SCIA ENGINEER

TUTORIAL

STEEL FRAME

Table of contents

General Information

Welcome

Welcome to the SCIA Engineer Tutorial Frame Concrete. SCIA Engineer is an integrated, multi-material structural analysis and design software for all kinds of structures. Its wide range of functionality makes it deployable for any construction type: design office buildings, industrial plants, bridges or any other project, all within the same easy-to-use environment.

The program treats the calculation of 2D/3D frameworks, design and check of reinforcement included. Besides frames, it is also possible to dimension plate structures, inclusive of advanced concrete calculations.

The complete process of calculation and design has been integrated in one program: input of the geometry, input of the calculation model (loads, supports ...), linear and non-linear calculation, output of results, reinforcement design and checks according to various codes, generating the calculation report, etc.

SCIA Engineer is available in three different editions:

License version
The license version of SCIA Engineer is secured with a 'dongle', a hardlock, which you apply to the USB gate of your computer or a softwarematic license in your network.
SCIA Engineer is modular and consists of various modules. The user chooses from the available modules and composes a custom design program, perfectly tuned to his needs.
In the general product overview of SCIA Engineer you will find an overview of the different modules or module editions that are available.

Viewer mode
If the program doesn't find a licence it can be used as a viewer only. That means that any project can be opened, properties of entities can be checked, if the calculation has been done also results can be seen and report can be printed.
However, no change of the model is possible, no calculation can be run, no new output can be created.

Student version
The student version has the same possibilities as the license version for all of modules. This version is also secured by a softwarematic protection.
The output contains a watermark "Student version". Projects that are stored in the student version cannot be opened in the license version.

SCIA Engineer Support

You can contact the SCIA Engineer support service

By e-mail
Send an e-mail to support@scia.net with a description of the problem and the concerning *.esa file, and mention the number of the version you are currently working with.
By telephone
For various phone numbers to different offices visit our page https://www.scia.net/en/contact/offices
Via the SCIA Customer Portal website
http://www.scia.net/en/portal

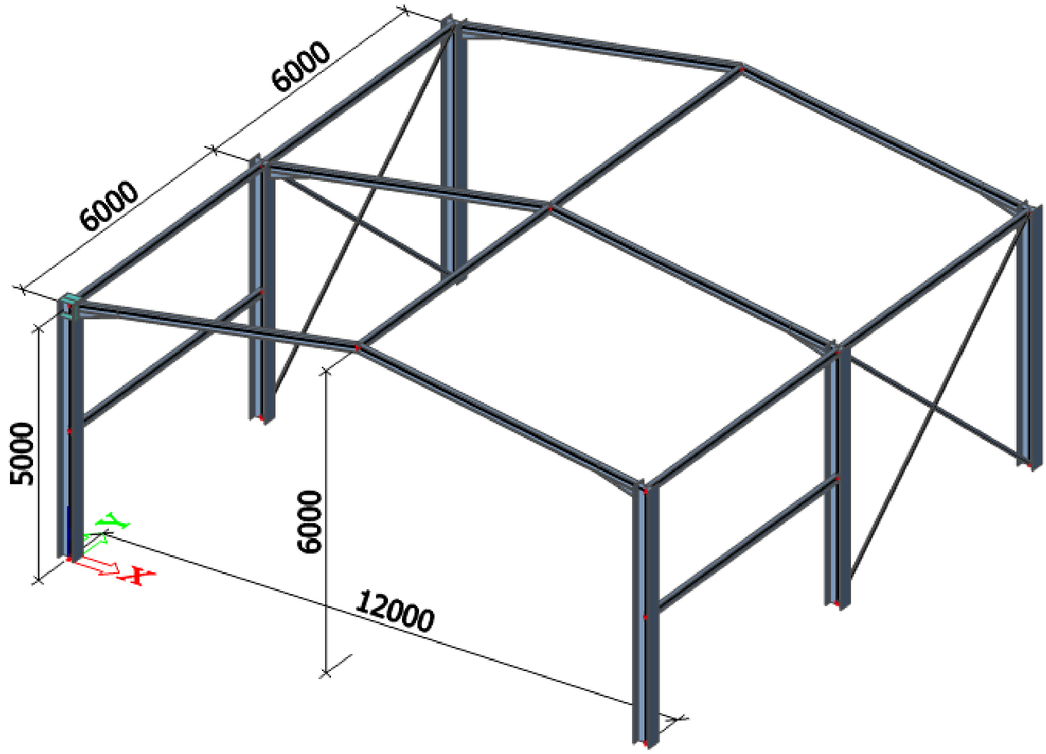
Websites

Link to Manuals and Tutorials
https://www.scia.net/en/support/downloads/scia-engineer-manuals-tutorials
Link to eLearning
http://elearning.scia.net/
Link to Web help
http://help.scia.net/

Introduction

This Tutorial describes the basic functions of SCIA Engineer, the input, analysis and design of a 3D steel frame.
Before you start, you must be familiar with your operating system: for instance working with dialogues, menu bars, toolbars, status bars, handling the mouse, etc.
First, we will explain how to create a new project and how to setup your structure. After the geometry and load input, the structure will be calculated and the results can be viewed.
Next, we will discuss the input of the buckling parameters and we will perform the steel check, the profile optimization and calculate steel connection.
The Tutorial ends with a brief introduction to the calculation report.

The figure below shows the calculation model of the structure to be designed:



Getting started

Starting a project

Before you can start a project, you need to start the program first.

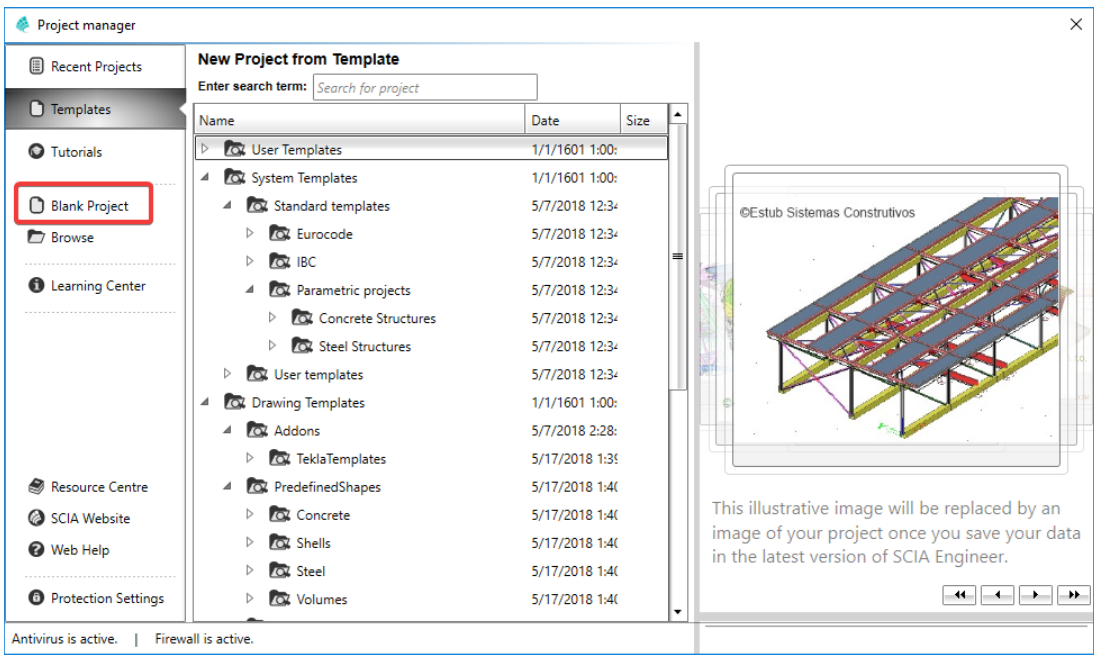
1. Double-click on the SCIA Engineer shortcut in the Windows Desktop, or
2. If the shortcut is not installed, click **[Start]** and choose **All apps > SCIA Engineer 18.0 > SCIA Engineer 18.0**.

If the program does not find any protection, you will see a dialogue indicating that no protection was found. You are offered to run Protection setup and select appropriate protection type (e.g. try-out), or run the program in Viewer mode.

For this Tutorial, you must start a new project with standard licence.

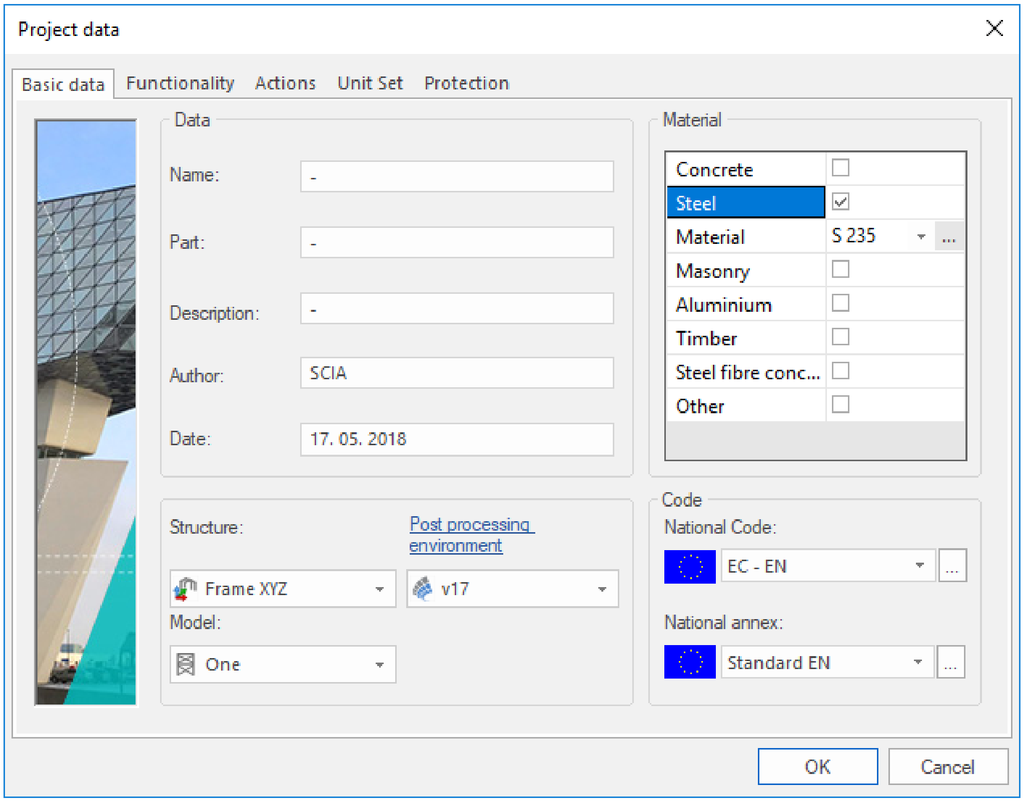
Starting a new project

1. When the **Project manager** dialogue appears, click **Blank project**.



2. You can also start new project with an icon  in the toolbar or with a key combination **Ctrl+N**.

   Now, the **Project data** dialogue is opened. Here, you can enter general data about the project.



3. In the **Basic data** group, enter your preferred data. These data can be mentioned on the output, e.g. in the report and on the drawings.
4. Choose the **Structure: Frame XYZ** (to limit input possibilities to 1D members in 2D plane only) and **Model: One**.
5. In the **Material** group, tick **Steel** checkbox.

   Material is the only required setting to proceed

   Choose **S235** from the combo-box.
6. In the Code frame select **National Code EC-EN** and **National annex: Standard EN**
7. Confirm your input with **[OK]** button.

Note:
*On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program. We don't need any additional functionality for this tutorial example.*

Project management

Save, Save as, Close and Open

Before entering the construction, we first discuss how to save a project, how to open an existing project and how to close a project. When running a project of this Tutorial, the project can be saved at any time. That way you can leave the program at any time and resume the project from there afterwards.

Saving a project

Click on  in the toolbar or press **Ctrl+S**.

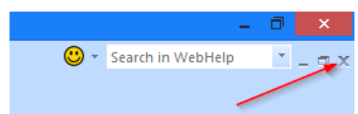
If a project has not yet been saved, the dialog box **Save as** appears. Click on the arrow in the list **Save** to choose the drive you want to save your project in. Select the file in which you want to put the project and click on **[Open**]. Select the subfolders. Enter the file name in **File name** and click on **[Save]** to save the project.

If you choose **File > Save as** in the main menu, you can enter a new/other drive, folder and name for the project file.

*Note: Autosave function creates a backup file every 15 minutes by default. These backup projects can be found in folder c:\Users*username*\Documents\ESA16.0\Autosave*

Closing a project

To close a project, choose **File > Close** in the main menu or click the smaller X button on top right corner of the application.



A dialog box appears asking if you really want to save the project. Depending on your choice, the project is saved and the active dialog is closed.

Opening a project

Click on  to open an existing project.

A list with projects appears. Select the desired project and click [**OK]** (or double-click on the project to open it).

Start project manger

Click on  to open project manager. Here the recently closed project can be found, as well as sample projects.

Geometry input

Input of the geometry

If you start a new project, the geometry of the structure must be entered. The structure can be entered directly, but you can also use for instance templates with parametric blocks, DXF files, DWG files and other formats.

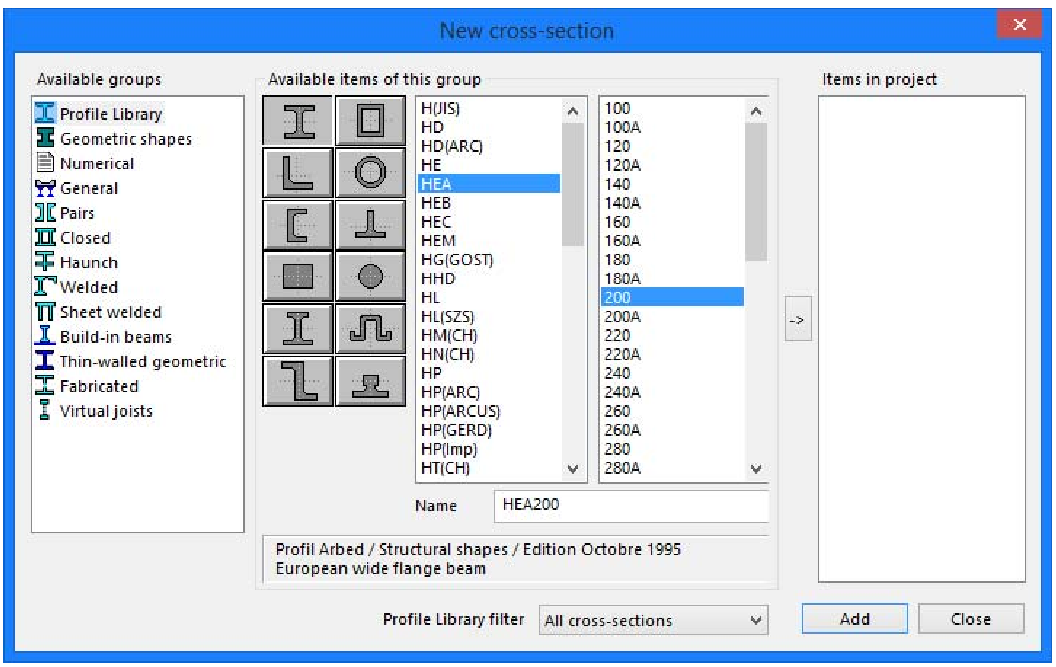
Profiles

When entering one or more 1D structure elements, a profile type is immediately assigned to each member. By default, the active profile type is represented. You can open the profile library to activate another profile type. If you want to add a structure part before a profile type has been defined, the profile library will automatically be opened.

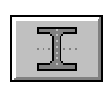
Adding a profile

1. Click on the **Cross-Sections**  icon in the toolbar.

   The cross-sections manager is opened. If no profiles have been entered in the project, the **New cross-section** window will be automatically opened.

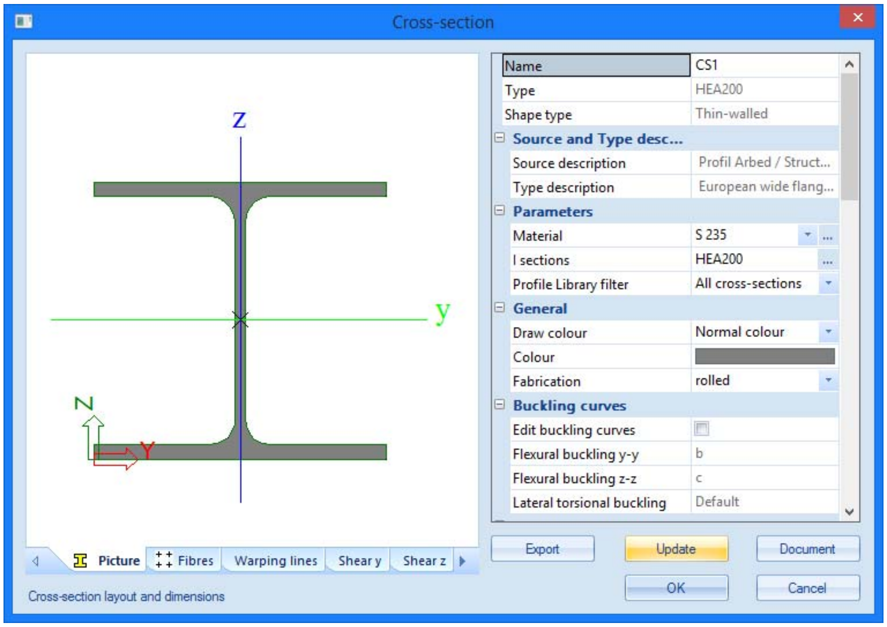


2. Click **Profile library** in the group **Available groups**.

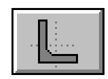
3. In the **Available items of this group**, you can choose I profile  . Choose **HEA 200** from the list.

4. Click **[Add]** or  to add the profile to the project.

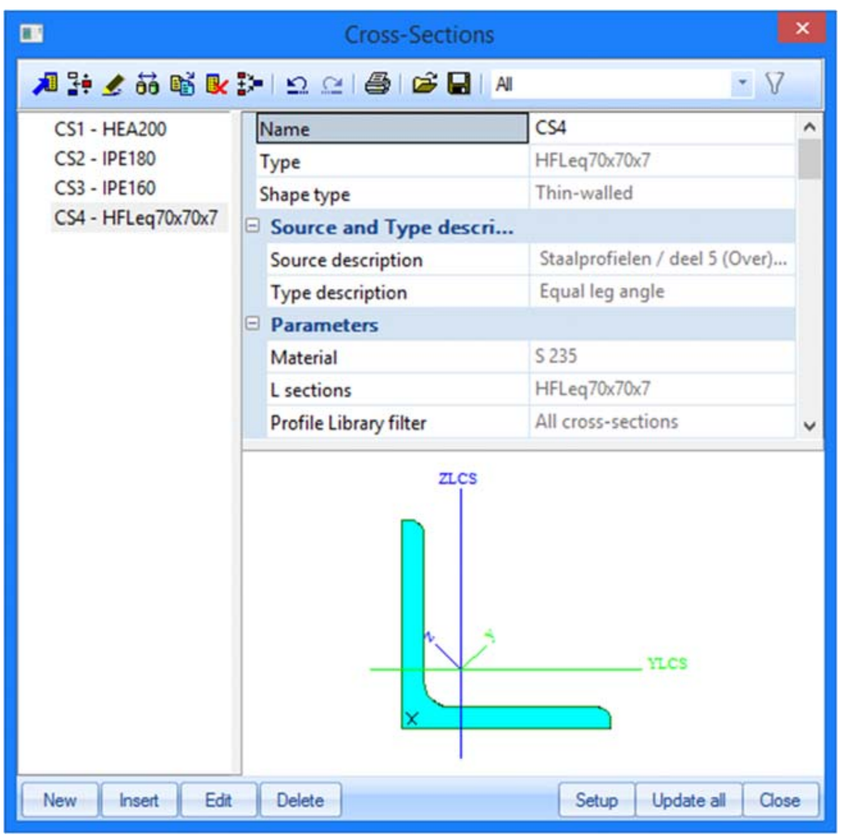
5. The **Cross-section** window appears.



6. Click **[OK]** to confirm, the profile is added to the **Items in project** frame. Add **IPE 180** and **IPE 160** in a similar way.

7. In the **Available items for this group**, you can choose an angle section  . Choose **HFLeq 70x70x7** from the list.

8. Click **[Add]** or  to add the profile to the project. Click **[OK]** to confirm, the profile is added to the **Items in project** frame.
9. Click **[Close]** in the **New cross-section** window, the **Cross-Sections** manager appears.

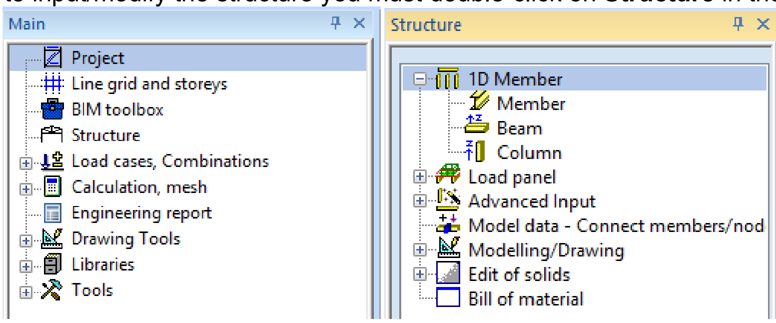


10. Click **[Close]** to close the **Cross-Sections** manager and to return to the basic modelling view.

Geometry

Structure menu

1. When a new project is started, the **Main** tree is automatically opened on left hand side. If you want to input/modify the structure you must double-click on **Structure** in the **Main** window.

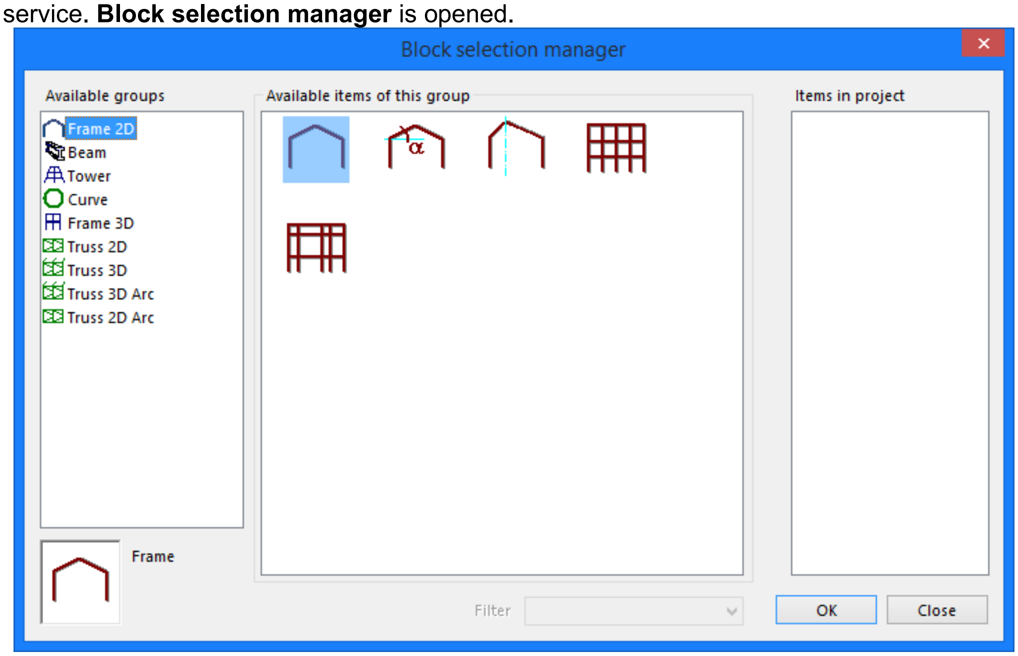


2. In the **Structure menu**, you can choose different structural elements to enter the structure.

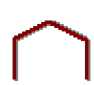
To model the structure you must enter the first frame. Then, this frame will be copied and the wind bracings and the horizontal beams will be added.
You can use columns and bars to enter the frame. SCIA Engineer however offers multiple Catalogue blocks, allowing for a smooth and simple input of the structure.

Entering a frame using a Catalogue Block

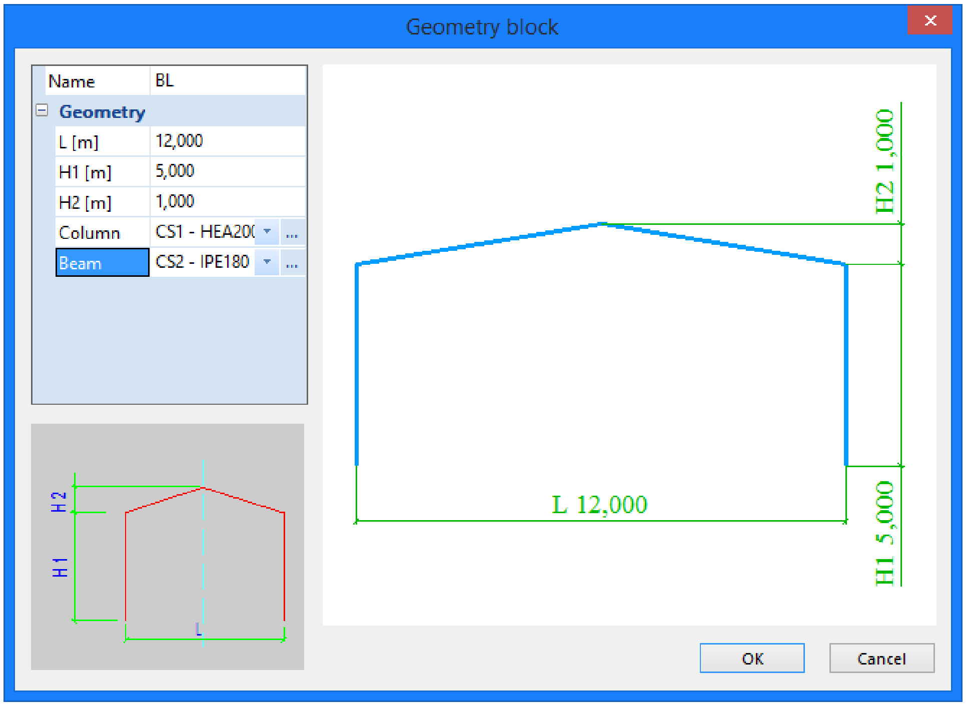
1. To enter a new frame, use the option **Advanced input > Catalogue Blocks** in the **Structure** service. **Block selection manager** is opened.



2. In the **Available Groups** group choose the first option **Frame 2D**

3. In the **Available items of this group** you can choose the first shape  .

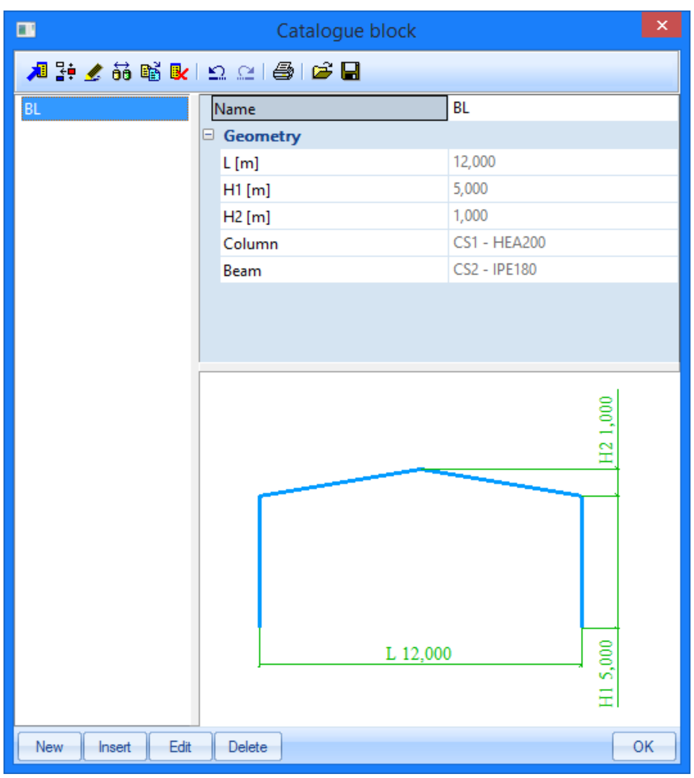
4. Confirm your choice with **[OK]**. The **Geometry block** window appears.



5. Now, enter the frame dimensions: **L = 12 m**, **H1 = 5 m** and **H2 = 1 m**

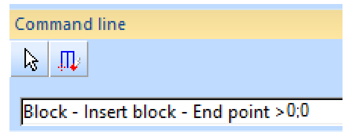
6. In the combo-box with cross-section selector choose **HEA 200** for the **Column** and **IPE 180** for the **Beam**.

7. Confirm your input with **[OK]**. The **Catalogue block** manager appears.



8. Click **[OK]** to return to the modelling view. The frame is now graphically represented by thin blue lines attached to mouse cursor. You are now asked to select insert point.

9. The frame is positioned with the left column in the origin of the coordinate system. Type the coordinates **0;0** in the **Command line** and press **<Enter>** to confirm your input.



10. Finish the input with the **<ESC>** key.

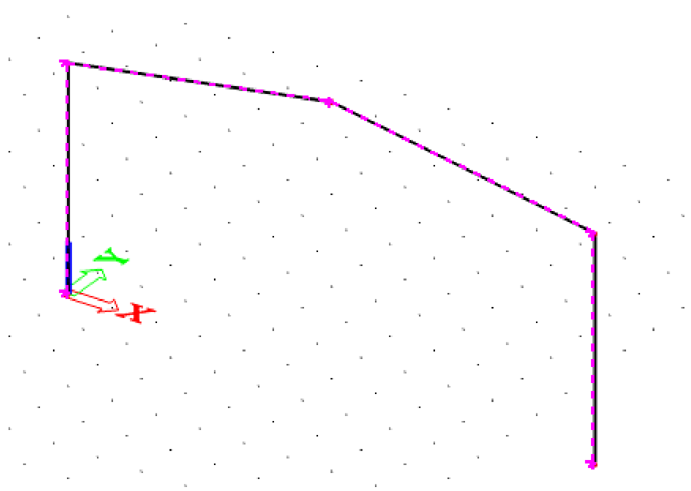
Notes:
*The properties of selected elements are shown and can be modified in the **Properties** window on the right hand side of the user interface.*
*If no cross-section has been defined in the project, the **New cross-section** window will automatically appear as soon as you try to enter a structural element (column, beam…).*
*With **Zoom All** button in the toolbar, or double-click with the mouse wheel, you can visualize the entire structure.*
*The use of **,** or **;** to separate coordinates depends on the regional settings of Windows. You can also use spacebar to define coordinates instead of the two.*

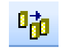
After input of the first frame, it can be copied to obtain the hall frames easily. As you need two copies, you can use the **Multiple copy** function .

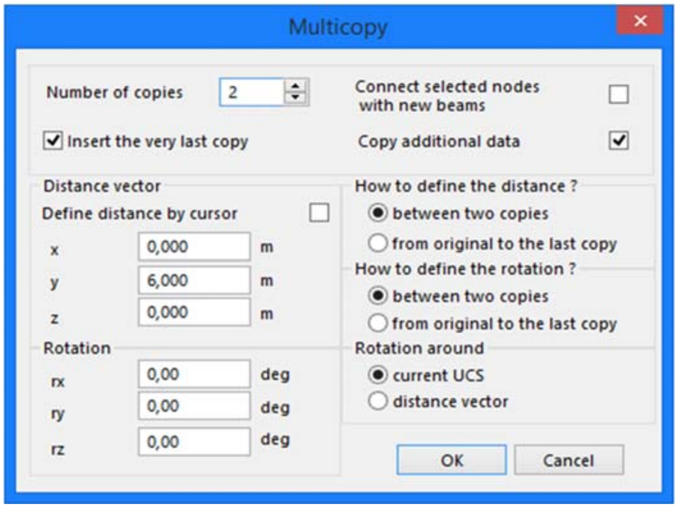
Create multiple copies

1. First select all (already modelled) entities to be copied. As you must copy all members, you can use the **Select All** icon 

   This way all bars and nodes are selected; this is represented by dashed violet line:



2. Now, you can use the **Multiple copy** option  (**Modify** > **Multiple copy**).

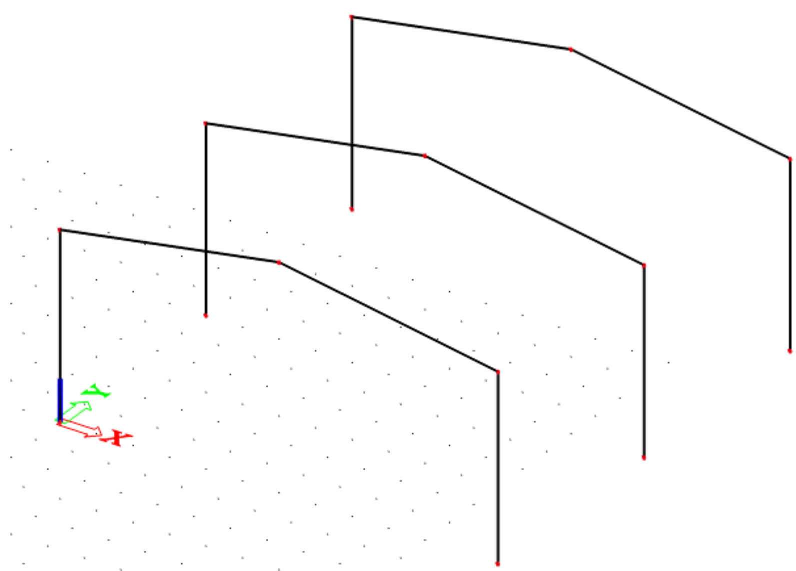


3. In the **Number of copies** field enter **2**.

4. To manually set the distance between the frames, deactivate the **Define distance by cursor** option. Now, you can enter the distance **6 m** in direction Y.

5. Click **[OK]** to confirm your input. New frames are modelled.

6. Press **<ESC>** to cancel the selection.

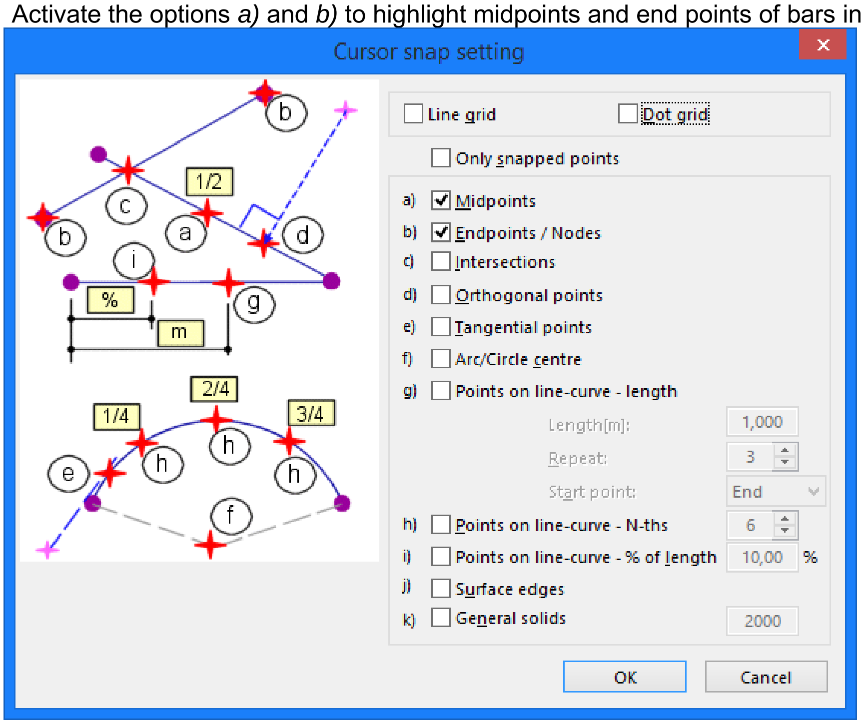


Connecting beams of the frames can be entered when the frames are entered. The start and end nodes of the beams are already known, i.e. begin and end nodes of the entered members. Therefore you do not have to enter the beams by means of coordinates; instead of that you can use the **Cursor snap settings.**

Cursor snap settings

1. Double-click on the **Cursor snap settings** icon in the Command line or click on the button Snap mode at the lower right corner of the application frame. The **Cursor snap settings** window is opened.

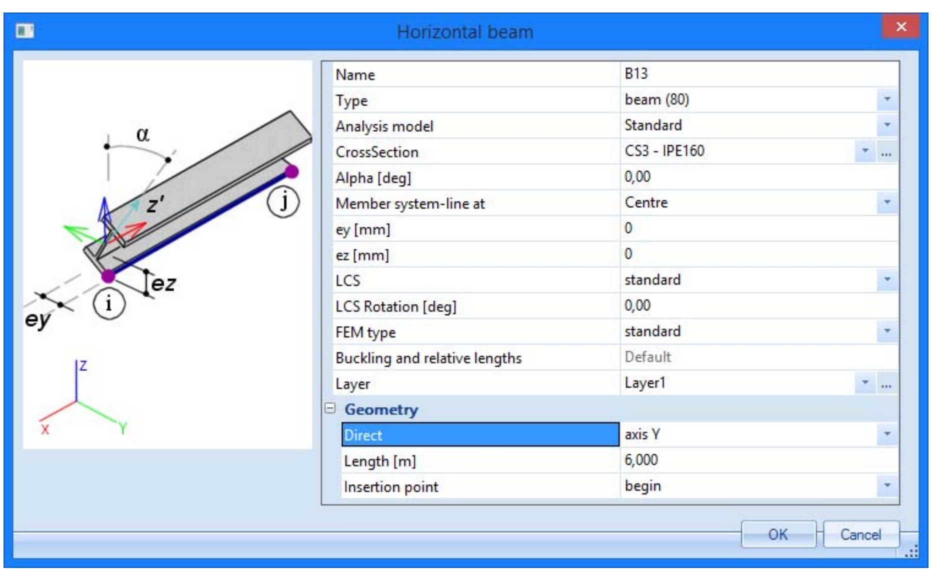
2. Activate the options *a)* and *b)* to highlight midpoints and end points of bars in this project.



3. Click **[OK]** to confirm your setup. Now, you can input the beams.

Entering a beam

1. To enter a new beam, use the **Beam** command in the **Structure** service.
2. In the **CrossSection** field, choose the third section, **CS3 - IPE160**.



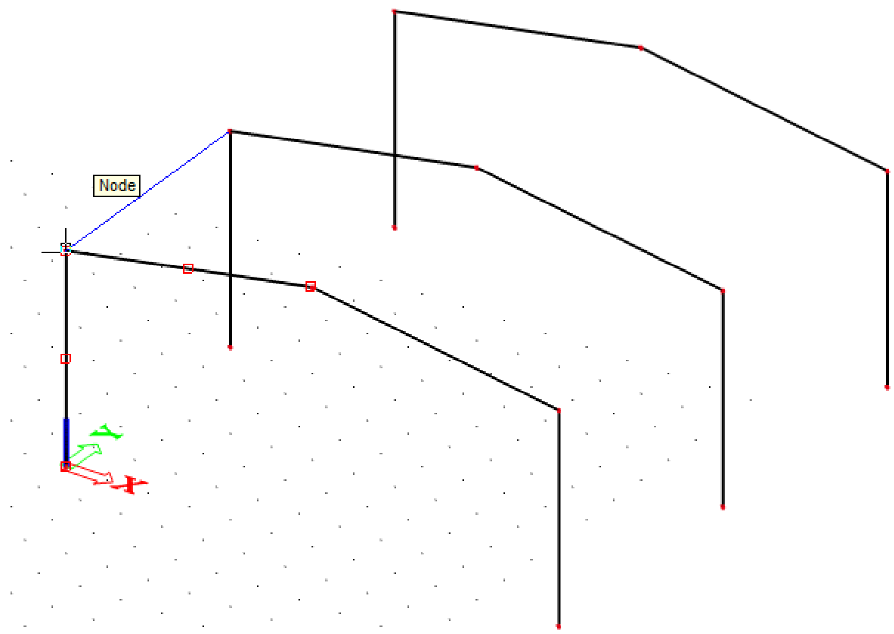
3. As the structure type **Frame XYZ** has got two horizontal axes (i.e. X and Y), you must indicate the right direction for the horizontal beam in the **Direct** field. Choose **axis Y** possibility.

4. The beam length is **6 m**.

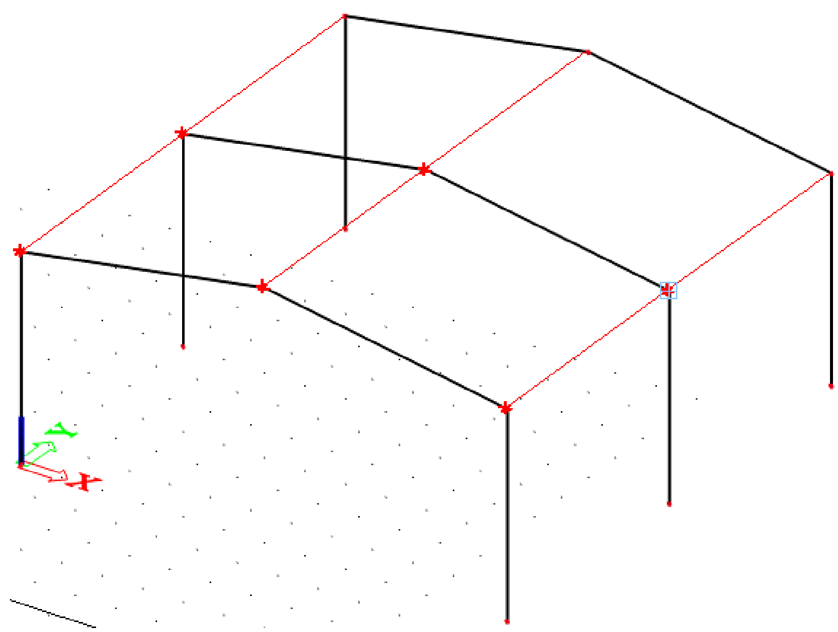
5. The insertion point is (as default) set to **begin** so that the left point determines the position of the beam.

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

7. Now, you can enter the beam by clicking with your mouse on the top node of the left-hand side column of the first frame and similar node on the middle frame:



8. Enter the other beams of the roof in a similar way, always by clicking the top nodes of columns or inclined beams.



9. Press **<ESC>** to finish the input.

10. Press **<ESC>** once more to cancel the selection.

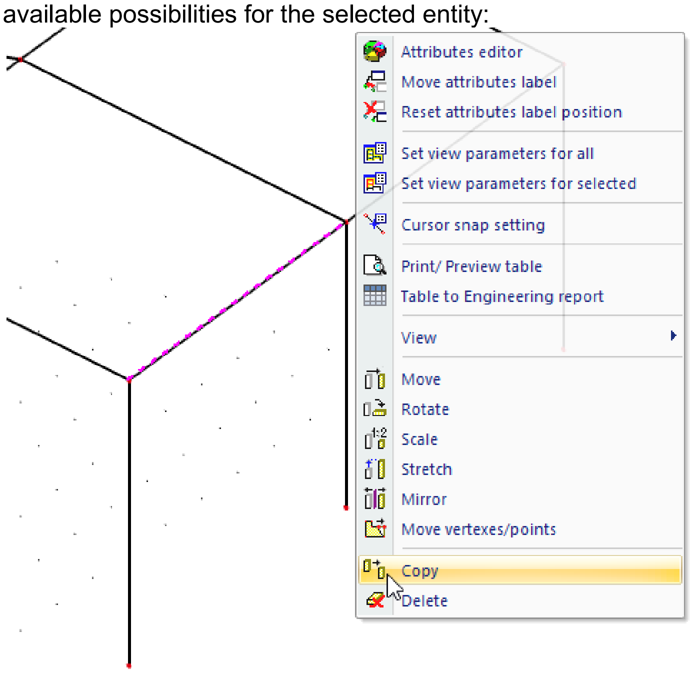
Note:
*The **Multiple copy** option also allows for the automatic input of the beams connecting the frames.*

In the first span (between the first and second frame) two more horizontal beams are located. To enter these beams you could use the **Beam** command. SCIA Engineer however enables copying these entities manually.

Copying entities

1. First select the entity to be copied. As this is a horizontal beam, you can select one of the entered beams with the left mouse button. Violet colour indicates that the bar has been selected. Properties of the bar are shown in the **Properties** window.

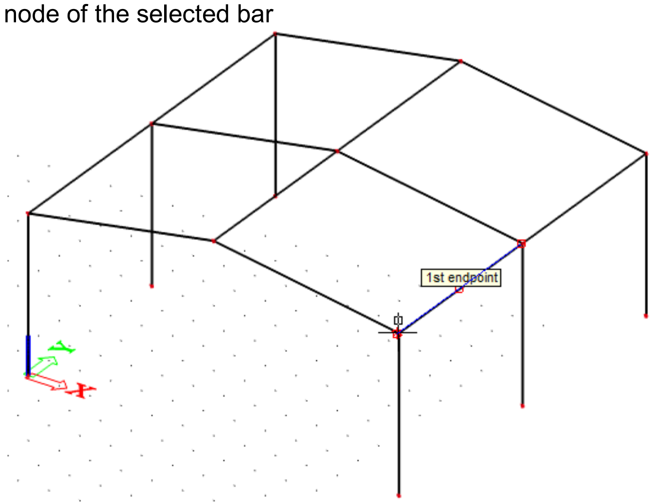
2. Click with the right mouse button anywhere in the user environment. Context menu lists the available possibilities for the selected entity:



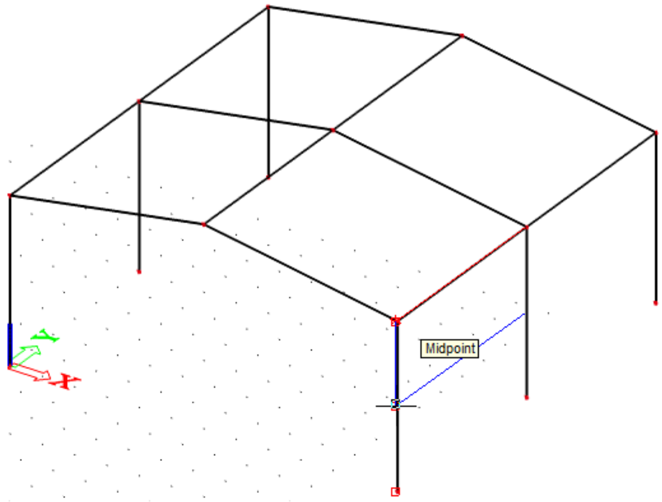
3. In this menu, choose the option **Copy**.

Note: Instead of points 2 and 3 you can also use CTRL+C key shortcut.

4. The program asks the **Start point** of the copy. Click with the left mouse button on the start node of the selected bar



5. Now you must enter the **End point**, i.e. the position where the starting point should be copied. As the new beams start in the middle of the columns, the midpoint of one of the columns in the first frame is selected.



As Midpoints option was already activated for the **Snap settings**, you can simply pick the centre point of the column.

6. When the first beam is copied, the command remains active until you press **<ESC>**, so that you can also pick the midpoint of the second column of the first frame to enter a horizontal beam at that position.

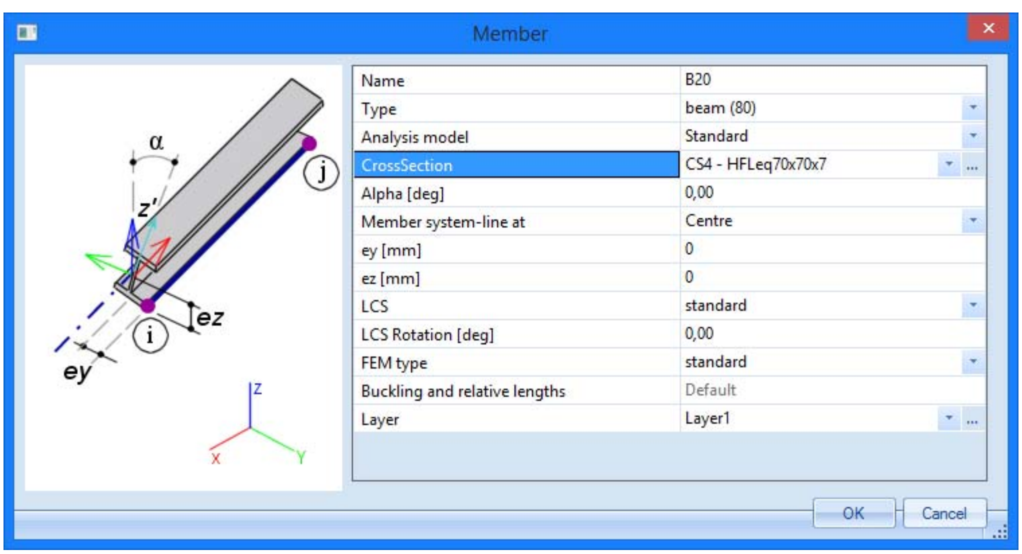
7. Press **<ESC>** to finish the input.

8. Press **<ESC>** once more to cancel the selection.

After input of the horizontal beams, you can enter the bracings.
The bracings are not vertical columns or horizontal beams but rather arbitrary 1D members. Therefore you must use the **Member** command in the **Structure** service**.**

Entering bracings

1. To enter a new bracing, use the **Member** command in the **Structure** service**.**

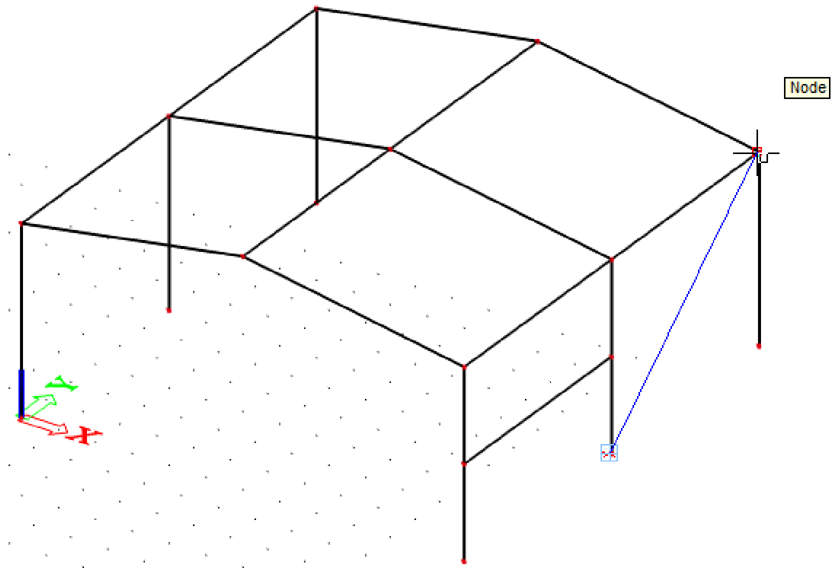


2. In the **Type** field, choose **Wall Bracing (0)**. This specification is considered for the STRUCTURAL model only and does not influence the calculation model or the results.

3. In the **CrossSection** field, choose the fourth section, **CS4 - HFLeq 70x70x7**. Note that no length or direction is requested this time, as you will define the geometry afterwards.

4. Confirm your input with **[OK]**.

5. Now, the bracings can be entered between the second and third frame. Click on the start and end nodes of the columns to draw diagonal:

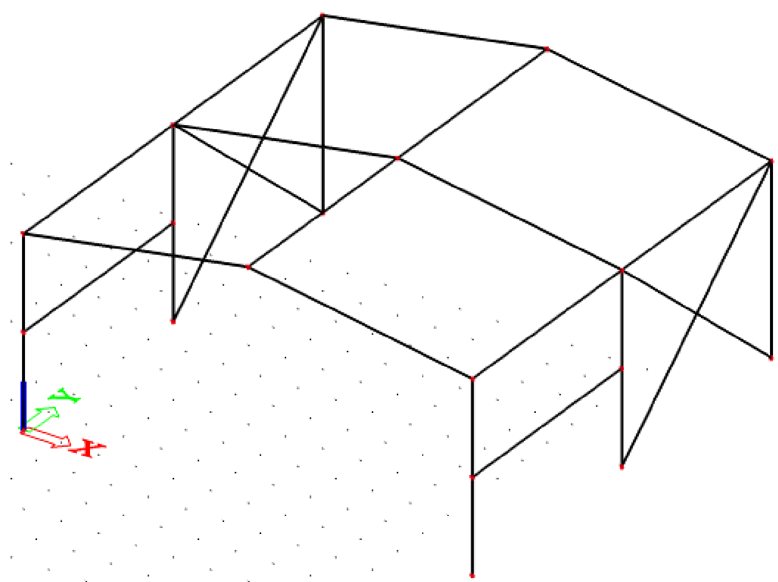


6. Repeat selecting start nodes and end nodes until all bracings are modelled.

7. Press **<ESC>** to finish the input.

8. Press **<ESC>** once more to finish the selection

The structure is completely set up. Now, you can finish the geometry input by adding end conditions, i.e. enter haunches, hinges and supports.



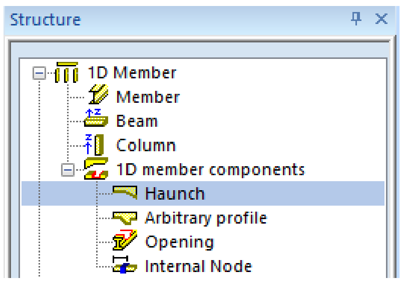
Haunches

In SCIA Engineer every member is regarded as prismatic, with constant cross-section, until a haunch is entered. Haunches are entered on the roof beams in this project, at column sides. Haunch is defined by the following parameters:

- A section with variable height
- A length, over which the variable height must vary up to 0

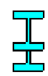
Entering Haunches

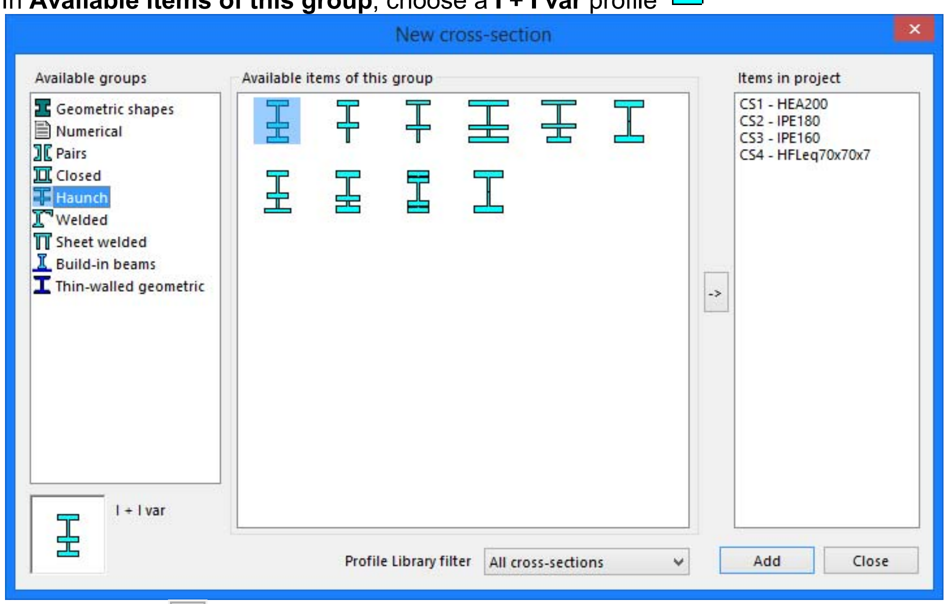
1. To enter a new haunch, use the **1D member > 1D member components > Haunch** command in the **Structure** menu.



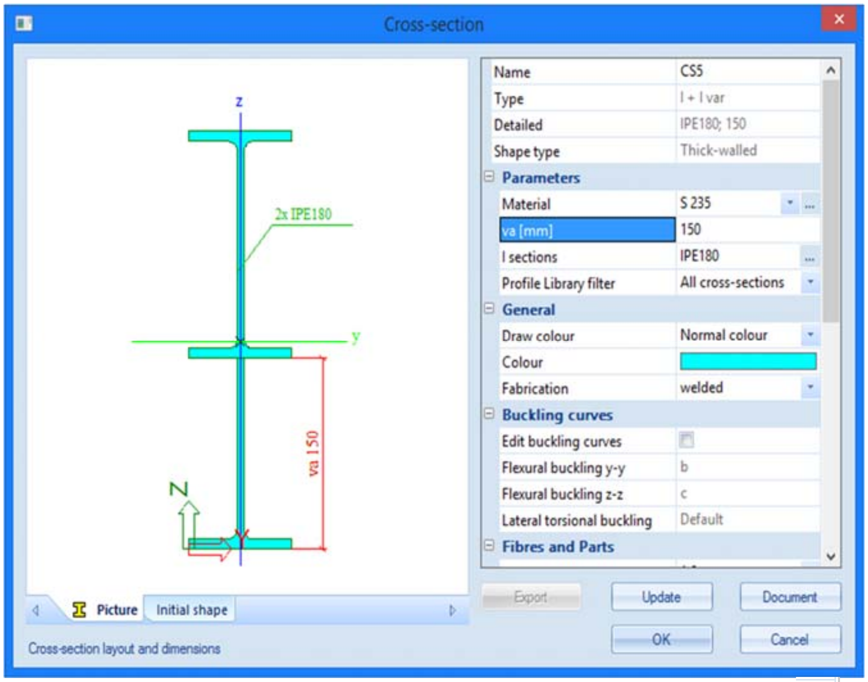
2. As indicated, a haunch requires a cross-section with a possibility to create variable dimension(s). Since this project does not contain any variable profiles yet, the **New cross-section** window automatically appears.

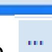
3. Select in the **Available groups** the **Haunch** group.

4. In **Available items of this group**, choose a **I + I var** profile 



5. Click **[Add]** or  to add the profile to the project. The **Cross-Section** window appears. Here, the properties of the variable section can be changed.



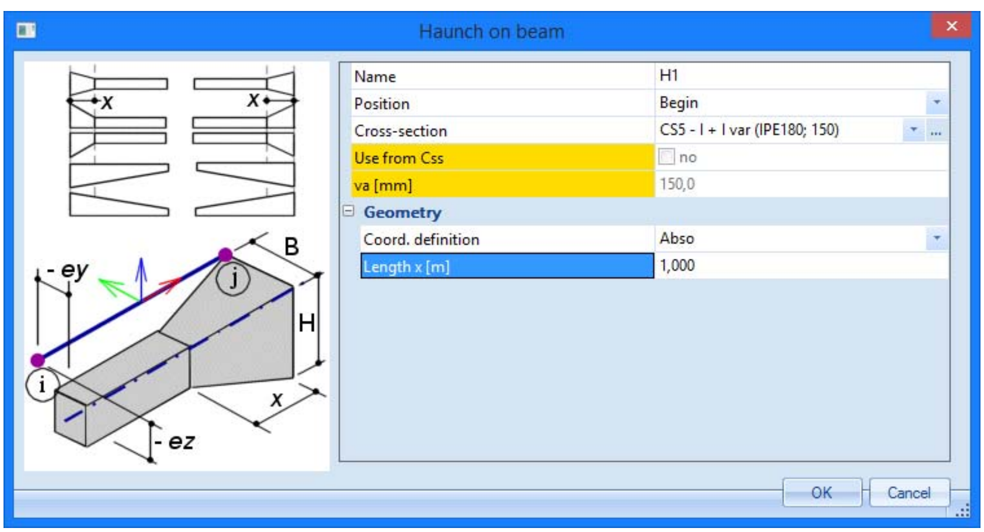
6. In the **I sections** field, change the section in an **IPE 180** by clicking the  button behind the section type.

7. When the correct I-section is set, the variable height **va (mm)** is set to **150mm**

8. Confirm your input with **[OK]** and use button **[Close]** to close the **New cross-section** dialogue.

9. The **Cross-Section manager** appears; click **[OK]** to close this window as well.

10. Now, the **Haunch on beam** window is opened.



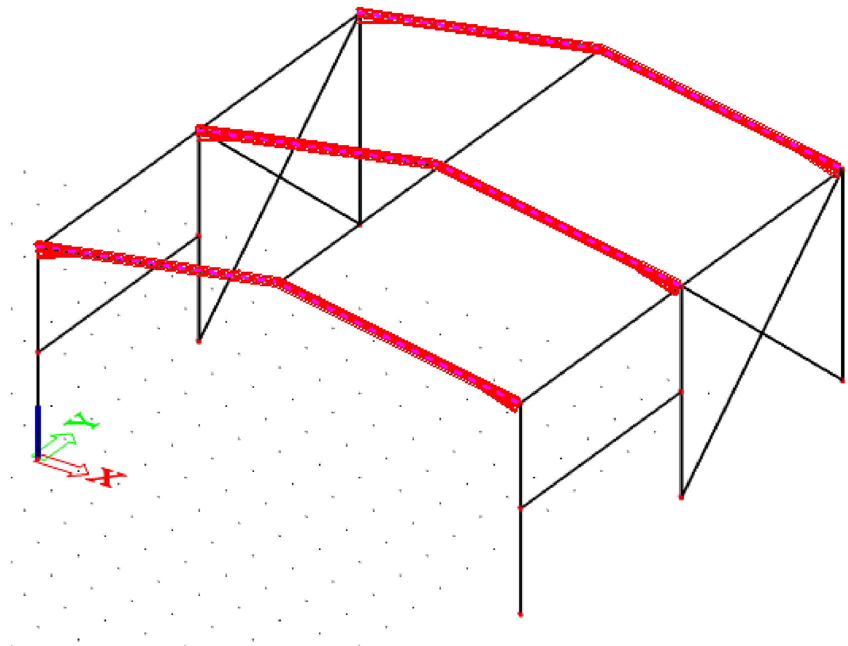
11. In the **Position** field, choose **Begin** to position the haunch at the start node of the member.

12. In the **Coord. definition** field, choose the option **Abso** to indicate that the length, over which the variable height must vary, can be entered in absolute units, i.e. in meter.

13. When the Coordinate Definition is adapted, the length of the haunch can be entered in the **Length x [m]** field. For this project, enter length **1 m**.

14. Confirm your input with **[OK]**

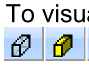
15. Now, the program asks to indicate the members on which a haunch must be entered. Select the 6 roof beams with the left mouse button:



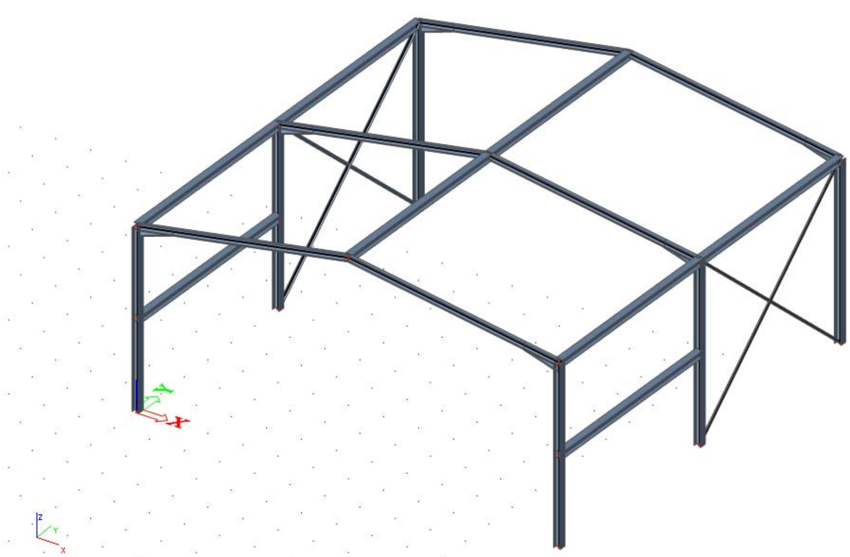
16. Press **<ESC>** to finish the input.

17. Press **<ESC>** once more to cancel the selection.

To visualize this model, you need to click the following buttons in the command line:



- **Show/hide surfaces** to show the surfaces of the sections.
- **Render geometry** to obtain a rendered view of the members.



In the command line, click **Show/hide surfaces**  and **Render geometry**  to deactivate both options and visualise the system lines of the members again.

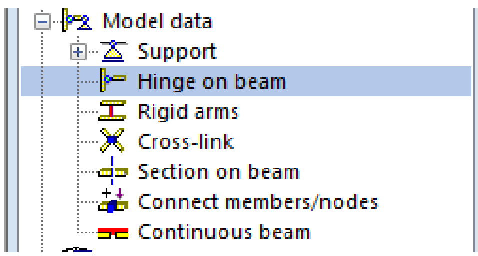
Note:
A haunch overwrites specification of the original cross-section. For this project, this specifically means that the profile of the roof beam is replaced by the I + I var profile. If the haunch is removed, the I + I var profile will be maintained instead of the I-section of the roof profile.

Hinges

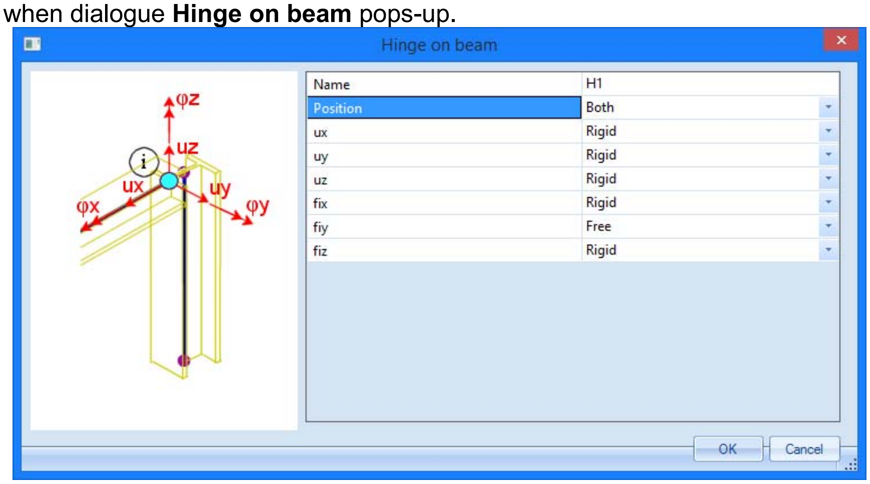
In SCIA Engineer, every node where two or more members connect is regarded as fixed, until a hinge is entered and some rotations are released.
In this project, the diagonals should be connected with the other members in a hinged way. Therefore, you must enter hinges manually.

Entering hinges

1. To enter hinges, use the **Model data > Hinge on beam** command in the **Structure** menu.



2. The hinges are put to both ends of the diagonals; therefore choose **Both** for the **Position** when dialogue **Hinge on beam** pops-up.



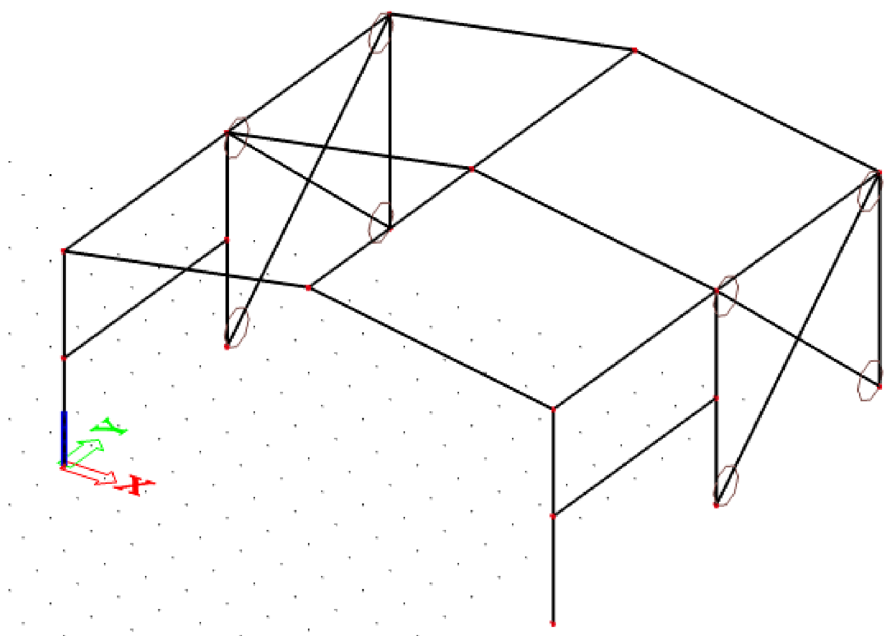
3. To obtain a hinge, the rotation **fiy** is set to **Free**, the translations and the other rotations remain **Rigid**. In this way, the diagonals will be exclusively hinged in the planes of the sidewalls.

4. Confirm your input with **[OK]**.

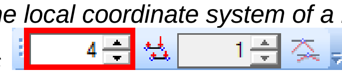
5. The hinges are added when you click the diagonals with the left mouse button.

6. Press **<ESC>** to finish the input.

7. Press **<ESC>** once more to finish the selection.



Note:
Hinges are always defined with regard to the local coordinate system of a member.

Increase the scale for input data in toolbars  *if you feel that the circle sign of hinge is too small.*

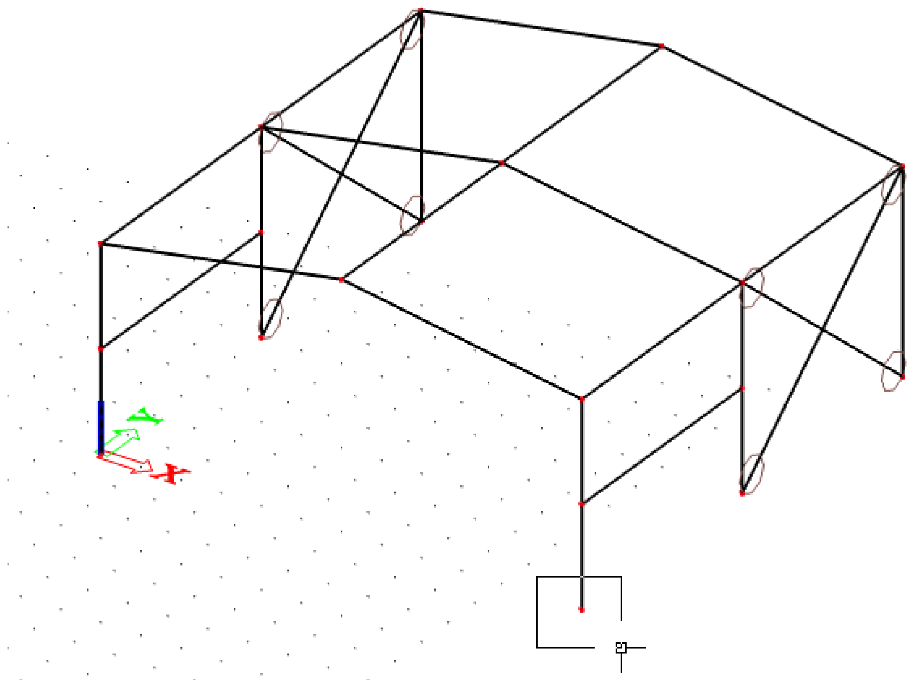
Supports

The geometry input can be completed with supports. The column bases are modelled with hinges.

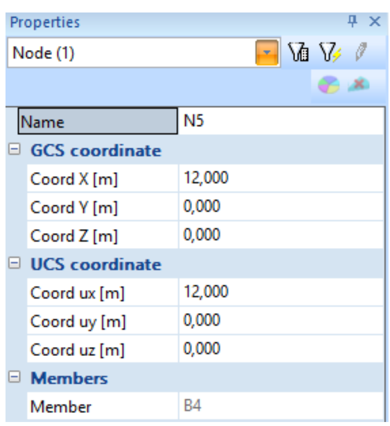
Before adding the supports, you first can select the nodes to position the supports (for this Tutorial). You can select these nodes manually, one by one, but SCIA Engineer offers a simple method to select multiple entities based on common property.

Selecting elements by property

1. To select all column bases, select one of the bottom nodes, by drawing a frame from the left to the right with the mouse cursor.

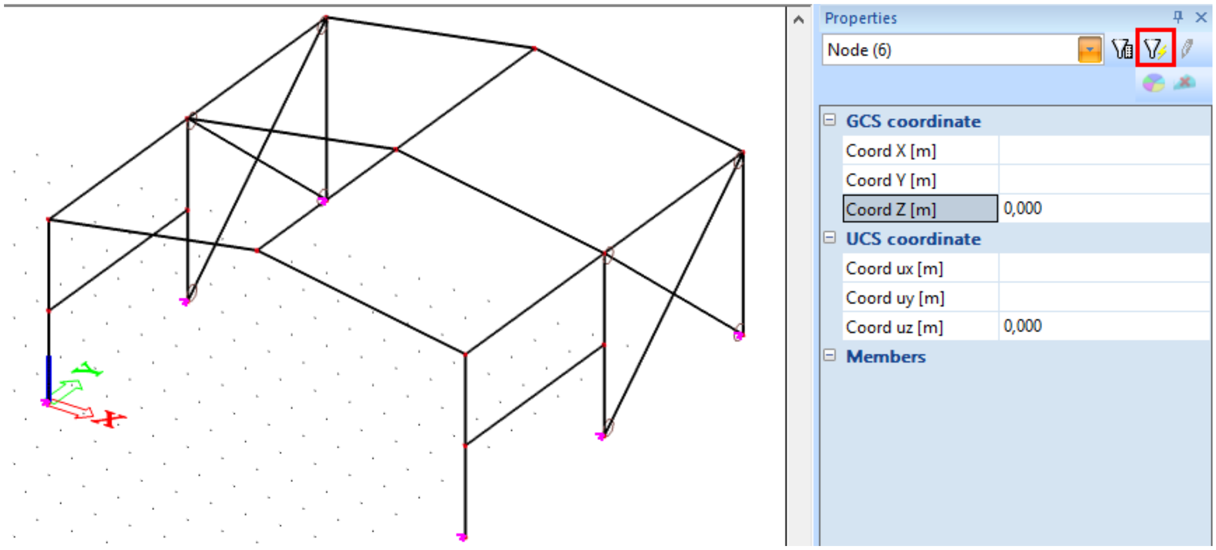


2. The **Properties window** shows the properties of this node:



3. Now, choose the property to be used for the selection of the entities. For this project, you want to select all bottom nodes. The common property of these nodes is their coordinate in global Z direction.
   Click with the left mouse button on the **Coord Z (m)** property to select appropriate row. The table cell is highlighted by blue colour.

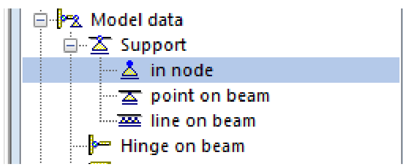
4. Choose the **Select elements by property** button . The program will search all entities with the same property. In this example, the program will select all nodes, for which the **Coord Z (m)** property corresponds to **0 m.**



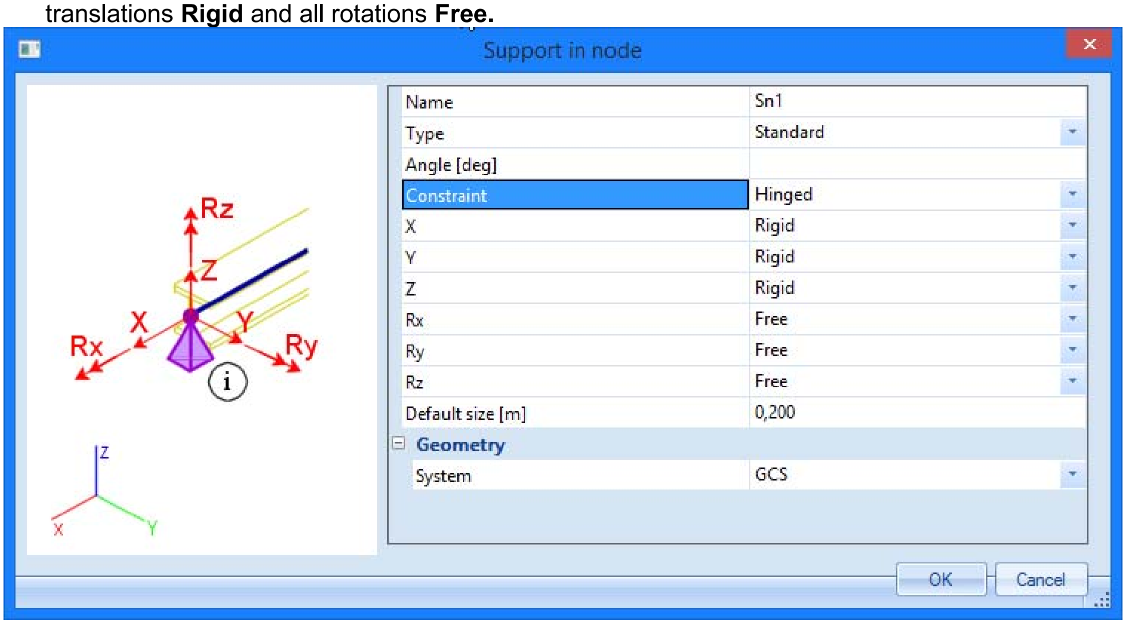
The column bases are selected; now, supports can be added to these nodes.

Entering supports

1. To enter supports, use the **Model data > Support > in node** option in the **Structure** menu.

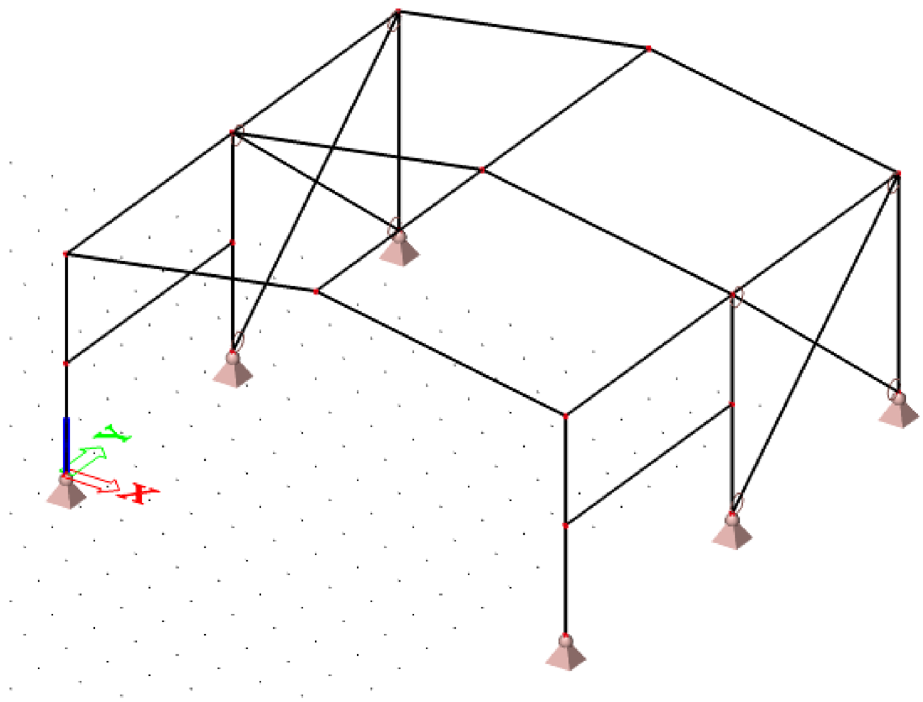


2. You can easily define all 6 end conditions by choosing **Constraint Hinged**, so that all translations **Rigid** and all rotations **Free.**

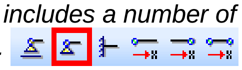


3. Confirm your input with **[OK]**. The supports are automatically attributed to the selected nodes.

4. Press **<ESC>** to finish the selection.



Notes:
If you draw the box from the left-hand side to the right-hand side with the mouse cursor, only entities which are completely inside will be selected. If you draw the rectangle from the right-hand side to the left-hand side, the entities which are completely in the rectangle as well as the entities that intersect with the rectangle will be selected.

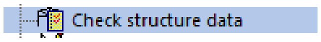
The ***Command line*** *includes a number of predefined supports. For this project, you could have used the* ***Hinged support***  *icon.*

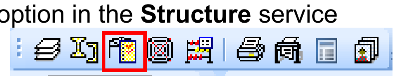
Similar to the selection of supports, you could have used the ***Select elements by property***  *option to select all diagonals, where hinges had to be entered. The determinant property here would have been the CrossSection.*

Check Structure data

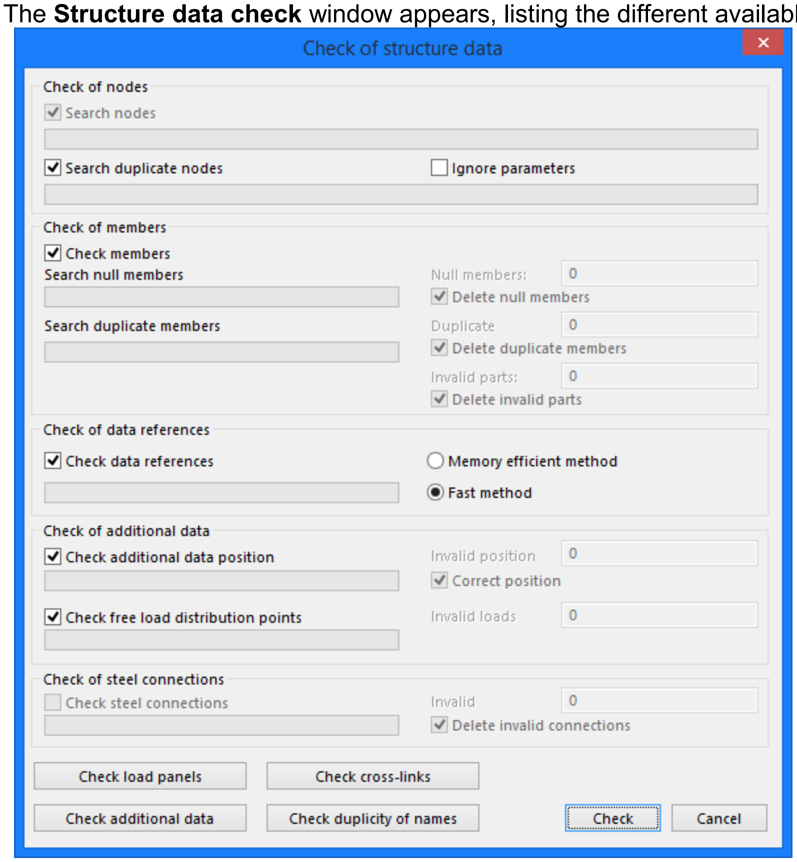
After input of the geometry, the input can be checked for errors by means of the option **Check Structure data.** With this tool, the geometry is checked for duplicate nodes, zero beams, duplicate members, wrong references of hinges or supports etc. However, this tool does not check if the structure is correctly supported or if it is a mechanism.

Checking the structure

1. Double-click on the **Check structure data** option in the **Structure** service 

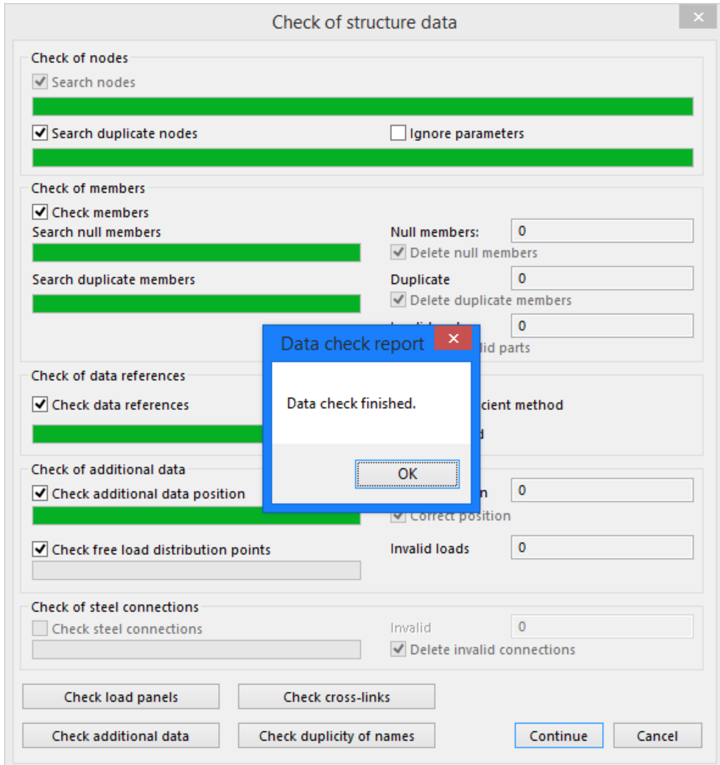
or click on the  icon in the toolbar.

2. The **Structure data check** window appears, listing the different available checks.



3. Click **[Check]** to perform the checks.

4. The **Data Check Report** window appears, indicating that no problems were found.



5. Close the check by clicking **[OK]**.

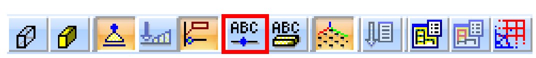
6. In case of any problem SCIA Engineer can automatically correct the structure data (delete duplicated entities, correct wrong reference, etc.)

Connecting entities

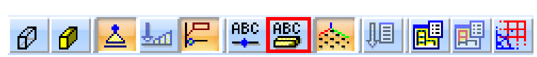
A column and a roof girder have one common node. The end node of the column (for instance) is the begin node of the roof girder. This girder is connected to the column automatically.
The two girders modelled in the middle of columns are not touching the column in nodes. The end nodes of the beams are located in-between the column nodes and therefore are not yet connected to the columns. In this paragraph, we will explain how to connect the bars to each other. It might be especially important for future editing and smooth calculation.

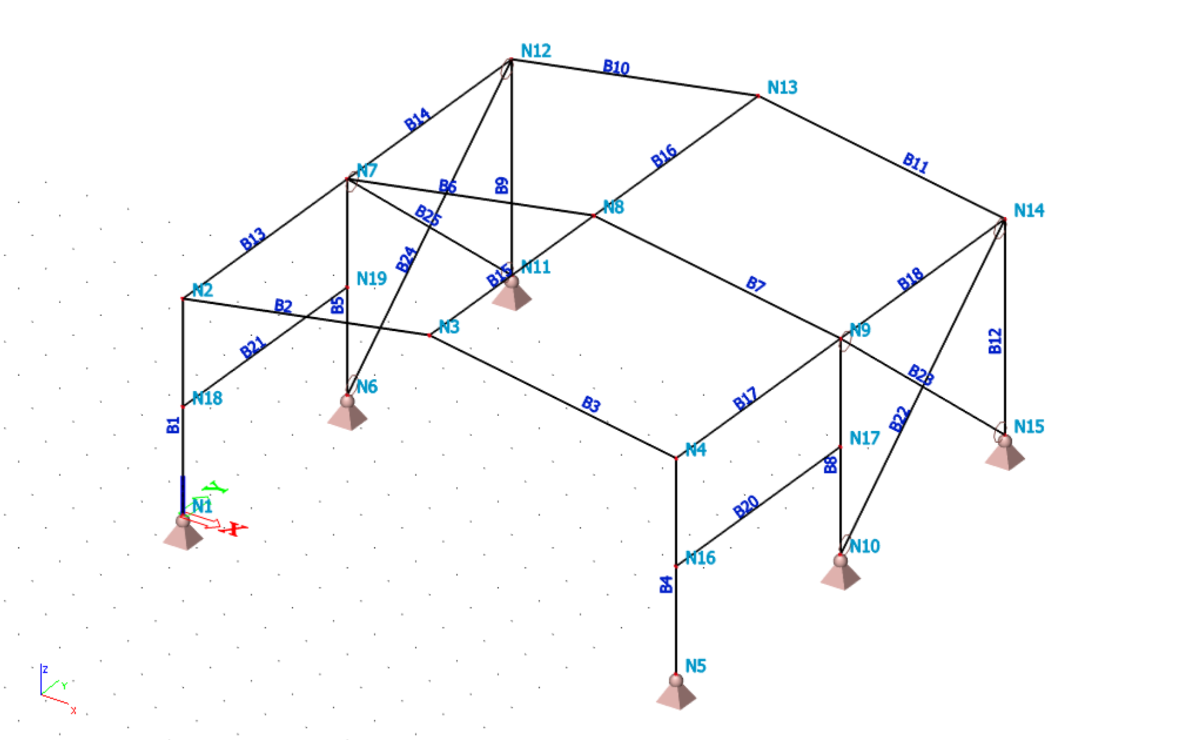
To display the names of the bars and nodes, you can activate the labels by means of the buttons in the **Command line.**

Activating node labels

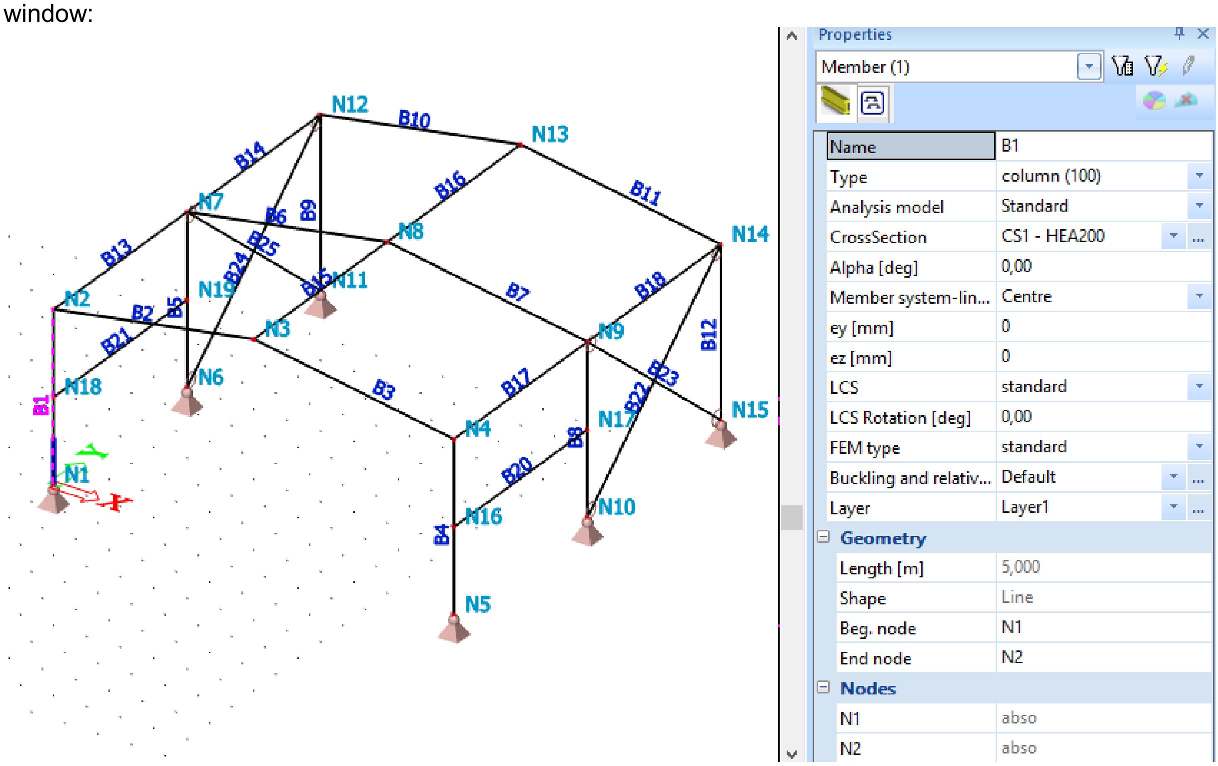
Node labels are activated by means of the  icon at bottom of the modelling window.

Activating member labels

Member labels are activated by means of the  icon at bottom of the modelling window.



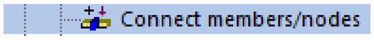
When you select column **B1** with the left mouse button, the properties are displayed in the **Properties** window:



This window indicates that the start node is **N1** and the end node **N2**. Node **N18** is not part of the column. To connect beam **B21** to the columns, use the **Connect members/nodes** option.

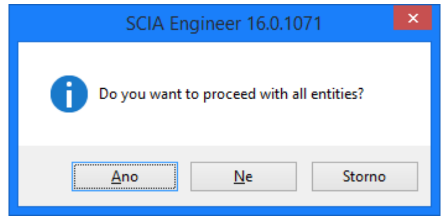
Connecting entities

1. Press **<ESC>** or click the **Cancel selection** icon to deactivate any selection of entities.

2. Double-click on the **Model data > Connect members/nodes** option in the **Structure** service 

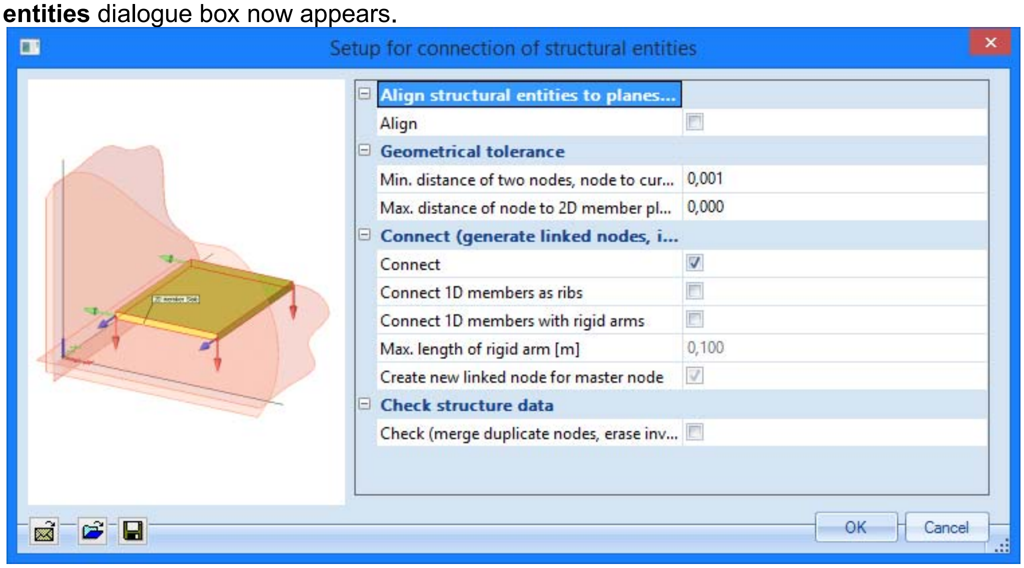
or click the  icon in the toolbar.

3. A dialogue asks if all nodes must be connected to bars:



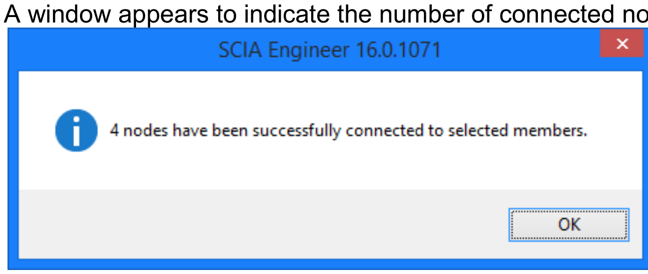
4. Click **<Yes>**.

5. The **Setup for connection of structural entities** dialogue box now appears.

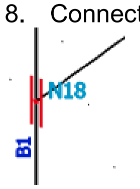


6. Confirm the settings by clicking **<OK>**.

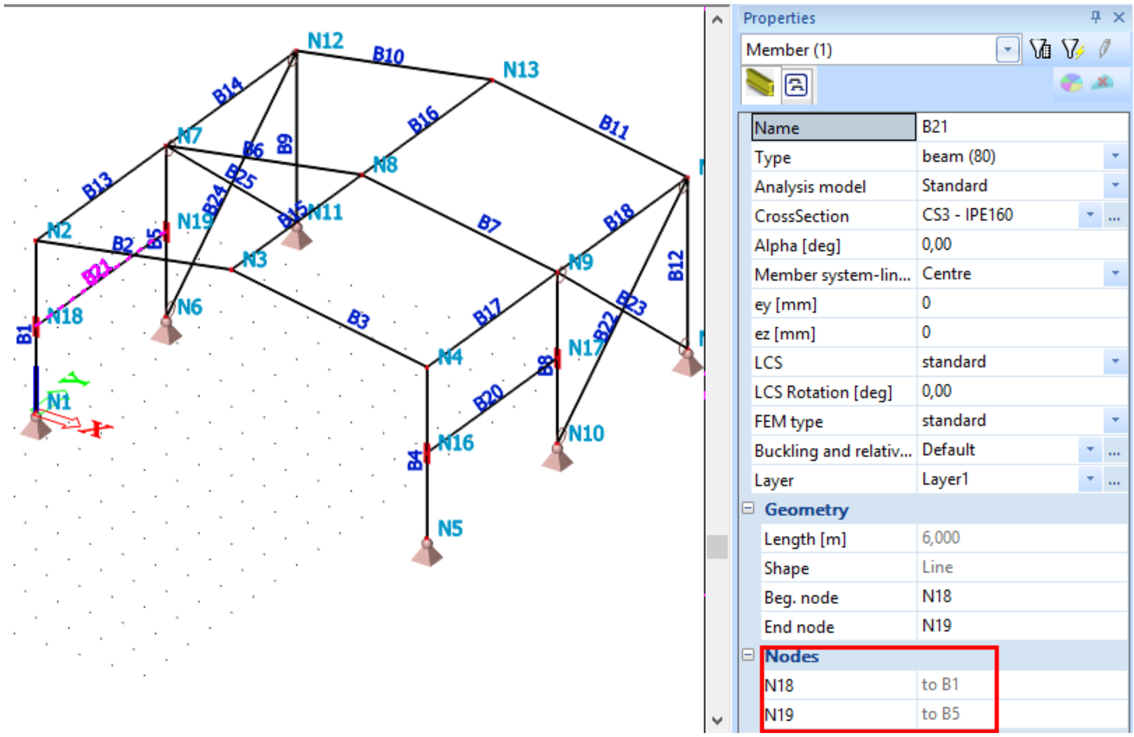
7. A window appears to indicate the number of connected nodes:



8. Connected nodes are represented in the graphical screen by means of double red lines:



When you select for instance girder **B21**, the **Properties window** will show that node **N18** connects the girder with column **B1** and that node **N19** connects the girder with column **B5**.



Note:
*If a possible active selection is not deactivated when the **Connect members/nodes** command is used, program will only search the nodes to be connected in this selection only and not in the entire structure. It is also possible to run the two previous operations at once. Therefore you have to check the option **Check (merge duplicate nodes, erase invalid entities)** in the **Setup for connection of structural entities** dialogue box.*

9. Click **[Close]** below the **Structure** menu to return to **Main** tree.

Graphic representation of the structure

Edit view

Within SCIA Engineer there are several possibilities to edit the graphic representation of the construction. Below you will find the most important options:

- Edit the view point on the model
- Set a view direction
- Use the magnifier
- Edit view parameters through the menu **View parameters**

Editing the view point on the model

Set view point through the wheels. Bottom right of the graphic window there are three wheels; two are horizontal and one is vertical. With these **wheels** you can **zoom in** on the construction or **turn** it.

1. To be able to zoom in on the construction or to turn the model, click on the wheel (the cursor will change into a hand), keep the left mouse button pressed and move the wheel

OR

Set the view point by combining the buttons and mouse:

2. Press CTRL + right mouse button at the same time and move the mouse to **turn** the construction.
3. Press SHIFT + right mouse button at the same time and move the mouse **move** the construction.
4. Press CTRL + SHIFT + right mouse button at the same time and move the mouse to **zoom in** or **out** on the construction.

Remark:

If the structure is being turned while a node is selected, the structure will turn around the selected node.
*You can also easily **zoom in** and **zoom out** with the mouse wheel. The same mouse wheel can be used to **move** the model in case you press it and hold. Double-click of the wheel zooms the structure so that it can be seen completely (the whole modelling windows is filled by the structure).*

Setting a view direction with regard to the global coordinate system

1. Click on the button **View in direction X**  for a view in the X-direction.
2. Click on the button **View in direction Y**  for a view in the Y-direction.
3. Click on the button **View in direction Z**  for a view in the Z-direction.

Remark:

*You can also type the letter X, Y, or Z into command line and click **<Enter>** to activate the view in desired direction.*

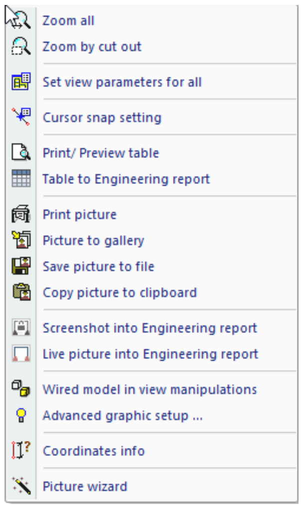
The magnifier

- Use  to enlarge.
- Use  to decrease.
- Use  to zoom in on a window.

- Use to view the whole structure.
- Use to zoom in on the selection of modeling entities.

Editing view parameters through the menu View parameters

Click in the graphic window on the right mouse button. The following shortcut menu appears:



Remark:

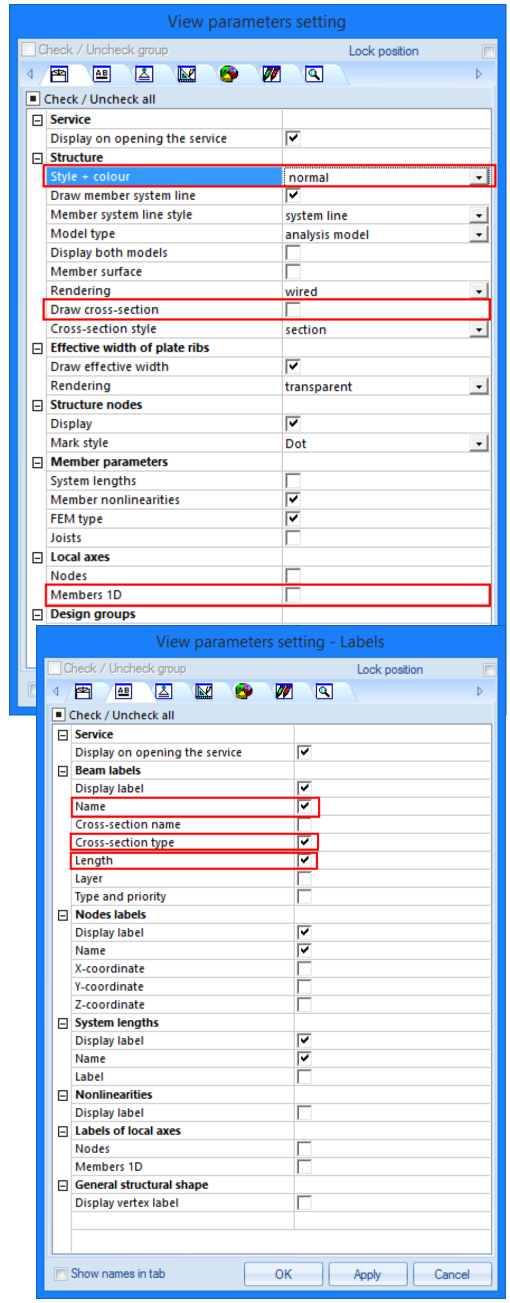
If an entity was selected previously, you can define a setting that only applies to the selected elements. (An adapted shortcut menu appears).

Choose the option **Set view parameters for all**. The window **View parameter setting** appears. The menu consists of various tabs for various data. You can set the view parameters for all entities or just for the selected entities.

View parameters – Structure

Through the tab Structure the representation of the different entities can be adapted. In the group **Structure** the following items are important for this project:

- **Style and colour:** You can display the colour per layer, material, cross-section, structural type or design group.
- **Draw cross-section:** With this option checked the symbol of the cross-section is displayed on every 1D member.
- **Local axes:** With this option the local axes of the elements are activated.



View parameters – Labels

Through the tab **Labels,** the labels of different entities can be displayed. In the group **Beam labels** the following items can be displayed in the label:

- **Name:** Show the name of the cross-sections in the label (e.g. CS.)
- **Cross-section type**: Show the cross-section type in the label (e.g. Rectangle (500; 500)).
- **Length**: show the length of the member in the label (e.g. 6,000 m).

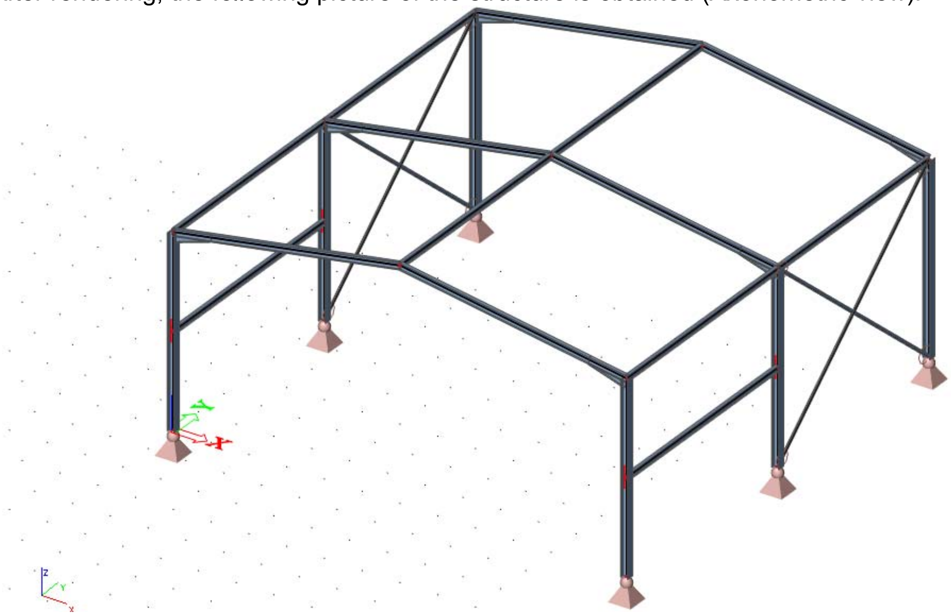
View parameters – shortcuts

In the tool bar above the **Command line**, several frequently used options are grouped among which:

- **Show/hide surfaces** to show the surfaces of the cross-sections.
- **Render geometry** to view the rendered members.
- **Show/hide supports** to show supports and hinges.
- **Show/hide load** to show the load case.
- **Show/hide other model data** to show other model data (like hinges, internal nodes, …).
- **Show/hide node labels** to view the label of the nodes.
- **Show/hide member labels** to view the label of members.
- **Set load case for view** to edit the active load case.

- **Fast adjustment of view parameters on the whole construction**  to quickly access to the options from the menu View parameters.

After rendering, the following picture of the structure is obtained (Axonometric view):



Loads and combinations

Load Cases and Load Groups

Each load is attributed to a **load case**. One load case can contain different load types. To each load case, properties are attributed which are determinant for the generation of combinations. The action type of a load case can be permanent or variable.
Each variable load case is associated with a **load group**. The group contains information about the category of the load (service load, wind, snow…) and its appearance (default, together, exclusive). In an exclusive group, the different loads attributed to the group cannot act together in a single combination. For default **combinations**, on the other hand, the combination generator allows the simultaneous action of the loads of a same group.
The way in which load cases are defined is decisive for the load combinations created by the generator. We recommend that you thoroughly read the chapter about loads and combinations in the reference manual.

Two load cases are entered in this project:

LC1: Permanent Load Case: Self weight of the bars + Roof weight
LC2: Variable Load Case: Side wind on the frames

Defining a Permanent Load Case

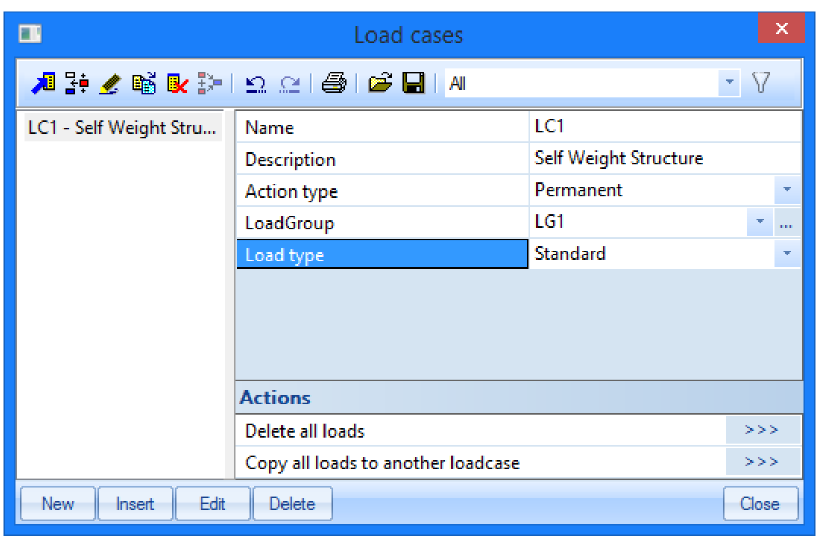
1. Double-click on Load in the **Main tree**.

2. Before you can define loads, you must enter load cases first. Since this project does not contain any load cases yet, the **Load Cases** manager will automatically appear.

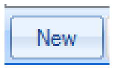
3. By default, the load case named **LC1** is created. This load is a permanent load of the **Self Weight** load type. The self weight of the structure is automatically calculated, although not graphically displayed.

4. Since you will also manually enter loads in the first load case of this project (Roof Weight), you must change the Load Type to **Standard**.

5. In the Description field, you can describe the content of this load case. For this project, enter the description **"Self Weight Structure"**.

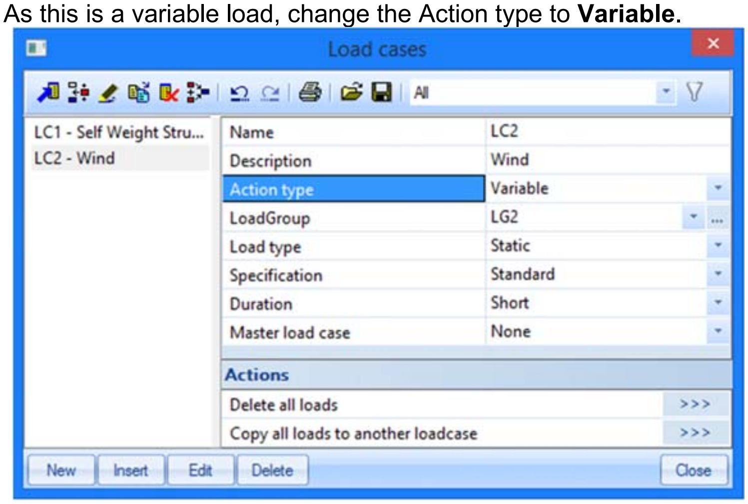


Defining a Variable Load Case

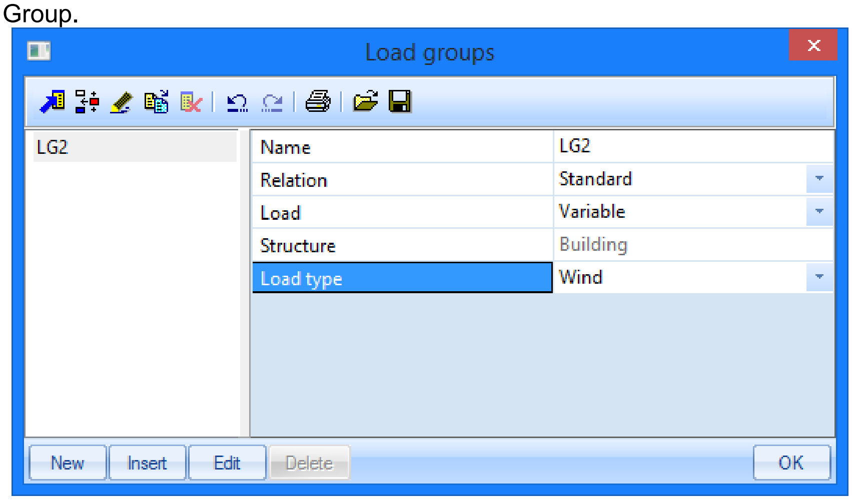
1. Click  or  to create a second load case.

2. Enter the description **"Wind"**.

3. As this is a variable load, change the Action type to **Variable**.



4. The Load Group LG2 is automatically created. Click to display properties of the Load Group.



The Load type determines the composition factors that are attributed to the load cases in this load group. In this project choose **Wind**.

5. Click **[OK]** to close the **Load group manager** and to return to the **Load cases** manager.

6. Click **[Close]** to close the **Load cases** manager.

Remark:
Load groups

Each load is classified in a group. These groups influence the combinations that are generated as well as the code-dependant coefficients to be applied. The following logic is adopted.
Variable load cases that are independent from each other are associated to different variable groups. For each group, you set the load category (see EN 1991). The combination factors from the Eurocode are generated from the available load groups. When a generated combination contains two load cases belonging to different groups, reduction factors will be applied for the transient loads.

If the load is divisible, its different components are entered as individual load cases. As long as the load combination does not contain any variable load belonging to another group, no reduction factors may be applied. The different load cases of a divisible load are therefore associated to one variable group. Load cases of the same type that may not act together, are put into one group, which is made exclusive, e.g. "Wind X" and "Wind -X" are associated to one exclusive group "Wind" to avoid simultaneous action.

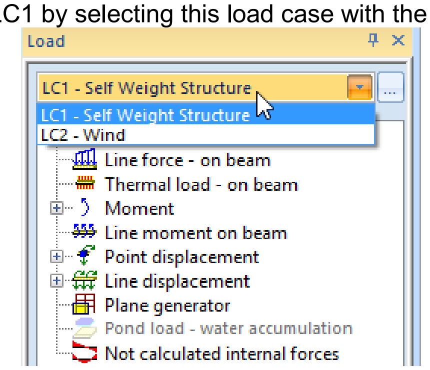
Loads

After input of the Load cases, the **Loads menu** will automatically appear:

The first load case (**LC1**) includes two loads:
- Self weight of the bars
- Roof weight

Switching between load cases

Activate LC1 by selecting this load case with the mouse pointer in the combo-box:

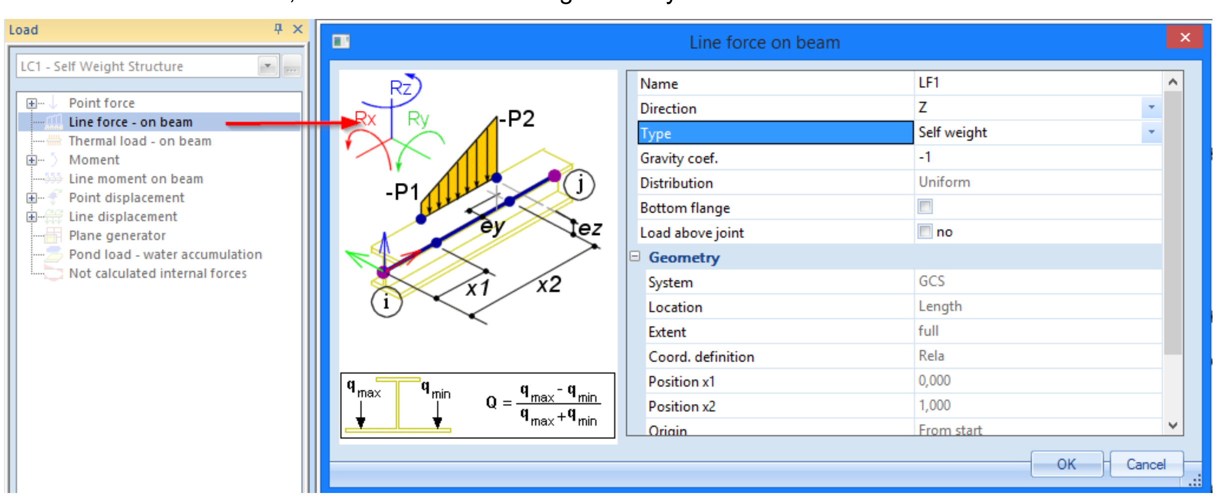


Entering the self weight as linear load

1. Cancel any possibly active selection by pressing **<ESC>**.

2. Click on **Line Force - on beam** in the **Loads** menu. The dialogue **Line Force on beam** appears.

3. In the field **Type**, choose **Self Weight.** The Direction is the global Z-direction and the Gravity coefficient is set to –1, so that the load is acting vertically downwards.



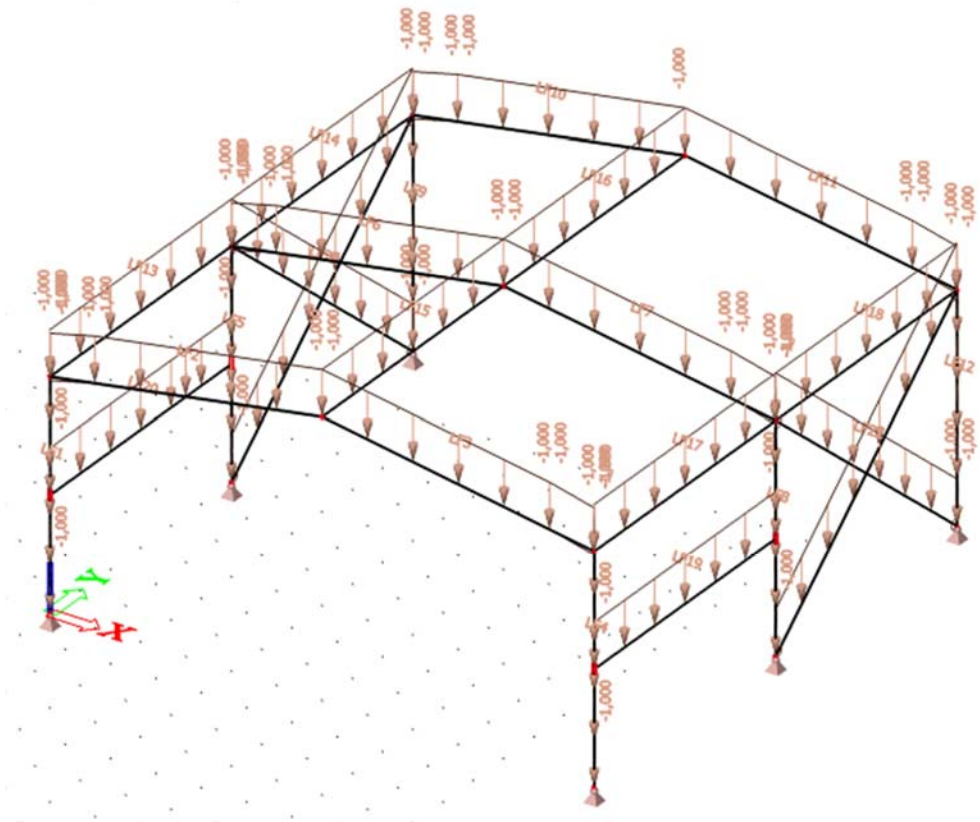
4. Confirm your input with **[OK]**.

5. Select all the bars by means of the **Select all**  icon in the toolbar.

6. Press **<ESC>** to finish the input.

7. Press **<ESC>** once more to finish the selection.

The self weight load is represented in brown:



Next, the roof weight is entered as a surface load of 1,5 kN/m². Only the roof girders are loaded directly.

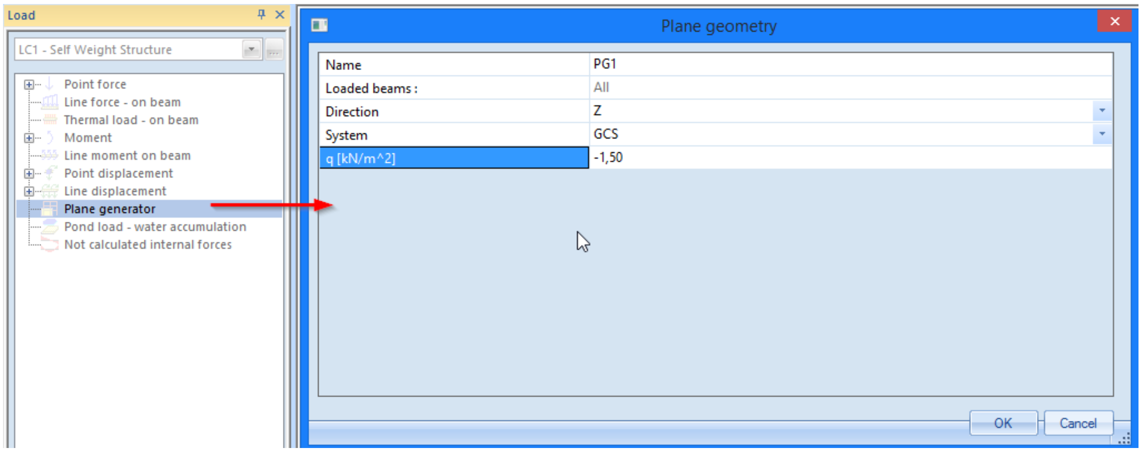
Entering the roof load as a surface load

In order to enter a **Surface load on beams**, an option called **Plane generator** will be used. It enables us to insert plane load in kN/m^2 even though there is no plate (2D) member. And program redistributes this load into linear load in kN/m.

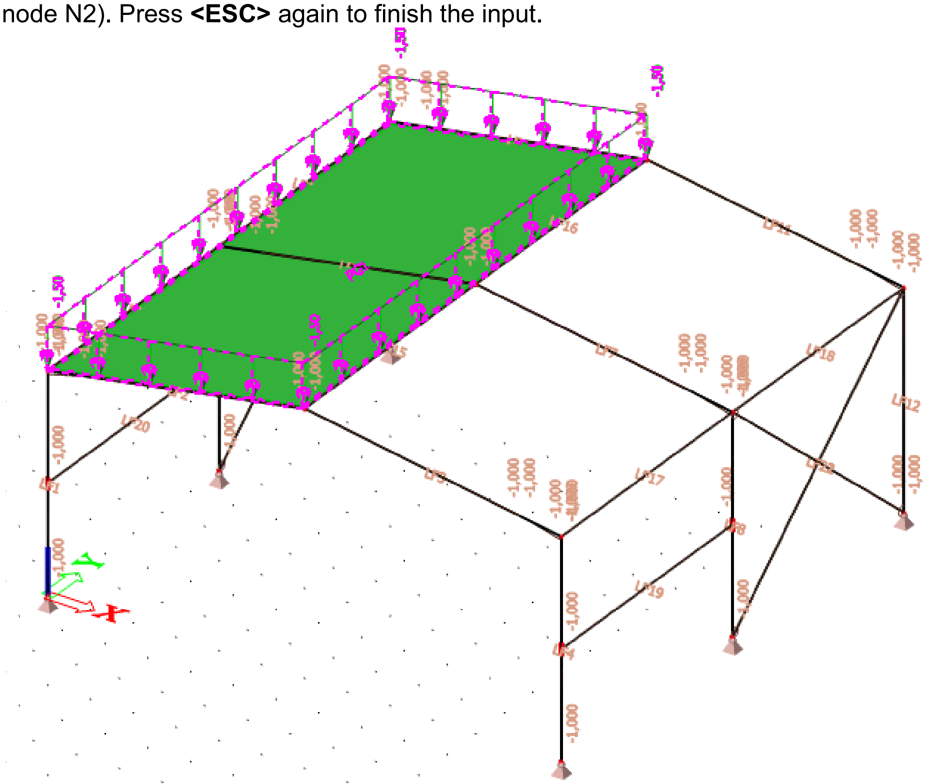
1. Click on **Plane generator** in the Load menu. The **Plane geometry** dialogue appears

2. The load **Direction** is **Z** and the **System** is the global coordinate system **GCS**. In this way, the load acting vertically downwards.

3. Change the **Value** to **–1,5 kN/m²** and press **[OK]**.



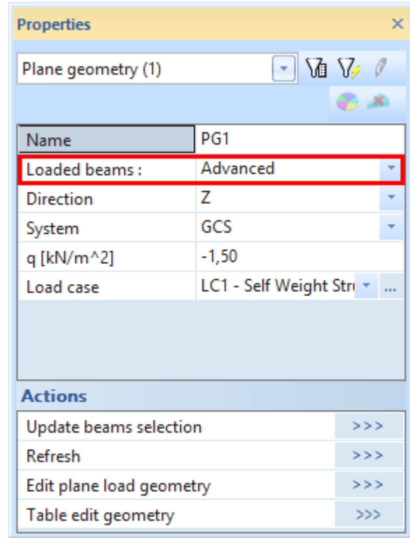
4. Click on nodes N2, N3, N13 and N12 consecutively to define the rectangle on one of the roof planes. Press **<ESC>** and the rectangle will be finished (the last node N12 is linked to the first node N2). Press **<ESC>** again to finish the input.

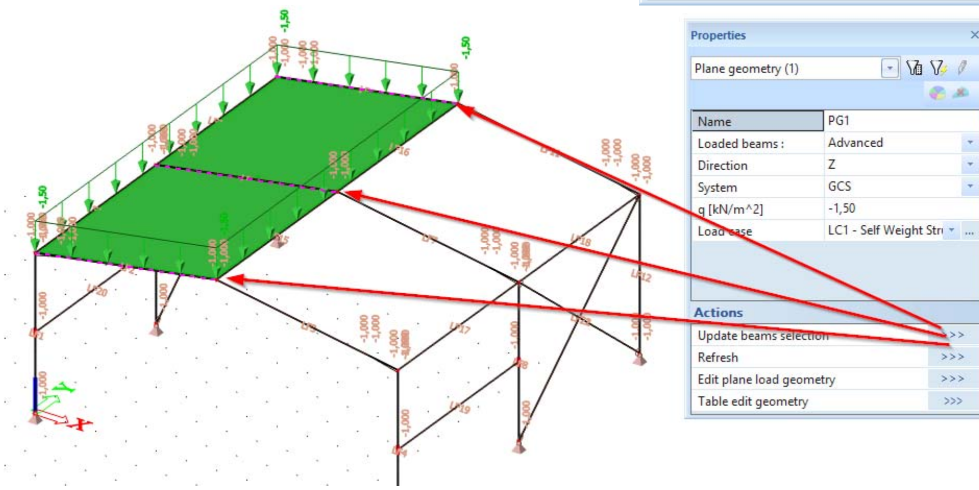


5. The load stays in violet colour which means it is still in active selection. Click on the combo-box in the **Properties window** > **Loaded beams** and change the option from **All** to **Advanced.**

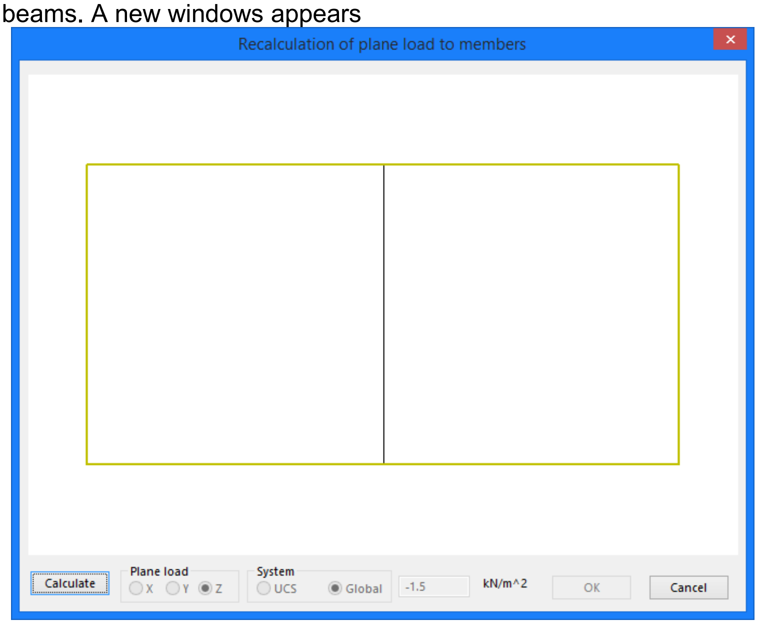
6. In the **Actions** buttons choose the option **Update beams selection** to indicate that only the girders and not the longitudinal beams will be loaded directly by this surface load.

7. Click on beams **B2 ,B6** and **B10.**

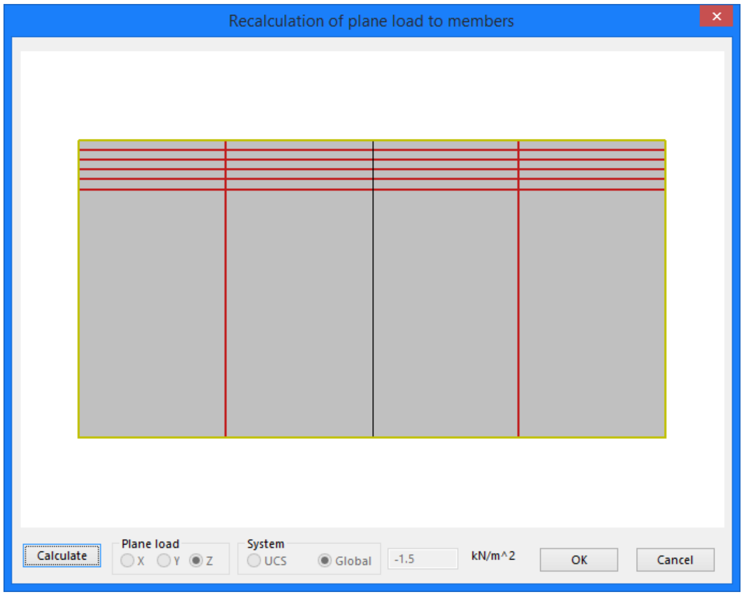




8. Press **<ESC>** to confirm the selection.
9. In the **Actions** buttons hit **Refresh** to generate/recalculate the surface load to line loads on beams. A new windows appears

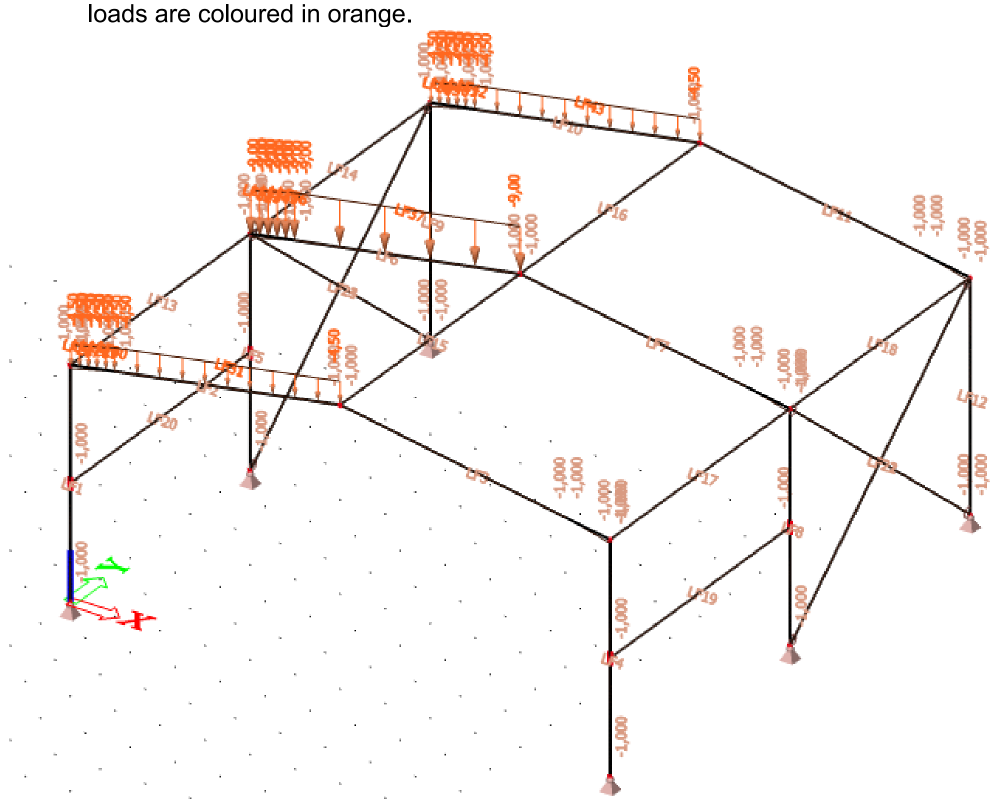


10. The **Recalculation of plane load to members** – window appears. Hit **Calculate** button in bottom left corner.

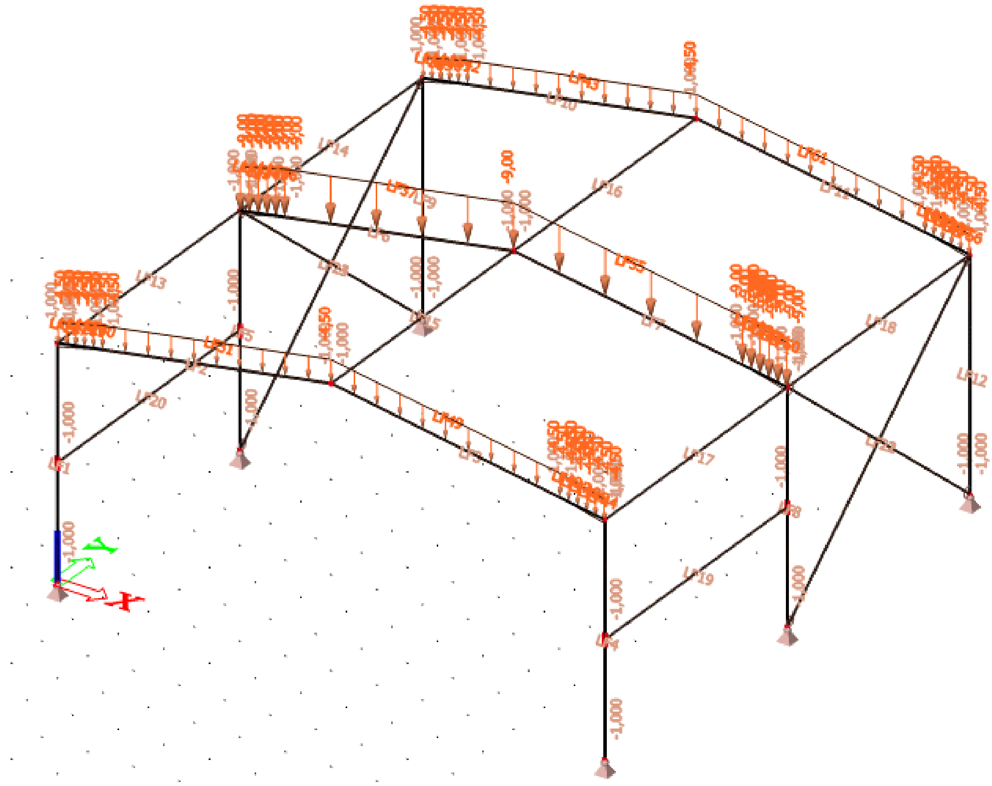


You will see a refinement of the calculated line loads at the beginning of the girders. This happens because of the presence of the haunches. Haunches are refined automatically.

11. Click <**OK**> to confirm. You will see now the recalculated line loads on beams. Recalculated loads are coloured in orange.

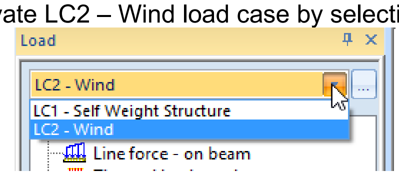


Repeat steps above to procedure identical load on the right-sided roof plane.



Entering roof load as second load case

Activate LC2 – Wind load case by selecting it with the mouse pointer in the combo-box:



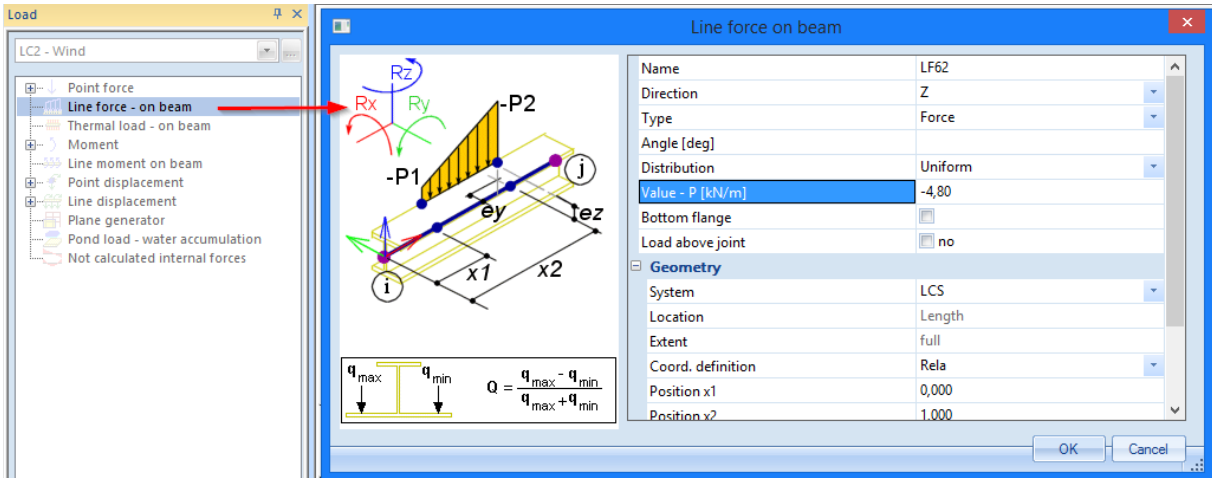
Entering a linear load

1. Click on **Line Force - on beam** in the **Load Menu**. The dialogue **Line Force on beam** appears.

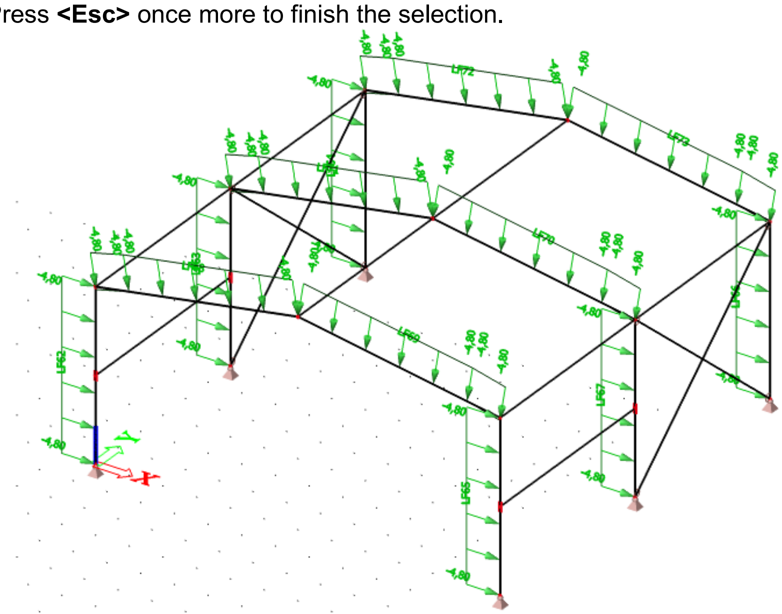
2. Change the **Type** to **Force.**

3. The load **Direction** is **Z** and the **System** is the local coordinate system **LCS**. The linear loads are acting in accordance with the local Z-axes of bars.

4. Change the **Value** to **-4,8 kN/m.**



5. Confirm your input with **[OK]**.
6. Select the bars where this load must be positioned: the roof girders and the columns.
7. Press **<Esc>** to finish the input.
8. Press **<Esc>** once more to finish the selection.

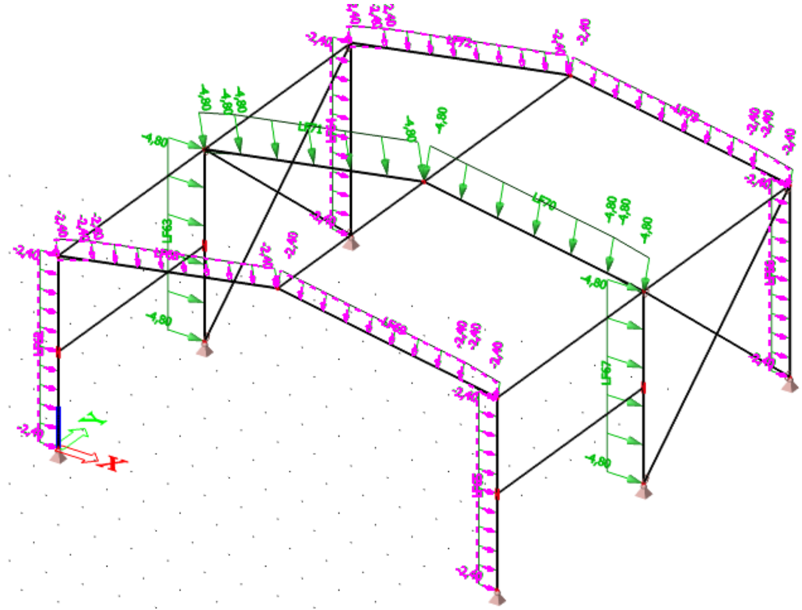


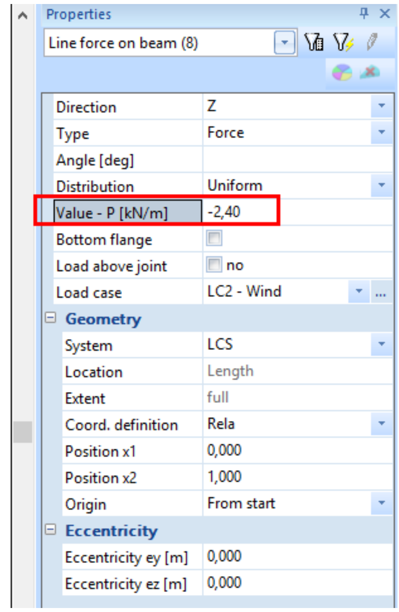
Adapting a load

1. Select the linear loads on the roof girders and the columns of the first and last frame by clicking with the left mouse button on these loads. Mentioned members must have only half of the load value because only half of the loading width, affected by wind, belongs to them.

2. The common properties of the 8 series are displayed in the **Properties window**.

3. Change the **Value** from **–4,8 kN** to **–2,4 kN** in the Properties window.



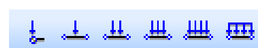


4. Confirm the modification with **<ENTER>**.

5. Press **<ESC>** to finish the selection.

Click **[Close]** to quit the **Loads menu** and to return to the **Main tree**.

Note:

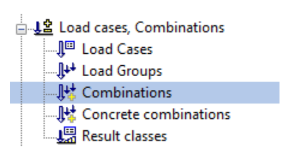
*The **Command line** includes a number of predefined loads:* *, which enable a fast and simple input of loads.*

Combinations

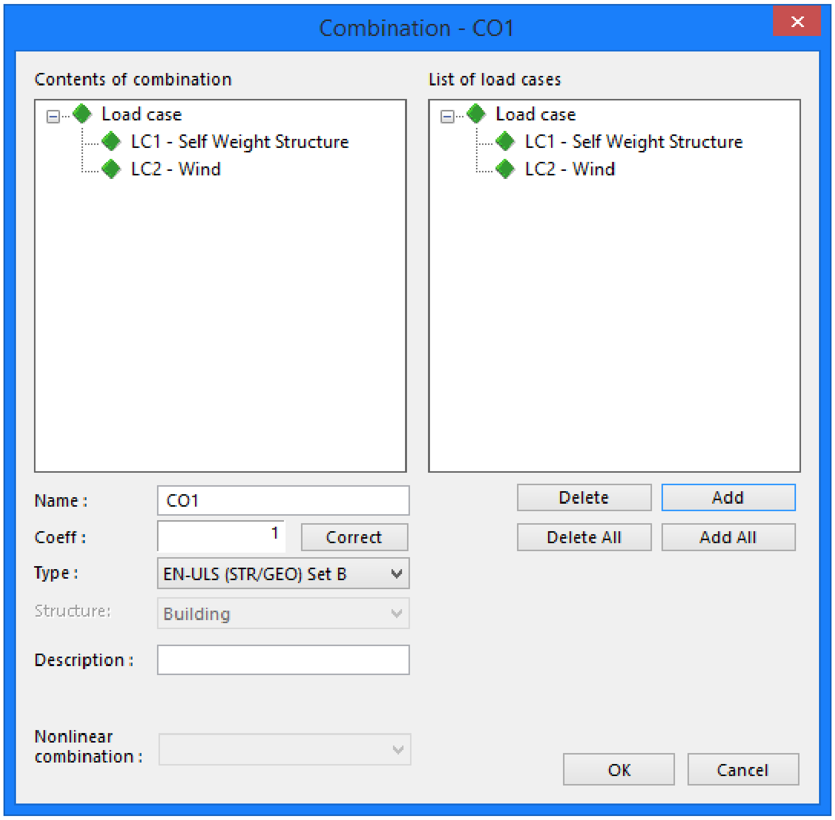
After input of loads and load cases, the latter can be grouped in combinations. In this project, two code combinations are created, one for the Ultimate Limit State and one for the Ultimate Serviceability State.

Defining Combinations

1. Double-click on Combinations in the **Main tree**.

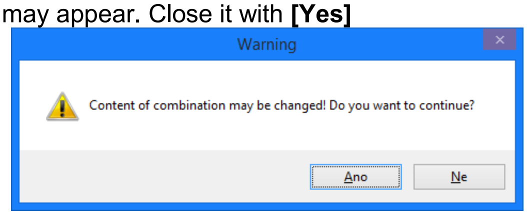


2. Since no combination has been entered yet, the window to create a new combination automatically appears.



3. The Type of the combination is changed to **EN – ULS (STR/GEO) Set B**. With this envelope combination type SCIA Engineer will automatically generate linear combinations in accordance with the complex composition rules of the Eurocode.

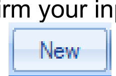
4. A warning message that controls the content of code combinations with respect to load type may appear. Close it with **[Yes]**



5. With the button **[Add all]**, all load cases can be added to the combination. Otherwise you can manually drag&drop load cases from the list of load cases (right frame) to the contents of combinations (left frame).

6. Type “ULS” into Description row to distinguish the combination from the second one.

7. Confirm your input with **[OK]**. The **Combinations manager** is opened.

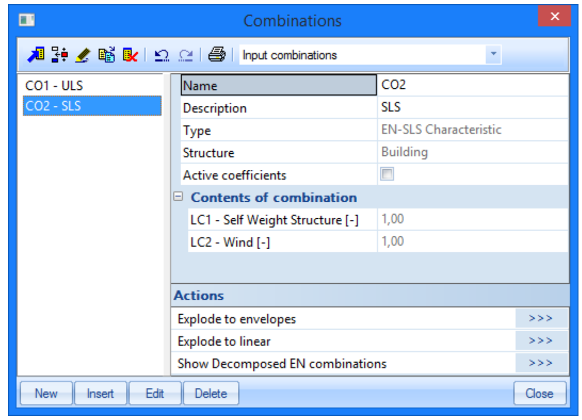
8. Click 

or  to create a second combination.

9. Change the **Type** of the combination to **EN-SLS Characteristic**. Type “SLS” into Description row to distinguish the combination from the first one.

10. Confirm your input with **[OK]**.

11. Click **[Close]** to close the **Combination manager**.



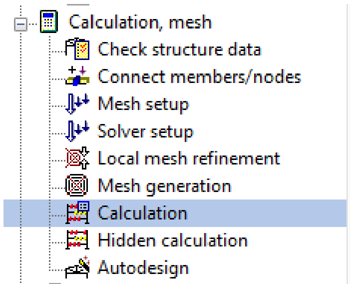
Calculation

Linear Calculation

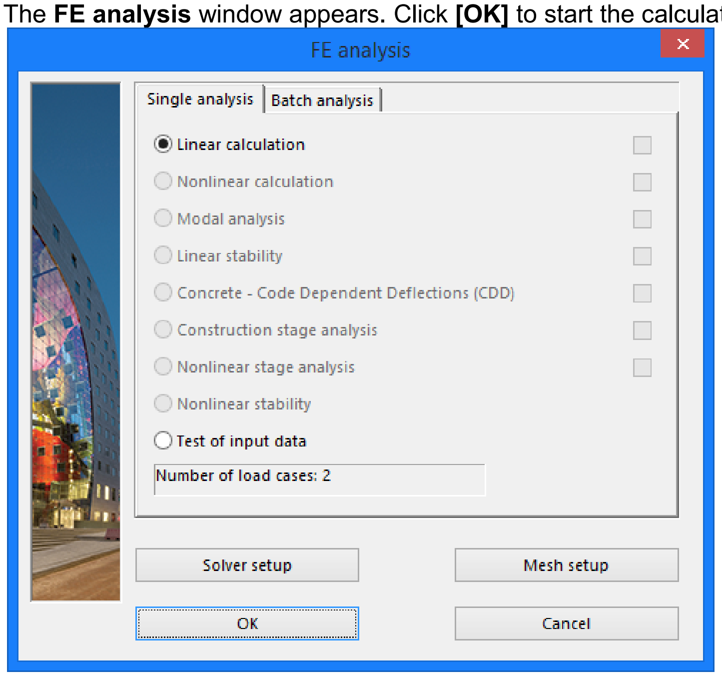
As the calculation model is completely ready, you now can start the calculation.

Executing the Linear Calculation

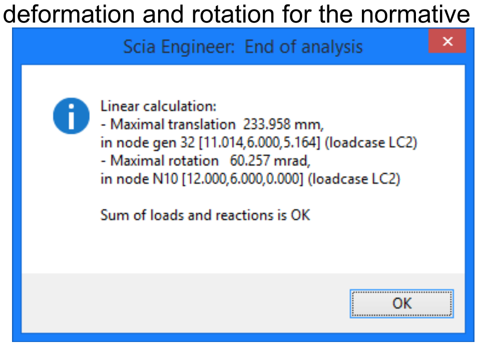
1. Double-click on Calculation in the **Main window,** or use identical icon  in toolbars.



2. The **FE analysis** window appears. Click **[OK]** to start the calculation.



3. After the calculation, a window announces that the calculation is finished and the maximum deformation and rotation for the normative load case is shown. Click **[OK]** to close this window.



Results

Viewing results

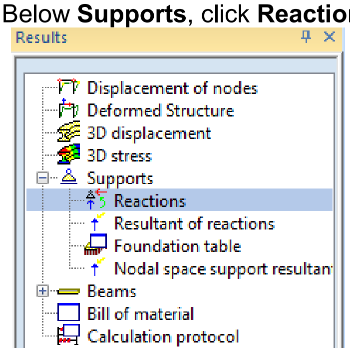
When the calculation is completed, results can be viewed. New service appears in the **Main** tree and also Properties window announces that Linear calculation is finished.

Viewing the Reaction Forces

1. Double-click on 

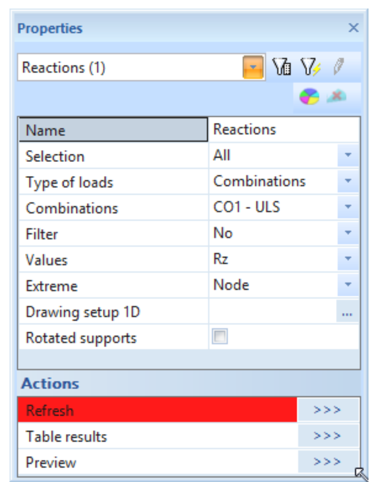
in the **Main tree**. The **Results menu** appears.

2. Below **Supports**, click **Reactions**.



3. The options in the **Property Window** are configured in the following way:

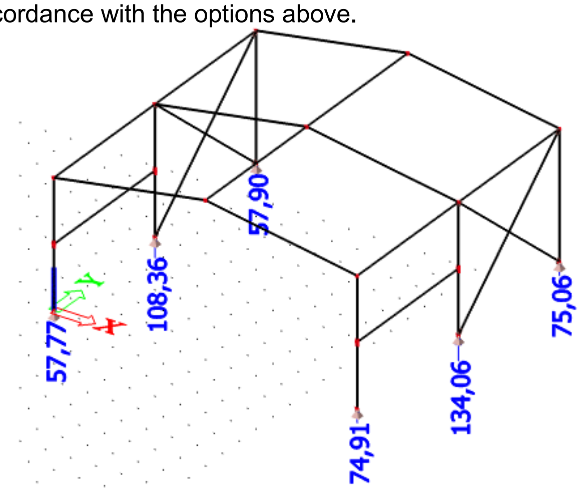
- The Selection field is set to **All**.
- The Load type is set to **Combinations** and the Combination to **CO1 - ULS**.
- The **Values** are wanted for **Rz**.
- The **Extreme** field is changed to **Node**.



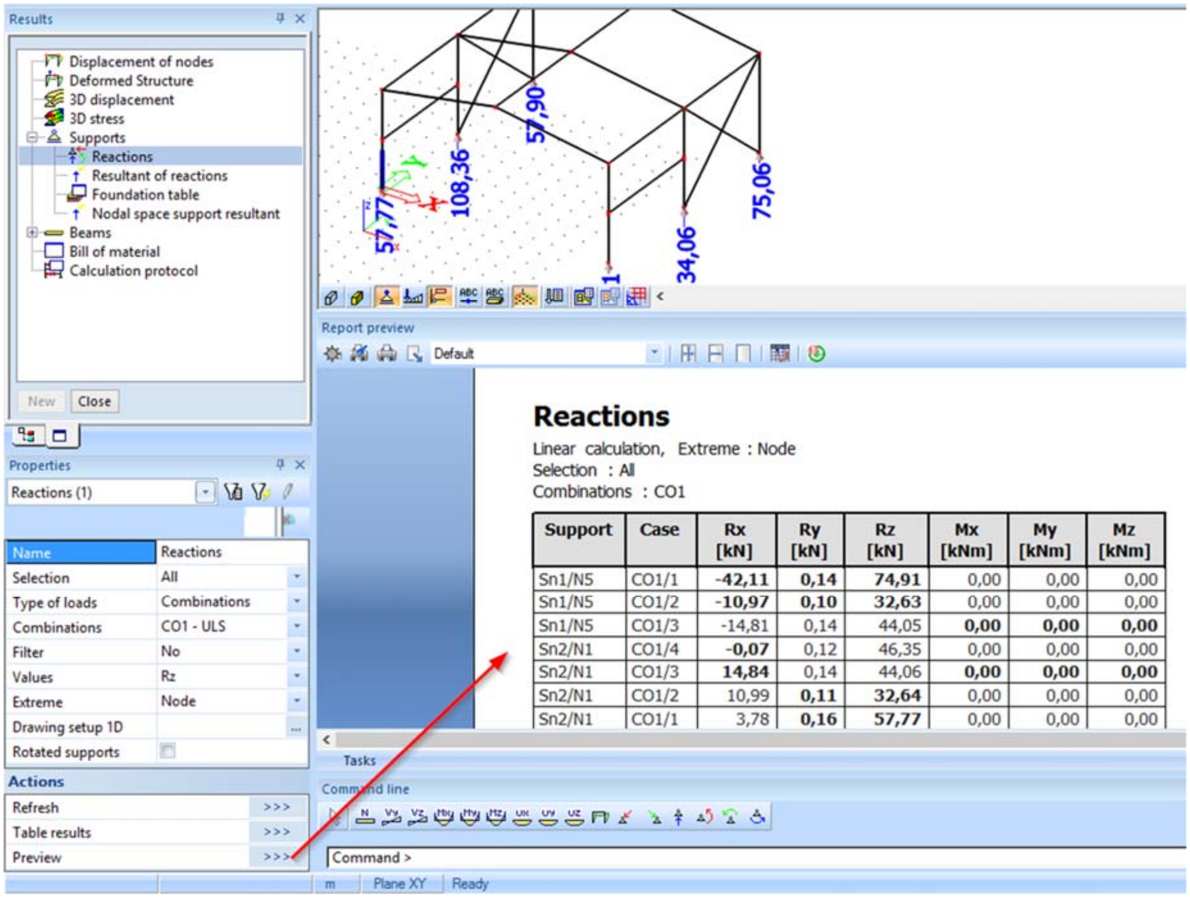
4. The action button **Refresh** has a red highlight, i.e. the graphical screen must be refreshed.

Click on the 

button next to **Refresh** to display the results in the graphical screen in accordance with the options above.

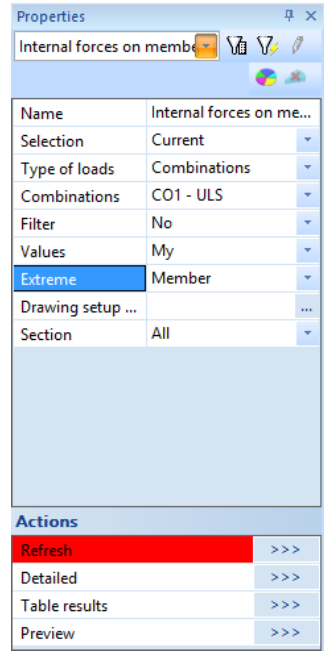


5. To display these results in a table, the **Preview** action is used. Click on the >>> next to **Preview** to open Report preview.



Note:
The Report preview appears between the Graphical Screen and the Command line. This screen can be maximised to display more data at once.

Viewing internal forces on beam

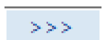


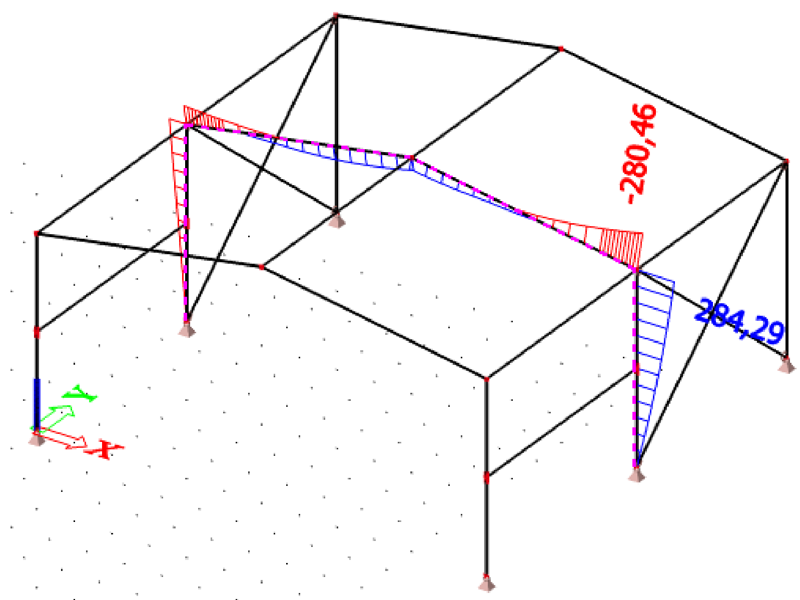
1. In the **Results** menu, open the **Beams** group and select **Internal forces on beams.**

2. The options in the **Property Window** are configured in the following way:

    - The Selection field is set to **Current**.
    - The Load type is set to **Combinations** and Combination to **CO1 - ULS**
    - The Values are wanted for **My**.
    - The Extreme field is changed to **Global**.

3. Select columns and the roof girders of the centre (middle) frame using the left mouse button.

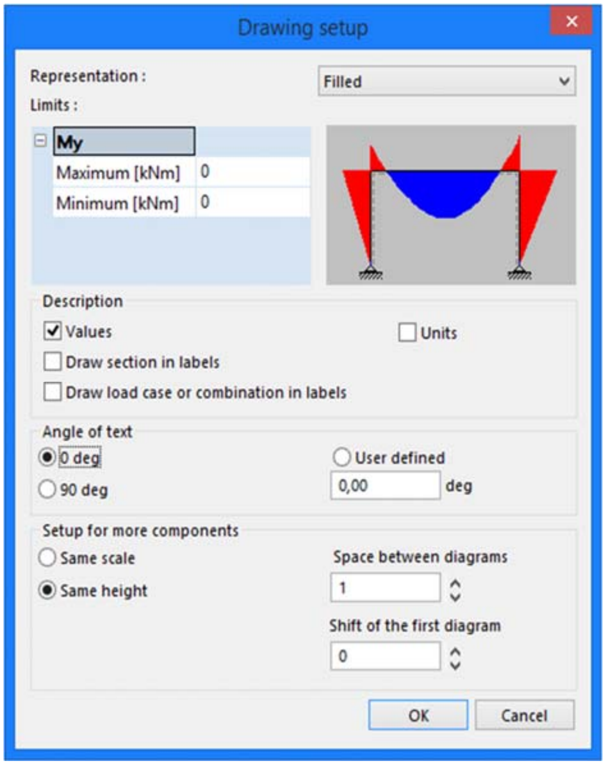
4. Click on the  button next to **Refresh** to display the results on the graphical screen in accordance with the set options.



To change the display of the results, the settings of the Graphical Screen can be adapted, as described in the following chapter.

Configuring the Graphical Screen

1. In the **Properties** window, click the icon next to **Drawing Setup**. The options for the graphical screen are opened.

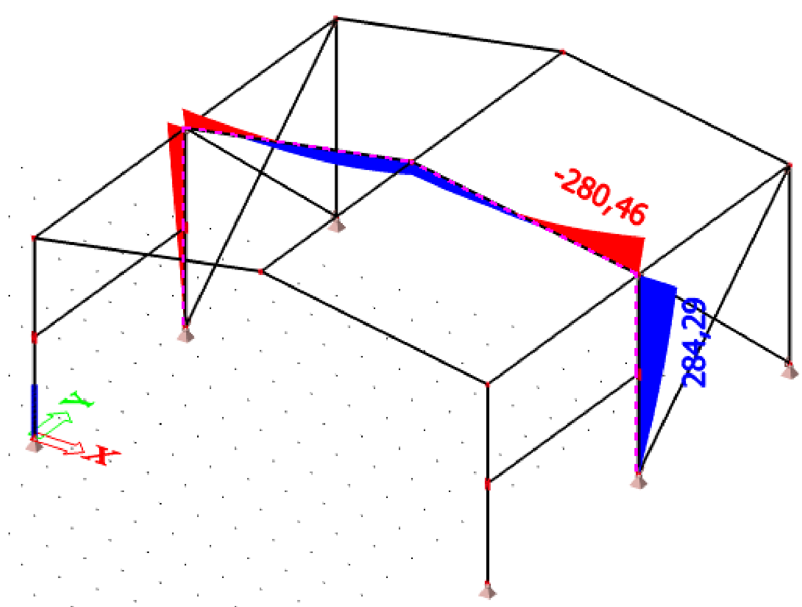


2. In the **Representation** field, choose **Filled.**

3. The **Angle of text** is set to **0°.**

4. Click **[OK]** to confirm your input.

5. In the **Property** window, click the >>> button next to **Refresh** to display the results in the graphical screen in accordance with the set options.



6. Then click **[Close]** to leave the **Results Menu**.

7. Press **<ESC>** to cancel the selection.

Note:

To change the font size of the displayed results, you can use the ***Setup > Fonts*** *menu. In this menu, the different sizes of the displayed labels can be changed.*

Code check

SCIA Engineer steel modules include a number of powerful tools to perform the steel calculations in accordance with the chosen design code.
The possibilities are as following:

- Input of advanced steel data per member
- Simple input and edition of buckling data
- Input of reinforcements, stabilities against lateral-torsional buckling, cladding
- Unit check of the cross-section
- Optimisation of the cross-section
- Fire-resistance check of a member
- Input and calculation of frame connections
- Input and calculation of diagonal connections
- Automatic generation of sectional drawings
- Automatic generation of assembly drawings and anchorage plans
- Relative deformation unity check
- etc.

We will only explain the basics of the steel design in this Tutorial. For more information regarding advanced steel calculations we refer to the Advanced Steel Training.

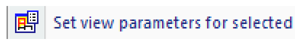
Before the steel calculations can be started, the buckling parameters of the members need to be checked. By means of the view parameters, the buckling lengths of the members can be visualised.

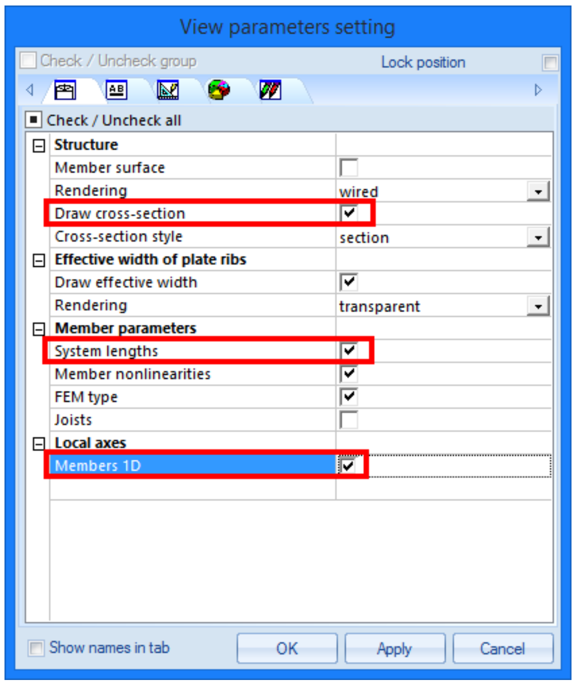
Buckling parameters

Displaying the system lengths

1. Select with the left mouse button column **B1**, the left column of the first frame.

2. Click the right mouse button at an arbitrary position in the workspace. Context menu lists the possibilities for the selected entity.

3. In this menu select the  option. Limited **View parameter settings** window appears.

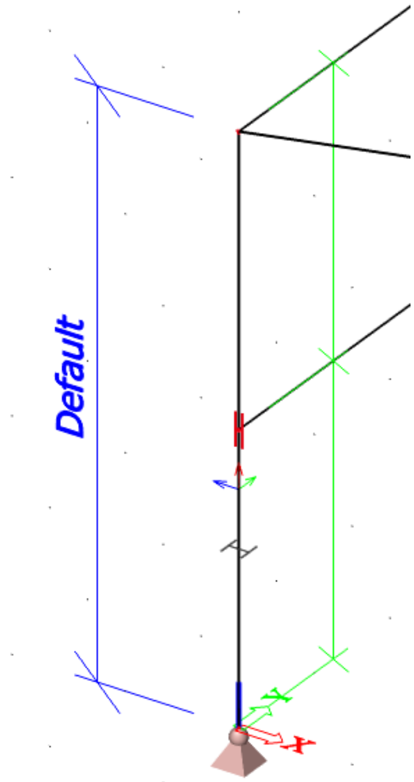


4. Activate the **System lengths** and **Draw cross-section** options to display the reference lengths and the section of the bar.

5. Activate the **Local axes – Members 1D** option to display the local coordinate system of the bar.

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

7. Press **<ESC>** to cancel the selection.

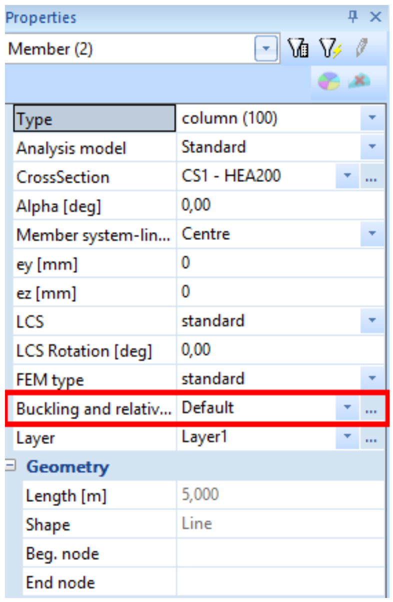


The figure shows that (default) system length Ly (blue line) for buckling around the strong axis (y-y) is total height of the column and Lz (green line) for buckling around the weak axis (z-z) is half of the height. The girder in the middle of the column therefore supports the column for buckling around the weak axis, i.e. for bending in the Y direction.
To modify the buckling data of a bar use the option **Buckling and relative lengths** in the **Property window** of the beam, if selected.

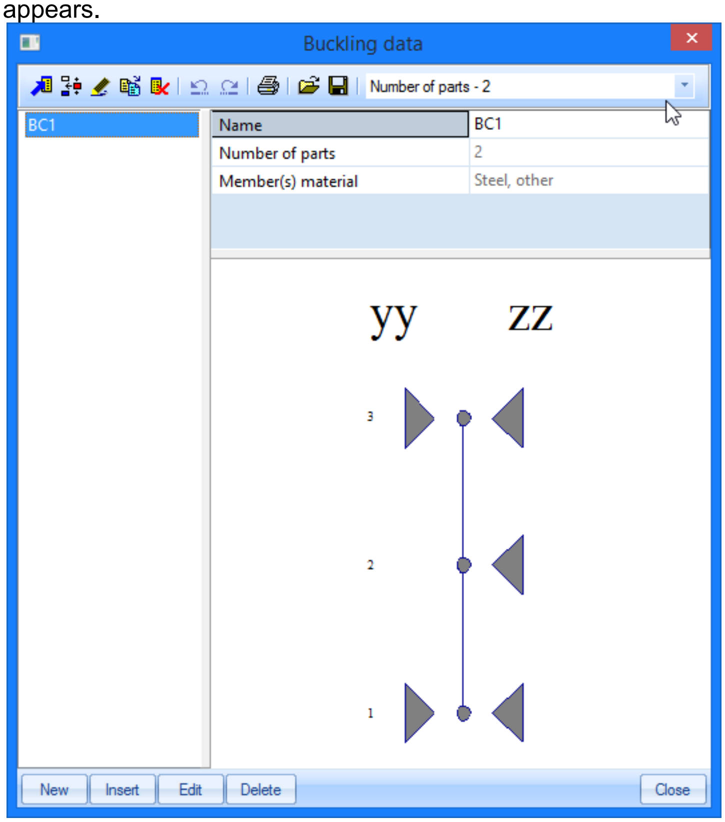
Setting the Buckling Parameters

1. Select both columns of the first frame with the left mouse button.

2. The **Properties window** shows the common properties of both entities. The **Buckling and relative lengths** are set to **Default**.

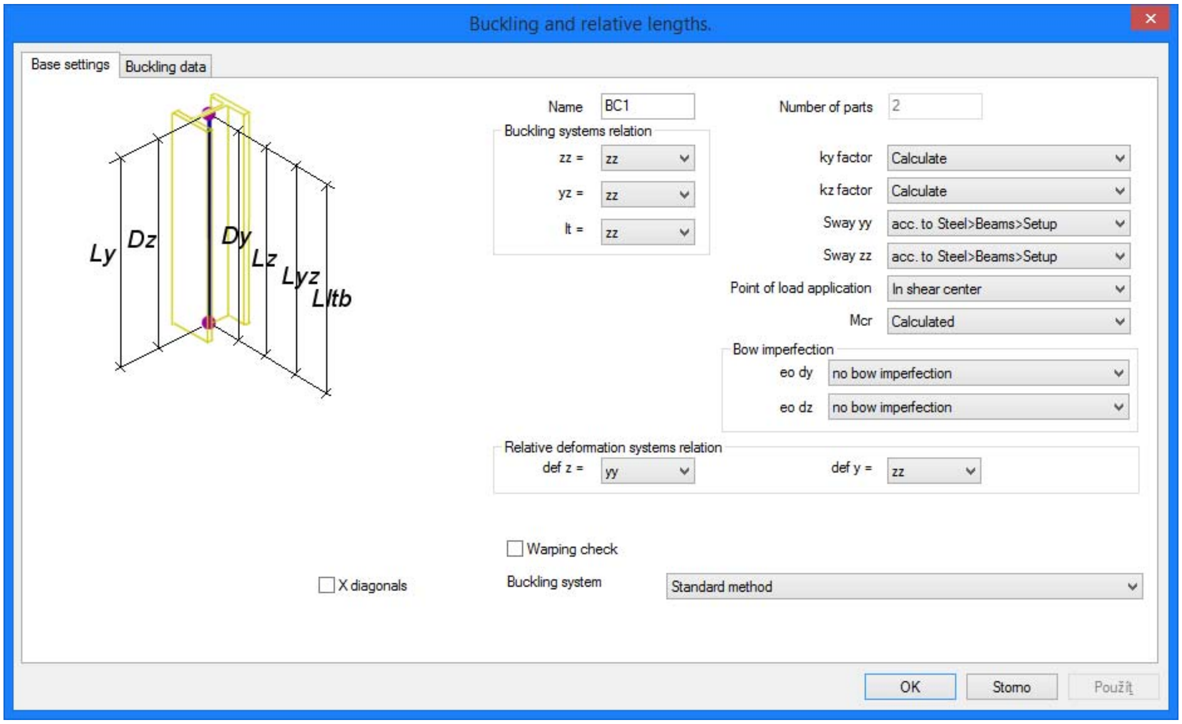


3. Click the ... icon next to **Buckling and relative lengths**. The **Buckling data** window appears.



This window shows that the column is supported for buckling around the weak axis (zz) in the middle and that it is not supported for buckling around the strong axis (yy) in the middle – the grey triangle is missing on left hand side.

4. Click **[Edit]** to change the buckling data. The **Buckling and relative lengths** window appears.



5. On the **Base Settings** tab, several data can be changed.

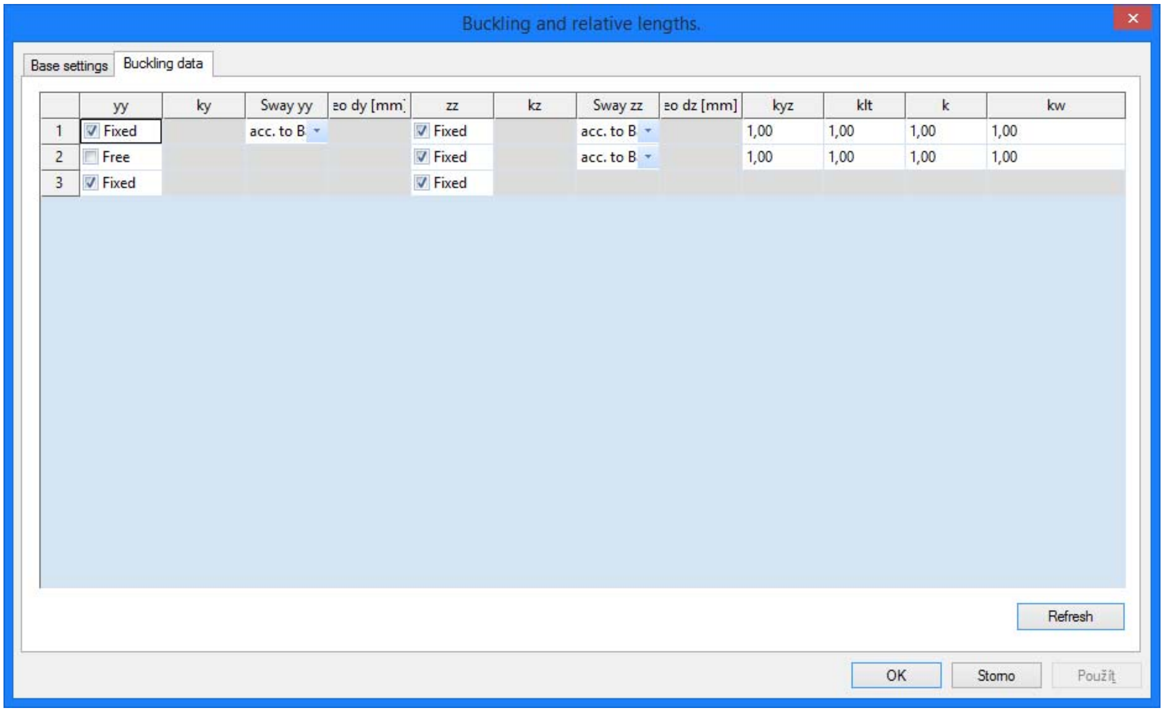
- The **Name** field contains the name of the buckling parameter, in this case **BC1**.
- **ky factor** and **kz factor**: in these fields you can indicate that the program should calculate the buckling factor or you can choose a manual input of this factor. A third option allows for a manual input of the buckling length (instead of factor).
- **Sway yy** and **Sway zz**: in these fields, you can indicate if the bar is braced or not in the direction regarded. When you choose option **acc. to Steel > Beams > Setup** option, the default settings are used.

  Note:
  The default settings for the buckling parameters are displayed below ***Steel > Beams > Steel****. The structure is by default non-braced for buckling around the strong axis and braced for buckling around the weak axis. In other words, a frame in non-braced in the plane and braced out of plane, taking the presence of wind bracings outside of the plane into account.*

- **Buckling systems relation**: in these fields you can define the system length to be used for, amongst other things, torsional buckling and lateral-torsional buckling.
- **Relative deformation systems relation**: in these fields you can define the system length to be used for the relative deformations (SLS check).

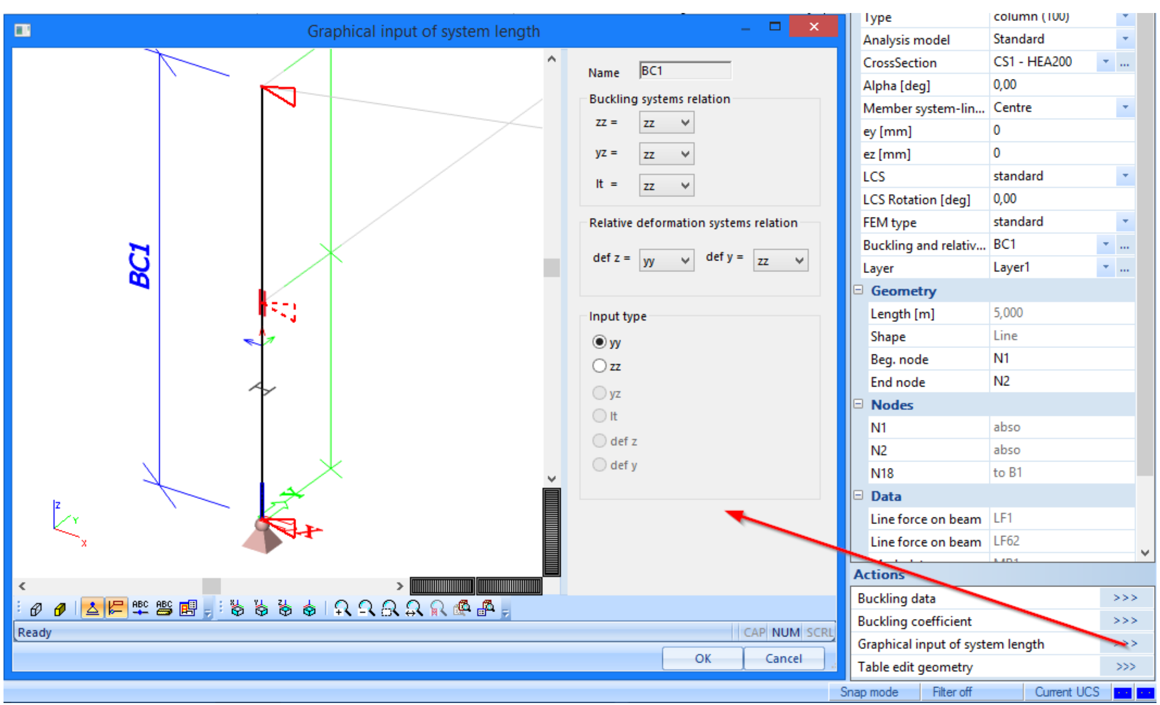
6. On the **Buckling data** tab you can edit the parameters in detail. The column consists of 2 components, i.e. 3 positions are available: (1) at the start, (2) in the middle at the horizontal girder and (3) at the end, at the roof girders.

- For instance, by modifying the **Free** option on position (2) for yy to **Fixed**, buckling of the column in the middle around the strong axis would be influenced as well. This would mean the system length around this axes would become also half of the total length (= 2,5 m). For this tutorial, the default options are kept.



7. Click **[OK]** to close this window.
8. **Buckling data** window re-appears. Click **[Close]** to close this window.
9. **Properties** window now indicates that the buckling parameter **BC1** is used for the columns of the first frame.
10. Press **<Esc>** to cancel the selection.

Remark:
You can double check the buckling system setting by action button Graphical input of system length. Here you can also change free nodes to fixed (by clicking the red triangles at specific locations at the members) and the other way round, or change relations between buckling systems.



When the buckling parameters are set, you can continue with the steel check. Before you proceed deactivate the **System lengths** and **Local axes** representation by means of the **Fast adjustment of viewflags on whole model** option.

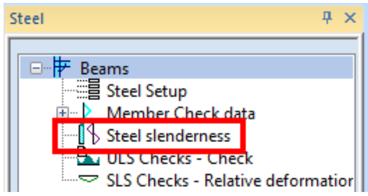
Steel code check

Double-click on 

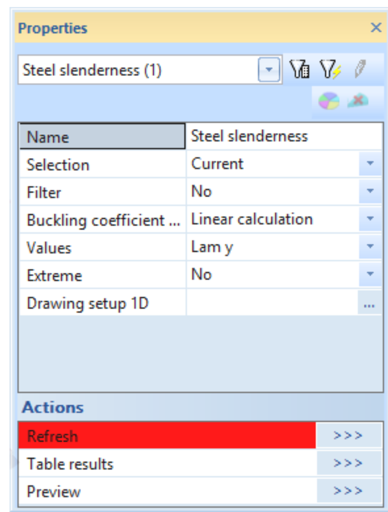
in the **Main** window to open the **Steel menu.**

Displaying the Slenderness and the Buckling Lengths

1. Click the **Steel slenderness** icon in the **Steel** menu.

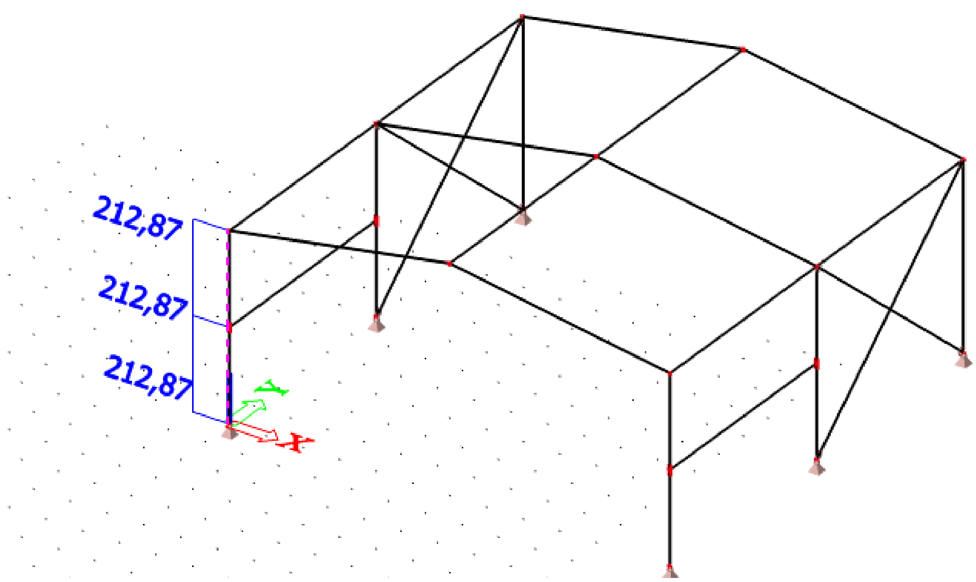


2. If this option is not available, you must restart the linear analysis using for example **Hidden calculation** icon in the Project toolbar.

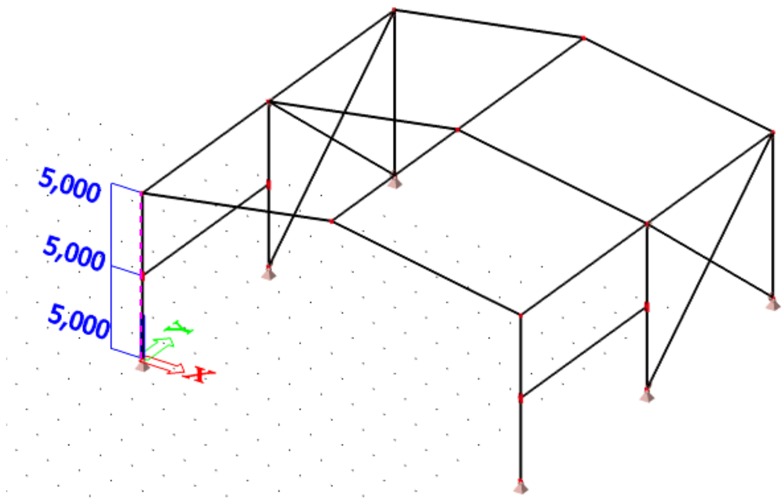


3. The options in the **Properties window** are configured in the following way:
   - **Selection** field is set to **Current**.
   - **Values** are wanted for **Lam y**, i.e. the slenderness around yy axis.
   - **Extreme** field is modified to **No**.

4. Select column **B1**, the left column of the first frame and hit **Refresh** >>> button in **Actions**.



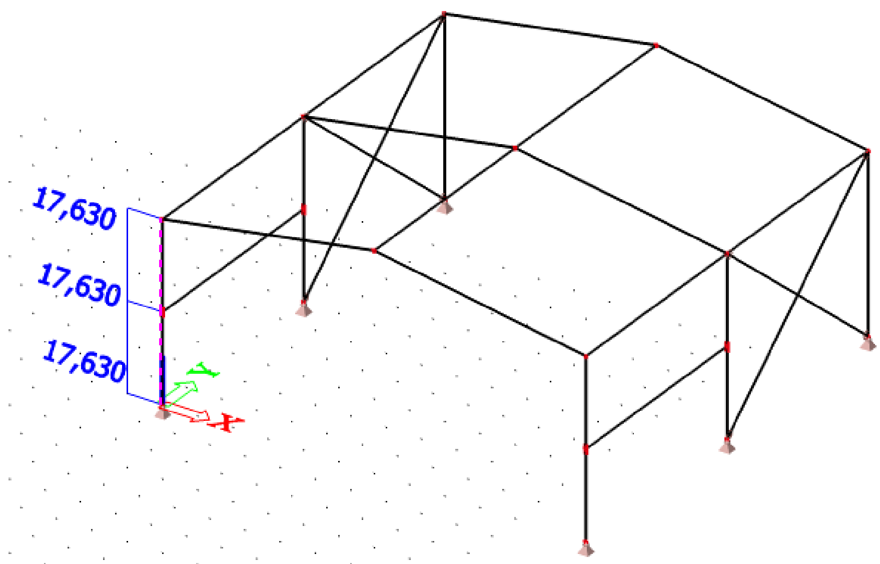
5. Change the **Values** field to **Ly** to display the reference length for buckling around the strong axis. Hit **Refresh** >>> button again.



As already indicated in the buckling parameters, the reference length is 5m.

6. Change the **Values** field to **ly** to display the buckling length for buckling around the strong axis.

7. Hit **Refresh** >>> button again.



The buckling length is determined as reference length **Ly** multiplied by buckling factor **ky**.

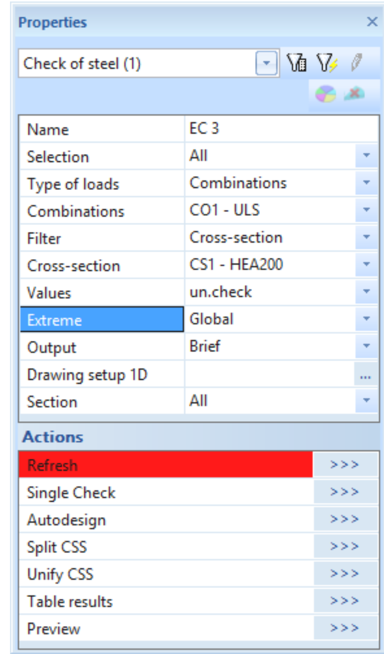
Since the default determination of buckling length is used in this project we always recommend checking the final slenderness by the steps described above. You can proceed to the steel check now. A unit check is carried out in accordance with the standard. The unit check includes both a capacity and a stability check.

Steel Code Check – Ultimate limit state

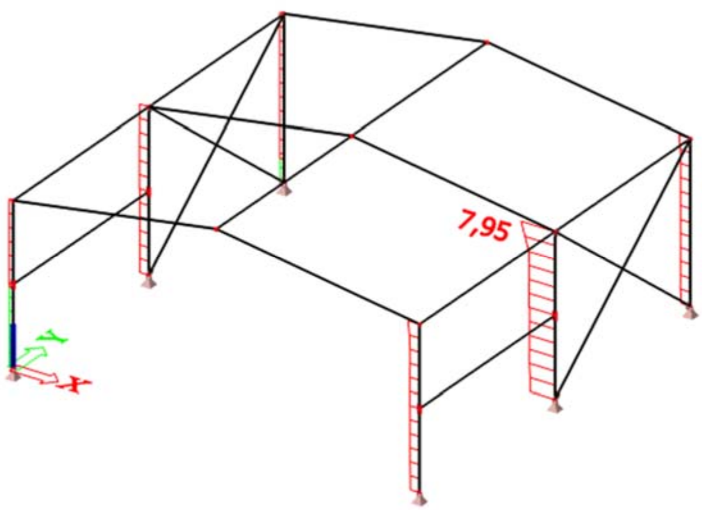
1. Click ULS Checks - Check in the **Steel** menu

2. The options in the **Properties** window are configured in the following way:

- **Selection** field is set to **All**.
- **Type of loads** is set to **Combinations** and the Combination to **CO1 - ULS**.
- **Filter** is changed to **Cross-Section.**
- For the **Cross-Section** choose **CS1 - HEA200** to ensure that only the results for columns are displayed.
- For the **Values** choose a **un. check.**
- **Extreme** field is changed to **Global**.

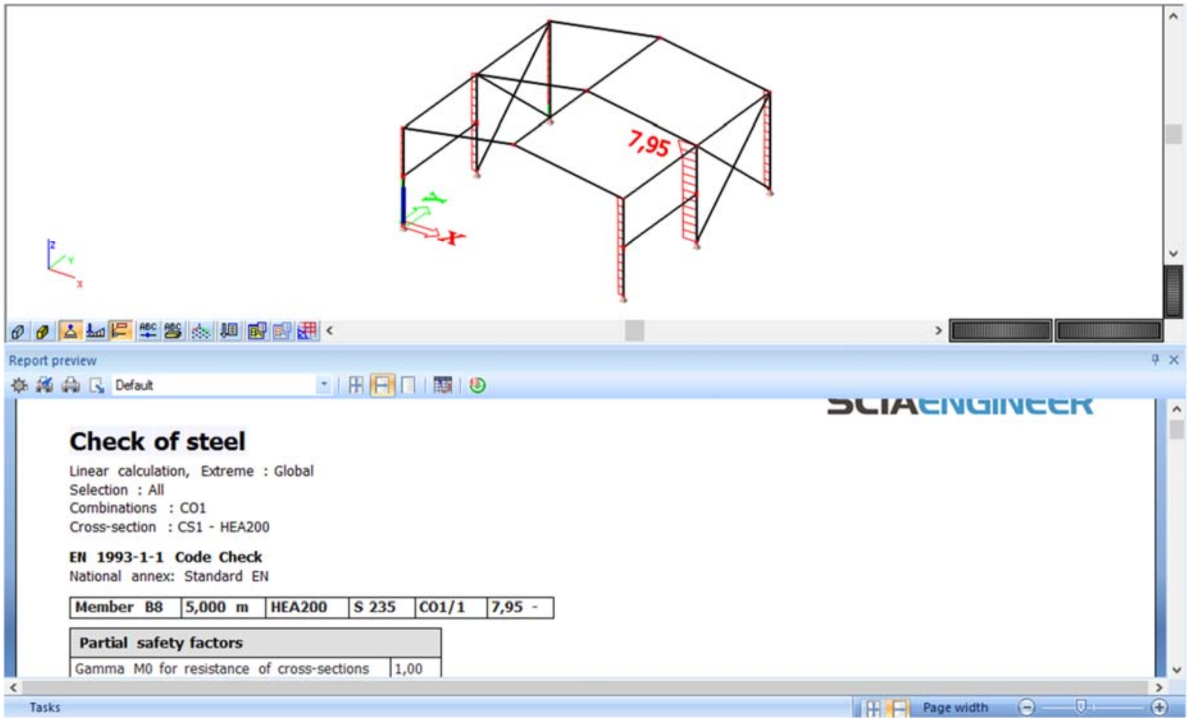


3. In the **Property** window, click the  button next to **Refresh** to display the results in the graphical screen in accordance with the above parameters.



Graphical screen shows that the maximum unity check occurs for one of the middle columns. To understand the reason behind you can open the **Report preview** with a detailed representation of the performed steel check. You can find all input data, calculated resistances, code formulas and notices and warnings in the detailed output that usually contains a few A4 pages.

4. Before opening the preview, set the **Output** option in **Properties** window to **Detailed**. Click the  icon next to **Preview** to open **Report preview**.



Check of steel

This detailed report reveals that the column does not comply with the Combined bending, axial force and shear force check according to article EN 1993-1-1 : 6.2.9.1. and formula EN 1993-1-1: (6.41) in this particular project; different cross-section with larger inertia is required. Below is only part of the complete check:

Combined bending, axial force and shear force check
According to EN 1993-1-1 article 6.2.9.1 and formula (6.41)

Mpl,y,Rd	100,85	kNm
Alpha	2,00	
Mpl,z,Rd	47,88	kNm
Beta	1,00	

Unity check (6.41) = 7,95 + 0,00 = **7,95** -

Note: Since the shear forces are less than half the plastic shear resistances their effect on the moment resistances is neglected.
Note: Since the axial force satisfies both criteria (6.33) and (6.34) of EN 1993-1-1 article 6.2.9.1(4) its effect on the moment resistance about the y-y axis is neglected.
Note: Since the axial force satisfies criteria (6.35) of EN 1993-1-1 article 6.2.9.1(4) its effect on the moment resistance about the z-z axis is neglected.

The member does NOT satisfy the section check!

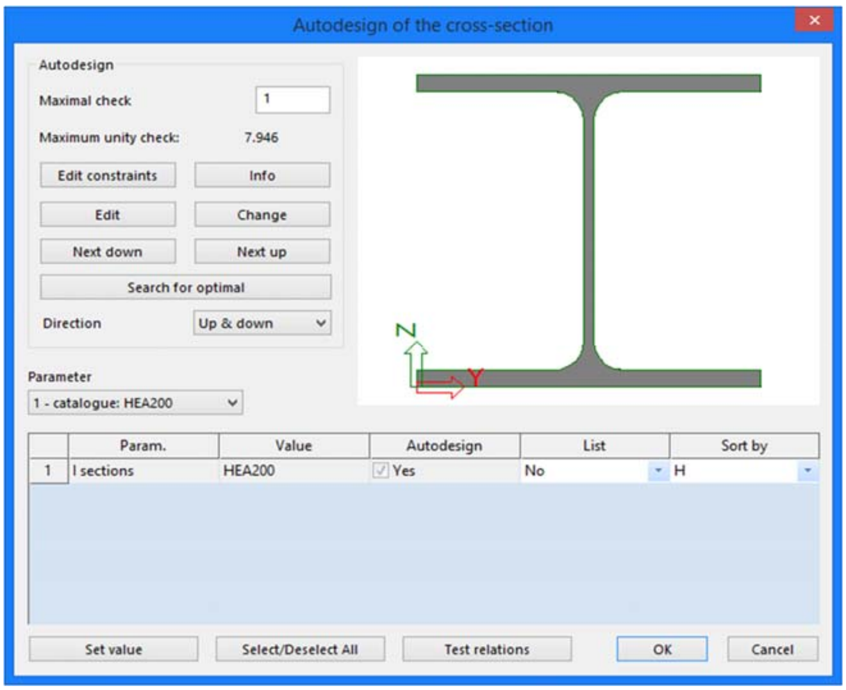
SCIA Engineer allows for a simple and smooth optimization of the steel section, whether it does not satisfy of whether it is too “heavy” and overdesigned. The program will automatically propose a cross-section section which complies with the unit check.

Optimisation of the Steel Section

1. In the **Properties** window, click the >>> icon next to **Autodesign** action. The settings of the **Properties** window are maintained, so that HEA200 will be optimised.

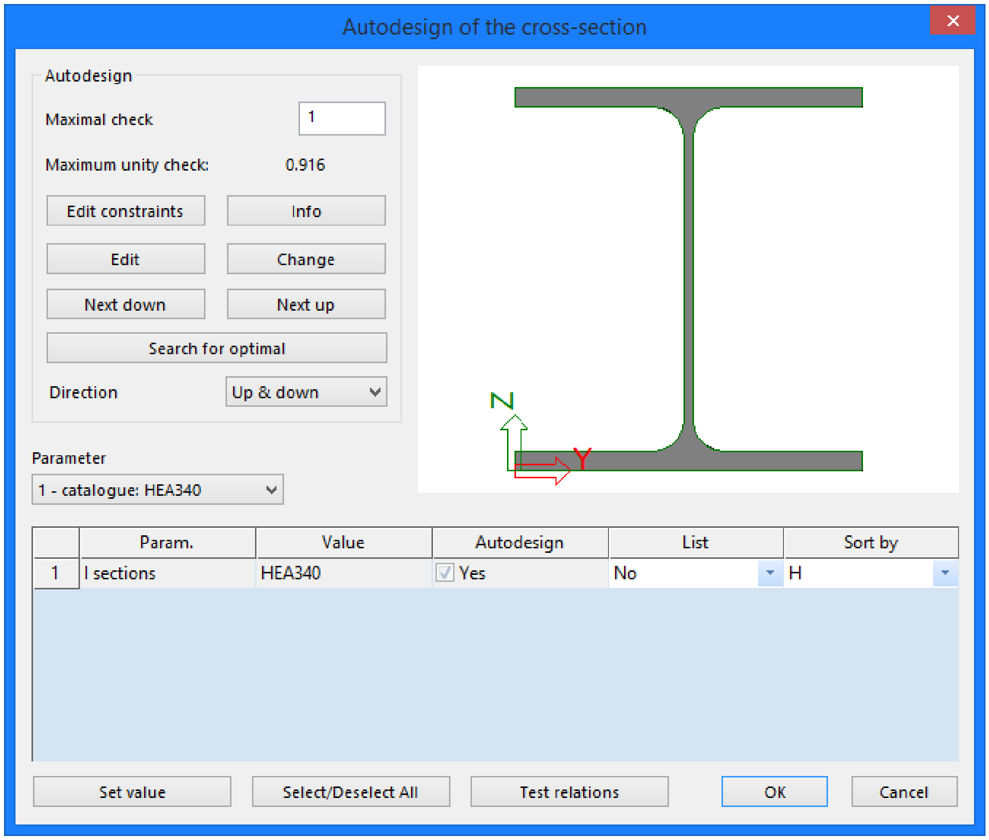
   Note: Steel optimisation works with cross-sections, not particular members. Therefore once the profile is changed, all members with that profile are changes as well. Using action button for Autodesign is therefore conditioned by using filter set to cross-section.

   The **Autodesign of the cross-section** dialogue looks like below



2. This window again displays the maximum unity check for all members with CS1 - HEA200 which is **7,946.** Just above this value there is a maximum unity check to be reached (but not exceeded, which is by default 1.

3. Click the **Search for optimal** button. The program will search within the profile library for a new cross-section and stops with the first one that satisfies the unity check.



It appears that HEA340 complies with the requirements: maximum unit check **0,916**

4. Confirm the optimisation with **[OK]**.

Note:
The project must be recalculated after the optimization. The changed cross-section modifies the self weight of the structure as well as the stiffness of the whole model, which will lead to a different distribution of the internal forces.
This specifically means that, after optimisation and recalculation of the structure, the profile concerned could possibly reveal inappropriate. In that case, you must re-execute the optimisation in order to find a solution in an iterative manner.

5. To quickly restart the calculation after an optimisation, use the hidden calculation command. Click on the **Hidden Calculation** icon in the Project toolbar.

6. Click **[Close]** to quit the **Steel** service.

Whole structure can be optimised using the steps above again for another cross-section (CS2, CS3,...). However, the procedure was described and repeating it for all the members is not an intention of this tutorial.

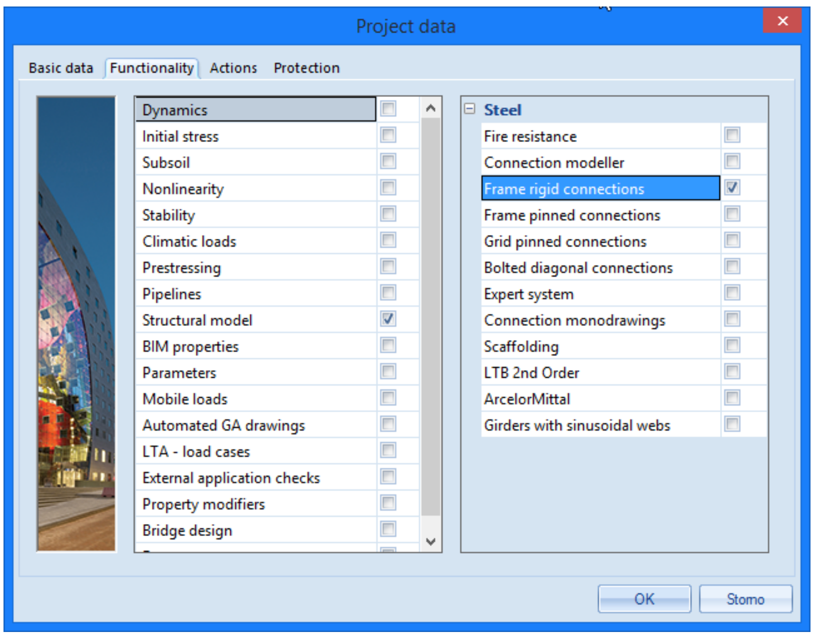
Steel connections

Steel connections in SCIA Engineer can be detailed in an advanced manner. Both rigid and hinged frame connections can be modelled, as well as bolted diagonal connections and grid connections. In this Tutorial, a rigid frame connection will be inserted between a column and a roof girder, as an isolated example.

Activating the Steel Connection Input

1. To obtain access to the steel connections, appropriate functionality needs to be activated first. Double-click on Project in the **Main** tree to open the Project data and click on the **Functionality** tab.

2. In the Steel part, activate the **Frame rigid connections** functionality. The functionality Structural model is automatically activated as well, as this one is required for the definition of the connection.



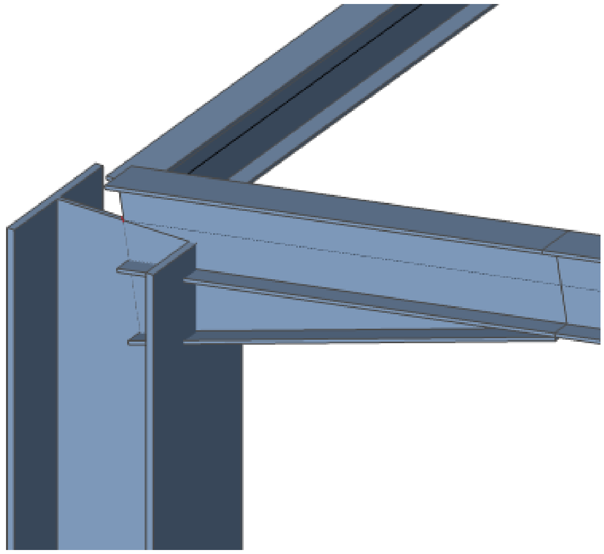
3. Confirm your choice with **[OK]** button.

When the required functionality is activated, the Structural mode which is crucial for definition of steel connection, can be activated. Structural model, unlike the analysis model which represents the nodes and system lines, shows surfaces of cross-sections and better reflect the real appearance of the structure.

Displaying the Structural model

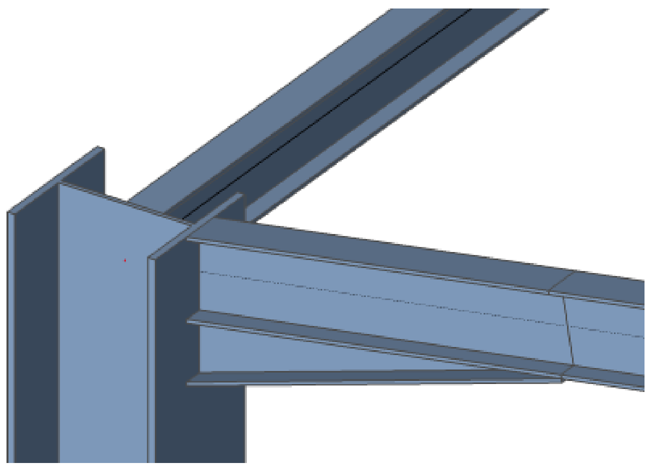
1. Zoom in on node **N2**, i.e. the node that connects column **B1** and roof girder **B2**.

2. In the **Command line** click on the **Show/hide surfaces** and the **Render geometry** icons. The program displays the calculation model like this:



3. In this analysis model, the bars are arriving in the same node, i.e. node **N2**. But in practice the girder is cut for instance at the column flange or vice versa. In SCIA Engineer, you can indicate this through the structural model.

4. Activate the structural model by clicking on view → Set view parameters → **Generate Structural model**  .



The structural model shows the structure as it will be realized in practice. On this model, you can introduce the connection because now its position is fully clear.

Note:
The structural model uses priorities of 1D members. The element with the highest priority value has priority over an element with a lower priority. By default, an element of the Column type has a higher priority (100) than an element of the Beam (80) type. Therefore, the beam is cut at the column flange and the column is a bit extended to be aligned with the top flange level of the beam.

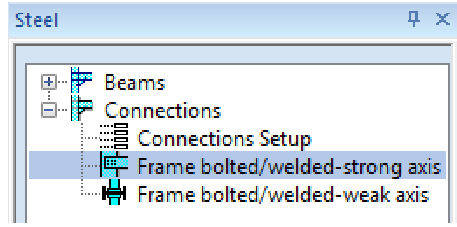
*With **Setup > Beam types (Structural)**, the priorities can be adapted.*

Steel connections are always based on the structural model. If the column continues, a connection with end plate on the girder is obtained; if the girder continues, a connection with end plate on the column is obtained.

Inputting a Steel Connection

1. Double-click on 

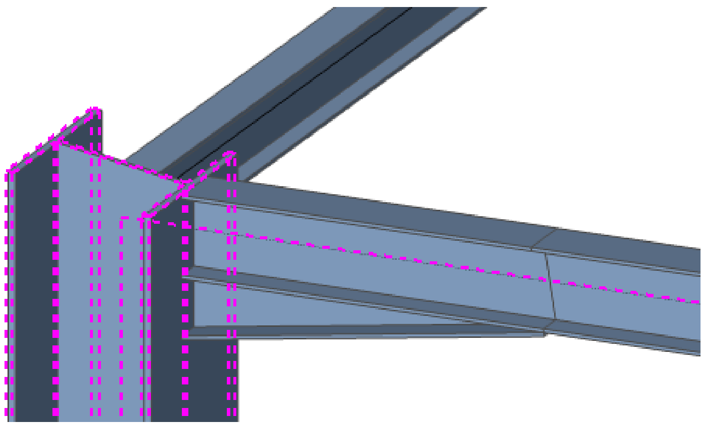
in the **Main** window to open the **Steel** service.



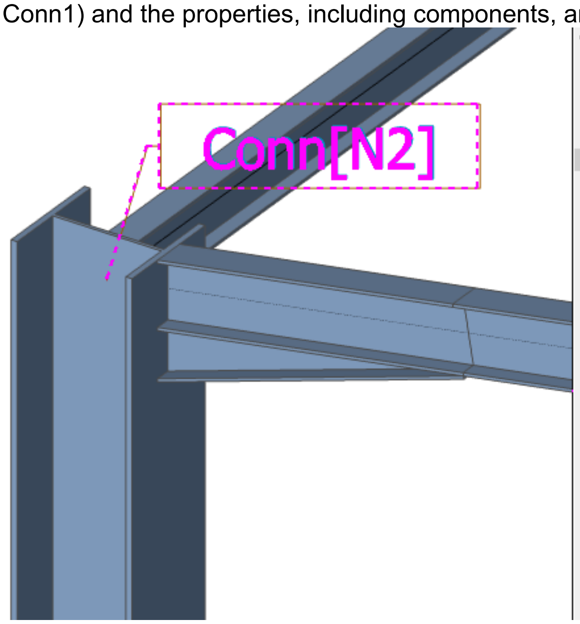
2. Double-click on Frame bolted/welded-strong axis under Connections to enter a new rigid frame connection.

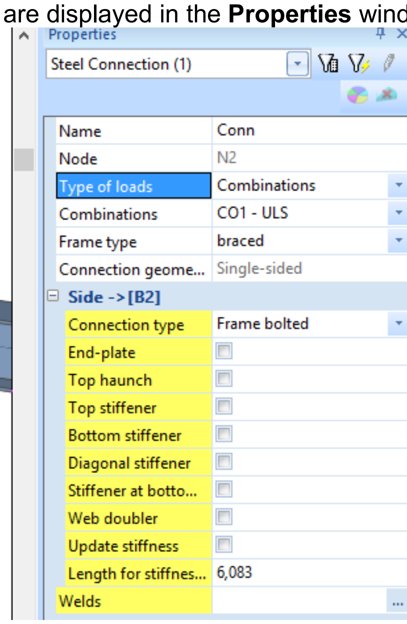
3. The program asks for a point of connection now, select node **N2.**

4. Now indicate the members between which the connection should be established. The program automatically selects (and highlights) all bars arriving in node **N2**. As the connection should be inserted between the column and the roof girder, deselect girder **B13**.
Press the **CTRL** (or **SHIFT**) key and click on the particular member with the left mouse button to deselect it.

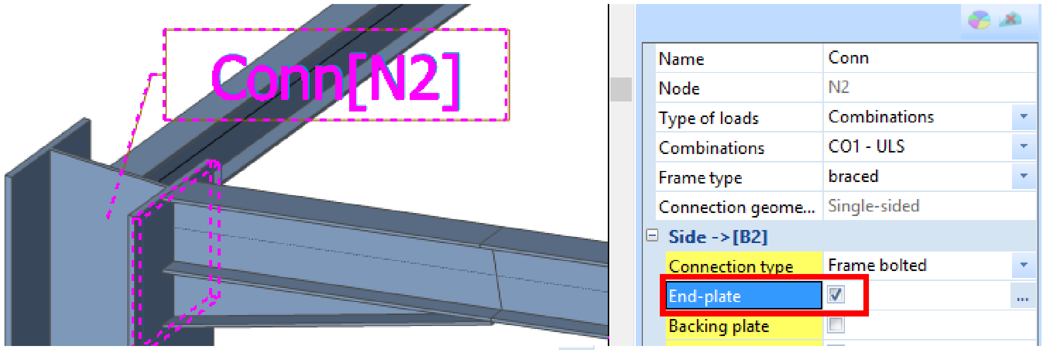


5. Press **<Esc>** to finish the selection. Connection is inserted (by the mean of a flag named Conn1) and the properties, including components, are displayed in the **Properties** window.

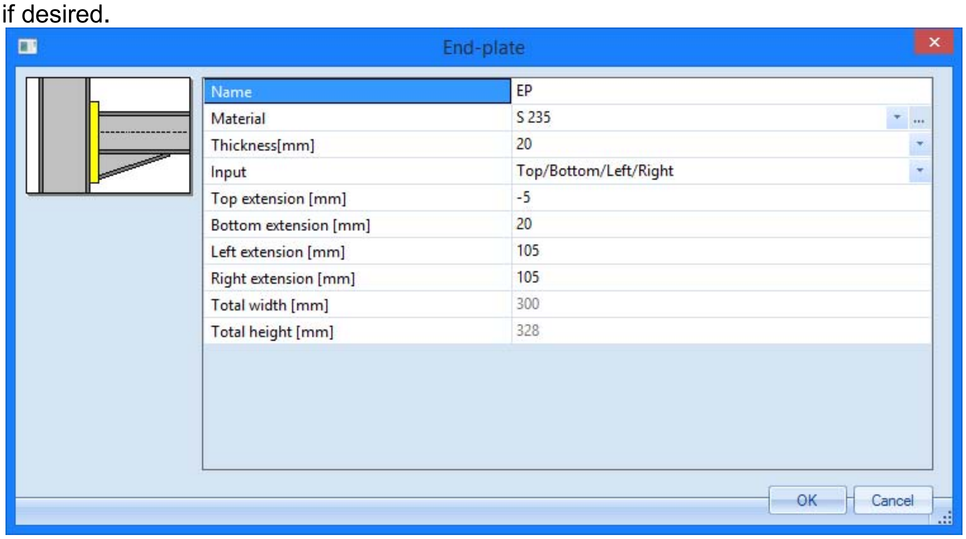




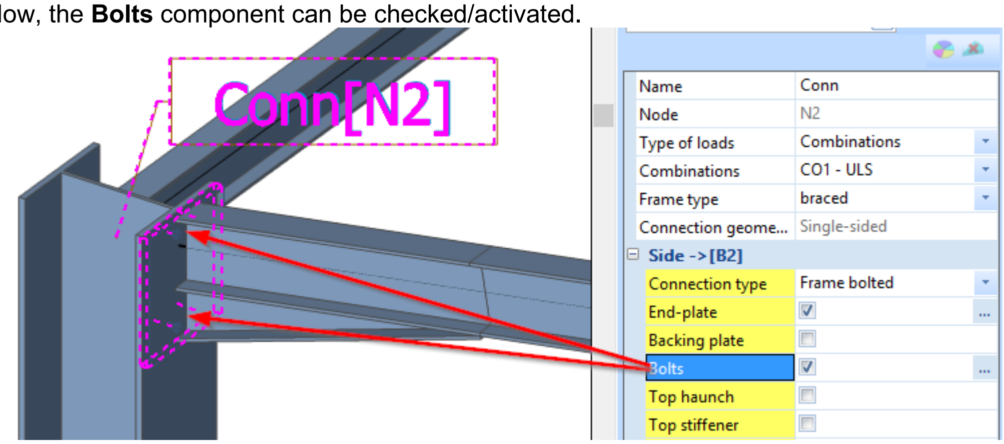
6. Now, the components of the connection can be entered. Activate (check) the **End-plate** option. The end plate is entered and immediately displayed in the graphical screen:



7. To change the properties of the end plate click the ... icon next to the **End plate** option in the **Properties** window. Properties of this particular component are displayed and can be modified, if desired.



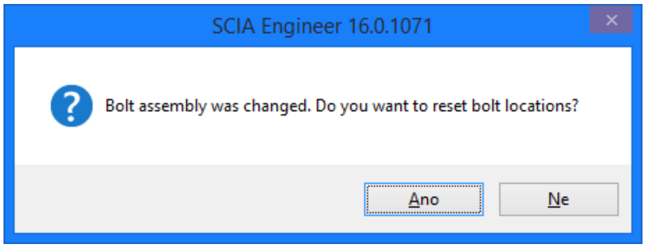
8. Click **[OK]** to close this window.
9. Now, the **Bolts** component can be checked/activated.



The bolts are automatically displayed in the graphical screen.

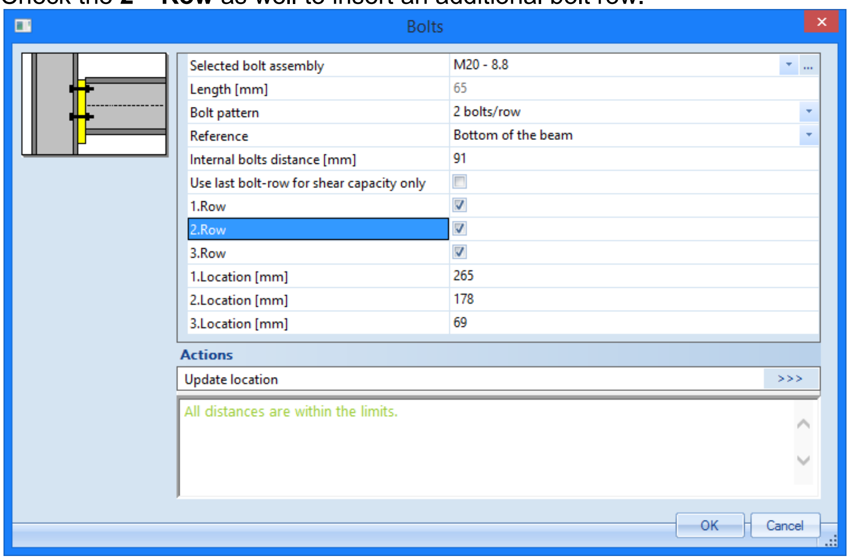
10. To change the properties of bolts, click the ... icon next to **Bolts** option in the **Properties** window.

11. For the **Selected bolt position**, an **M20 – 8.8** is chosen. A window informs you that the bolt position has changed.

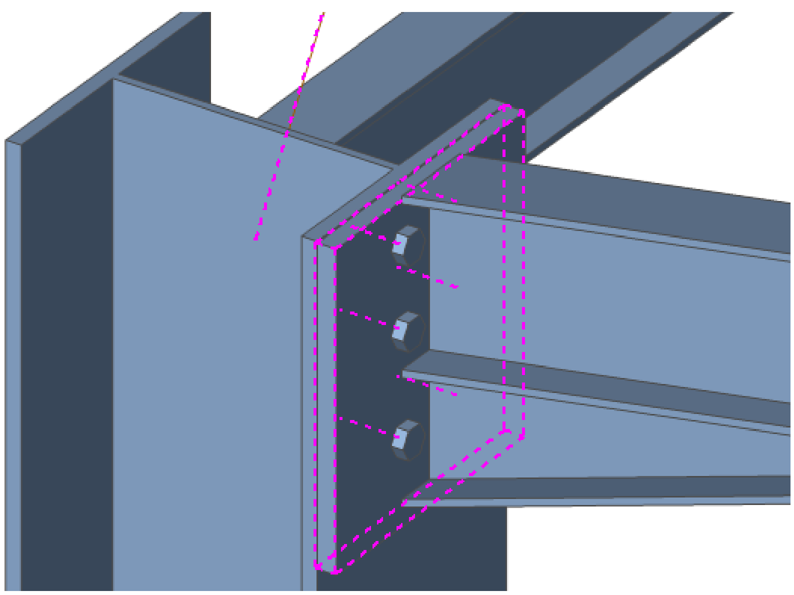


Click **Yes**: the bolt positions, intermediate distances, edge distances etc. are automatically adapted to the new bolt type.

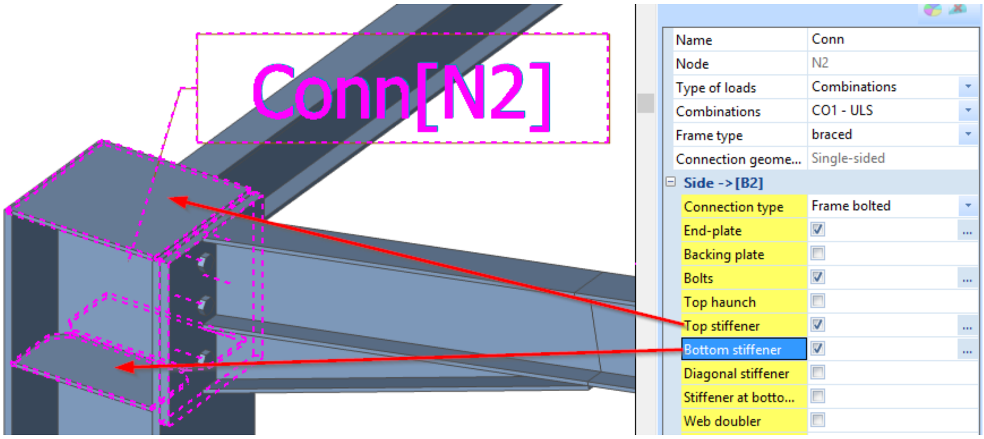
12. The window furthermore shows that 2 bolt rows are active: the **1st Row** and the **3rd Row**. Check the **2nd Row** as well to insert an additional bolt row.



13. Click **[OK]** to confirm your input. The bolts are displayed in the graphical screen.

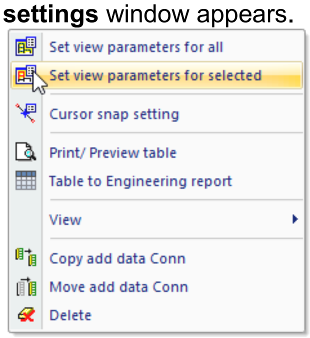


14. To complete the connection, check the **Top Stiffener** and **Bottom Stiffener** components.

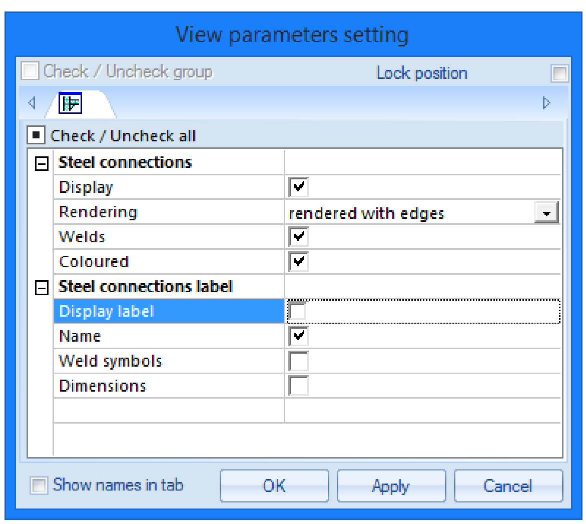


Connection is now displayed in the same colour as the model. To change this view, use the **View parameters** menu.

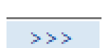
15. Click the right mouse button at an arbitrary location in the workspace. The menu lists the available possibilities for the selected entity.
16. In the context menu hit **Set view parameters for selected** command. The **View parameters settings** window appears.



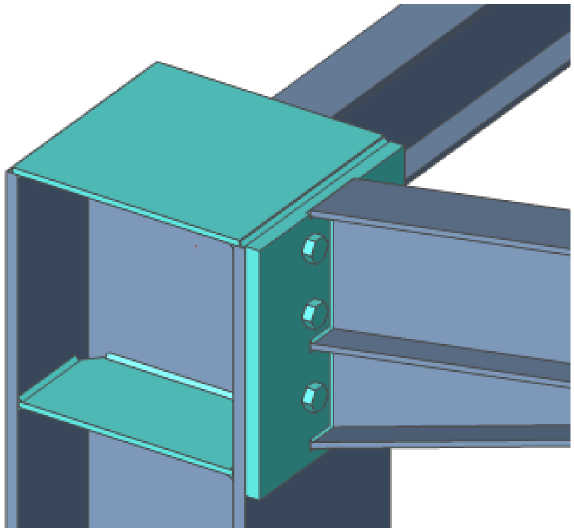
17. In this menu, check the **Welds** and **Coloured** options. You can also disable (untick) **Display label** option.



Close this menu with **[OK]**.

18. In the **Properties window**, click the 

icon next to **Refresh** to regenerate the input in the graphical screen. The connection is displayed in a rendered style:



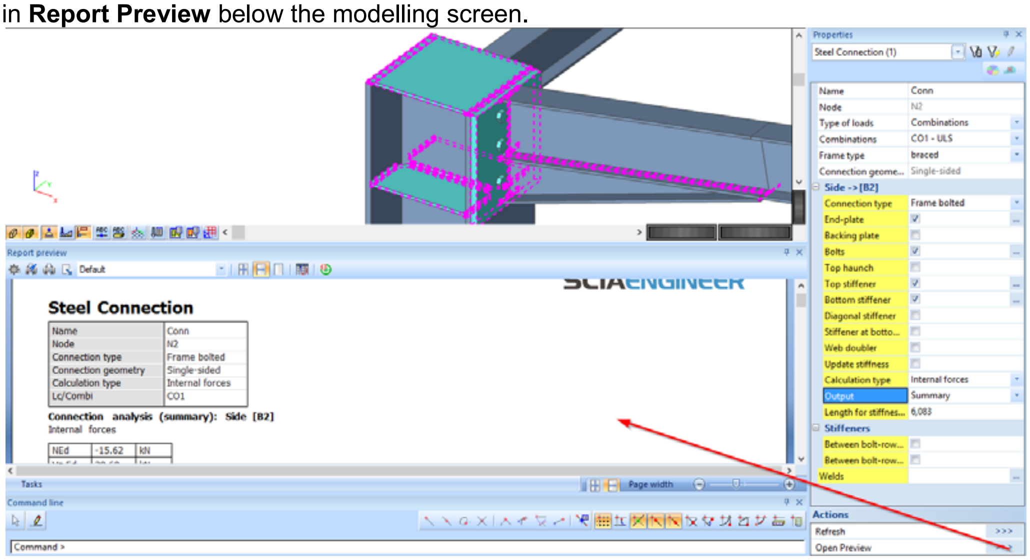
The connection now is completely modelled and you can open the results.

Checking the connection

1. Select the connection by mouse cursor and in the **Properties** window set the following:

   - **Type of load** is set to **Combinations** and **Combinations** to **CO1 - ULS**.
   - For the **Frame type**, choose **braced.**
   - **Output** is set to **Summary**.

2. In the **Property** window, click the  button next to **Open Preview** to display the results in **Report Preview** below the modelling screen.



Output shows the internal forces, design resistances, results of unity checks and moment-rotation diagrams. The most significant part of the output says that this particular connection does not satisfy the check, therefore we have to change the configuration of the connections.

...::RESULTS::...
Unity checks

My,Ed/Mj,y,Rd	1.25
Mz,Ed/Mj,z,Rd	0.02
NEd/Nj,Rd	0.06
Vz,Ed/Vz,Rd	0.30
Vy,Ed/Vy,Rd	0.00
Vz,Ed/Vz,Rd + Vy,Ed/Vy,Rd	0.30
My,Ed/Mj,y,Rd + Mz,Ed/Mj,z,Rd	**1.27**

In this case, the limit part is the Column Flange in bending (Ft,fc,Rd), as it is stated in the report as well. The user must change the used cross-section for the column, because the flange of the column cannot take the bending moment for this connection.

3. Press **[Close]** below the **Steel** menu to return to the **Main** tree.

Notes:

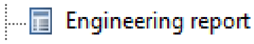
Detailed output with all intermediate calculations can be also displayed if necessary. It covers many A4 pages for just one connection to enable hand verification of the provided results.

Connections are additional data, i.e. a connection can be copied to other nodes.

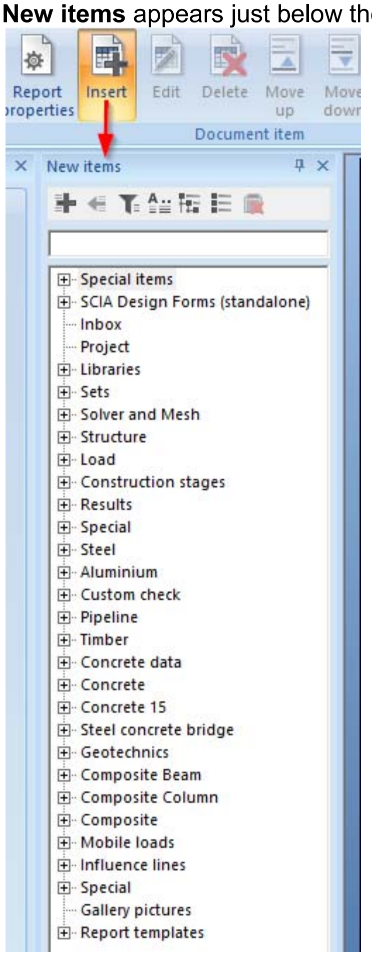
Document

In this final part of the tutorial, we will explain how to make nice report of the calculation and design.

Engineering report

1. Double-click 

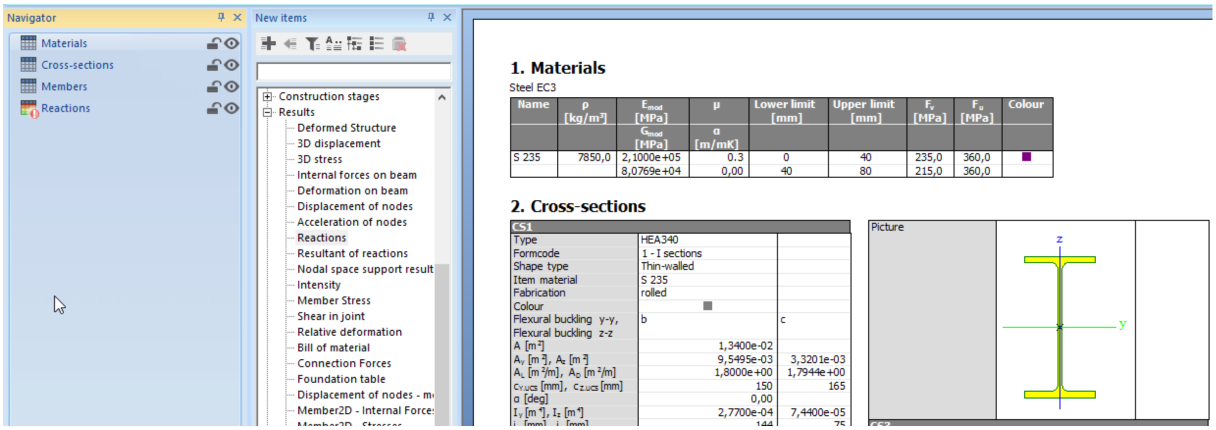
in the **Main Window** or click  in the toolbar. Because no report was created before, **Report_1** directly appears as a new application. This application is in a certain way independent on the SCIA Engineer application. That is significant also in the Windows main bar


2. Click **Insert** button in the ribbon to start inputting items in the report navigator. Windows with **New items** appears just below the **Insert** icon:


3. Using this window, various data can be added to the report.

   - Open the **Libraries** group and select **Materials**. Double-click in this item or hit  button to add this item to the document navigator.
   - Add also **Cross-Sections** one row above.
   - Open the **Structure** group and double-click **Members**.
   - Open the **Results** group and click **Internal forces on beam.**

4. You can directly see these items in the Navigator and on the paper preview as well:

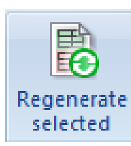


Drag the items with the mouse to change their order.

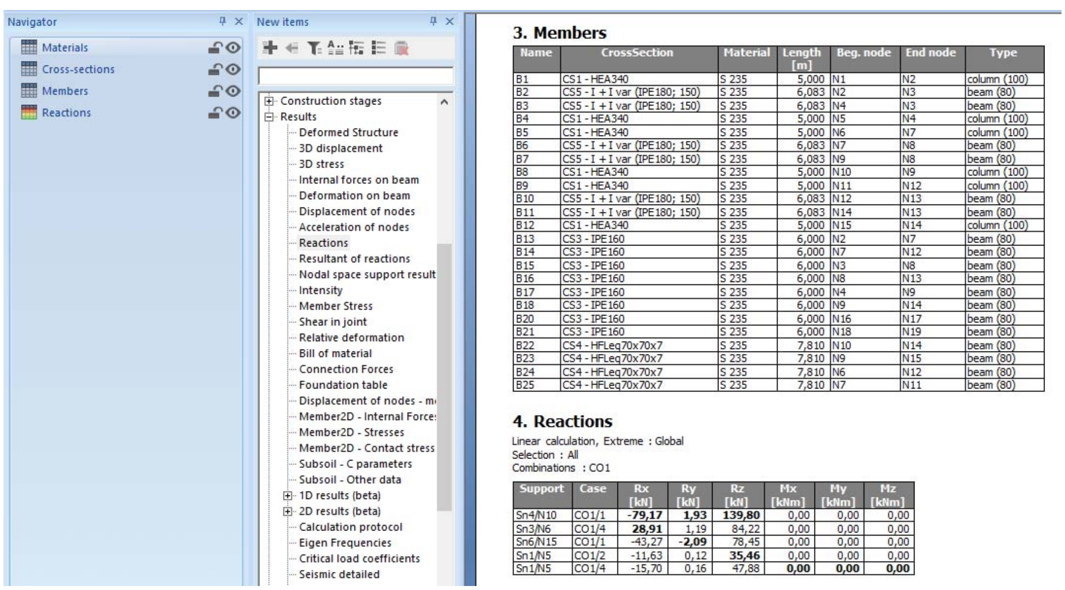
Displaying results in the document

1. In the **Navigator** click **Internal forces on beam**. The red exclamation mark both in Navigator and preview indicates that the values presented are not up-to-date. In the **Properties window** the setting of this table is displayed. Parameters for displaying the results in the **Engineering Report** are configured in the same way as the parameters for viewing the results in the **Results Menu** of the SCIA Engineer application.

   - **Selection type** is set to **All**.
   - **Type of load** is set to **Combinations** and the Combination to **CO1 - ULS**.
   - **Values** are set to vertical reaction **Rz**.
   - **Extreme** field is changed to **Global**.

2. Click the 

button in the top ribbon to display the table in accordance with the predefined options. Red exclamation mark disappears.

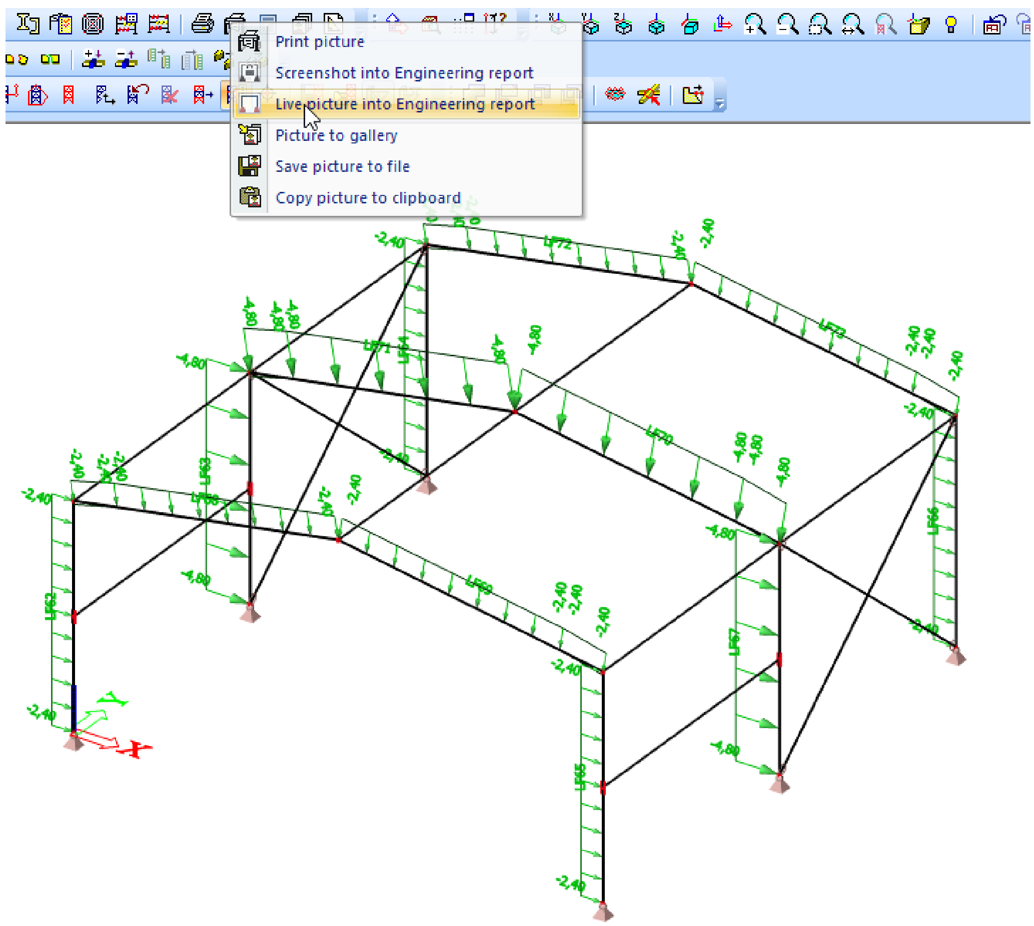


Adding an image to the document

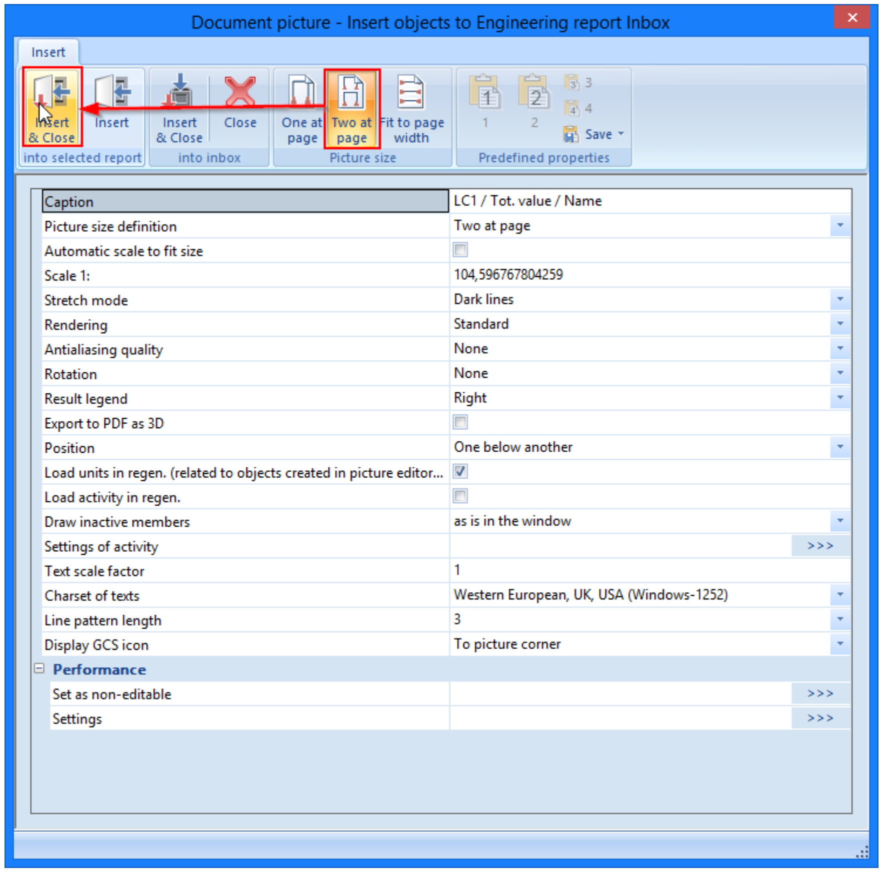
1. Any picture from SCIA Engineer application can be set to Engineering Report. Either as print-screen (that is unchanged for ever) or as live picture (that can be regenerated and is always up-to date).

2. Prepare any scene in the 3D modelling window, for example the analytic model with loads. You can use the icons above the **Command line** to hide surfaces and rendering and show loads:

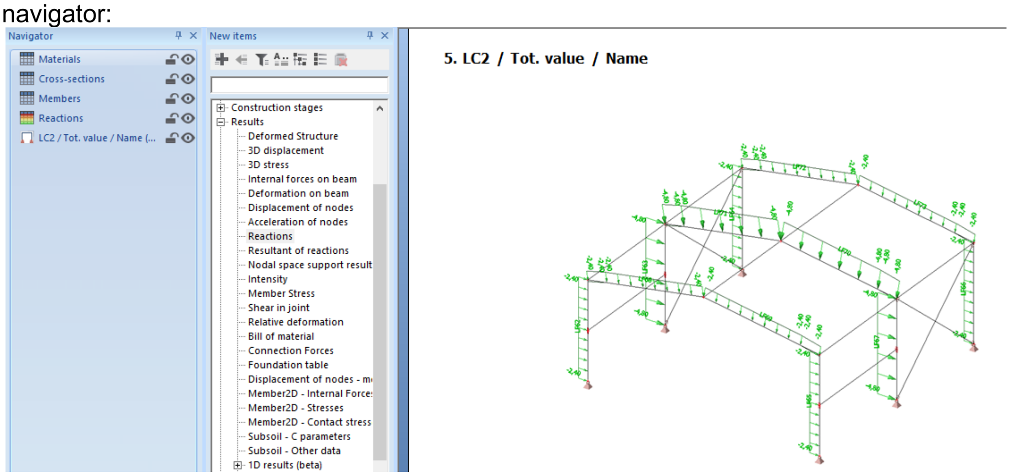

3. Click on button **Print Picture in toolbars** and select **Live picture into Engineering Report**



4. Document picture properties dialog is opened. Here you can arrange the picture caption, scale, size etc. Use button **Two at page** at the top ribbon and click on button **Insert & Close** into selected report

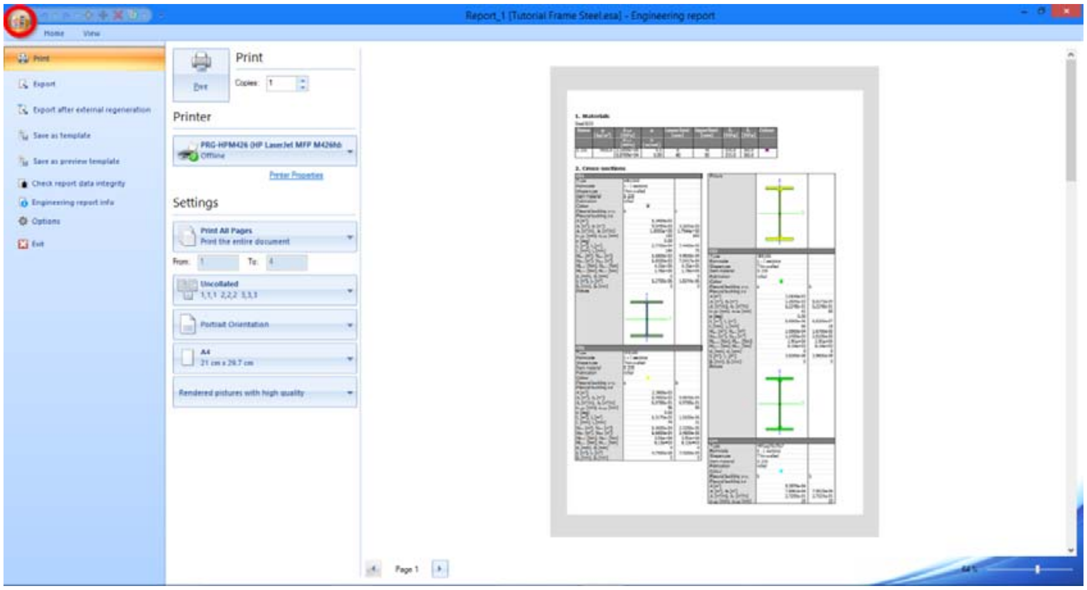


5. Switch to Engineering Report application again and see the picture as the very last item in the navigator:



Printing Engineering Report

Once the report is completed you can print it or export into various formats (e.g. PDF, RTF, HTML) by clicking the top left button of the window.



Epilogue

In this syllabus, the basic functionalities of SCIA Engineer for the input of a steel structure, including the steel calculation, were introduced by means of an example.

After reading the text and executing the example, the user should be able to model and calculate simple structures consisting of steel bars.

For more detailed information about steel calculations we refer to the Advanced Training Steel documentation or the dedicated Web help chapters.