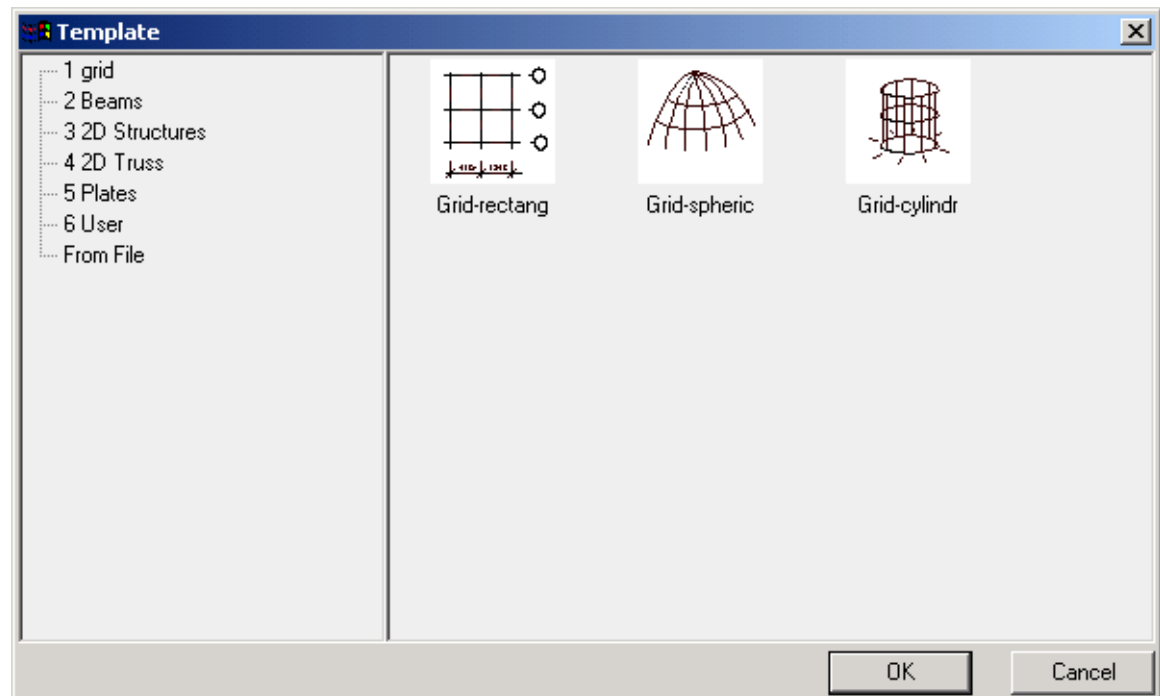
3. Select the rectangular grid (**grid-rectang**) and confirm with **[OK]**.

4. The **Line grid** dialogue is opened on the screen. Fill it in according to the picture:

    a. in X-direction set the dimension to x = 5 m and define 2 repetitions

    b. in Y-direction se the dimension to y = 4 m and define 4 repetitions

    c. in Z-direction se the dimension to z = 3 m and define 2 repetitions

	Name	X [m]	dx [m]	Rep	SL
1	A	0.000			☐
2	B	5.000	5.000	5	☐
3	C	10.000	5.000		☐
*		0.000	0.000	1	☐

	Name	Y [m]	dy [m]	Rep	SL
1	1	0.000			☐
2	2	4.000	4.000	4	☐
3	3	8.000	4.000		☐
4	4	12.000	4.000		☐
5	5	16.000	4.000		☐
*		0.000	0.000	1	☐

	Name	Z [m]	dz [m]	Rep	SL
1	a	0.000			☐
2	b	3.000	3.000	2	☐
3	c	6.000	3.000		☐
*		0.000	0.000	1	☐

Insertion point [m]

X 0.000   Y 0.000   Z 0.000   Rotation 0.00 deg   Refresh names

Name Grid1   Type Cartesian ▼

5. Confirm with **[OK]**.

6. Insert the grid to point (0, 0, 0).

7. From now on the vertices of the grid can be used to input other elements of the model.

Input of a planar frame - i.e. drawing a section through the hall

In the first step of our modelling you will create a planar frame that represents a transverse section through the hall.

☑ Procedure

1. First, create a new project in Allplan.

2. Start menu function: **Vorlage import**.

3. The **Template selection dialogue** appears on the screen.

4. In the left hand side of the dialogue select the template group: **2D Structures**.

5. In the right hand side of the dialogue choose template: **frame 05**.

6. Confirm with **[OK]**.

7. The Import parameters dialogue opens on the screen

8. Say that you want **Colours from ESA**.

9. Confirm the rest of default options with **[OK]**.

10. The dialogue with template properties is opened on the screen.
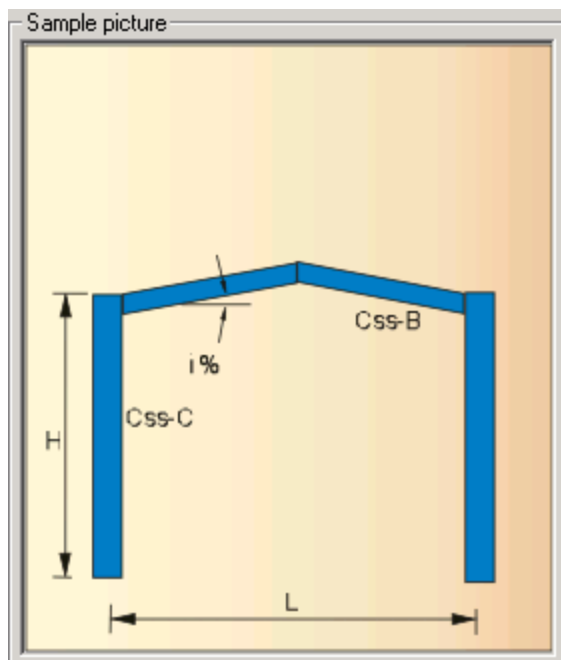
11. Specify the following parameters and confirm with **[OK]**:

    a. L = 10

    b. H = 4

    c. Css-B = HEA320

   d. Css-C = HEA500

   e. i = 10

12. Position the template into the line grid.
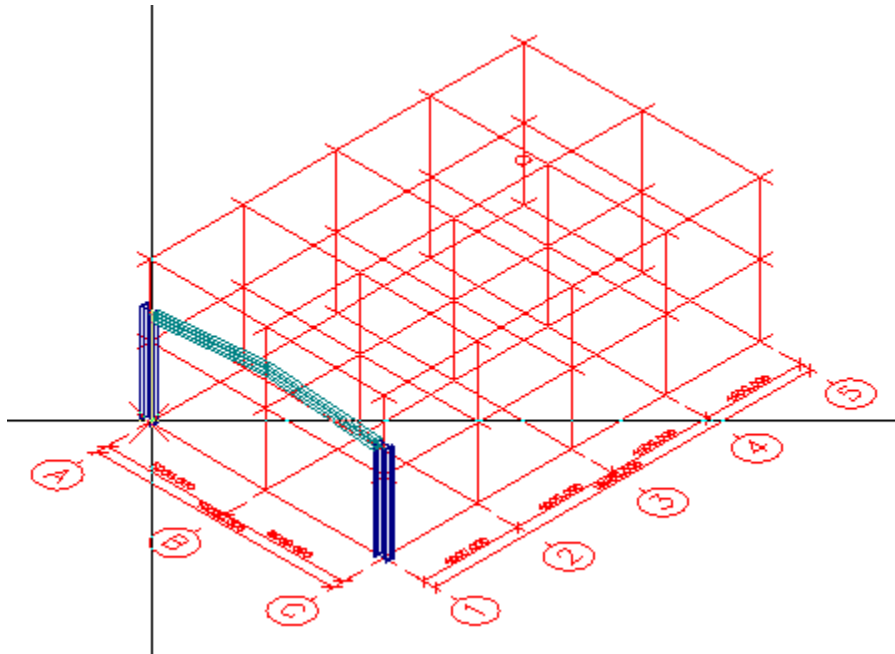
When hidden lines are not drawn, the structure (the already modelled part of it) may look like:

**13**
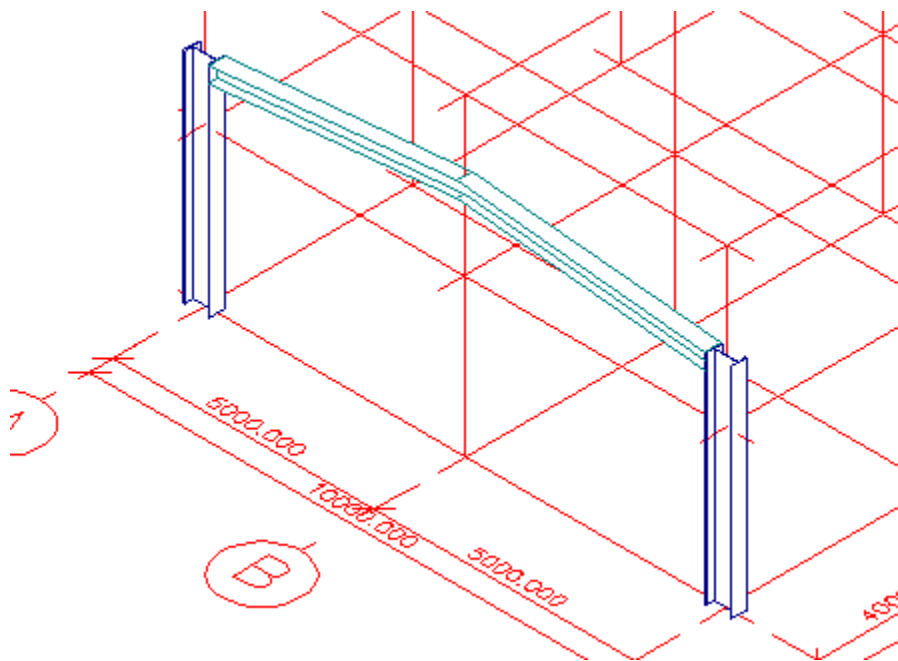
Creating the 3D structure

The intention of this sample project is to create a model of a rather simple steel hall. In the previous step, you have inserted a template of a planar frame that represents a section through that hall.

In this step, you will add more planar frames and thus create the skeleton of the hall.

In general, it is possible to use the way for the input of other "sections" as was used in the first step. In will be however more efficient to exploit another feature – copying.

☑ Procedure

1. Call standard Allpan function **Kopieren und einfügen**.

2. Start multi-select mode.

3. Select all the beams of the frame.
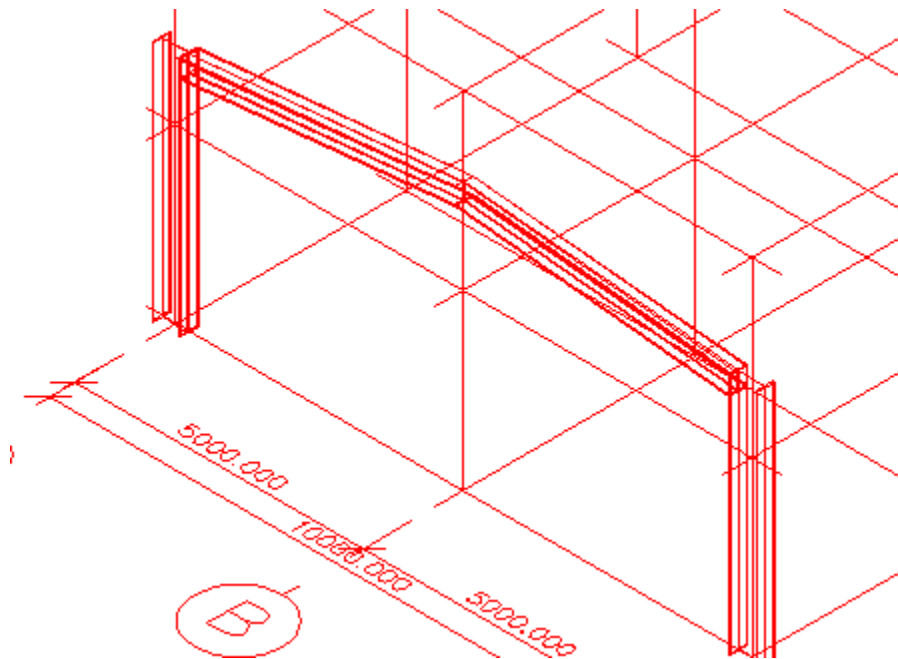
4. End the multi-select mode to start the function itself.

5. Pick the from-point, i.e. the foot of the column – it is convenient to pick the foot that is closer to you.
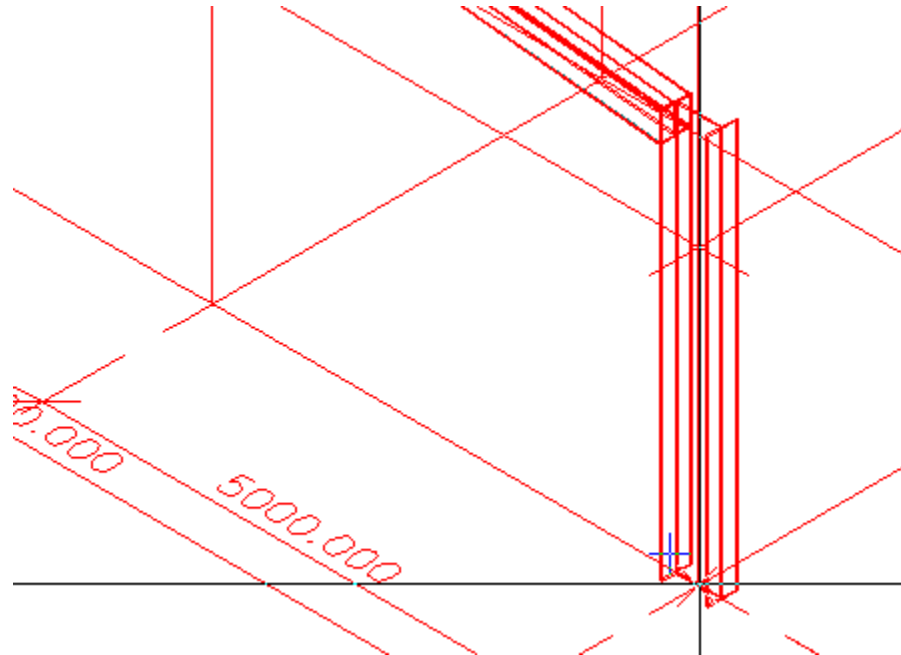
6.   Pick the destination point – use the line grid to define the destination point.

7.   Repeat this step five times to make the whole hall.
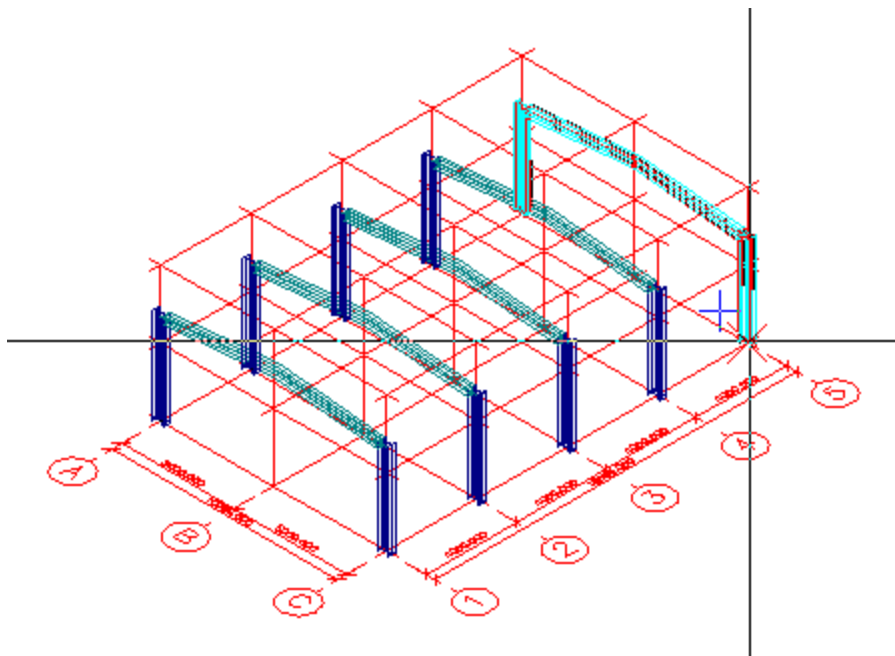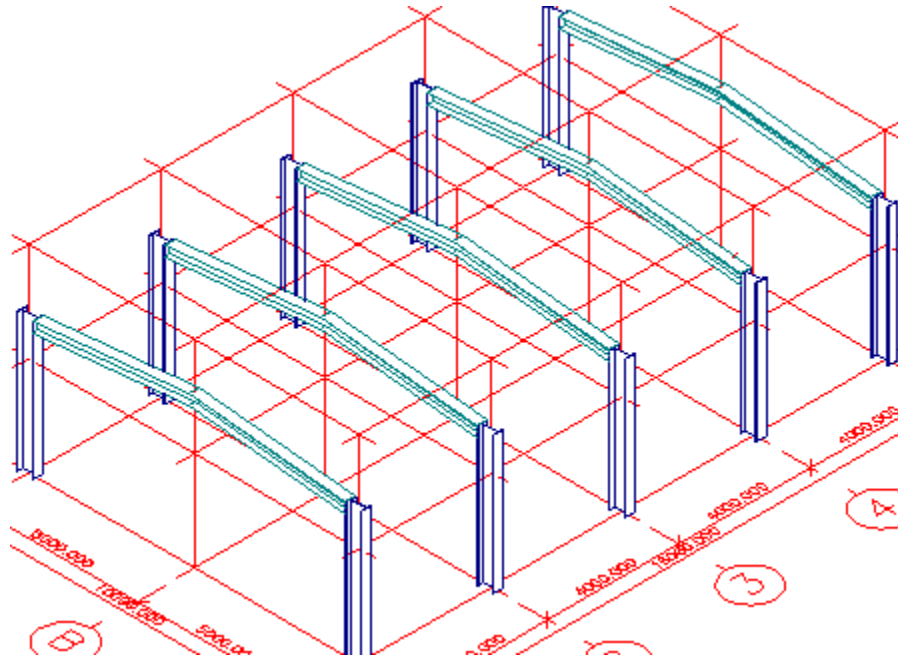
Modelling the roof structure

Also roof beams will be input by means of a template. This time it will not be a simple template, but an "intelligent one". The word "intelligent" means that the template is able to calculate some information from graphically provided input data.

You won't have to type all the dimensions of the roof. You will pick them on the screen by selecting the appropriate corners of the already defined hall.

☑ Procedure

1. Once again, start function **Vorlage import**.

2. Select group **2D structures**.

3. Select template **Array of beams by 3 points**.


Array of beams by 3points
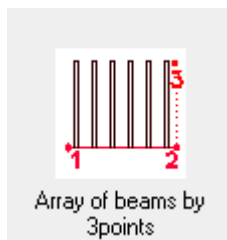
4. Say that you want **Colours from ESA**.

5. Confirm the rest of default options with **[OK]**.
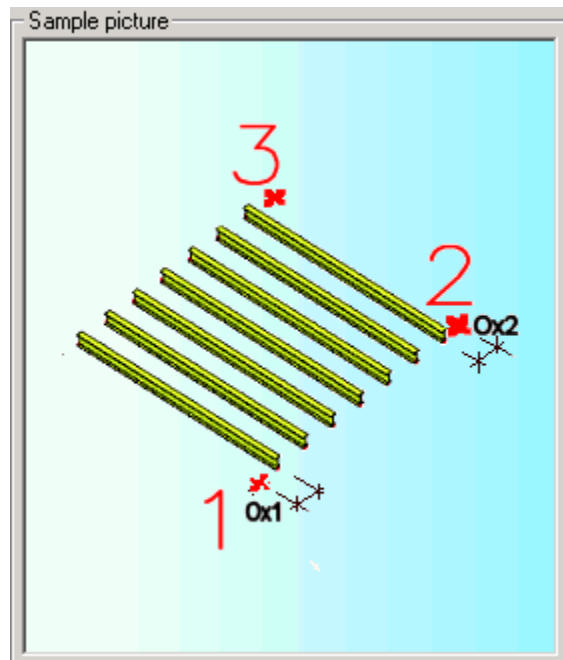
6. Select cross-section IPE 100 and you may let other input values as they are and confirm the input dialogue with **[OK]**.

7. And now comes the most important step. You must correctly input the three points that define the template location and size. The following picture shows, where the points are located in your model. The subsequent pictures then inform you where exactly you should put the cursor and click. Be careful with the definition of the input points. Zoom-in and zoom-out are very useful functions for this task.
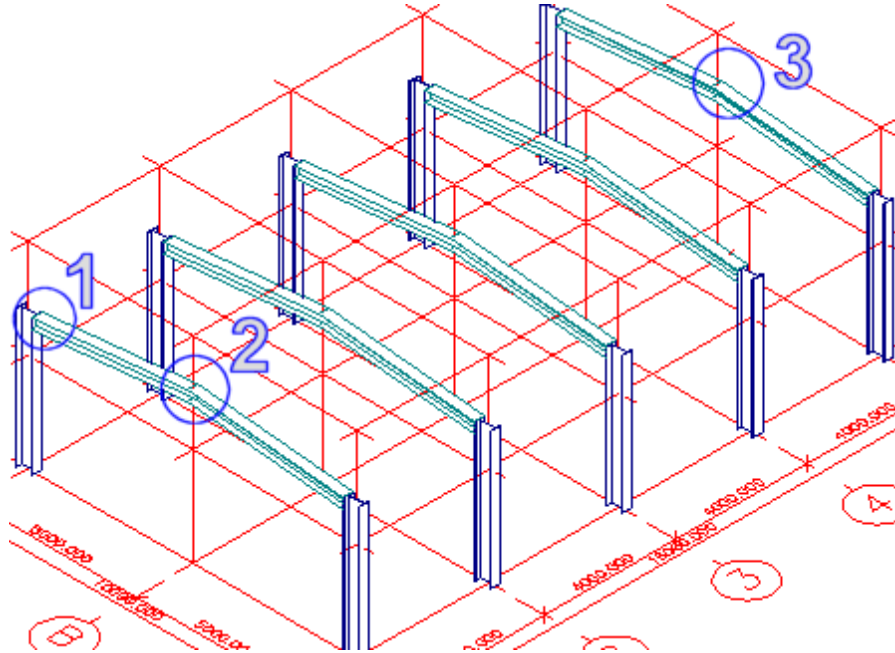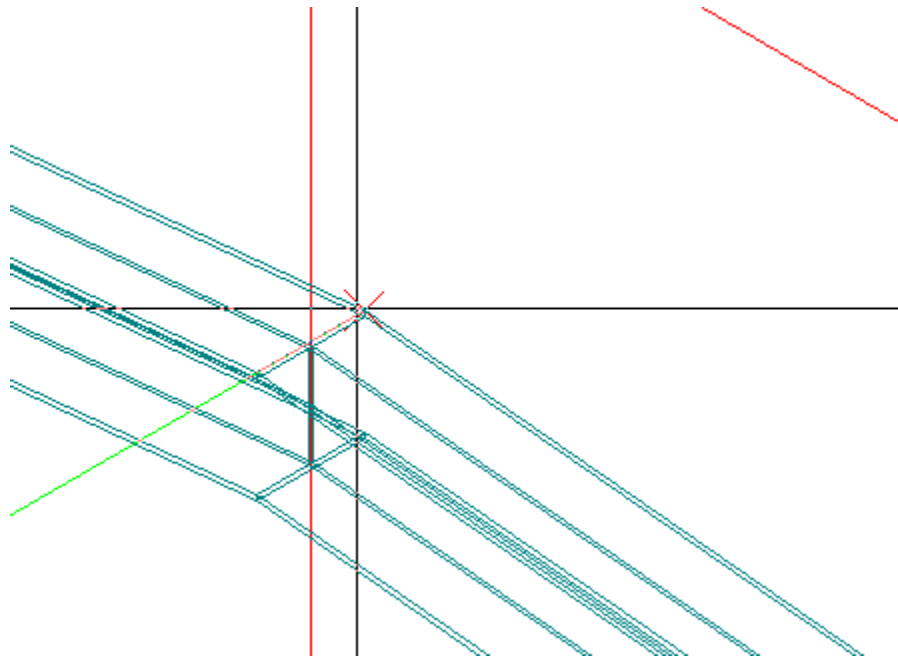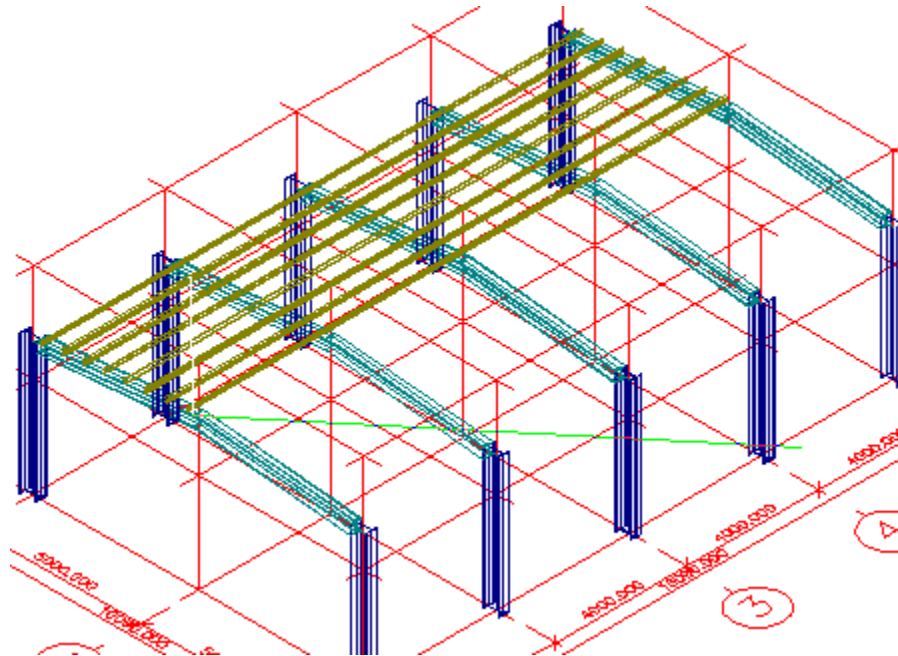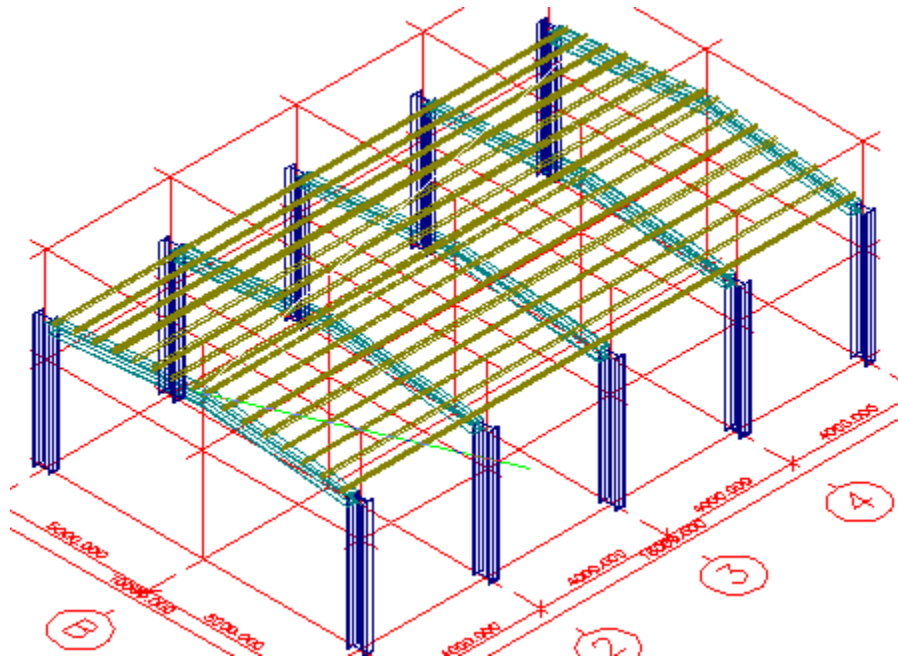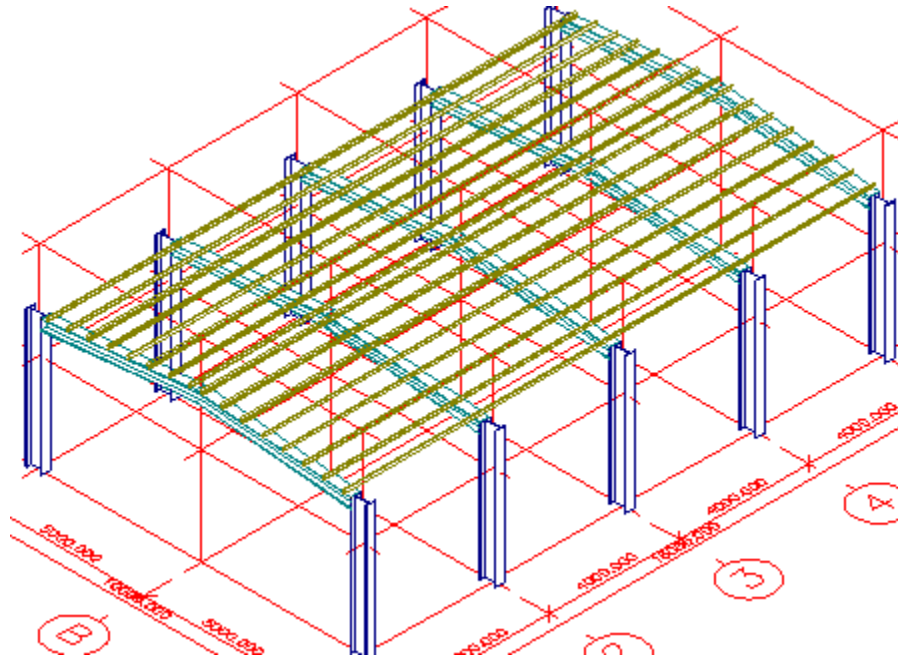

POINT 1:
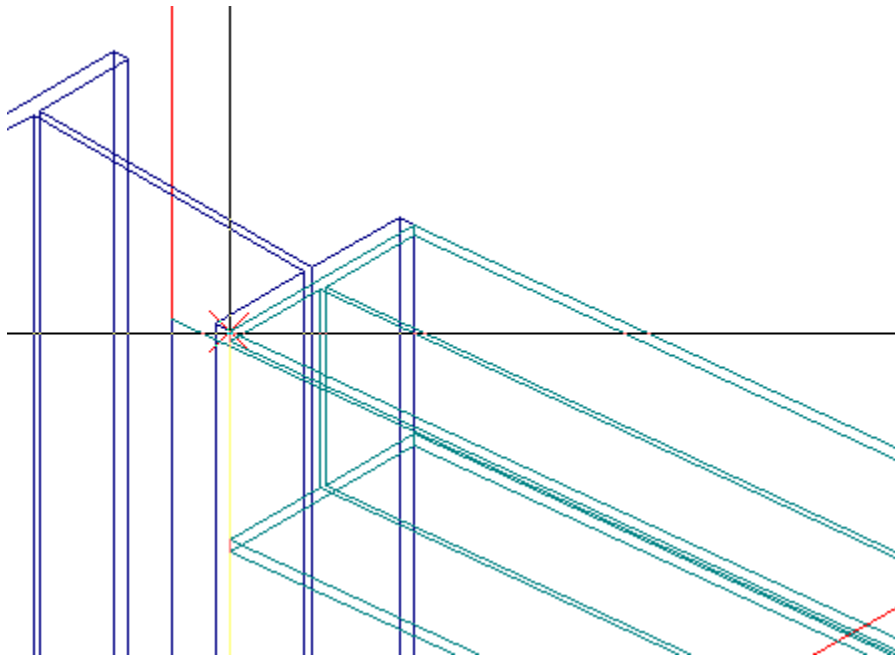
POINT 2:
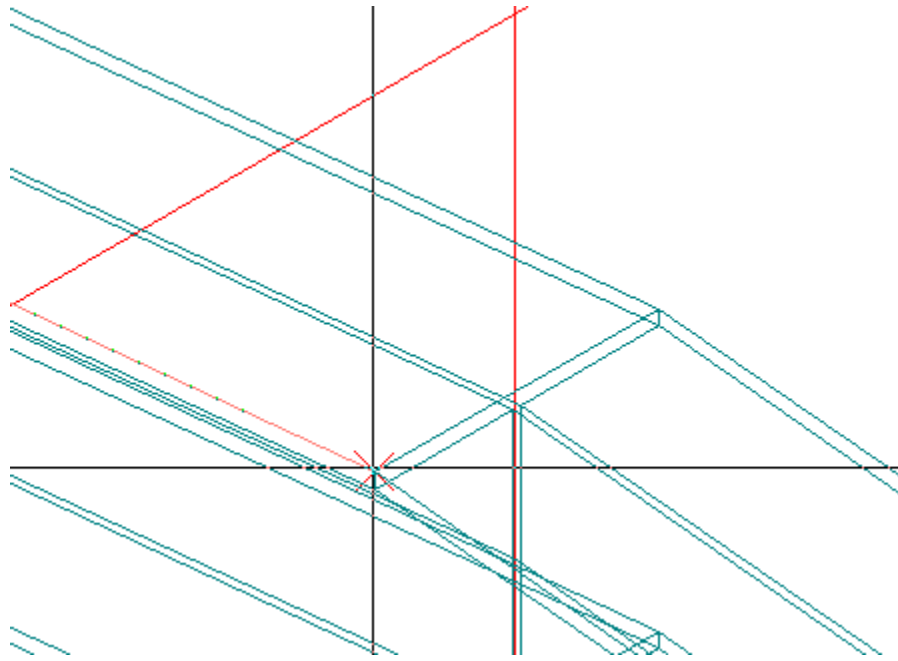
POINT 3:

8. The result is:

9. Use the same procedure to input the roof structure above the other half of the hall. Be careful with the definition of the input points. Once again, zoom-in and zoom-out are very useful functions for this task. You will obtain the following result.

Modelling the bracing

The hall as it is at the moment would be unstable when subject to a longitudinal impulse. Some kind of bracing is therefore necessary. There are numerous ways to do it in practice and there are several ways to input it in Steel design modeller. We will exploit one more template to input a simple X-like (cross) bracing. It will be again a "dynamic" template as in the case of roof – i.e. the input template beam will be stretched to fit into our model.

☑ Procedure

1. Once again, start function **Vorlage import**.

2. Select group **Beams**.

3. Select template **2pt-beam**.

4. Say that you want **Colours from ESA**.

5. Confirm the rest of default options with **[OK]**.
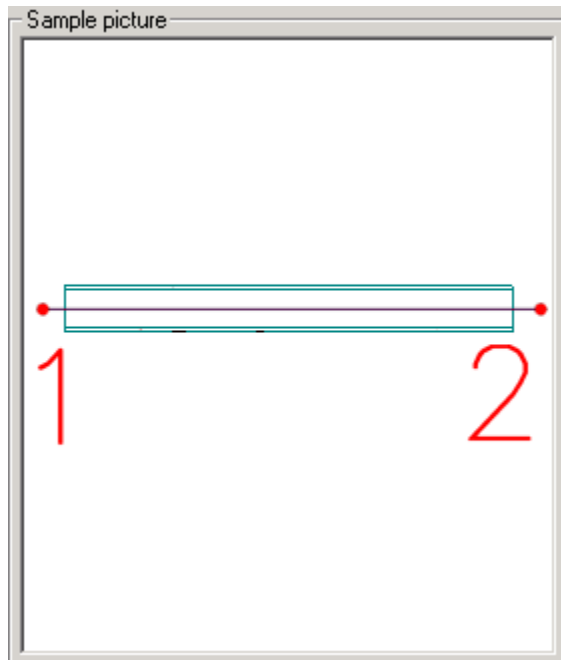
6. By default, the template contains a set of cross-sections, that do not necessarily must meet your needs. It may be necessary to input a new cross-section. It can be done via "three-dot" button next to the cross-section combo box (see the picture).

Name	B1
Type	beam (80)
CrossSection	CS16 - IPE27I ...
Alpha [deg]	0,00
LCS	standard
LCS Rotation [deg]	0,00
Layer	Standard ...
Priority definition	according to mem
Priority value	80
Perp. alignment	default
Eccentricity def.	whole member
Eccentricity y [mm]	0
Eccentricity z [mm]	0
End-cuts	Automatic
x-gap begin [mm]	0
x-gap end [mm]	0
Geometry	
Length [m]	8,062
Shape	Line
Beg. node	N2
End node	N3

7. When the "three-dot" button is pressed, a cross-section manager is opened. Select button **[New]** to input a new cross-section. In the list of offered types select **Profile library**. Browse for channel profile **UPE 160** and add to your project through button **[Add]**. Then close the dialogue and the cross-section manager.

8. Input the bracing beam from the foot of the first column to the head of the second one. Be careful with the definition of the points, zoom-in function is very useful here again. (The picture below shows the input of the second end point.)
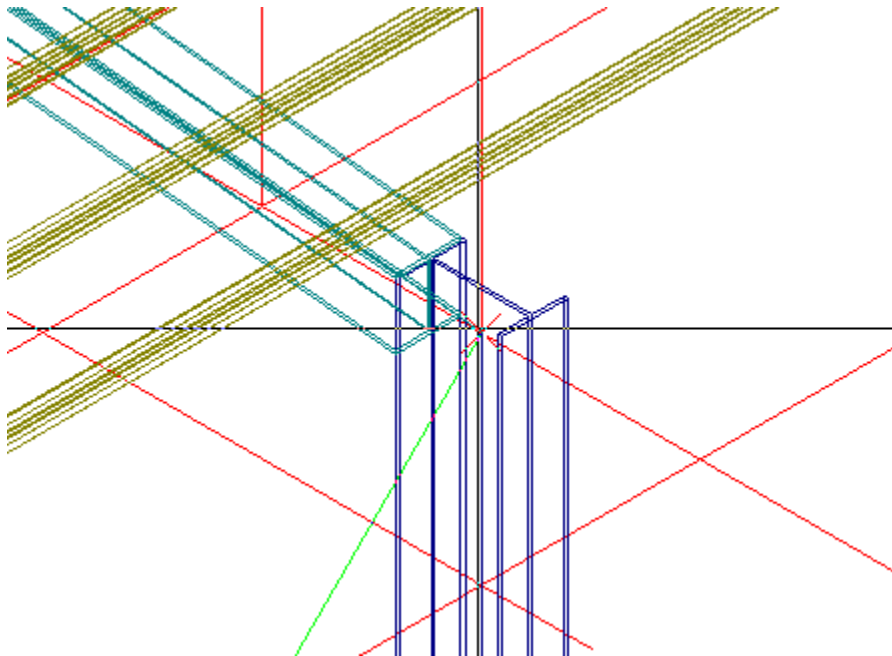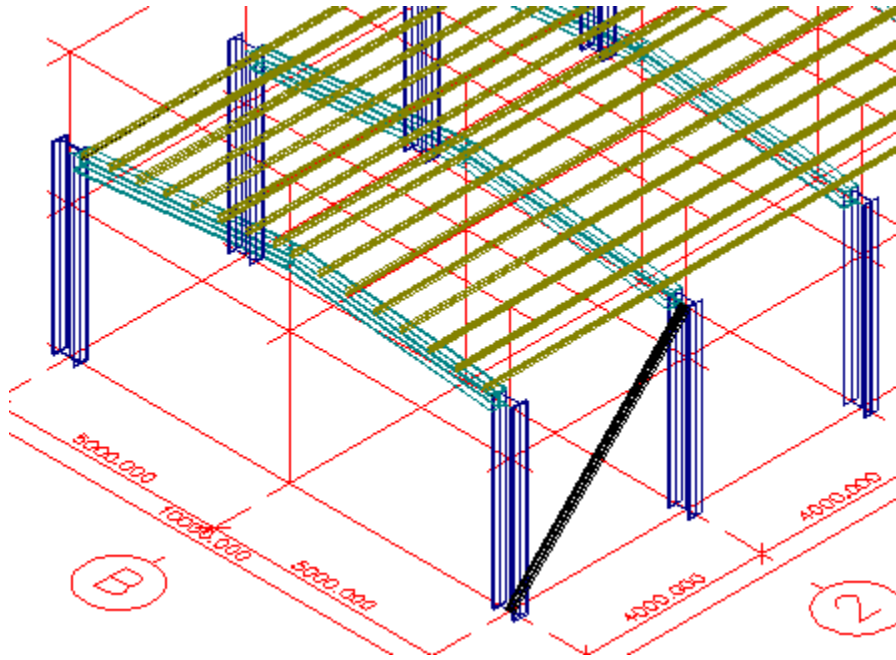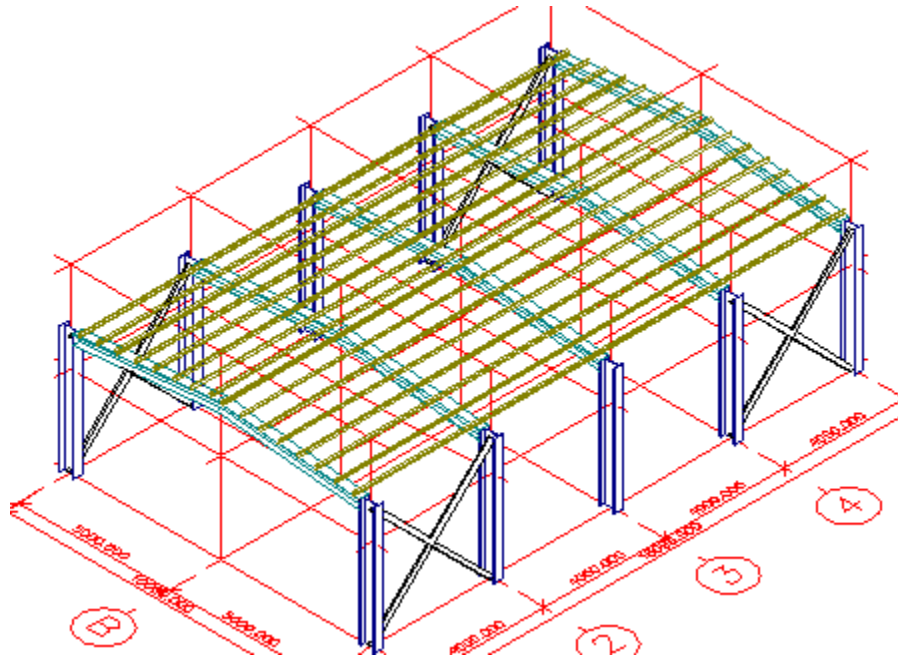
9. Input the other beam of the bracing "cross" the same way.
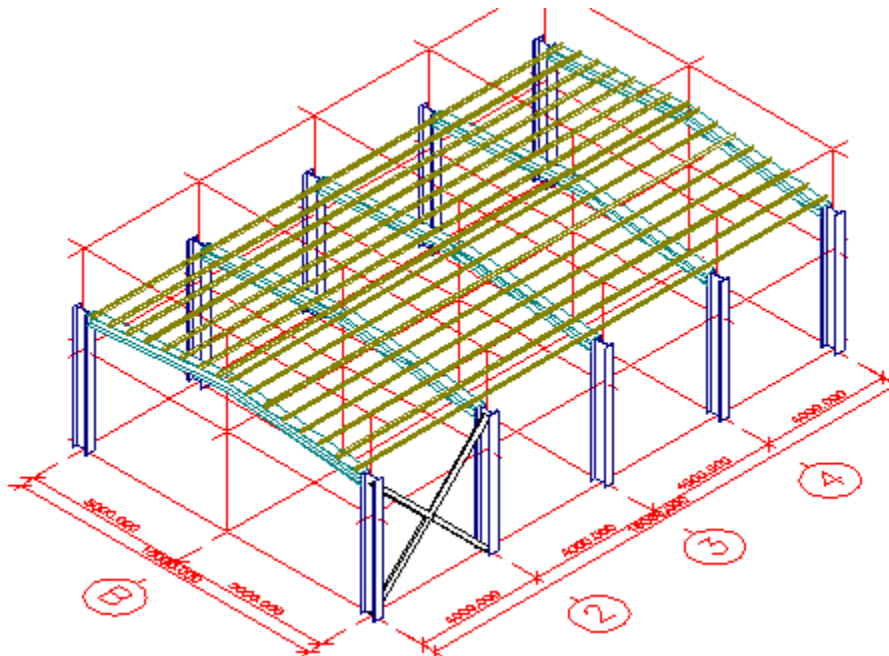
10. And finally, copy the bracing "cross" to all corner fields.

Editing the already inserted template

A template that has been inserted into the model can be edited almost any time later. When a template is edited, the same property dialogue is opened as during the input of the template.

📝 *Note: The editing presented in this sample project is done for didactical purpose. In practice you would have probably adjusted the parameters "properly" during the input phase.*

📝 *Note: The first sentence of this chapter says that "almost" any time later an inserted template can be edited. This suggests that there are some situations when the editing of the inserted template is not possible. It may happen that you decide to edit a part of the structure in SCIA·ESA PT. After such a modification if several templates are edited in SCIA·ESA PT at the same time or if several members from different templates are edited in SCIA·ESA PT at the same time, the affected templates may lose the information (either all or some) about themselves.*

In our sample project you have surely noticed that the channel profiles used for bracing are both oriented to the same side, which would lead to difficulties in designing the "intersection" of the bracing beams.
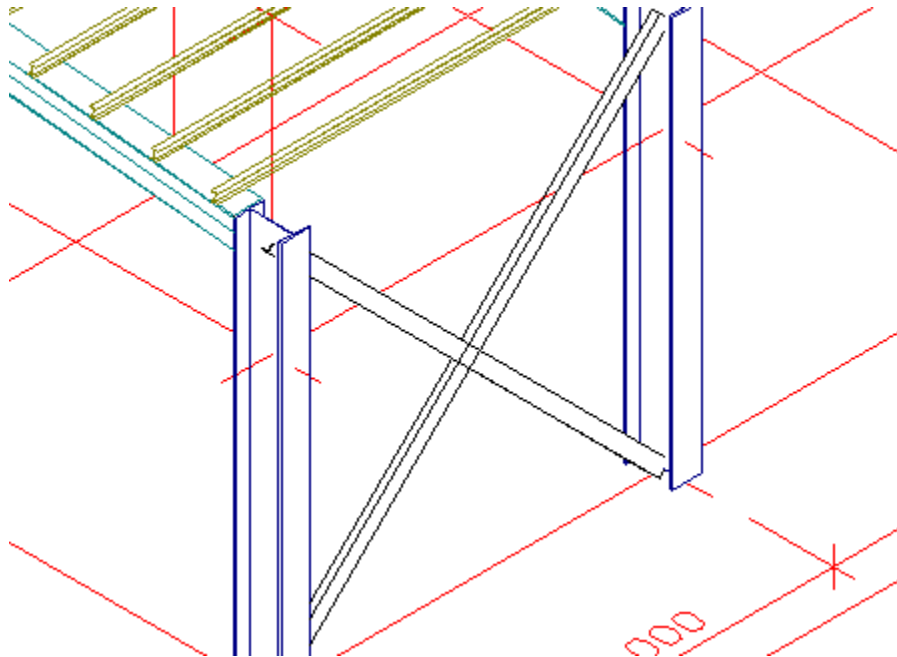
In practice one of the beams would be turned around (rotated by 180°) and the beams (or one of them) would be shifted in order to make the webs of the two channels to rest on each other. We will use edit function to provide for such a detail.

☑ Procedure

1. Start function **Vorlage modifizieren**.

2. Select the beam that goes from the foot of the first column to the head of the second column.

3. The template input dialogue is opened.

4. In order to rotate the profile, input 180 into LCS rotation field (see the picture below).

|	
Name	B1
Type	beam (80)
CrossSection	CS18 - UPE16 ...
Alpha [deg]	0,00
LCS	standard
LCS Rotation [deg]	180,00
Layer	Standard ...
Priority definition	according to mem
Priority value	80
Perp. alignment	default
Eccentricity def.	whole member
Eccentricity y [mm]	48
Eccentricity z [mm]	0
End-cuts	Automatic
x-gap begin [mm]	0
x-gap end [mm]	0
⊟ **Geometry**	
Length [m]	5,657
Shape	Line
Beg. node	N2
End node	N3

5. In order to shift the beam to make the webs to touch each other, we will define the eccentricity of the beam equal to the distance of the centroid from the surface. This distance can be found in the cross-section manager. Press the "three-dot" button next to the cross-section name and select the channel. In the property list find **dy**. Remember this value, close the cross-section manager and type the value to field **eccentricity y** (see the picture above).

ly [m^4]	9,1100e-006
lz [m^4]	1,0700e-006
lw [m^6]	4,2211e-009
alpha [deg]	0,00
Wely [m^3]	1,1400e-004
Welz [m^3]	2,2600e-005
Wply [m^3]	1,3161e-004
Wplz [m^3]	4,2516e-005
cYLCS [mm]	23
cZLCS [mm]	80
dy [mm]	-48
dz [mm]	0

6. Confirm with **[OK]**.
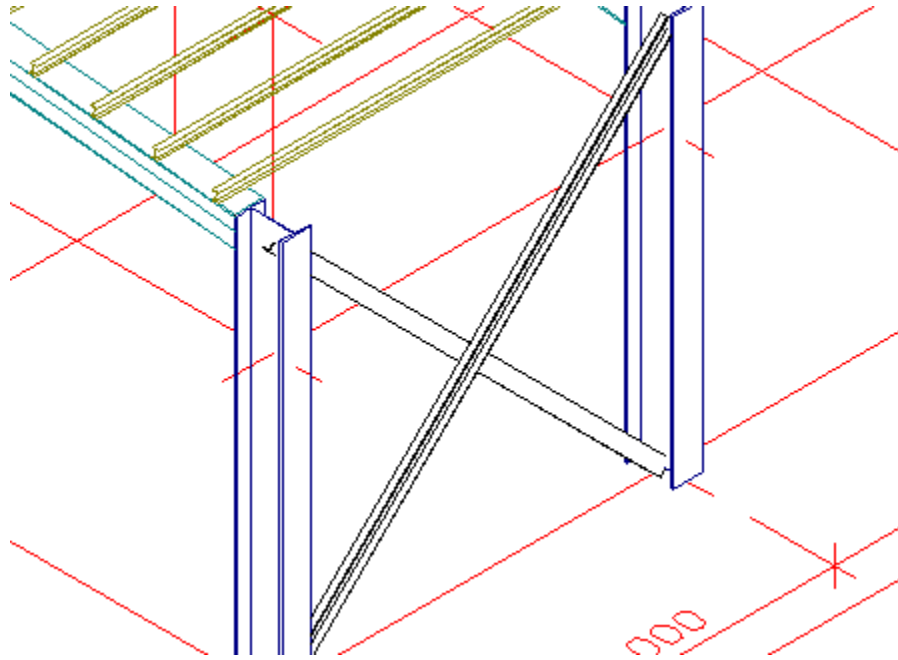
Editing the model in SCIA·ESA PT

The whole model or any part of it can be edited in the environment of SCIA·ESA PT. This procedure may be convenient for those who are familiar with that program. It is also efficient in some special modifications of the model that SCIA·ESA PT has been specifically designed for.

Once the modification is completed, you may return back to Allplan and all the changes are automatically transferred there.

Note: If members that belong to various templates are selected and edited in SCIA·ESA PT, this "group" of templates or members become a new "template" and it may happen that they forget the information about the original templates. It means that from now on they it may not be possible to edit them as original templates.

In this chapter the procedure is just general. You are kindly invited to follow it yourself and explore all the capabilities that this approach in editing offers you. If you are not familiar with SCIA·ESA PT, we suggest that you attend a specialised training for this software.

Procedure

1. Start function **Modifikation in ESA PT**.

2. Select the member or members that you want to edit in SCIA·ESA PT.

3. Once you finish the selection, SCIA·ESA PT is started and the selected part of the model is read into it.

4. After making the necessary changes, just close the project in SCIA·ESA PT and let the changes be saved to Allplan.