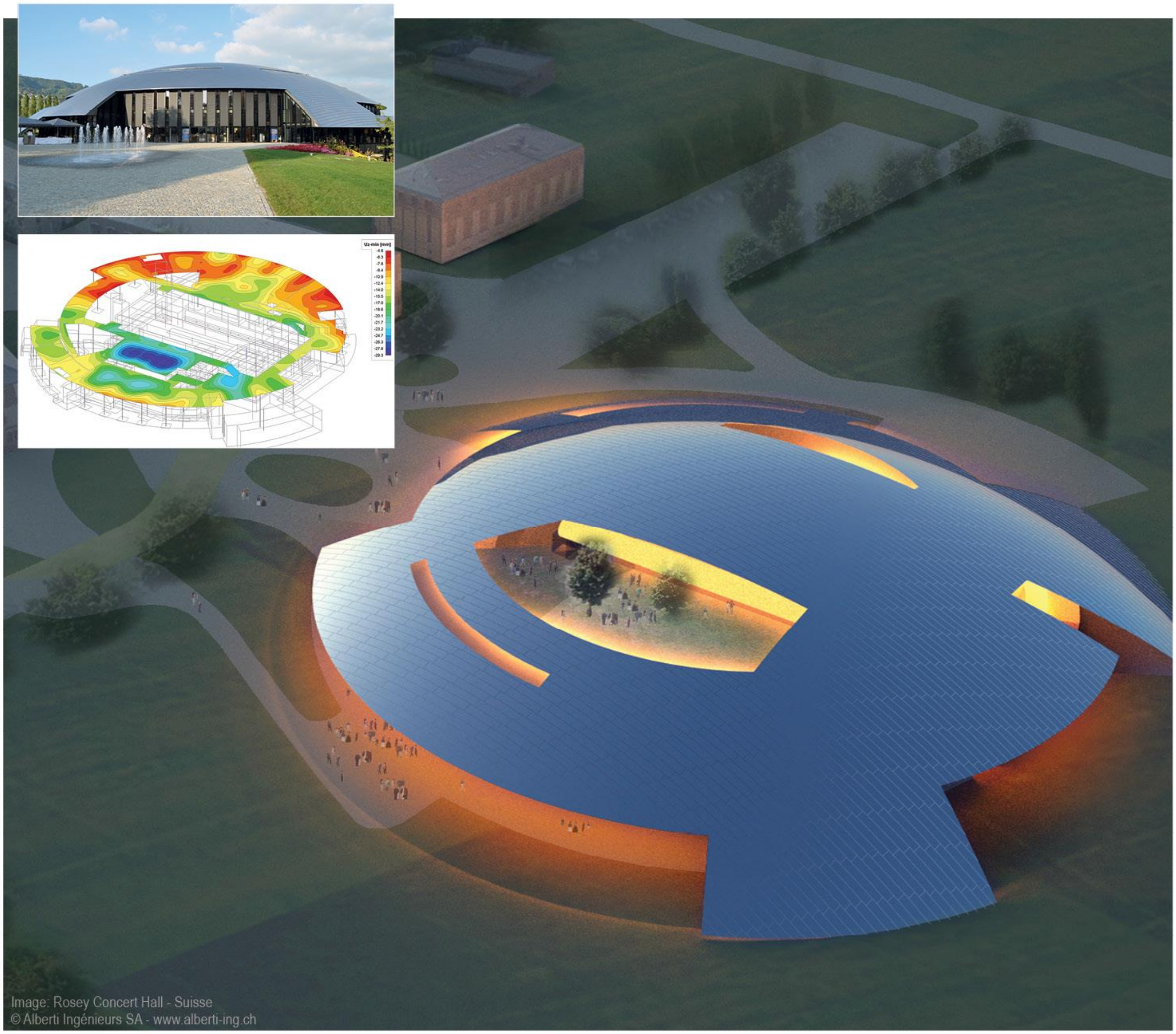
SCIA ENGINEER



Tutorial

Steel Hall (AISC 360-10)

Table of Contents

General Information 1

Introduction 2

Getting started 3

Starting a project 3

Project management 6

Save, Save as, Close and open 6

Saving a project 6

Closing a project 6

Opening a project 6

Project Unit Setup 7

Changing the units 7

..... 7

Geometry Input 9

Input of the geometry 9

Profiles 9

Geometry 10

Haunches 21

Hinges 26

Supports 27

Check Structure data 31

Checking the structure 31

Connecting entities 31

Graphic representation of the structure 36

Loads and combinations 41

Load Cases and Load Groups 41

Defining a Self Weight Load Case 41

Defining a Permanent Load Case 41

Defining a Variable Load Case 42

Loads 43

Combinations 50

Calculation 53

Linear Calculation 53

Results 54

Viewing results 54

Code check 59

Buckling parameters 60

Displaying the system lengths 60

Setting the Buckling Parameters 62

Steel code check 65

Displaying the Slenderness and the Buckling Lengths 65

Steel Code Check 67

Steel Design Check - Preview 69

Optimization of the Steel Section 70

Engineering Report 73

Formatting the Document 73

Epilogue 75

General Information

Welcome

Welcome to the Scia Engineer 15 AISC Steel Frame Tutorial. Scia Engineer is a design program in Windows with a broad application field: from checking/designing simple frames to the advanced design of complex projects in steel, concrete, cold formed steel and a variety of other materials.

The software allows engineers to model 2D and 3D structures which include flat or curved plates and beam members (straight or curved) as well as other advanced 3D geometry. The complete calculation and design process has been integrated into one program so that the input of geometry, input of calculation information (loads, combinations, and supports), linear and non-linear calculation, output of results, reinforcement design according to various codes and the generation of the calculation documentation are all completed in the same software.

Scia Engineer is available in three different editions all which require a license to operate:

- Concept
- Professional
- Expert

<u>License version</u>

A licensed version of Scia Engineer is secured with either a 'dongle', which you apply to the USB port of your computer or a software license on your company's network.

Scia Engineer is also modular and consists of various modules. The user chooses from the available modules and composes a custom design program, perfectly tuned to the desired needs of the company.

In the general product overview of Scia Engineer you will find an overview of the different modules that are available.

<u>Demo version</u>

If the program doesn't find a license on your computer, it will automatically start the demo version. The properties of the demo version are:

- All projects can be inserted however projects created in a demo version cannot be opened in a licensed version.
- The calculation is restricted to projects with 25 elements, 3 plates/shells and two load cases
- The output contains a watermark "Unlicensed software"

Scia Engineer Support

If you need assistance with the software, you can contact the Scia Engineer support service in the following manners:

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

From USA: 443-542-0637

Via the Scia Support website

http://www.scia-online.com/en/online-support.html

Website

Link to Tutorials

http://www.Scia-online.com > Support & Downloads > Downloads > input e-mail address > Scia Engineer > Scia Engineer Manuals & Tutorials

Link to eLearning

http://www.scia-online.com > Support & Downloads > eLearning

Link to Demo version

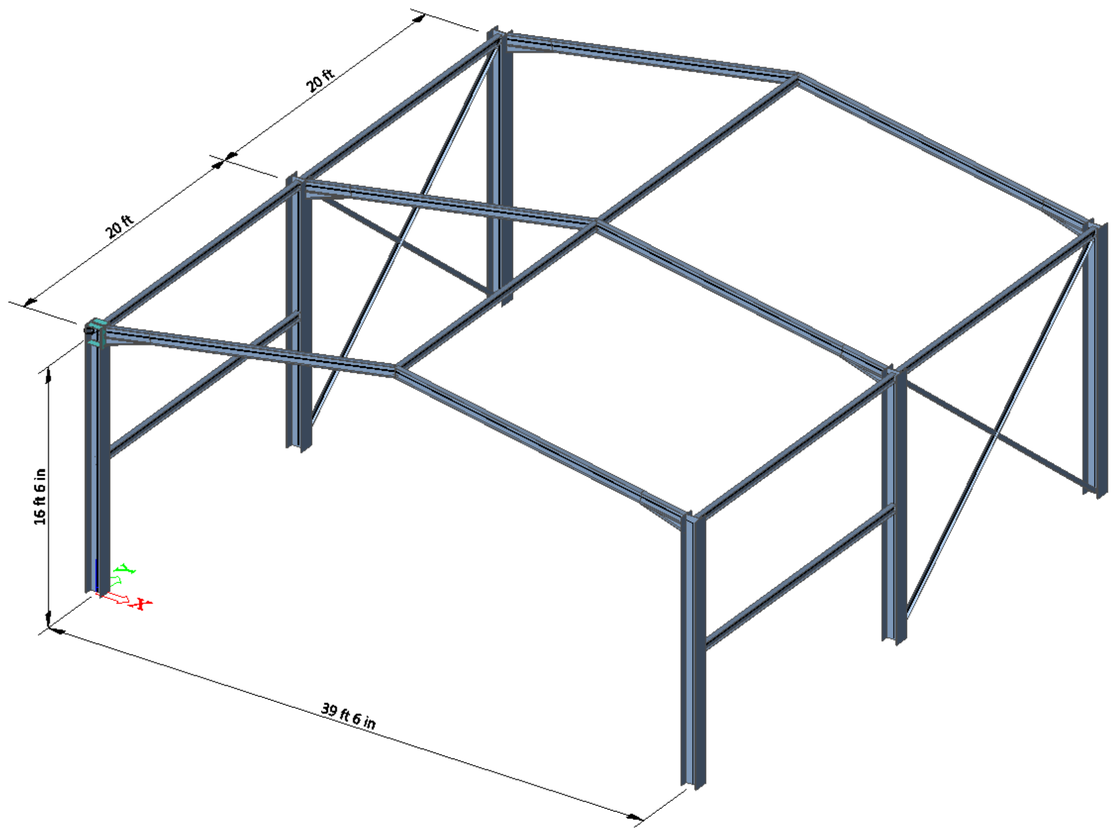
http://www.Scia-online.com > Support & Downloads > Secured Downloads > input username and password > Service Packs > Scia Engineer > Setup – Scia Engineer

Introduction

This tutorial describes the main functions of Scia Engineer 15, the input of design data and calculation of a 3D steel frame building.

The tutorial will begin with the creation of a new project and the modelling of the steel frame structure. After the input of all frame geometry and loads, the structure will be calculated and the results will be viewed. Next, the input of the slenderness and buckling parameters of the frame will be discussed, followed by the steel member calculations which contain the inclusion of unity checks and member optimization. Next, a rigid steel connection will be modelled and displayed within the structure. Lastly, the tutorial will discuss how to format an engineering report while properly displaying the calculation results.

The figure below shows the computational model of the structure that will be completed through this tutorial (the units are in the Imperial system, e.g. kips, feet):



Getting started

Starting a project

Starting the program

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

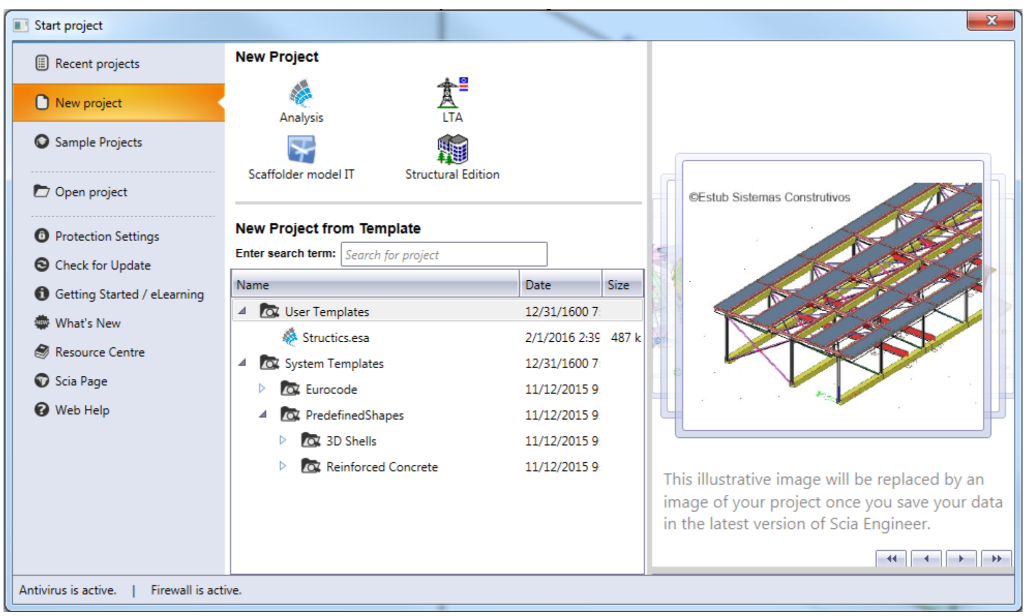
1. Double-click on the Scia Engineer shortcut in the Windows Desktop.
   Or:
2. If the shortcut is not installed, click **[Start]** and choose **Programs > Scia Engineer 15> Scia Engineer 15**.

If the software cannot locate a license file, you will receive a dialogue box indicating that no license was found. A second dialogue box will then list the restrictions of the demo version. Click **[OK]** in both windows.

For this Tutorial, you must start a new project.

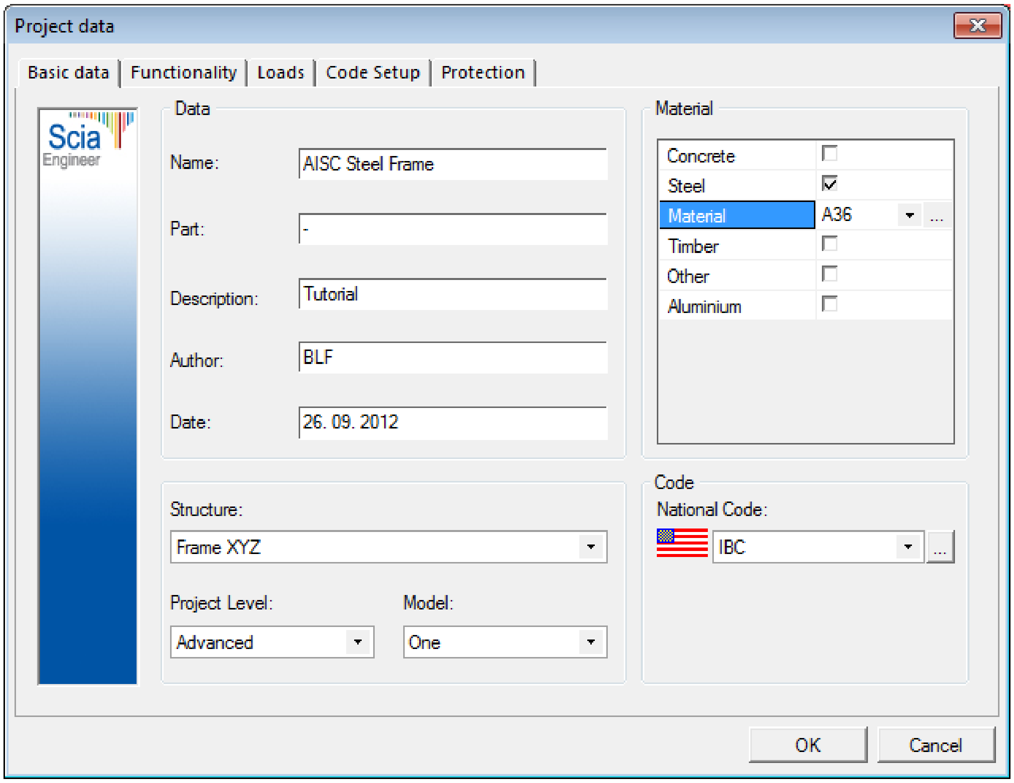
Starting a new project

1. If the dialogue **Open** appears, click **[Cancel]**.
2. Click the **New** icon  in the toolbar.



In the **Select New Project** dialogue box, choose the **Analysis** environment by clicking on the corresponding icon. Confirm your choice by clicking **[OK].**

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



3. In the **Basic Data** group, enter your preferred data. The data you enter will be displayed on the output produced by Scia Engineer, e.g. in the document and on the drawings.

4. Choose the **Project level: Advanced** and **Model: One**.

5. Click on the rectangular button below **National Code** to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the tutorial we will choose IBC (International Building Code). The window **Codes in project** is opened.

   - Click **[Add]**.
   - The dialogue box **Available national codes** are opened.
   - Select the USA flag and click **[OK]**.
   - You will return to the **Codes in project** dialogue and **IBC** is added.
   - Select the flag named **IBC.**
   - Select the **Active code** option and click **[Close]**.
   - You will return to the **Project data** window and **IBC** is the active code.

6. Select **Frame XYZ** in the **Structure** field.

   The structure type (Frame XZ, Frame XYZ, Plate XY, General XYZ, etc.) will restrict the input possibilities during the calculation.

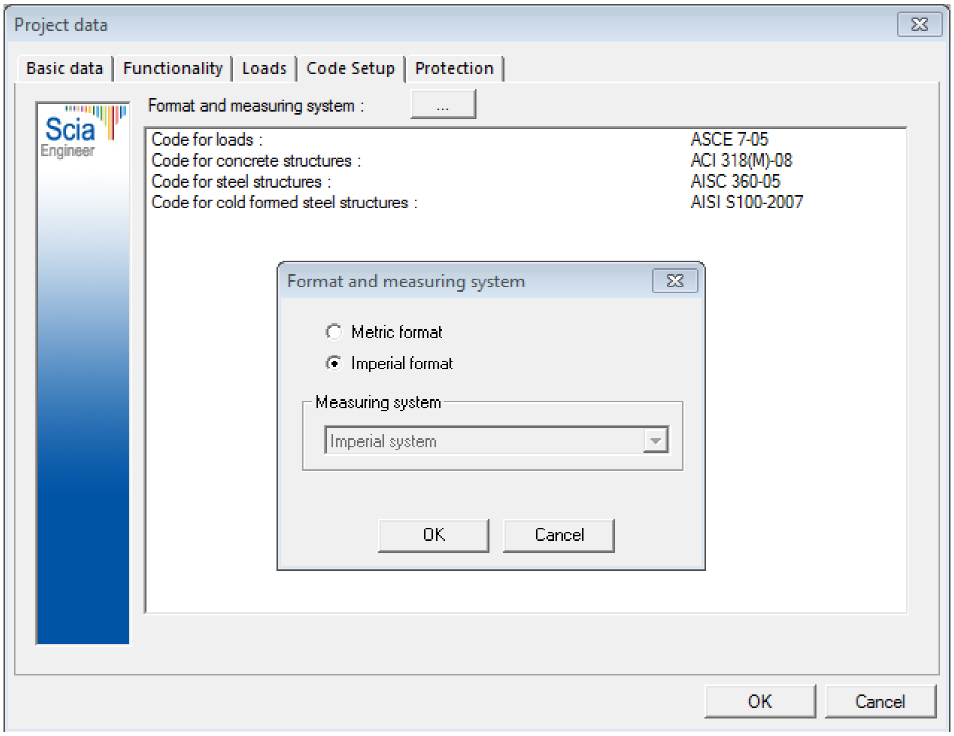
7. In the **Material** group, select **Steel**. Below the item **Steel**, a new item **Material** will appear.

8. Choose **A36** from the menu.

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

10. On the **Code Setup** tab, you can see the specific codes that are used for the generation of loads and the design of steel, concrete and cold formed steel structures. In addition to these codes, you can also choose the measurement system and formatting that will be used throughout the project.

11. Click on **Code Setup** tab and click on the [...] button beside **Code setup**. Now, the **Format and measuring system** dialogue box is opened. Choose **Imperial format** and click **[OK].**



<u>Notes:</u>

*On the **Basic data** tab, you can set a project level. If you choose "standard", the program will only show the most frequently used basic functions. If you choose "advanced", all basic functions will be shown.*
*On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program and the analysis.*
*On the **Load** tab, you will find the value for the acceleration of gravity and the applicable wind and snow loads that can be activated with the Climate loads functionality.*

.

Project management

Save, Save as, Close and open

Before entering the software to complete the frame construction, we will first discuss how to save a project, how to open an existing project and how to close a project. While completing the tutorial, the project can be saved at any time, that way you can leave the program at any time and resume the project from the save location later.

Saving a project

Click on  in the toolbar.

If a project has not yet been saved, the dialog box **Save as** appears. Navigate to the location or the drive where you want to save your project in. Select the folder or subfolder in which you want to save the project and enter the project file name in **File name** field. Once this is complete click on **[Save]** to save the project.

If you press  twice, the project is automatically stored with the same name. If you choose **File > Save as** in the main menu, you can enter a new file name or save location for the project file.

Closing a project

To close a project, choose **File > Close** in the main menu.

A dialog box appears asking if you 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.

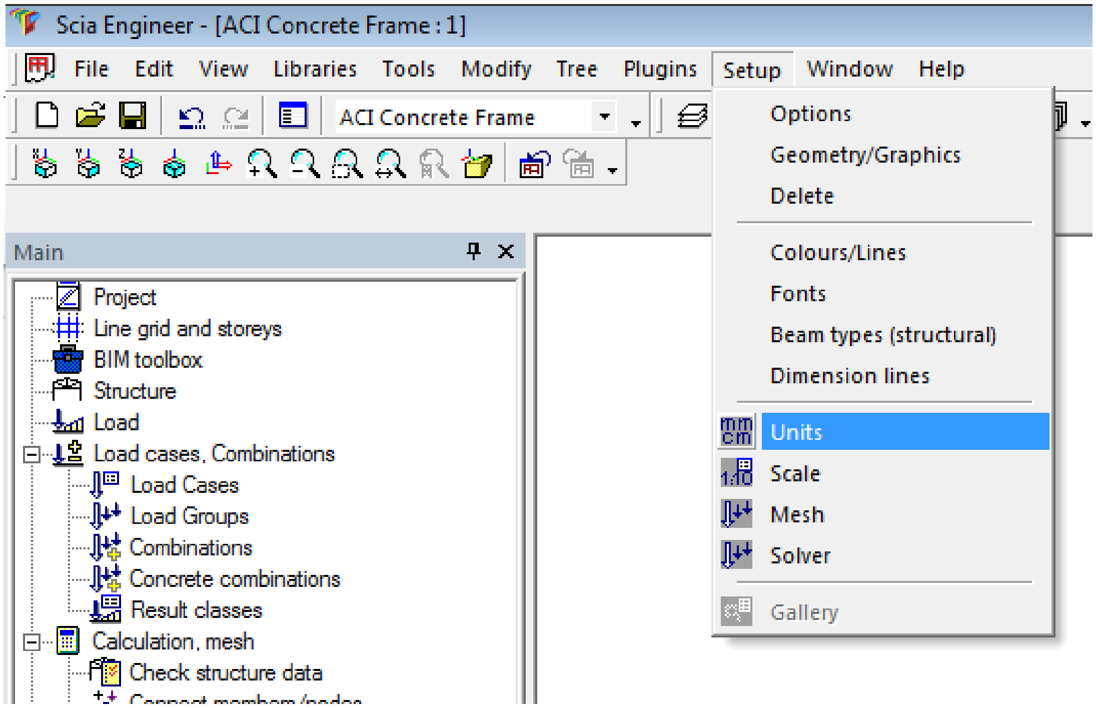
A list with Scia Engineer projects appears. Select the desired project and click **[OK]** (or double-click on the project to open it).

Project Unit Setup

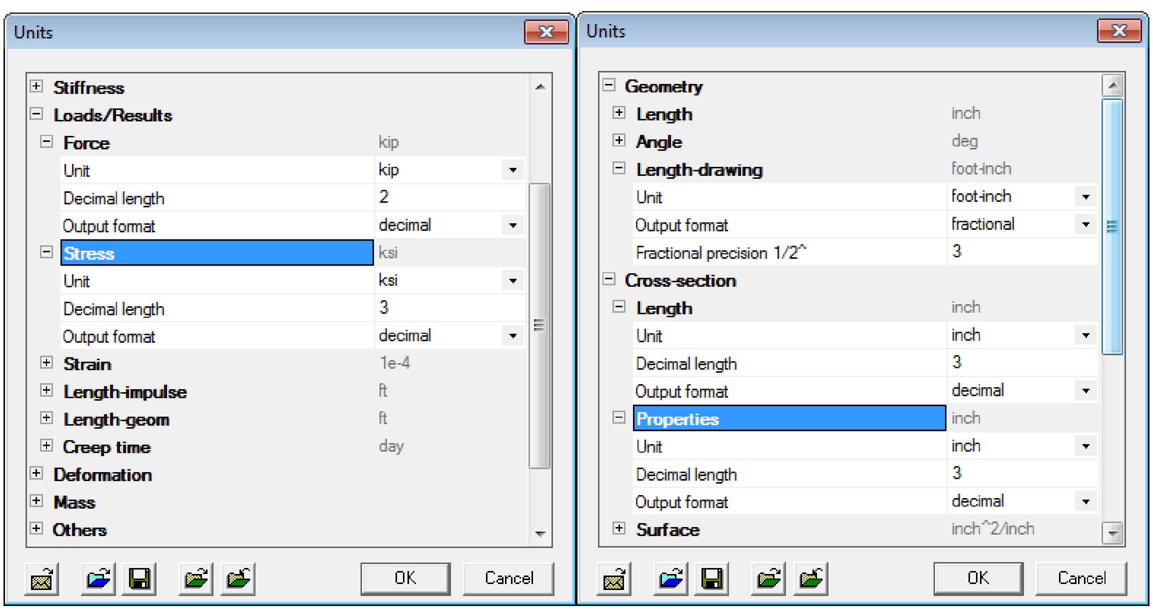
Changing the units

At the beginning of a new project, verify that the units used by Scia Engineer are compatible with your desired units. To check and modify the units:

1. Click on **Setup** in the main menu and select **Units**.



2. In the Units window you can view the units for different options and modify them to your desired unit system.



<u>Note:</u>

It is possible to create a default template file that contains user preferences for desired units, scales and fonts. A template file can be created and saved so that the user can load the desired template file before beginning a new project. The creation and use of template files is not discussed in this tutorial.

Geometry Input

Input of the geometry

When starting a new project, the specific geometry of the structure must be entered. The structure can be entered directly, or it is possible to add geometry using instance templates with parametric blocks, DXF files, DWG files and other formats.

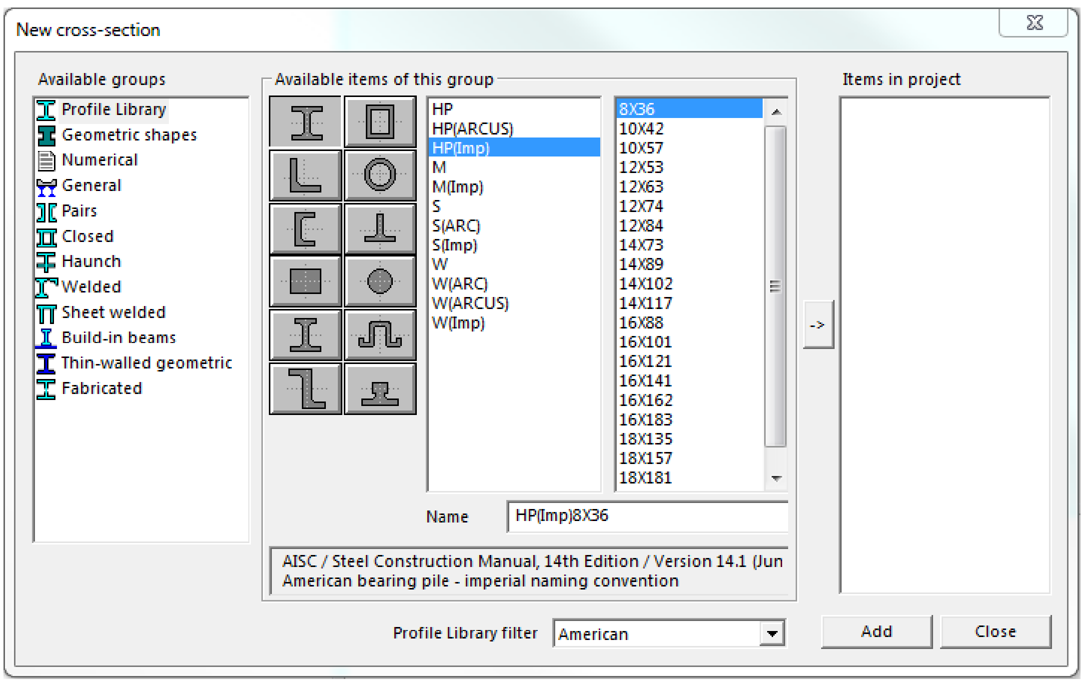
Profiles

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

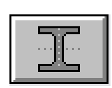
Adding a profile

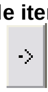
3. Click on the **Cross-Sections**  icon in the toolbar.

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

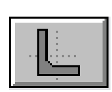


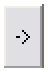
4. Click **Profile library** in the group **Available groups**.

5. In the **Available items of this group**, you can choose a I-shaped profile  . Choose **HP(Imp) 8X36** from the list.

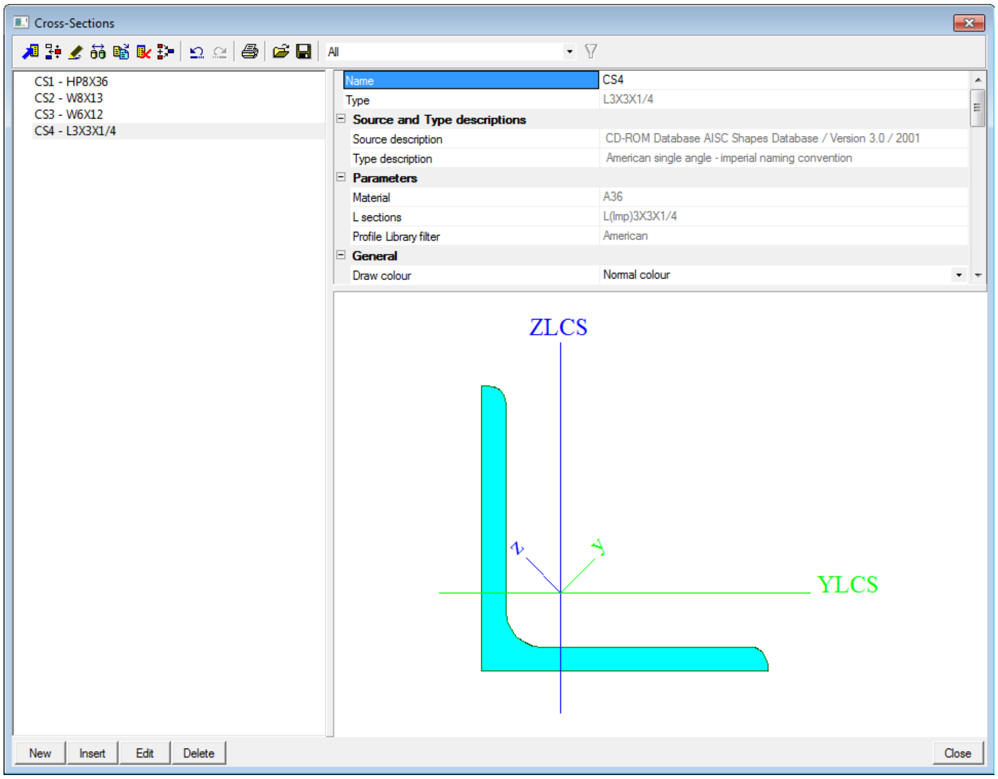
6. Click **[Add]** or  to add the profile to the project.

7. Add **W8X13** and **W6X12** in a similar way.

8. In the **Available items for this group**, you can choose an angle section  . Choose **L(Imp) 3X3X1/4** from the list.

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

10. Click **[Close]** in the **New cross-section** window, the **Cross-Sections manager** will appear.

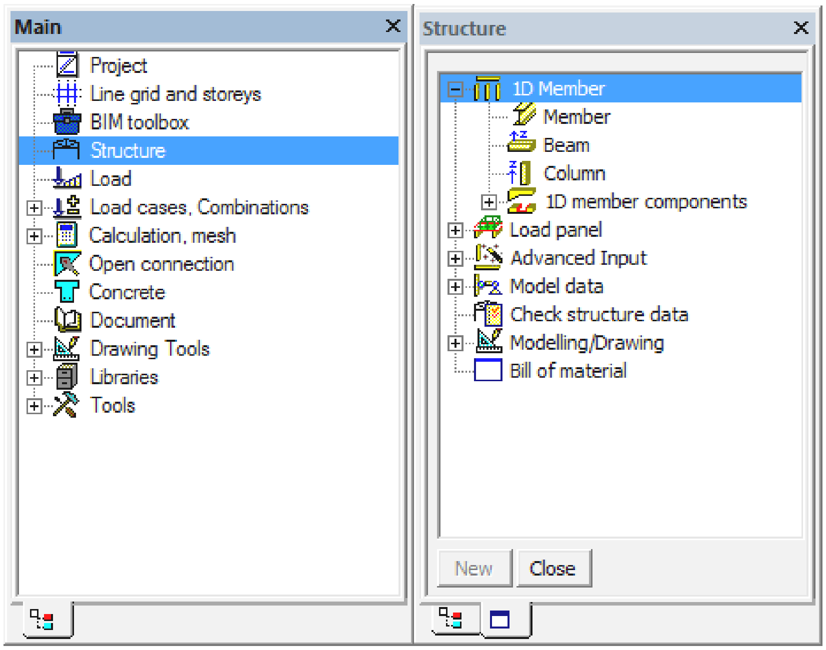


11. Click **[Close]** to close the **Cross-Sections manager** and to return to the project.

Geometry

Structure menu

1. When a new project is started, the **Structure menu** is automatically opened in the **Main window**. If you want to modify the structure at a later time, you must double-click on **Structure** in the **Main window**.



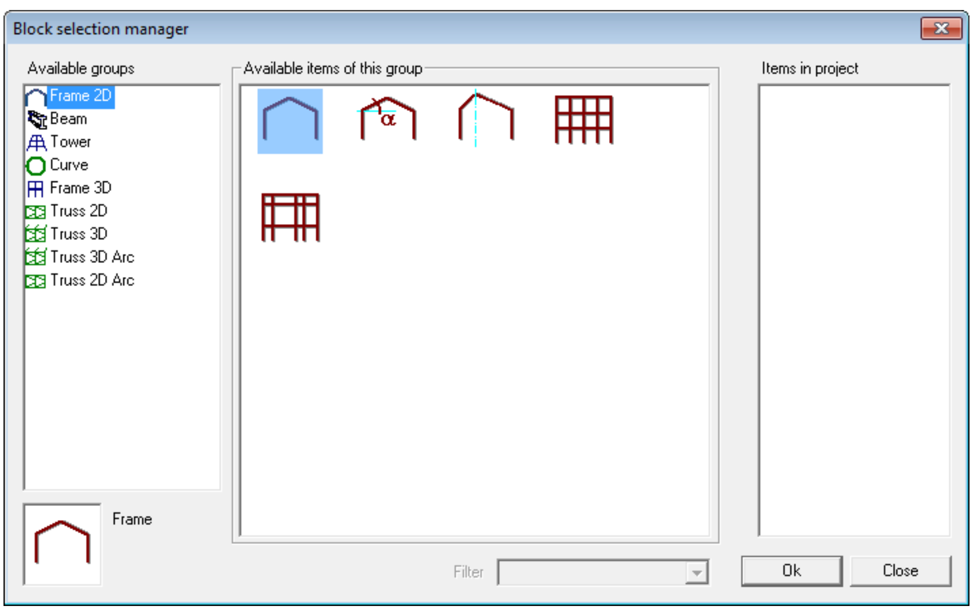
2. In the **Structure menu**, you can choose different structural elements to enter the structure.

To set up the structure, you must enter the first frame. After doing so, the frame can be copied and the wind bracings and the horizontal beams will be entered.

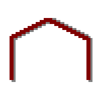
To enter the frame, you can simply add members such as columns and beams. Scia Engineer however offers multiple Catalogue blocks, allowing for a smooth and simple input of the structure.

Entering a frame using a Catalogue Block

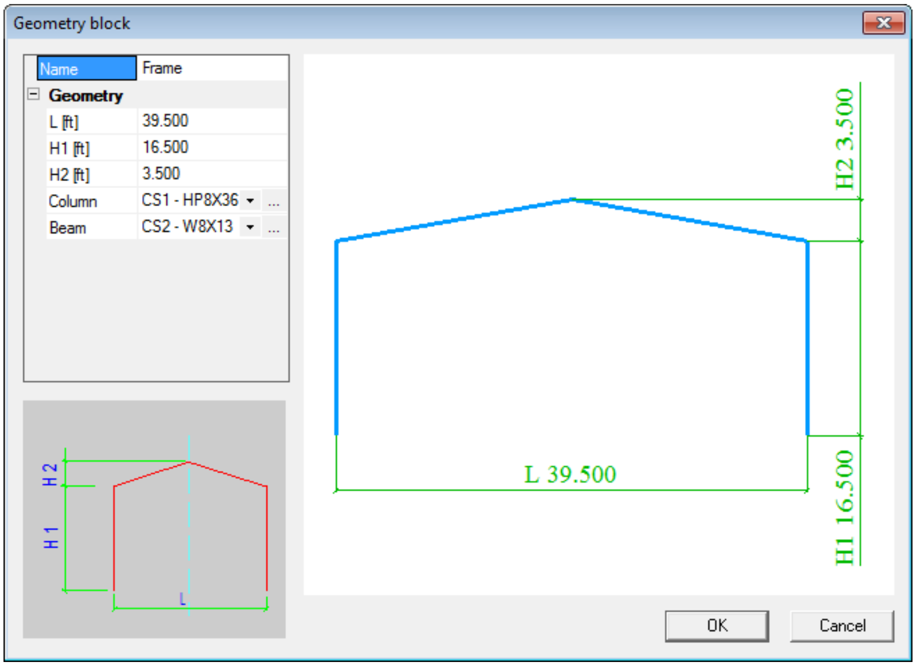
1. To enter a new frame, use the option **Advanced input > Catalogue Blocks** in the **Structure menu**. The **Block select manager** is opened.



2. In the **Available Groups** group, choose a **Frame 2D**

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

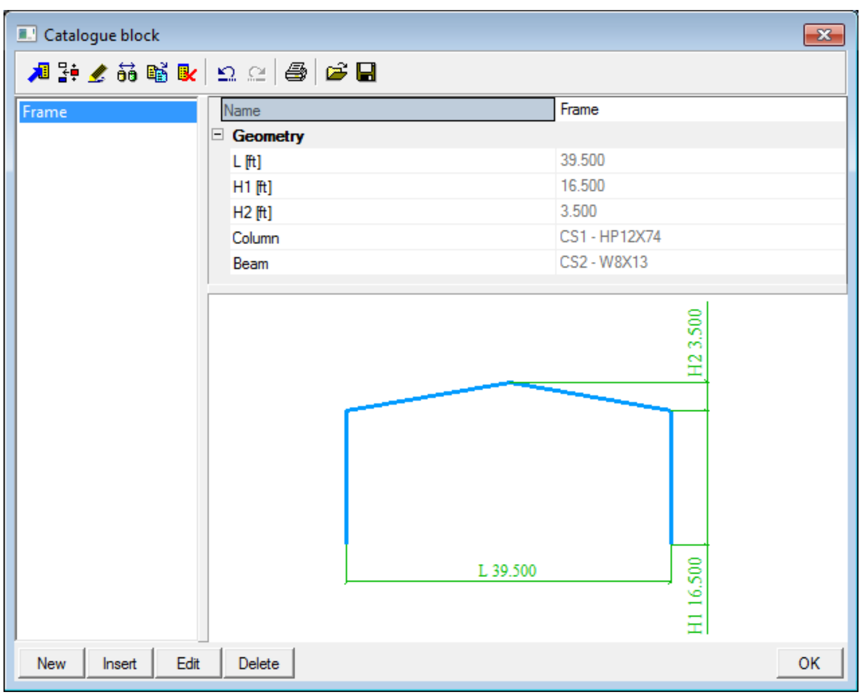
4. Confirm your choice with **[OK]**. The **Geometry block** window appears.



5. Now, enter the frame dimensions: **L = 39.5 ft**, **H1 = 16.5 ft** and **H2 = 3.5 ft**

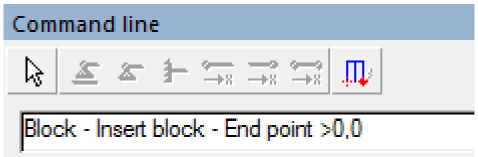
6. In the pull-down menu, choose **HP 8X36** for the **Column** and **W8X13** for the **Beam**.

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



8. Click **[OK]** to return to the project.

9. The frame is positioned with the left column at the origin of the coordinate system. To complete this, type the coordinates **0,0** in the **Command line** and press **<Enter>** to confirm your input.



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

<u>Notes:</u>

*The properties of selected elements are shown and can be modified in the **Properties window**. If no 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, brace, etc.). At any time, you can end your active input by pressing either the **<Esc>** key either the right mouse button.*

*To quickly visualize the entire structure, click **Zoom All***  *in the toolbar.*

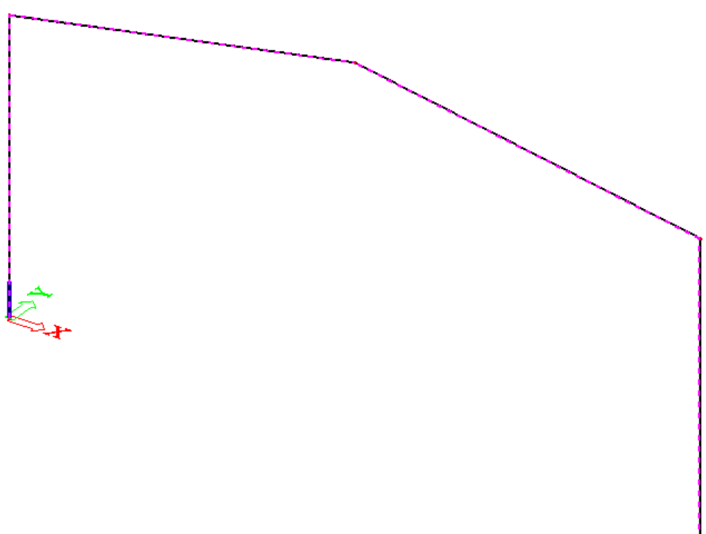
When entering coordinates using the command line, separate the desired coordinates using either a comma (,) or a space.

After input of the first frame, it can be copied in order to obtain the structure for the entire hall. Because the hall requires two additional steel frames, you can use the **multiple copy** option  to complete this task.

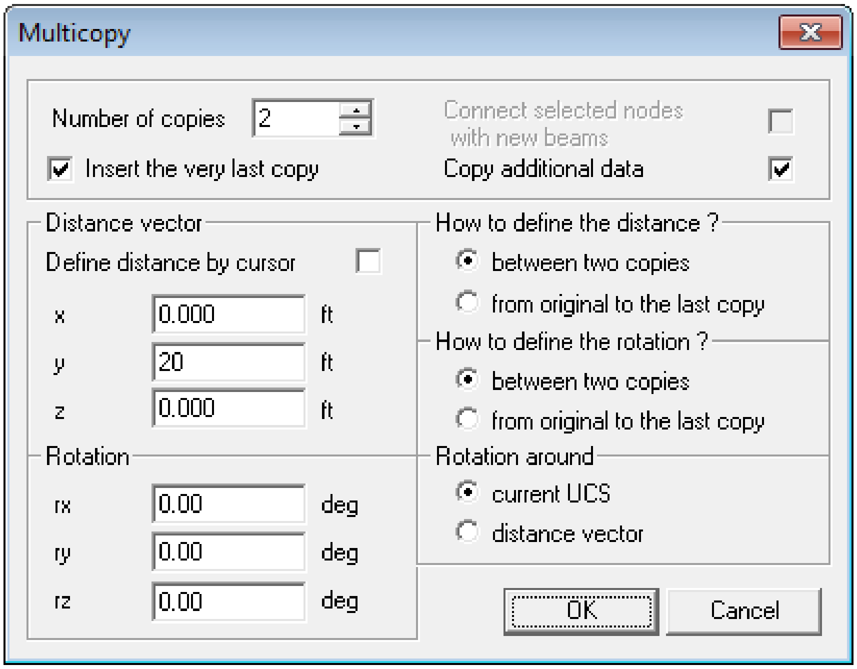
Create multiple copies

1. First select all entities to be copied. Since it is necessary to copy all the beams and columns in the frame, you can use the **Select All** icon .

   In this way, all beams and nodes are selected; this is represented in magenta:



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

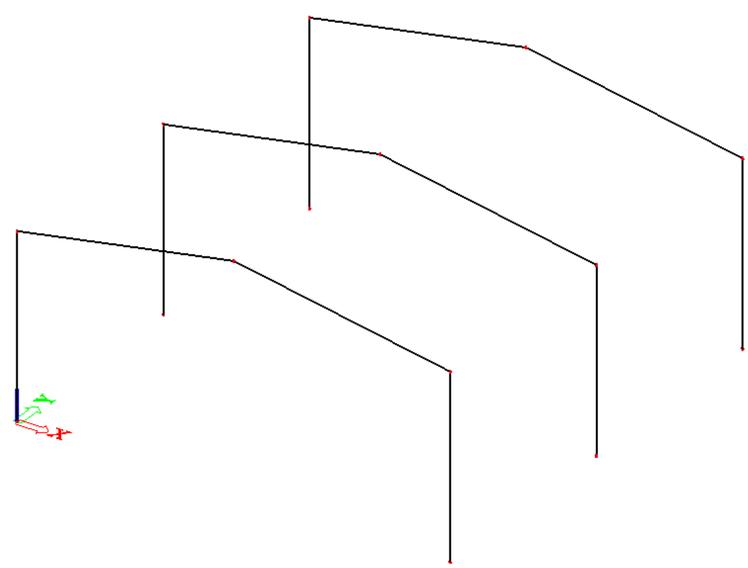


3. In the **Number of copies** field, enter **2**.

4. To manually set the distance between the frames, deactivate the **Define distance by cursor** option. Now, you can enter the distance **20ft** as the y distance vector.

5. Click **[OK]** to confirm your input. The frame has now been copied.

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



After the frames have been entered, the connecting beams of the frames can be entered. The start and end nodes of the beams are already known, i.e. the start and end nodes of the entered beams or columns. Therefore, you do not have to enter the beams by means of coordinates; instead you can use the **Cursor snap settings.**

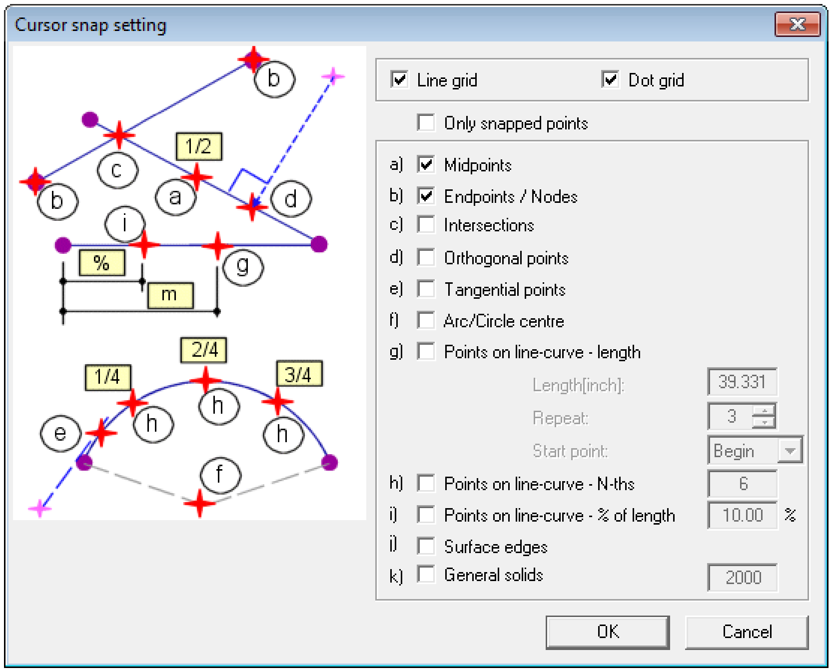
Cursor snap settings

1. Double-click on the **Cursor snap settings** icon  in the Command line or click on the button 

at the lower right of the screen. The **Cursor snap settings** window is opened:

2. Activate the options a) and b) to pick the midpoints and the end points of members in this project.

3. Click **[OK]** to confirm.

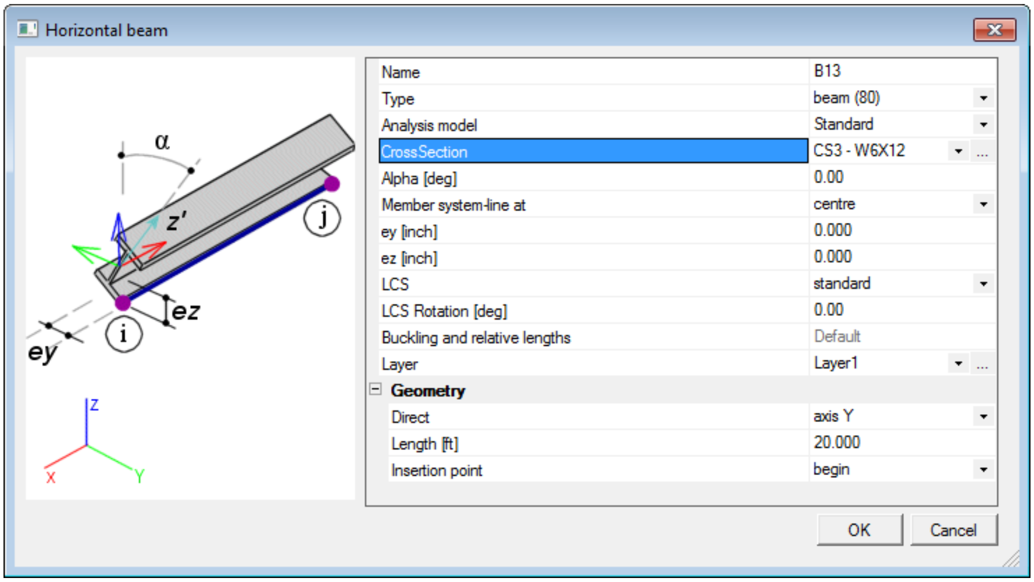


Now, you can enter the beams that connect the individual frames.

Entering a beam

1. To enter a new beam, you use the **Beam** option in the **Structure menu.**

2. In the **Cross Section** field, choose the third section, **W6X12**.



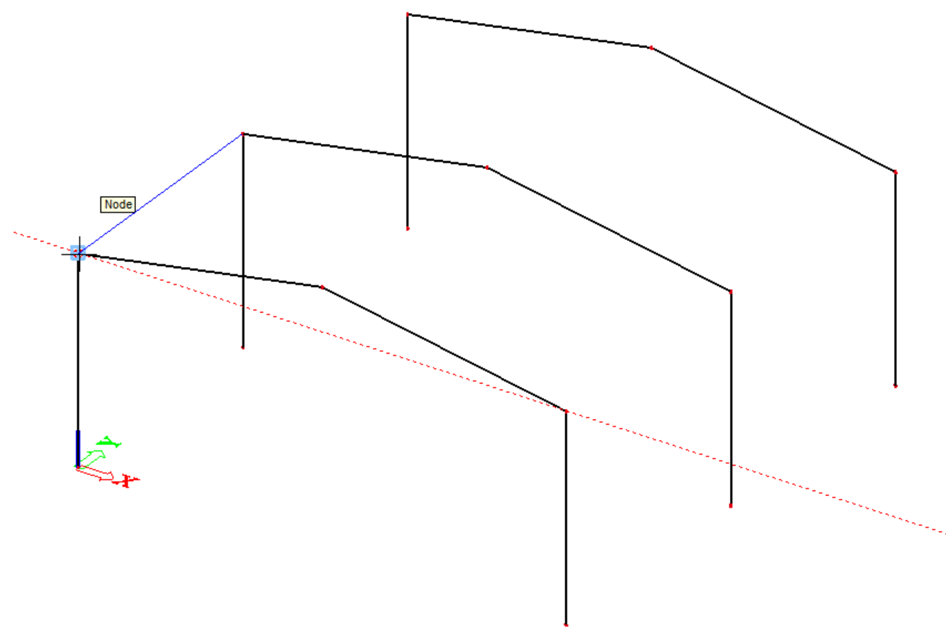
3. Since the structure type **Frame XYZ** has two horizontal directions (i.e. X and Y), you must indicate the right direction for the horizontal beam in the **Direct** field. Choose the **Axis Y** option.

4. Set the beam length to **20ft**.

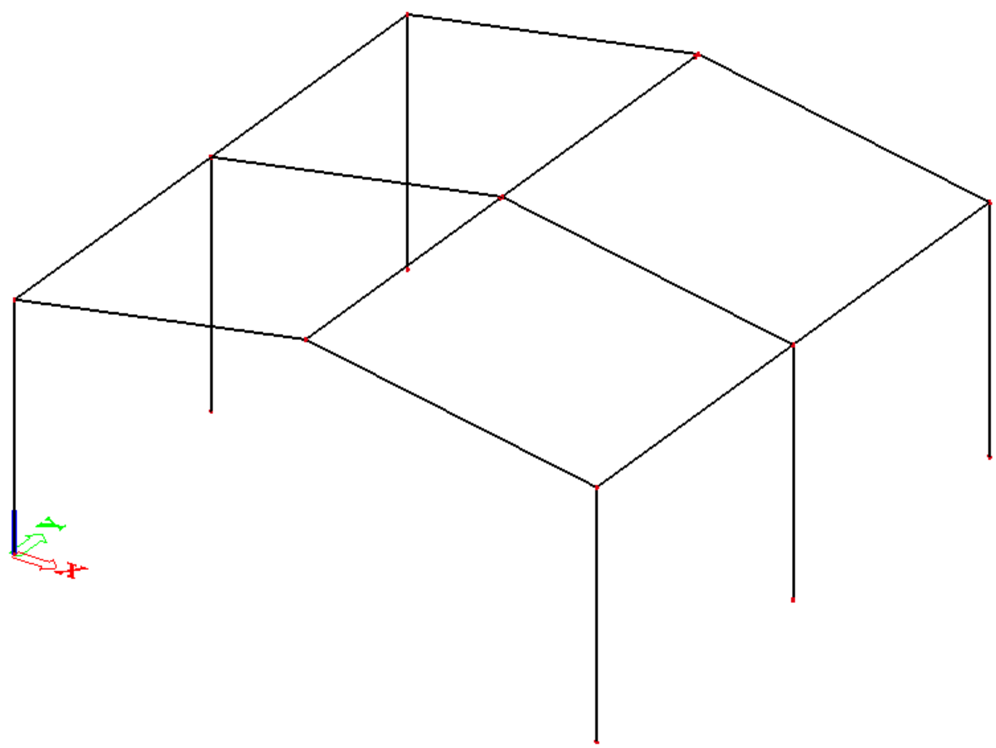
5. The default insertion point is set to **Begin** so that the left point determines the insertion position of the beam.

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:



8. Enter the other beams of the roof in a similar way, by always clicking on the top node of a column.



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

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

<u>Note:</u>

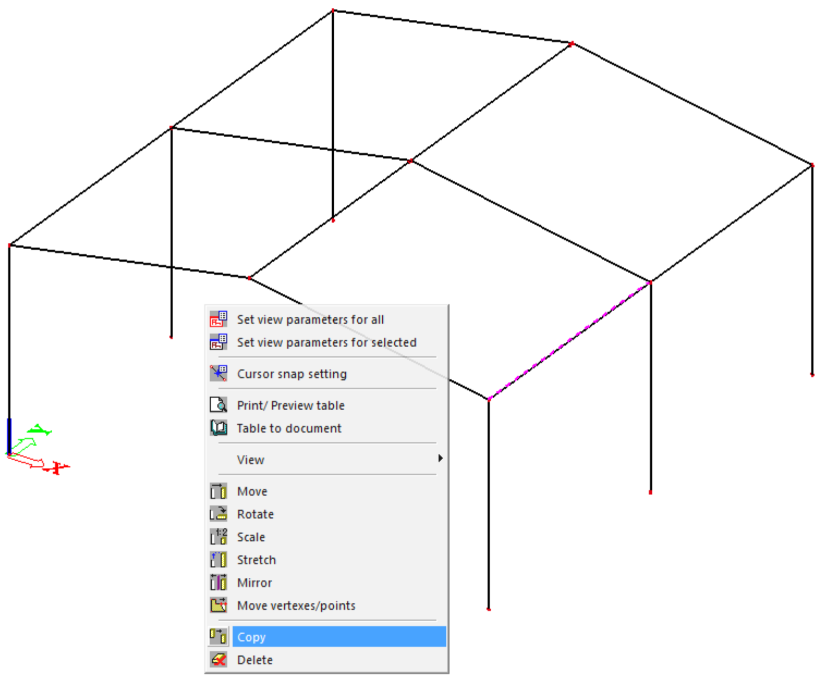
*The **Multiple copy** option also allows for the automatic input of the beams connecting the frames.*

In the location between the first and second frame, we will add two more horizontal beams. To enter these beams, you could use the **Beam** option. Scia Engineer however allows the user to copy these entities manually.

Copying entities

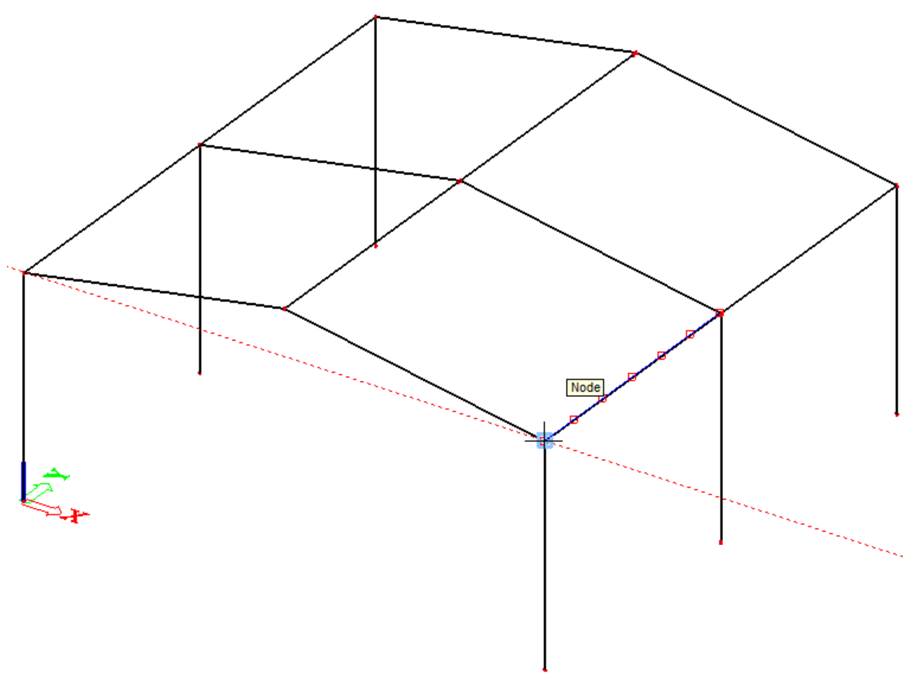
1. First indicate the entity that is to be copied. Since the beam we want to add is a horizontal beam, you can select any of the entered beams with the left mouse button. The magenta color indicates that the beam is selected. The properties of the beam are shown in the **Properties window**.

2. Click with the right mouse button somewhere in the working environment. The menu that lists the available possibilities for the selected entity appears:

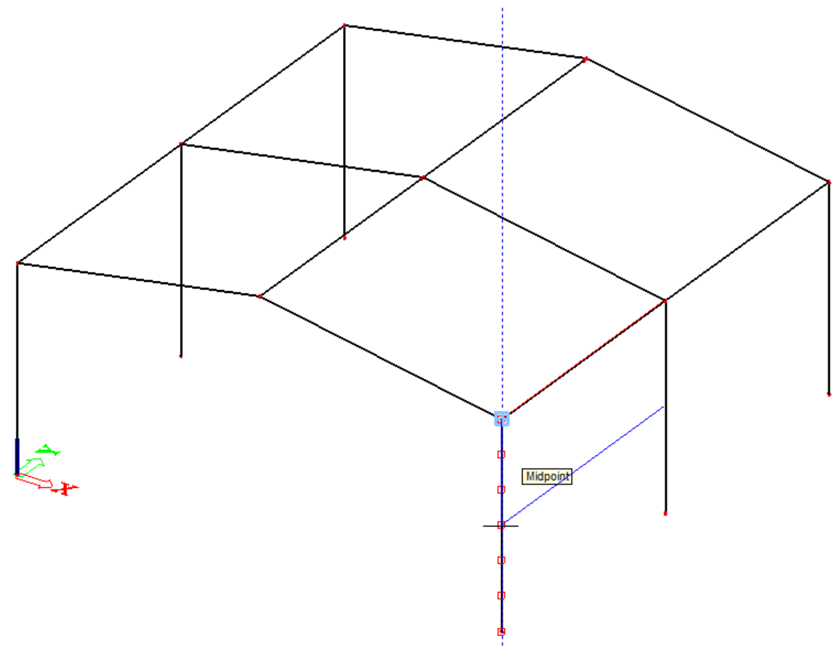


3. In this menu, choose the option 

4. The program asks for the **Start point** of the member being copied. Click with the left mouse button on the start node of the selected beam.



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 of the first frame is chosen.



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

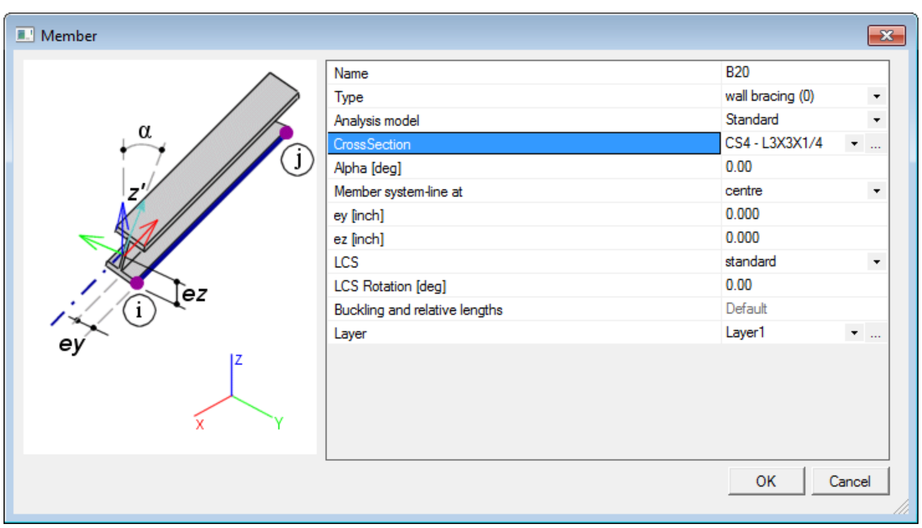
6. After the first beam is copied, the copy command remains active so that you can also pick the midpoint of the second column of the first frame in order to enter a horizontal beam at that position.
7. Press **<ESC>** to finish the input.
8. Press **<ESC>** once more to finish the selection

After the input of the horizontal beams, you can enter the required bracings.

The bracings are not columns or horizontal beams; therefore, you must use the **Member** command in the **Structure menu.**

Entering bracings

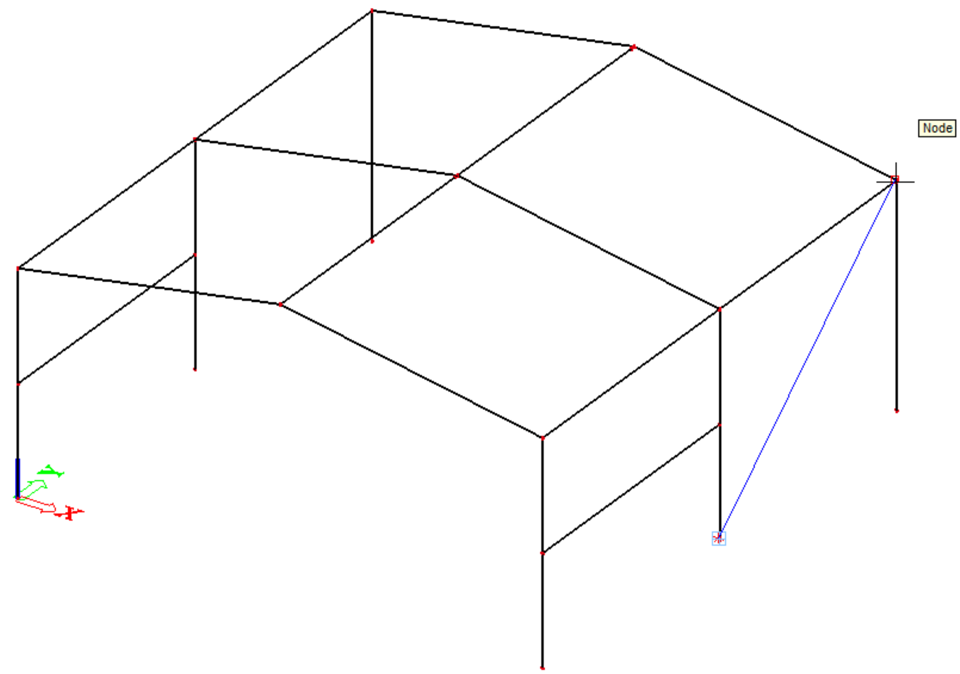
1. To enter a new member, use the **Member** option in the **Structure menu.**



2. In the **Type** field, choose **Wall Bracing (0)**. This member type is only considered for the STRUCTURAL model and does not influence the calculation model or the results.
3. In the **CrossSection** field, choose the steel angle section, **L3X3X1/4**

4. Confirm your input by selecting **[OK]**.

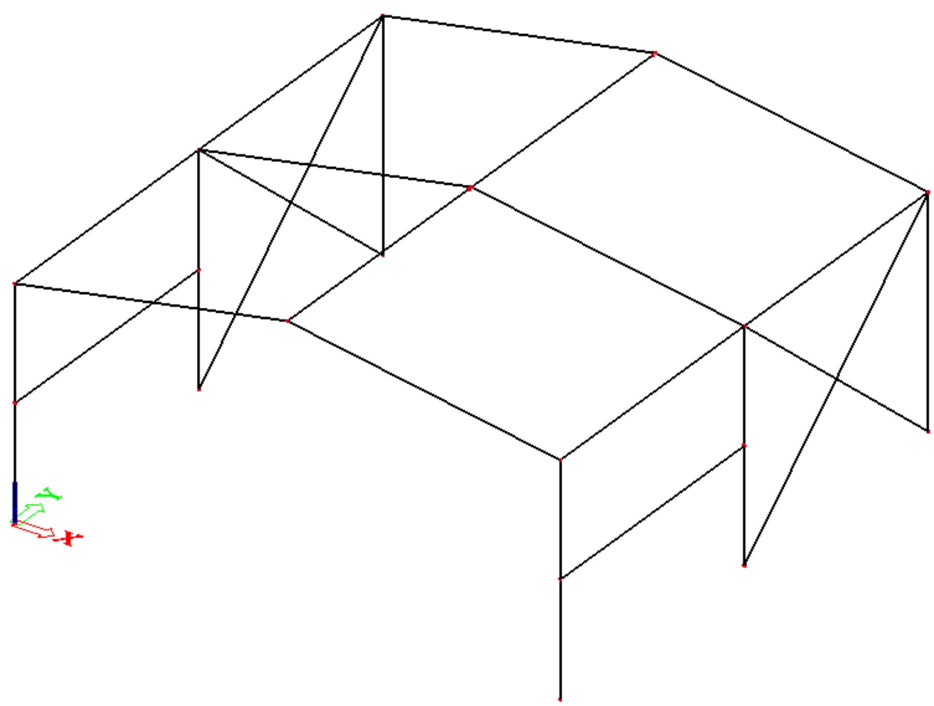
5. Now, the bracings can be entered between the second and third frame of the structure. To accomplish this, make sure to always click on the start and end nodes of the columns that you want to connect the braces with:



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

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

The structure has now been completely modelled. It is now possible to finish the structure by entering additional geometry, i.e. haunches, hinges and supports.



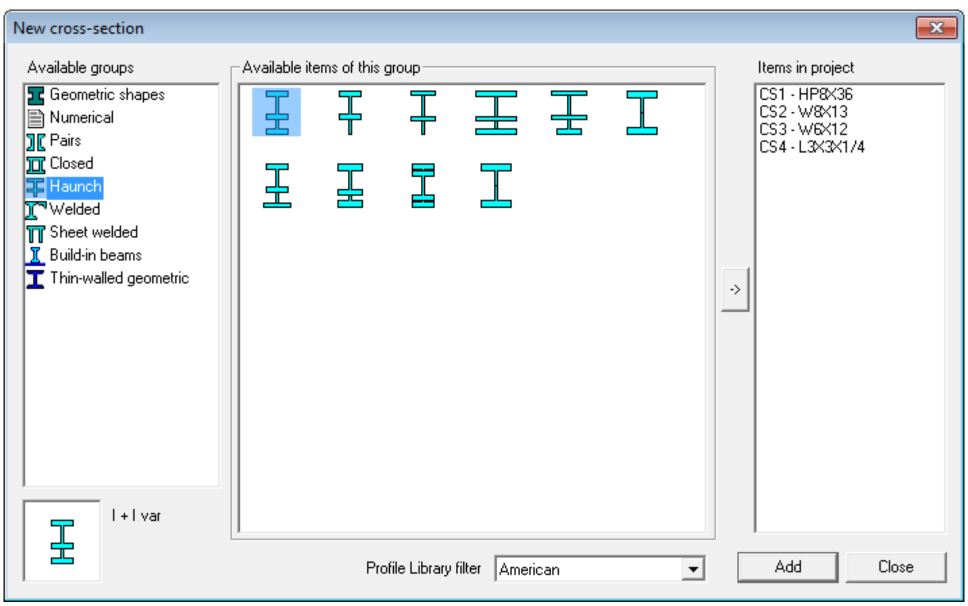
Haunches

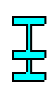
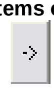
In this project, haunches are entered on the roof beams, at the connecting positions with the columns. In Scia Engineer a haunch is defined by the following parameters:

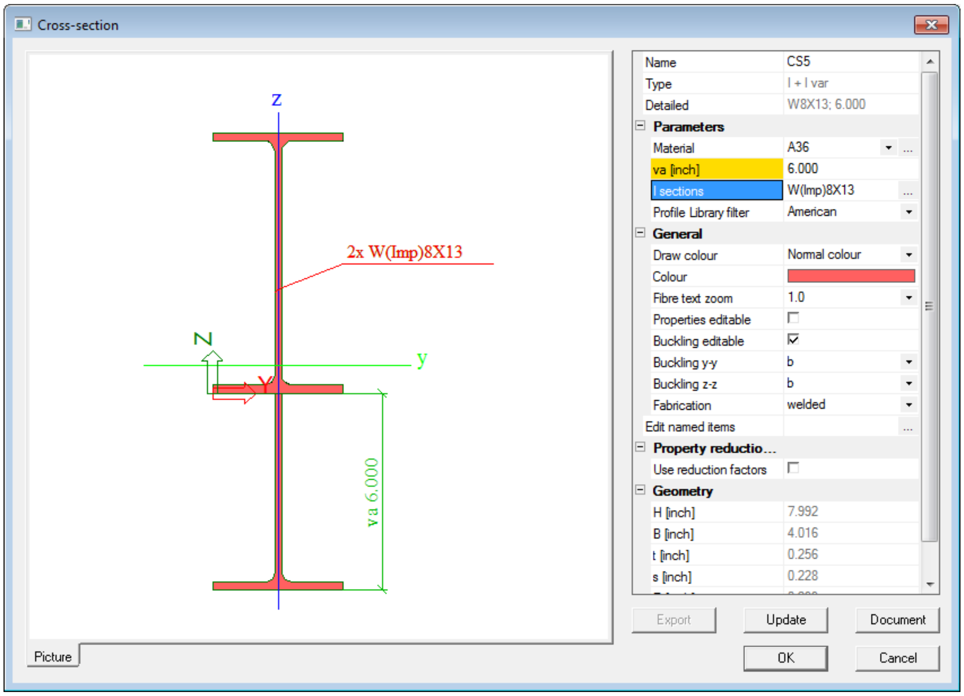
- A section with variable height
- A length, over which the variable height must vary up to 0

Entering Haunches

1. To enter a new haunch, use the **1D member components > Haunch** option in the **Structure menu**.
2. As indicated, a haunch requires a variable profile. Since this project does not yet contain any variable profiles, the **New cross-section** window automatically appears.
3. In the **Available groups** menu, select 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 then appears where the member properties attributed to the variable section can be modified.



6. In the **I sections** field, change the section to **W8X13** by clicking the button behind the section type.

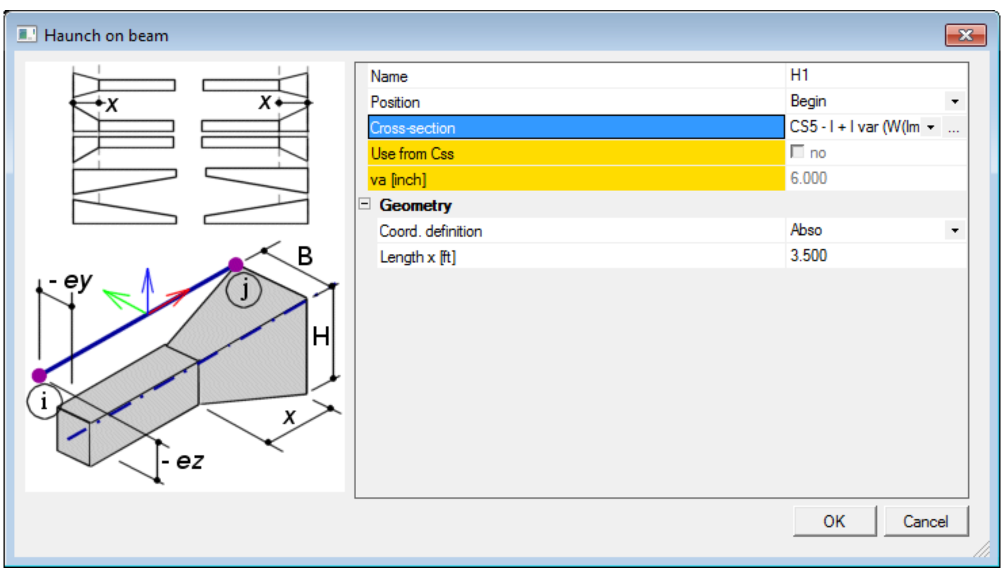
7. When the correct I-section is selected, the variable height **va [inch]** can then be set as **6 inches**.

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

9. The **New cross-section** window reappears; click **[Close]** to close this window.

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

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



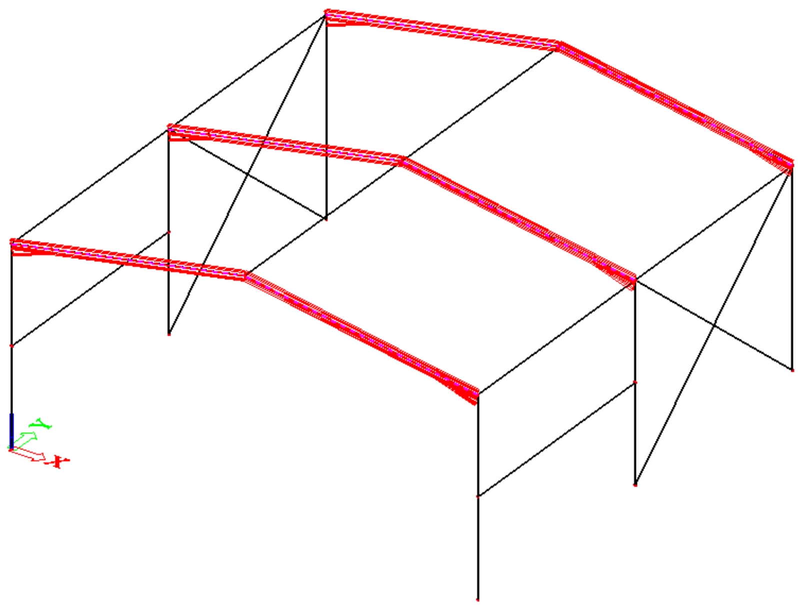
12. In the **Position** field, choose **Begin** to position the haunch at the start node of the member.

13. 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 feet.

14. When the **Coord. Definition** is adapted; the length of the haunch can be entered in the **Length x [ft]** field. For this project, enter length **3.5ft**.

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

16. The program will now ask you to indicate the members, on which a haunch must be entered. Select the 6 roof beams with the left mouse button:



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

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

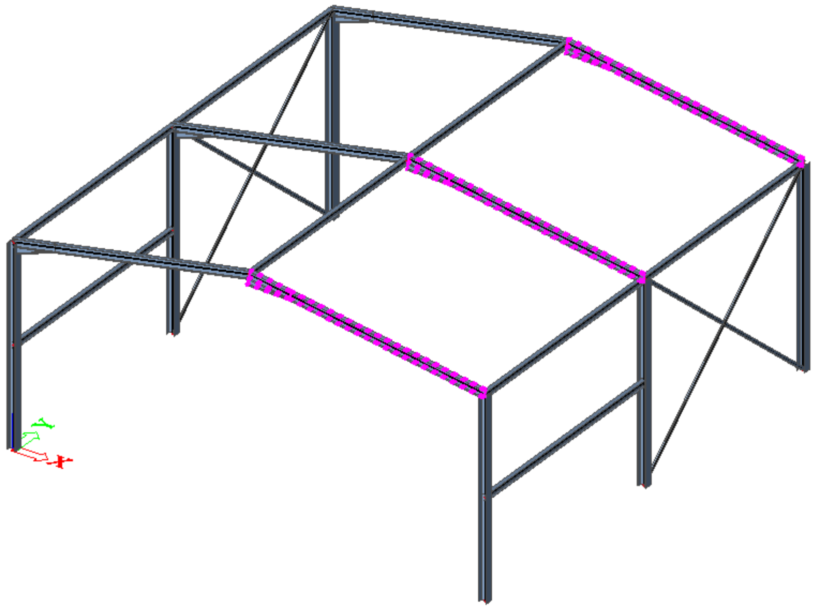
As the previous image already shows the haunches are located in the correct locations on all 6 roof beams. However, depending on how you input the roof beams initially and which nodes you chose as the Start nodes the haunches may be incorrectly positioned on one side of the hall. To visualize this situation, 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.

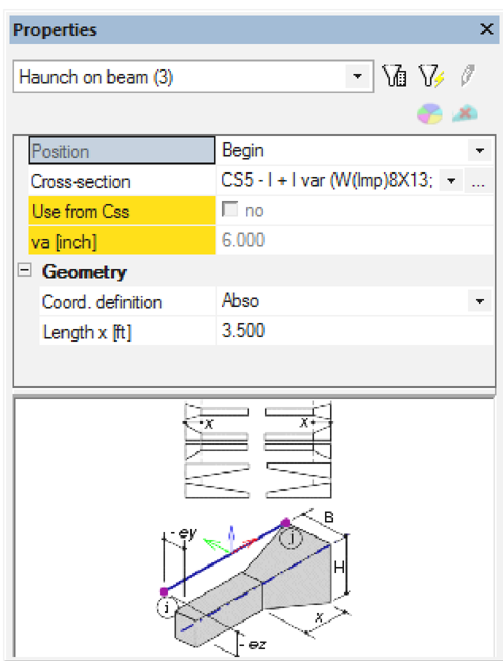
The position of the three haunches that were input incorrectly can be corrected in the **Properties window**.

Adapting Entities through the Properties window

1. Select the 3 haunches to be adapted with the left mouse button.

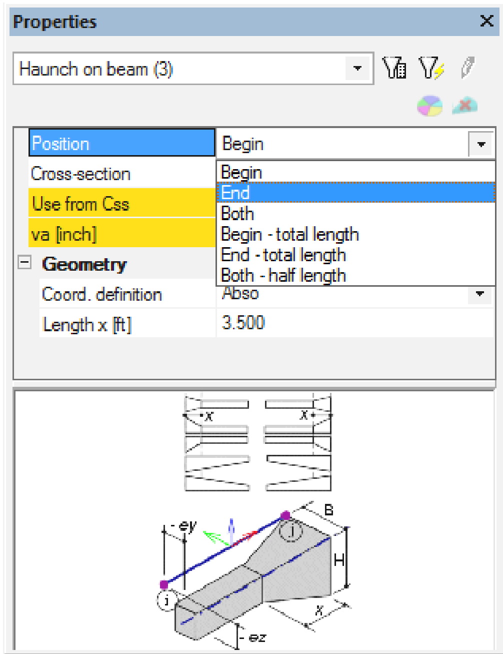


2. The **Properties window** shows the common properties of these 3 entities



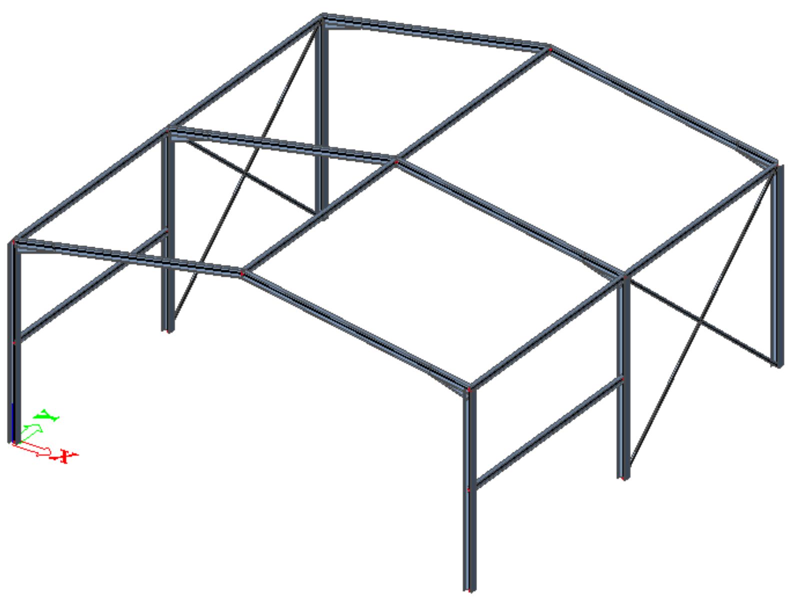
3. Here, you can see that the **Position** is set to **Begin**, in accordance with your input.

4. Change this **Position** to **End**



The modification is immediately reflected in the graphical window.

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



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

<u>Note:</u>

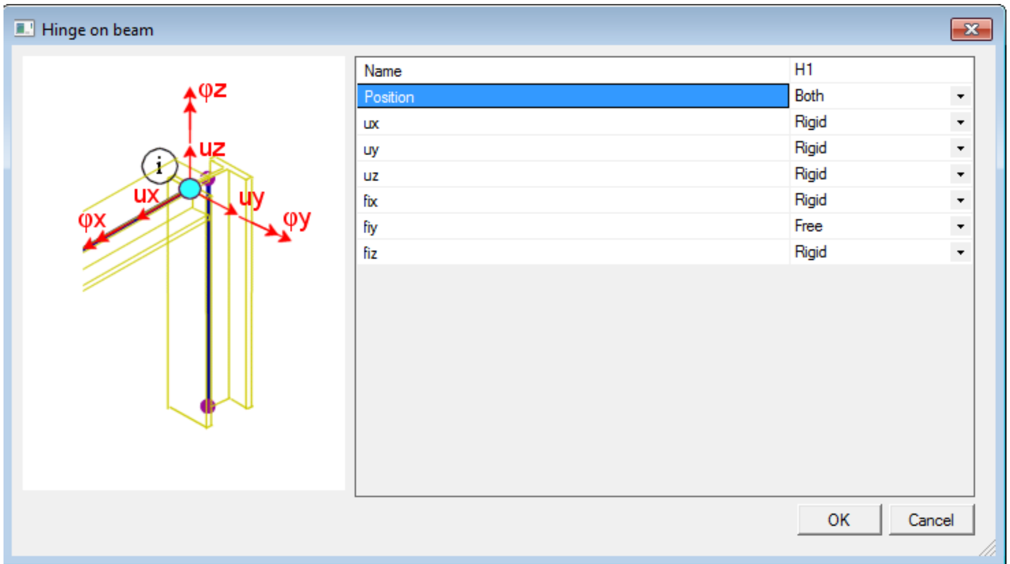
A haunch entered into the structure overwrites the original section that was chosen for a specific beam. For this project, this specifically means that the profile of the roof beam is replaced by the I + I var profile. If the haunch is removed, the I + I var profile will be maintained instead of the I-section of the roof profile.

Hinges

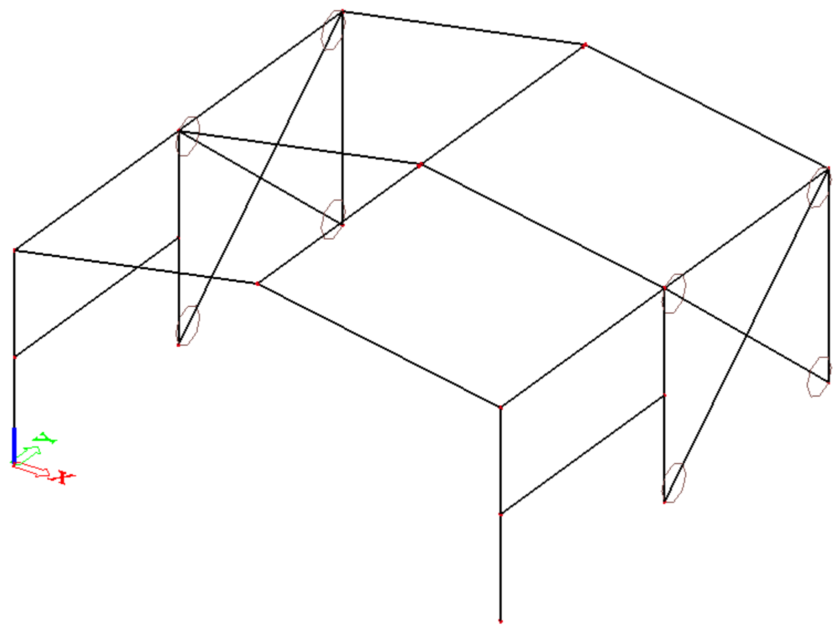
In this project, the braces are connected with the other members in a hinged manner. As the chosen structure type is Frame XYZ, the structural elements are connected to each other in a rigid manner. Therefore, you must enter hinges manually on the structure.

Entering hinges

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



2. The hinges are entered at both extremities of the diagonals; therefore, choose **Both** for the **Position**.
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.

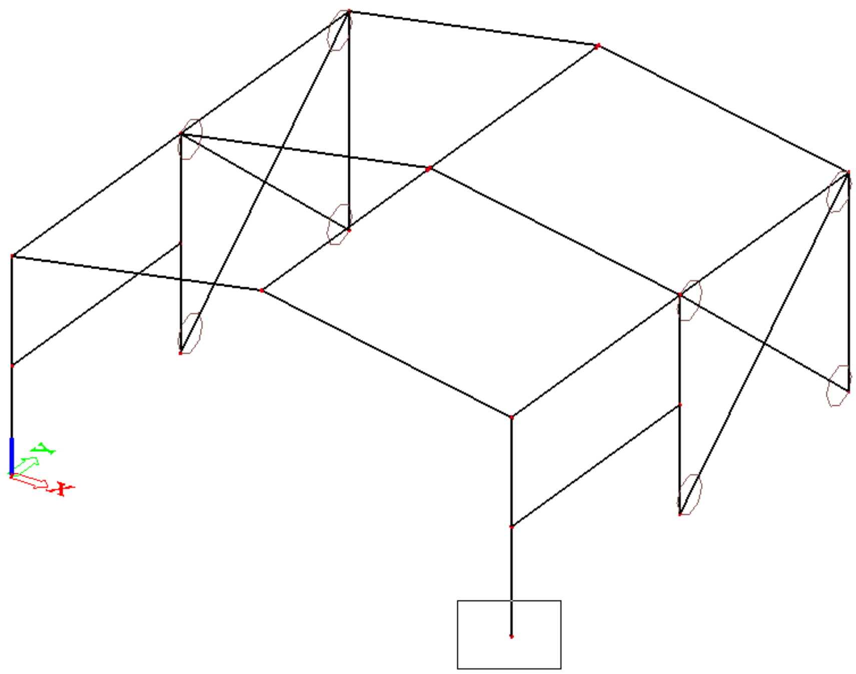
Supports

The geometry input of the structure will be completed by adding supports to the structure. The column bases of the structure will be modelled as hinged supports.

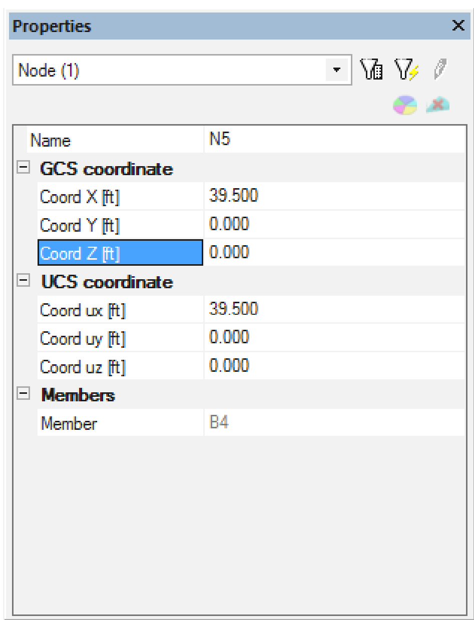
Before adding the supports, you first must select the nodes that the hinged supports will be added to. You can select these nodes manually, one by one, or Scia Engineer offers a simple method to select entities with a common property.

Selecting elements per property

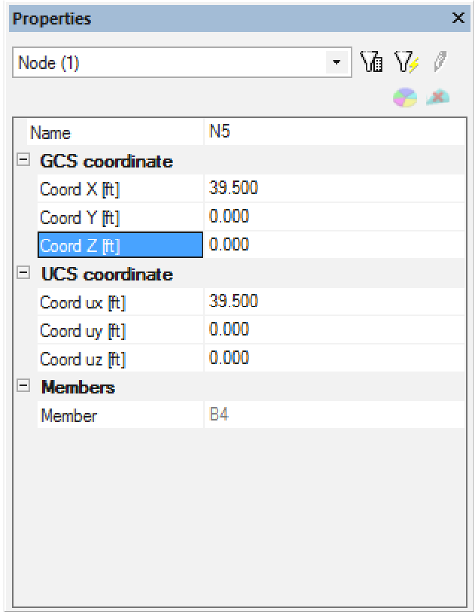
1. To select all column bases, select one of the lower nodes, by drawing a frame from the left to the right with the mouse.



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 lower nodes. The common property of these nodes is their coordinate in the global Z direction. To accomplish this, click with the left mouse button on the **Coord Z (ft)** property to select the field of this property.

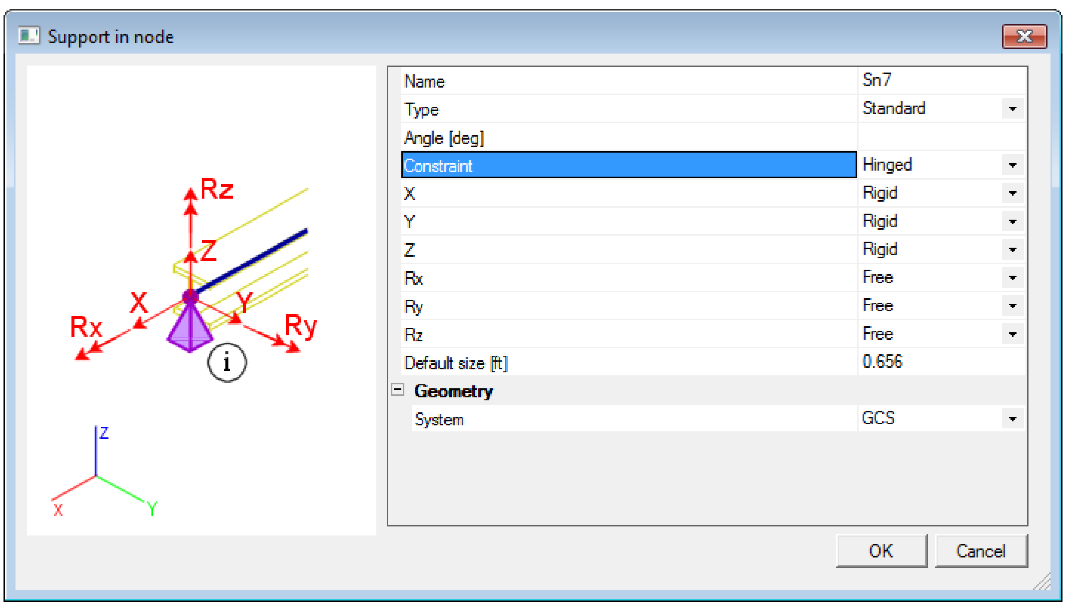


4. Choose the **Select elements by property** option. The program will search for all entities within the structure that have the same property. In this example, the program will select all nodes, for which the **Coord Z (ft)** property corresponds to **0 ft.**

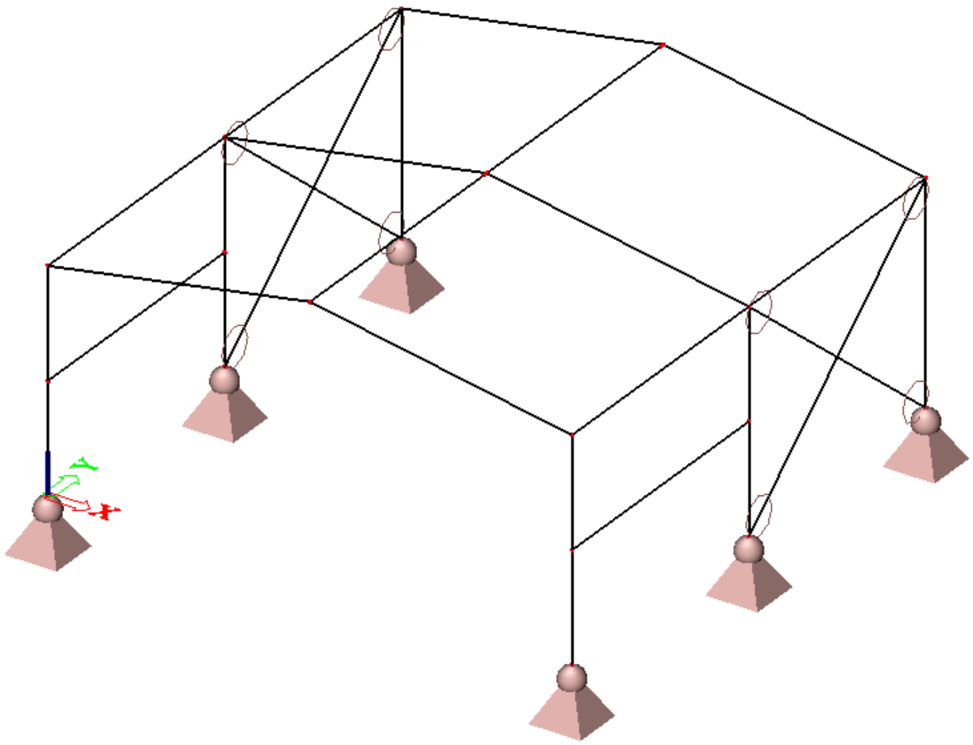
Upon completion of the previous step, all column bases are selected. It is now possible for supports to be added to these nodes.

Entering supports

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



2. To create a hinge, select the 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.



<u>Notes:</u>

If you draw the box from the left-hand side to the right-hand side, only entities, which are completely in the rectangle, 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 were to be entered. The determinant property here would have been the Section attributed to the braces.*

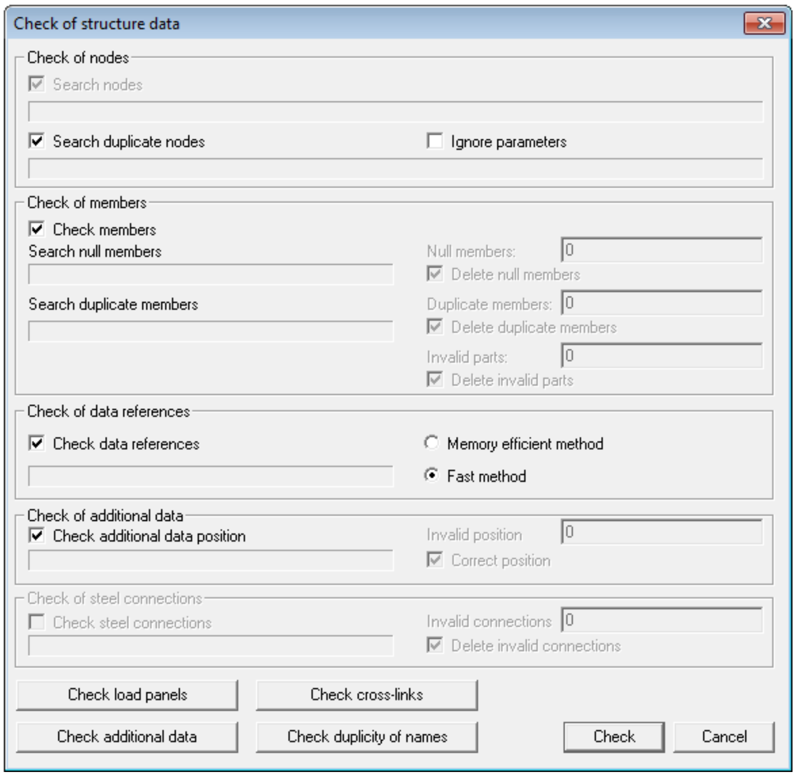
Check Structure data

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 beams, duplicate hinges and duplicate supports.

Checking the structure

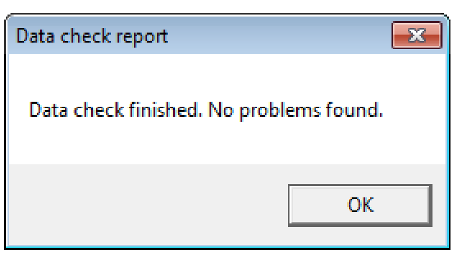
1. Double-click on the **Check Structure data** option in the **Structure Menu** or click on the  icon in the toolbar.

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]**.

Connecting entities

In the case of this structure, the columns and roof girders share a common node. The end node of the column for instance is the start node of the roof girder. Therefore the girder is automatically connected to the column.

The two beams connecting at the middle of the columns are not ending in nodes. The end nodes of the beams are located inside the column and therefore are not yet connected to the columns. In this paragraph, we will explain how to connect the beams to each other at points other than typical end or start nodes.

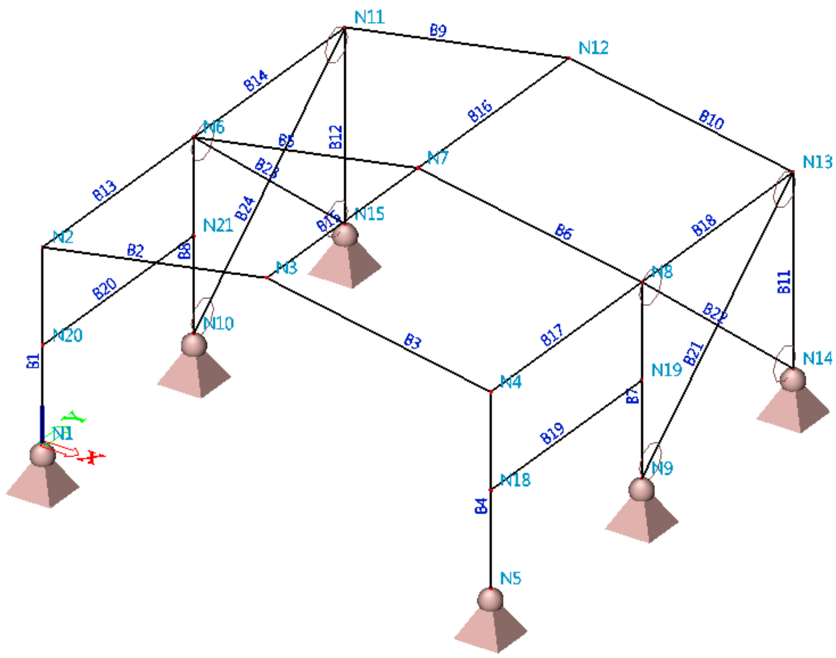
To display the names of the members and nodes, you can activate the labels by means of the buttons in the **Command line.**

Activating node labels

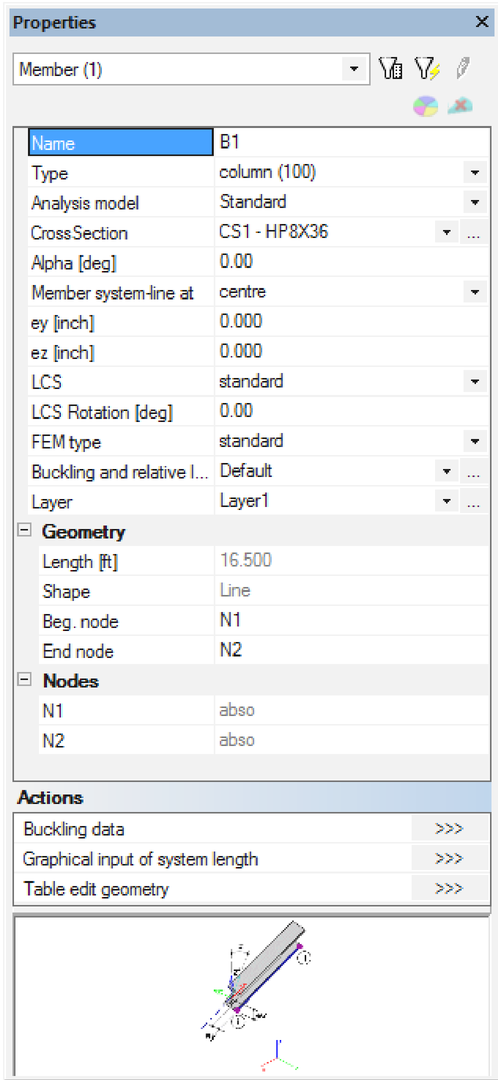
Node labels are activated by means of the  icon on top of the **Command line**.

Activating Member labels

Member labels are activated by means of the  button on top of the **Command line**.



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



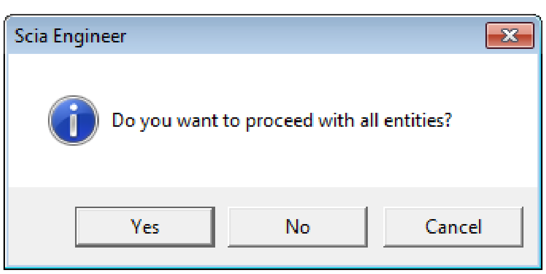
This window indicates that the start node is **N1** and the end node **N2**. Node **N20** is not part of the column. To connect beam **B20** to the columns, use the **Connect members/nodes** option.

Connecting entities

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

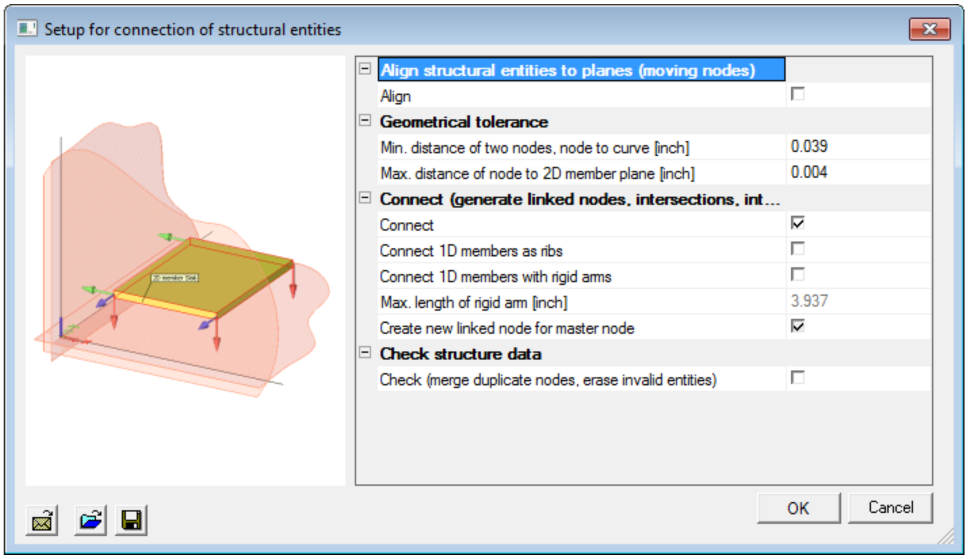
2. Double-click on the **Model data > Connect members/nodes** option in the **Structure menu** or click the icon in the toolbar.

3. A dialogue box asks if all nodes entities are to be connected:



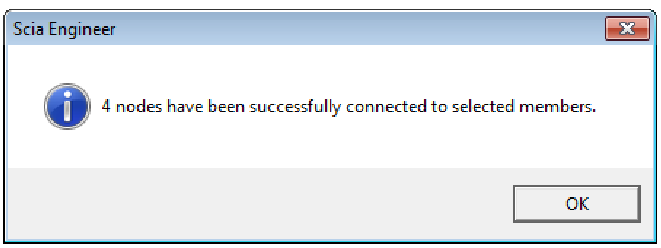
4. Click **<Yes>**.

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

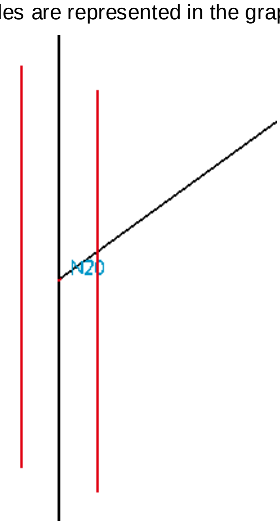


6. Confirm the settings by clicking **<OK>**.

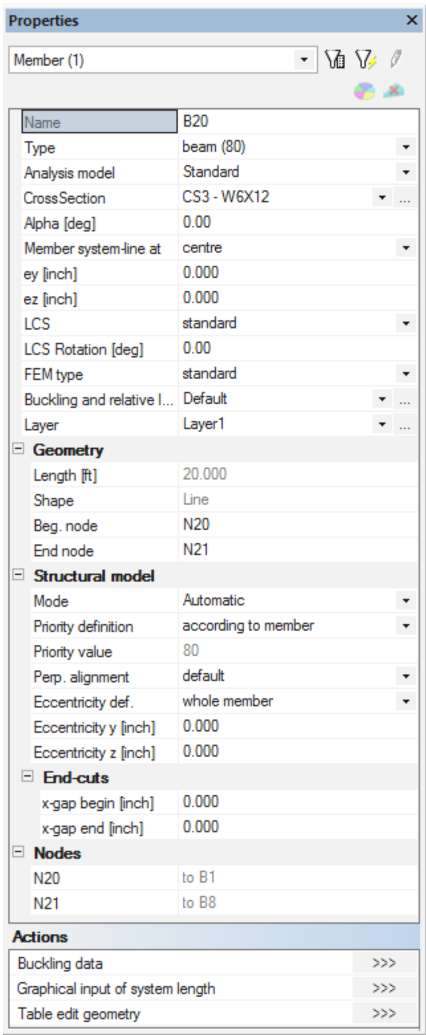
7. A window appears to indicate the number of connected nodes:



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



When you select for instance beam **B20**, the **Properties window** will show that node **N20** connects the girder with column **B1** and that node **N21** connects the girder with column **B8**



Note:

*If a possible active selection is not deactivated when the **Connect members/nodes** command is used, the program will only search the nodes to be connected in this selection and not in the entire project.*

*It is also possible to perform the two previous operations at once. To do this 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**.

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 Construction
- Set a view direction
- Use the magnifier
- Edit view parameters through the menu **View parameters**

Editing the view point on the construction

One of the methods for editing the view point of the structure is through the three wheels in the bottom right of the graphic window; two are horizontal and one is vertical. With these **wheels** you can **zoom in** on the construction or **turn** it.

1. To zoom in on the construction or to turn the structure, click on the appropriate 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.

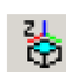
4. Press CTRL + SHIFT + right mouse button at the same time and move the mouse to **zoom in or out** on the construction.

<u>*Note:*</u>
If the structure is being turned while a node is selected, the structure will turn around the selected node.

Setting a view direction with regard to the global coordinate system

1. Click on the button **View in direction- X**  for a view the structure in the X-direction.

2. Click on the button **View in direction- Y**  for a view the structure in the Y-direction.

3. Click on the button **View in direction- Z**  for a view the structure in the Z-direction.

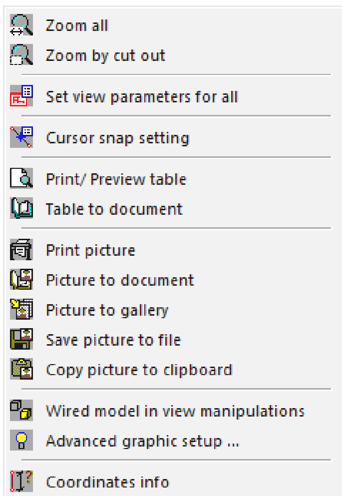
The magnifier

- Use  to enlarge the view size.
- Use  to decrease the view size.

- Use  to zoom in on a window.
- Use  to view the whole structure.
- Use  to zoom in on the selection.

Editing view parameters through the menu View parameters

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



<u>Note:</u>

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

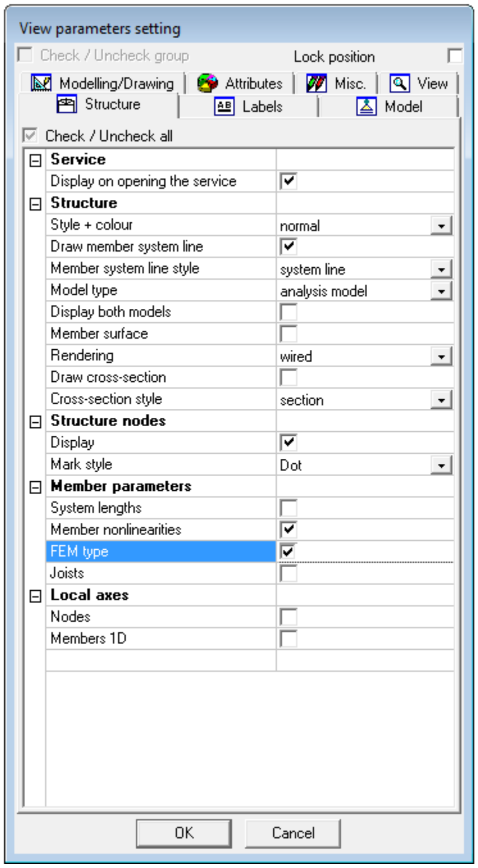
2. Choose the option **Set view parameters for all**. The window **View parameter setting** appears. The menu consists of various tabs. You can set the view parameters for all entities or just for the selected entities.

View parameters – Entities

Through the tab entities the representation of the different entities can be adapted.

In the group **Structure** the following items are important for this project:

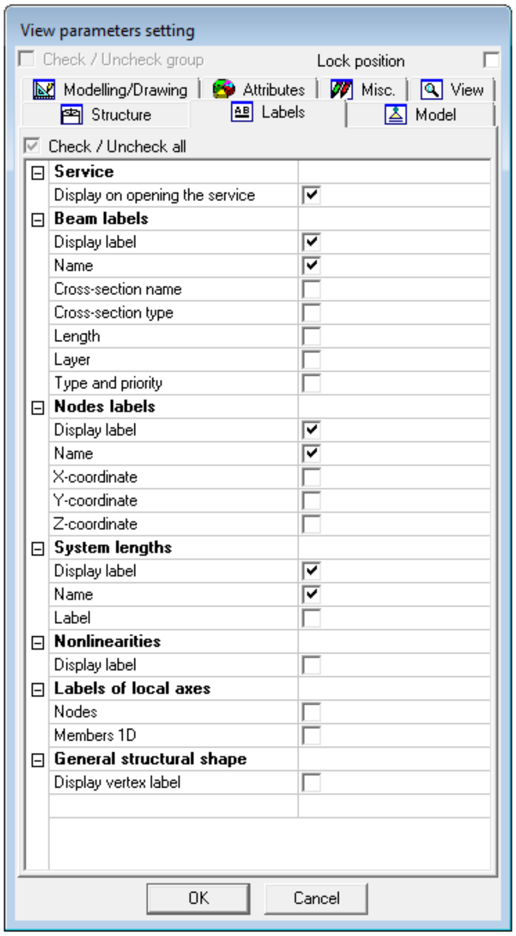
- **Style and color:** You can display the colors per layer, material, cross-section or structural type.
- **Draw cross-section:** With this the symbol of the cross-section is displayed on every member.
- **Local axes:** With this the local axes of the elements are activated.



View parameters – Labels and description

Through the tab **Labels,** the labels of different entities can be displayed. In the group **Members** the following items can be displayed as a label on the structure:

- **Name:** Show the name of the cross-sections in the label.
- **Cross-section type**: Show the cross-section type in the label.
- **Length**: show the length of the member in the label.



View parameters – shortcuts

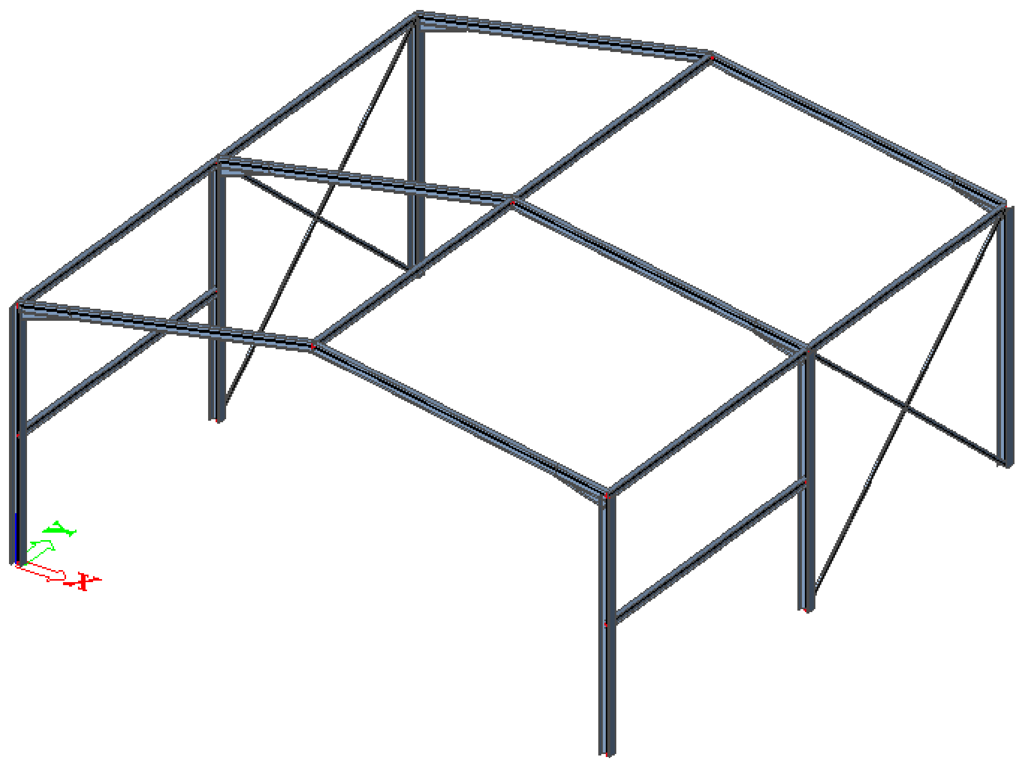
In the toolbar above the **Command line**, several frequently used options are grouped among which:

- **Show/hide surfaces**  to show the surfaces of the cross-sections.
- **Render geometry**  to view the rendered members in the structure.
- **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, etc.)
- **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 the options from the View Parameters menu.

After rendering the following structure is obtained:



Loads and combinations

Load Cases and Load Groups

Each load that is inserted into the project and added to the structure is attributed to a load case. A particular load case can contain many different load types.

Each load can is attributed properties which will determine the proper generation of load combinations. In addition, a specific load case will carry a specific action which can be set as permanent or variable.

In the case of variable load cases, each variable load has its own associated load group. The group contains information about the category of the load (service load, wind, snow, etc.) and its appearance (default, together, exclusive). In an exclusive group, the different loads attributed to the group cannot act together in a normal combination. Default combinations, on the other hand, will allow for simultaneous action of the loads in the same group within the load combination generator.

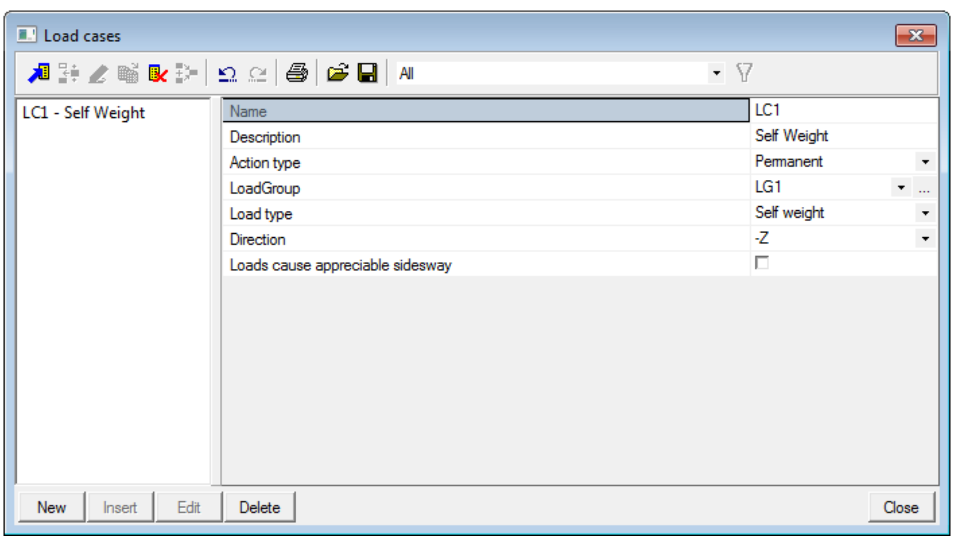
The way in which load cases are defined, is critical for the load combinations created by the generator. We recommend that you thoroughly read the chapter about loads and combinations in the reference manual found in the Help menu.

In this particular project, three load cases are entered:

- **LC1**: Permanent Load Case: Self weight of the steel members
- **LC2**: Permanent Load Case: Weight on the Roof (Roof Dead)
- **LC3**: Variable Load Case: Wind load on the Frame (Columns)

Defining a Self Weight Load Case

1. Double-click on Load in the **Main window**.
2. Before you can define loads, you first must enter load cases. Since this project does not contain any load cases yet, the **Load Cases Manager** will automatically appear.
3. By default, load case **LC1** is created. This load is a permanent load of the **Self Weight** load type. The self weight of the structure is automatically calculated depending on the length and type of steel cross sections entered.
4. In the Description field, you can describe the content of this load case. For this project, enter the description **"Self Weight"**.

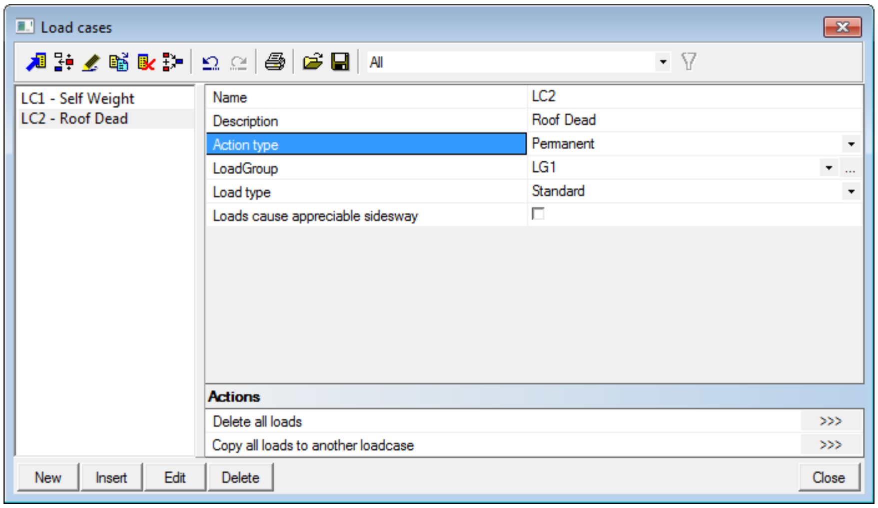


Defining a Permanent Load Case

1. Click New or  to create a second load case.
2. Enter the description **"Roof Dead"**.

3. As this is a permanent load, change the Action type to **Permanent**.

4. Verify that both LC1 – Self Weight and LC2 – Superimposed Dead are in the LG1 LoadGroup.



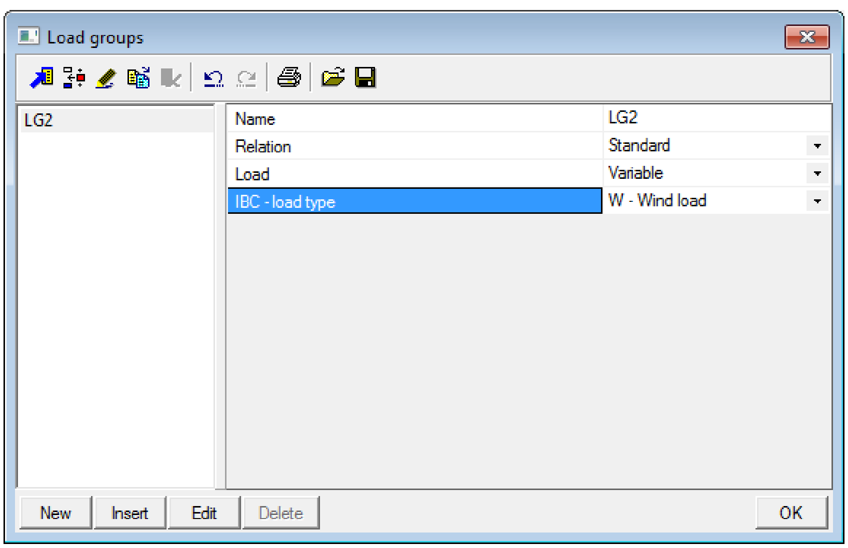
Defining a Variable Load Case

5. Click New or  to create a second load case.

6. Enter the description **"Wind"**.

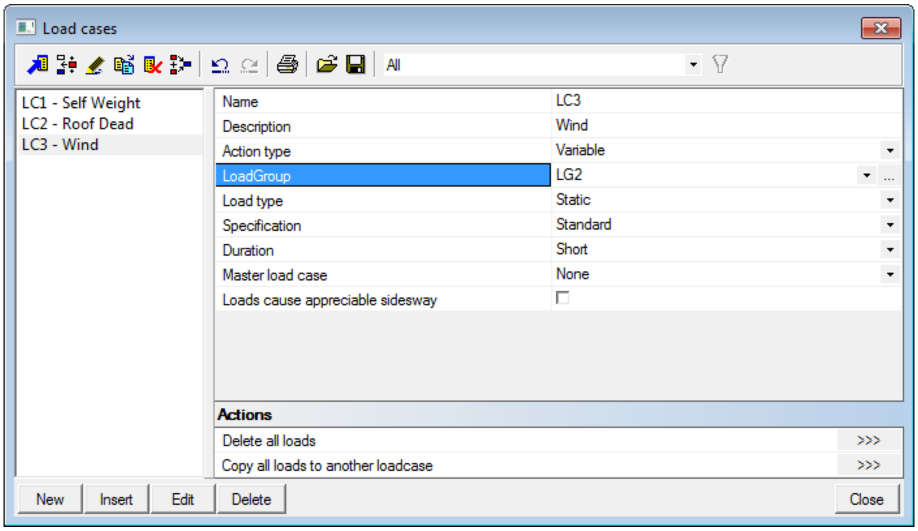
7. As this is a variable load, change the Action type to **Variable**.

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



The IBC (LRFD) - load type determines the load factor that is attributed to the load cases in this load group. In this project, choose **W - Wind**.

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



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

<u>Note:</u>

Load groups

Each load is classified in a group. These groups influence the combinations that are generated as well as the code-dependant factors to be applied. The following logic is adopted throughout the software:

Variable load cases that are independent from each other are associated to different variable groups. For each group, you set the load category (see ASCE 7-05*) and the combination factors from the AISC 360-05 or ASCE 7-05 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.*

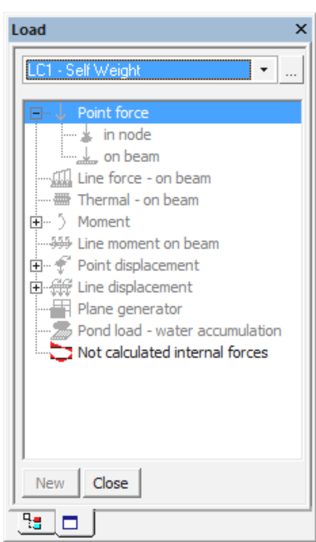
If the load is divisible, its different components are entered as individual load cases. As long as the load combination does not contain any variable load belonging to another group, no reduction factors will be applied. The different load cases of a divisible load are therefore associated to one variable 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".

Loads

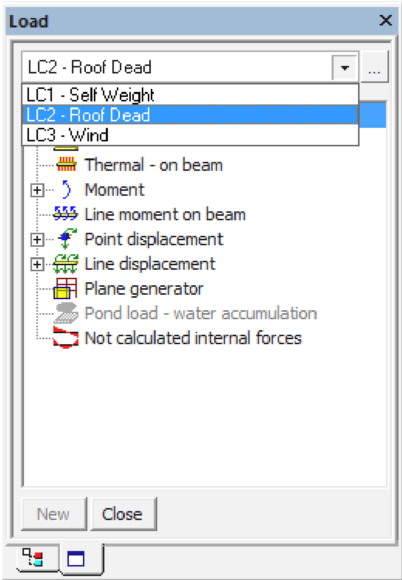
After the input of the Load cases, the **Loads menu** will automatically appear:

The first load case (**LC1**) includes the self weight of the steel members. As can be seen in the Loads menu, self weight is automatically accounted for and no point or line loads need to be added to the frame. The self weight is added to the structure based on the geometry and material properties of the sections.



Switching between load cases

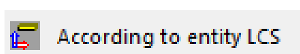
Activate LC2 by selecting this load case with the mouse pointer in the list box:



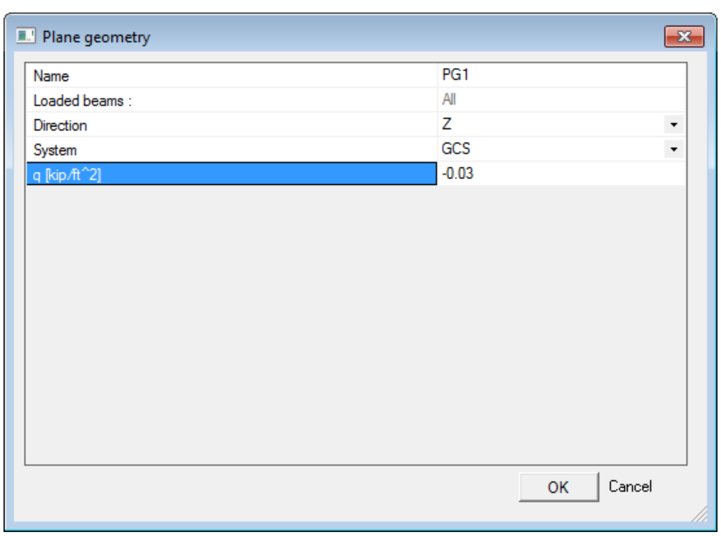
Once you have switched load cases, the roof dead loads can be entered as surface loads of 30lb/ft². For this example, only the roof girders will be directly loaded.

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. This plane generator can only be used in the current Workplane. Therefore, in order to input a roof load, the workplane will be changed to the roof.

1. Go to **Tools > UCS** and select the option 

2. Select member **B2**. The local z-axes of B2 (LCS) follows the inclination of the roof. Thus, the workplane will now also follow the inclination of the roof
3. Click on **Plane generator** in the Load menu. The **Plane geometry** dialogue appears



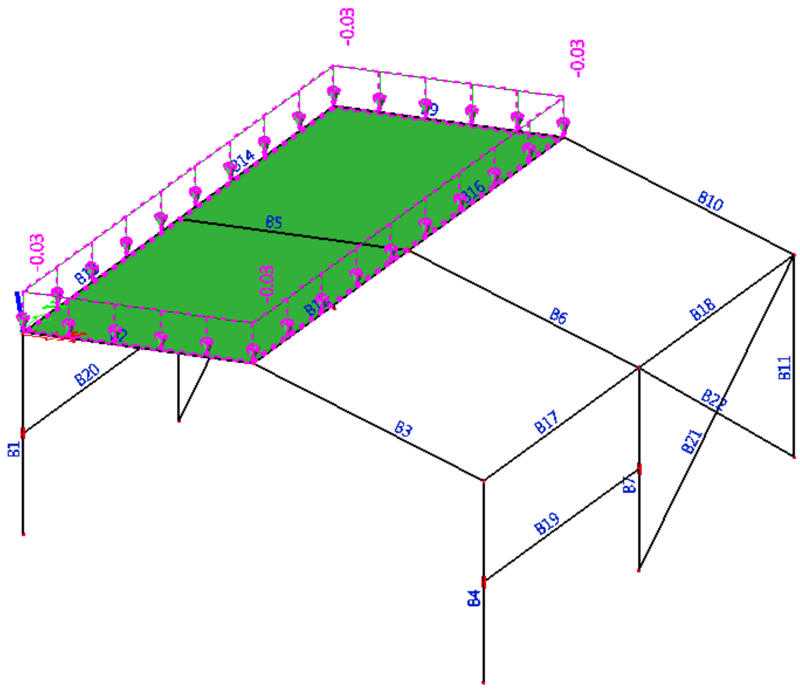
4. The load **Direction** is **Z** and the **System** is the global coordinate system **GCS**. In this way, the load is acting vertically downwards.

5. Change the **Value** to **–0.3 k/ft²**

6. Click **[OK]** to close the **plane geometry manager**.

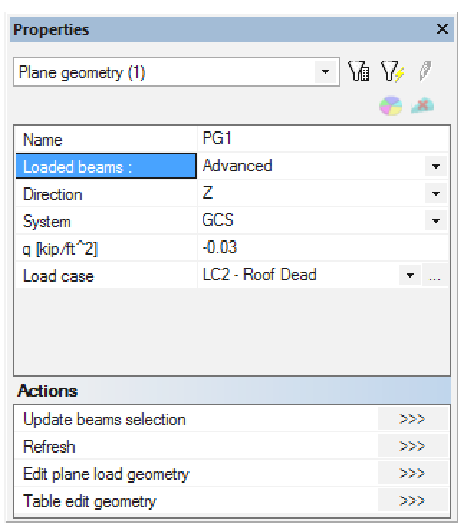
7. In the **Command Line**, select the option **New Rectangle**.

8. Click on node N2 (left node of the left roof plane) as the begin node and on node N13 (right node of the left roof plane) as the end node.



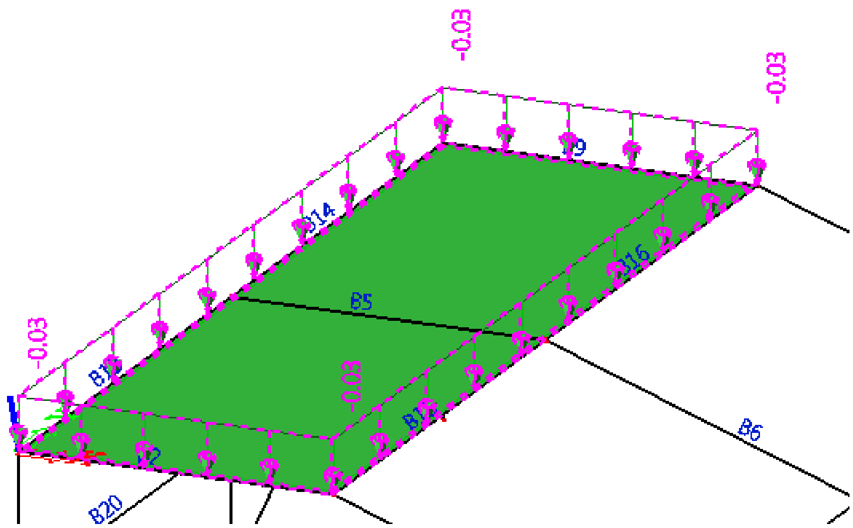
9. Press **<ESC>** to finish the input.

10. The load stays Magenta in color, which means it is still selected. Click on the combo-box in the **Properties window** > **Loaded beams** and change the option from **All** to **Advanced.**



11. In the **Actions – window**, select the option **Update beams selection** to indicate that only the girders and not the beams will be loaded directly by this surface load.

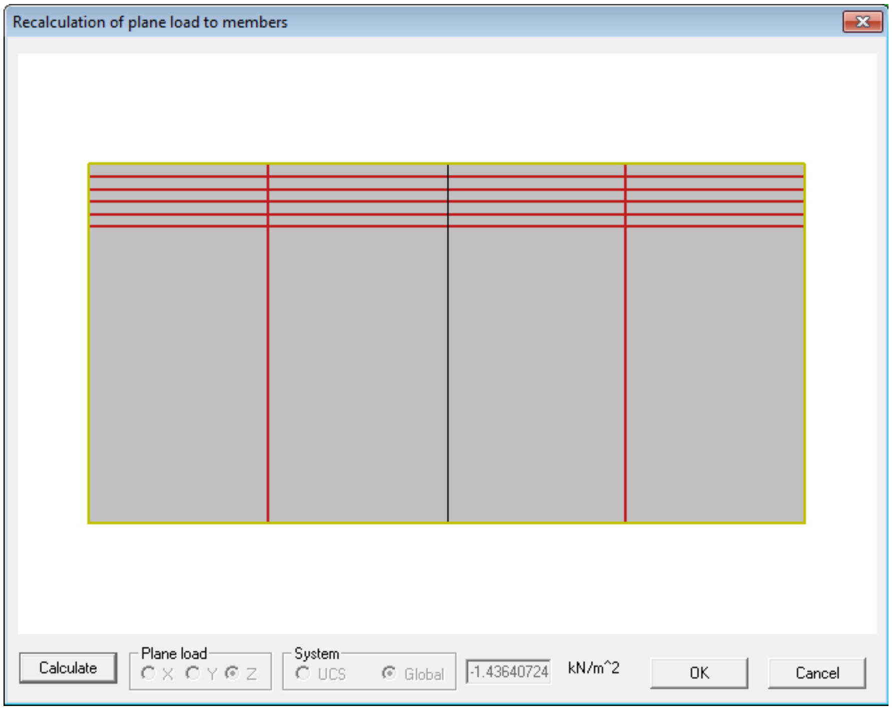
12. Click on beams **B2, B5** and **B9.**



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

14. In the **Actions-window**, click on **Refresh** to generate/ recalculate the surface load to line loads on beams.

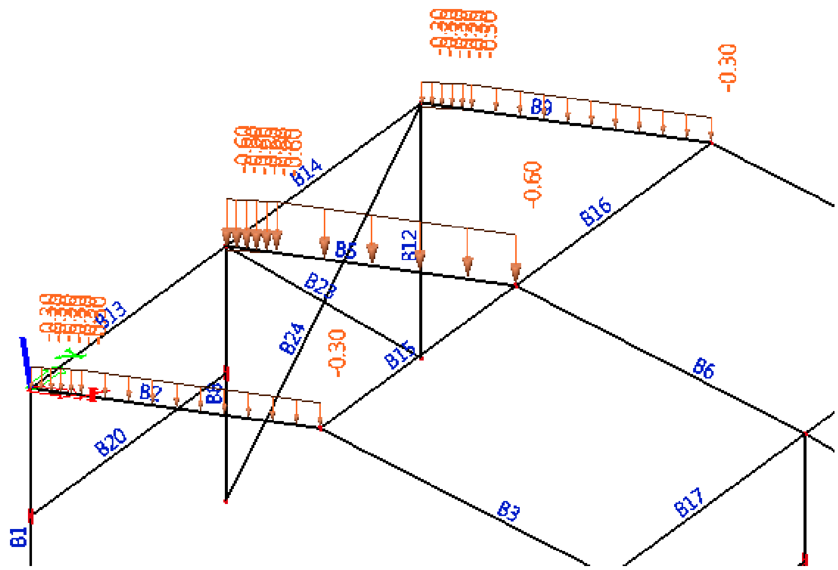
15. The **Recalculation of plane load to members** – window appears. Hit the **Calculate** –button.



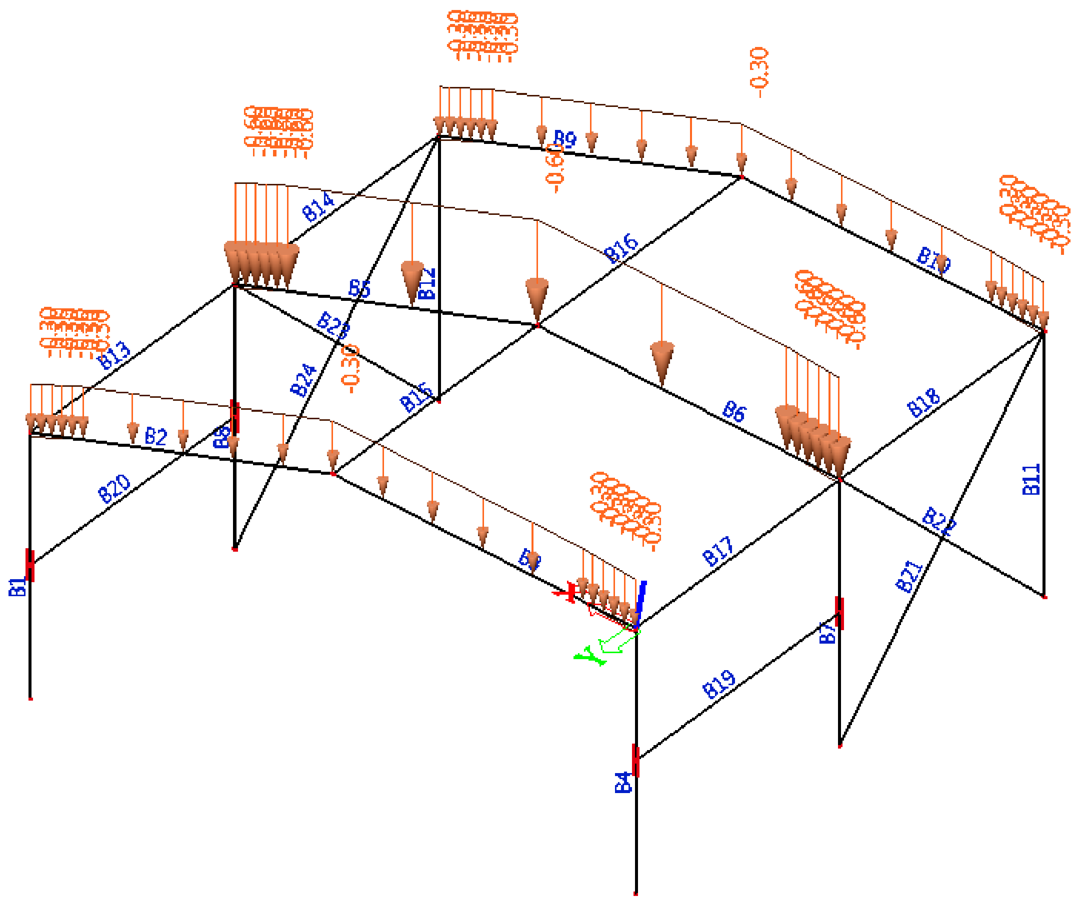
You will see a refinement of the calculated line loads at the beginning of the girders. This refinement occurs automatically because of the presence of the haunches at the beginning on the roof girders.

16. Click **[OK]** to confirme.

17. You will now see the calculated line loads on the roof girders

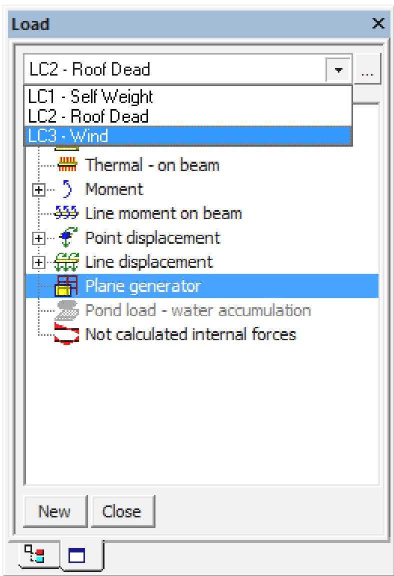


Continue the exercise by executing the same procedure on the right-sided roof plane.



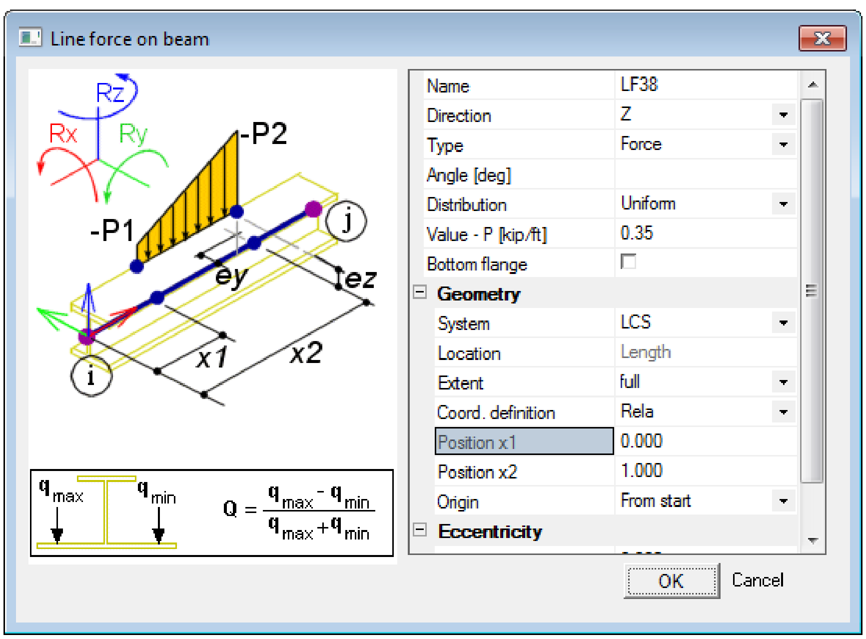
Switching between load cases

Activate LC2 by selecting this load case with the mouse pointer in the list 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 the members.

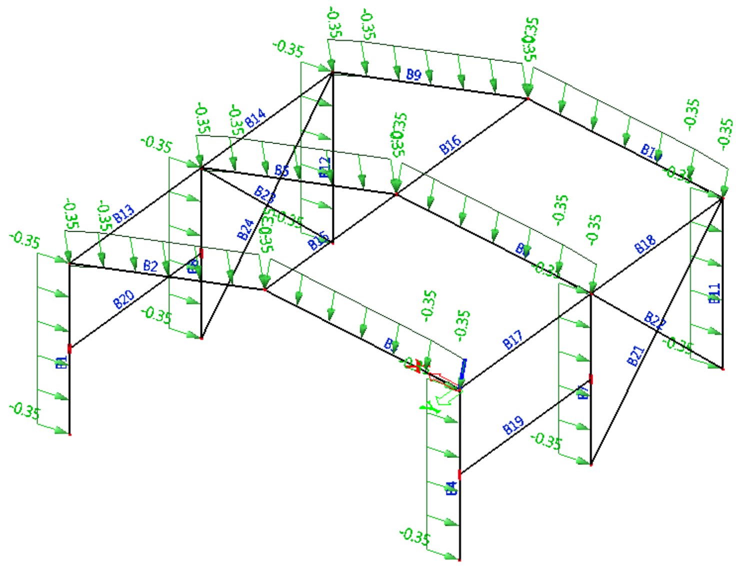
4. Change the **Value** to **–0.35 kip/ft**.

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

6. Select the members that the wind load will be applied to: 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

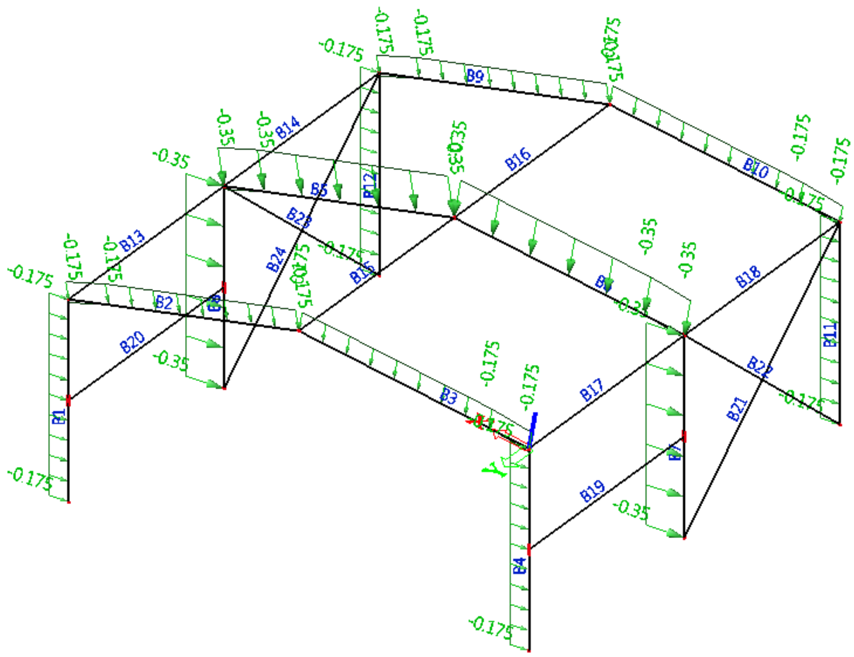
1. Select the linear loads on the roof girders and the columns of the first and last frame by clicking the loads with the left mouse button.

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

3. Change the **Value** from –**0.35 kip/ft** to –**0.175 kip/ft** in the Properties window.

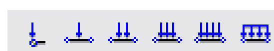
4. Confirm the modification with **<ENTER>**.

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



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

Note:

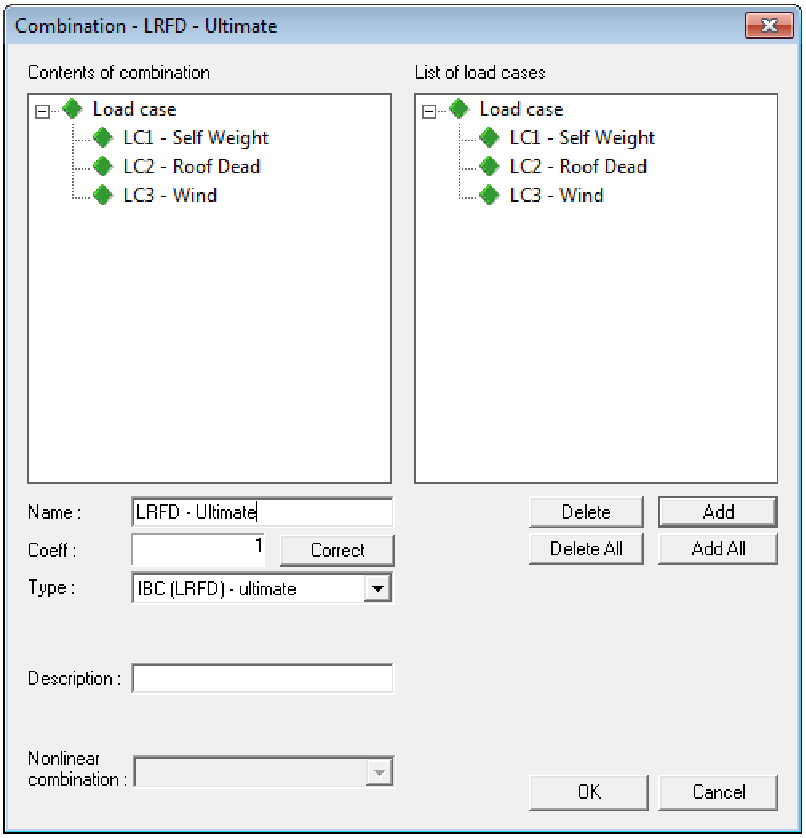
*The **Command line** includes a number of predefined loads:* *, which enable a fast and simple input of loads.*

Combinations

After input of the load cases, the latter can be grouped in combinations. In this project, two linear combinations are created, one for the Ultimate Limit State **(LRFD – Ultimate)** and one for the Ultimate Serviceability State **(LRFD – Service).**

Defining Combinations

1. Double-click on Combinations below Load cases, Combinations in the **Main window**.
2. Since no combination has been entered yet, the window to create a new combination will automatically appear.



3. The Type of the combination is changed to **IBC (LRFD) - ultimate**. With this combination type, Scia Engineer will automatically generate combinations in accordance with the complex composition rules of the ASCE.

4. By means of the button **[Add all]**, all load cases can be added to the combination.

5. Confirm your input with **[OK]**. The **Combination Manager** is opened.

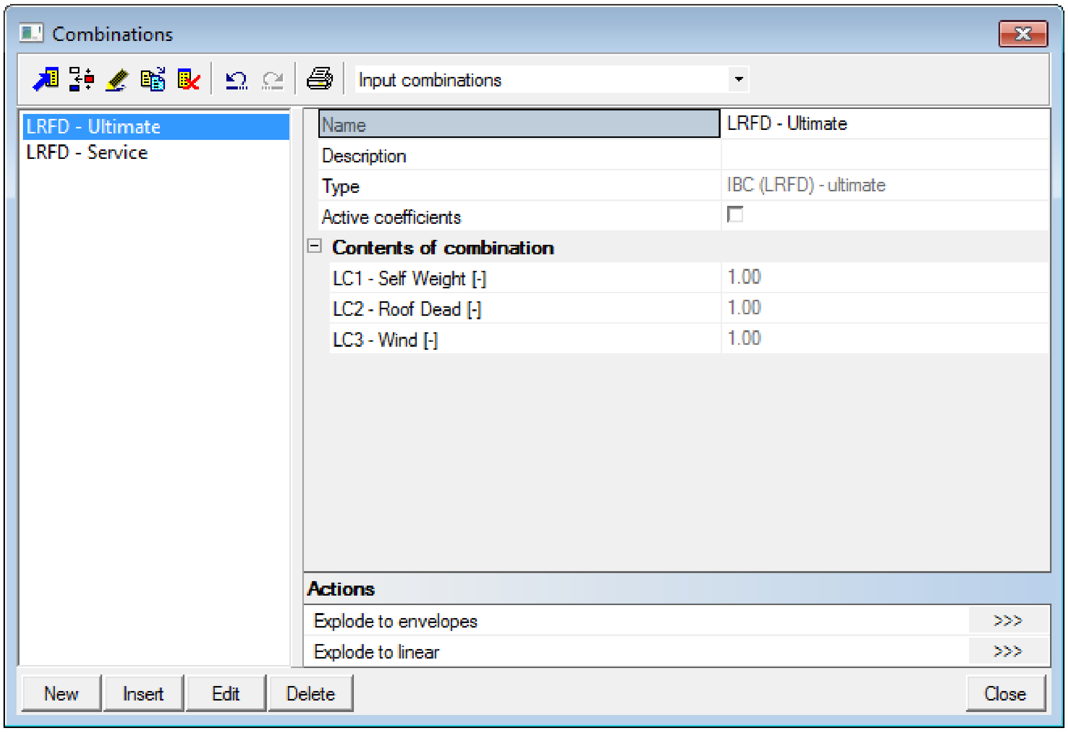
6. Click 

or  to create a second combination.

7. Change the **Type** of the combination to **IBC (LRFD) - serviceability**.

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

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


Combinations
Input combinations
LRFD - Ultimate
LRFD - Service
Name
LRFD - Ultimate
Description
Type
IBC (LRFD) - ultimate
Active coefficients
Contents of combination
LC1 - Self Weight [-]
1.00
LC2 - Roof Dead [-]
1.00
LC3 - Wind [-]
1.00
Actions
Explode to envelopes
>>>
Explode to linear
>>>
New
Insert
Edit
Delete
Close

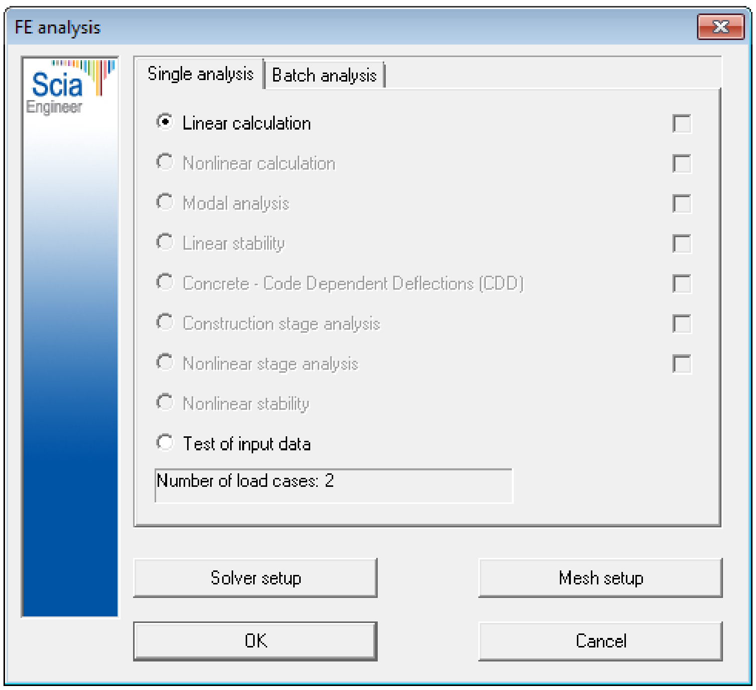
Calculation

Linear Calculation

If the calculation model is ready to be analyzed, it is then time to run the calculation of the frame and retrieve results.

Executing the Linear Calculation

1. Double-click on Calculation below Calculation,Mesh in the **Main window**.
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

After the calculation is executed, the results of the loading on the frame can be viewed.

Viewing the Reaction Forces

1. Double-click on Results in the **Main window**. The **Results menu** appears.

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

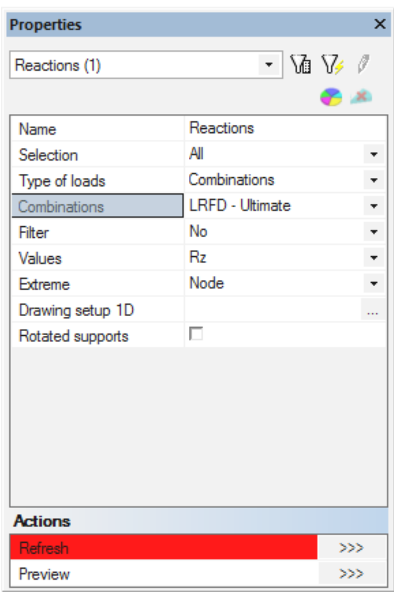
3. The options in the **Property Window** are configured in the following way:

   - The Selection field is set to **All**.

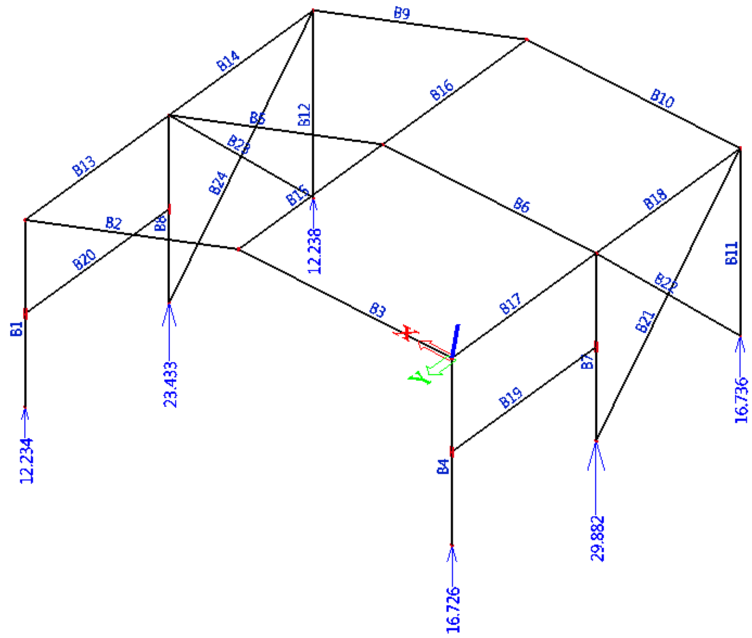
   - The Load type is set to **Combinations** and the Combination to **LRFD - Ultimate**.

   - The **Values** are wanted for **Rz**.

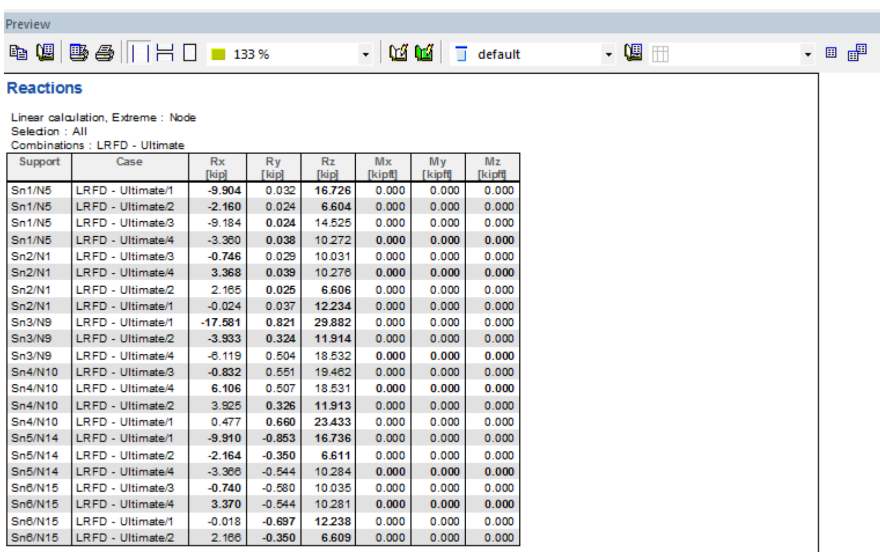
   - The **Extreme** field is changed to **Node**.



4. The action **Refresh** has a red background, i.e. the graphical screen must be refreshed. Click on the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options. The results for Rz displayed on the frame are in kip.



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



Reactions

Linear calculation, Extreme : Node
Selection : All
Combinations : LRFD - Ultimate

Support	Case	Rx [kip]	Ry [kip]	Rz [kip]	Mx [kipft]	My [kipft]	Mz [kipft]
Sn1/N5	LRFD - Ultimate/1	-9.904	0.032	16.726	0.000	0.000	0.000
Sn1/N5	LRFD - Ultimate/2	-2.160	0.024	6.604	0.000	0.000	0.000
Sn1/N5	LRFD - Ultimate/3	-9.184	0.024	14.525	0.000	0.000	0.000
Sn1/N5	LRFD - Ultimate/4	-3.360	0.038	10.272	0.000	0.000	0.000
Sn2/N1	LRFD - Ultimate/3	-0.746	0.029	10.031	0.000	0.000	0.000
Sn2/N1	LRFD - Ultimate/4	3.368	0.039	10.276	0.000	0.000	0.000
Sn2/N1	LRFD - Ultimate/2	2.165	0.025	6.606	0.000	0.000	0.000
Sn2/N1	LRFD - Ultimate/1	-0.024	0.037	12.234	0.000	0.000	0.000
Sn3/N9	LRFD - Ultimate/1	-17.581	0.821	29.882	0.000	0.000	0.000
Sn3/N9	LRFD - Ultimate/2	-3.933	0.324	11.914	0.000	0.000	0.000
Sn3/N9	LRFD - Ultimate/4	-6.119	0.504	18.532	0.000	0.000	0.000
Sn4/N10	LRFD - Ultimate/3	-0.832	0.551	19.462	0.000	0.000	0.000
Sn4/N10	LRFD - Ultimate/4	6.106	0.507	18.531	0.000	0.000	0.000
Sn4/N10	LRFD - Ultimate/2	3.925	0.326	11.913	0.000	0.000	0.000
Sn4/N10	LRFD - Ultimate/1	0.477	0.660	23.433	0.000	0.000	0.000
Sn5/N14	LRFD - Ultimate/1	-9.910	-0.853	16.736	0.000	0.000	0.000
Sn5/N14	LRFD - Ultimate/2	-2.164	-0.350	6.611	0.000	0.000	0.000
Sn5/N14	LRFD - Ultimate/4	-3.366	-0.544	10.284	0.000	0.000	0.000
Sn6/N15	LRFD - Ultimate/3	-0.740	-0.580	10.035	0.000	0.000	0.000
Sn6/N15	LRFD - Ultimate/4	3.370	-0.544	10.281	0.000	0.000	0.000
Sn6/N15	LRFD - Ultimate/1	-0.018	-0.697	12.238	0.000	0.000	0.000
Sn6/N15	LRFD - Ultimate/2	2.166	-0.350	6.609	0.000	0.000	0.000

<u>Note:</u>

The preview appears between the Graphical Screen and the Command line. This screen can be maximized to display more data.

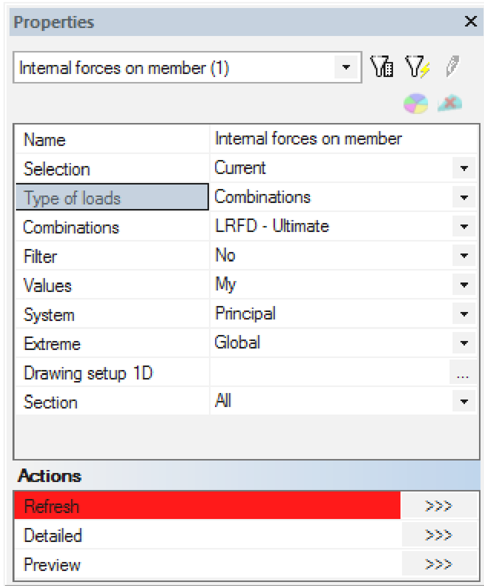
Viewing internal forces on beam

1. Click on **Internal forces on beam** below **Beams**

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

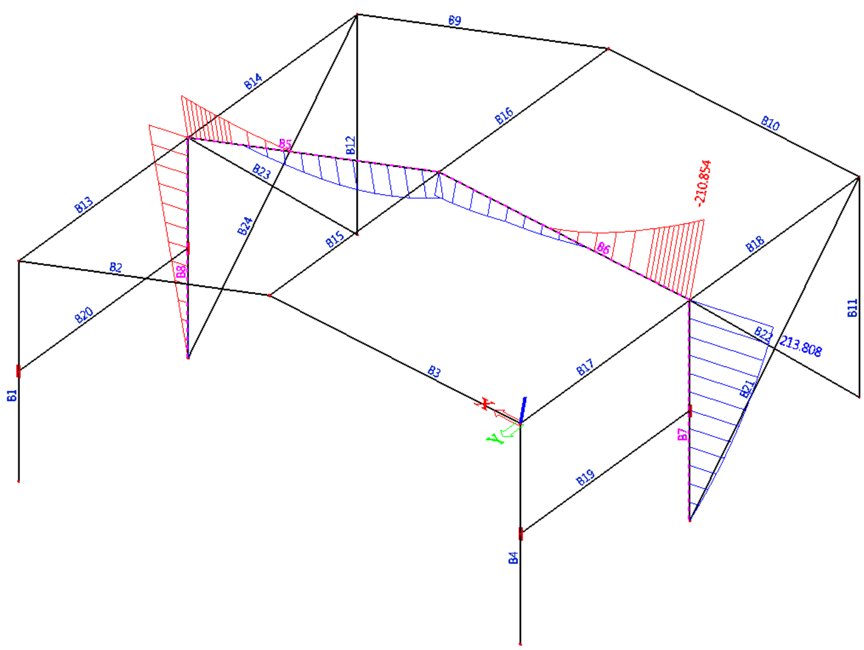
- The **Selection** field is set to **Current**.
- The **Load type** is set to **Combinations** and Combination to **CO1**
- The **Values** are wanted for **My**.

- The **Extreme** field is changed to **Global**.



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

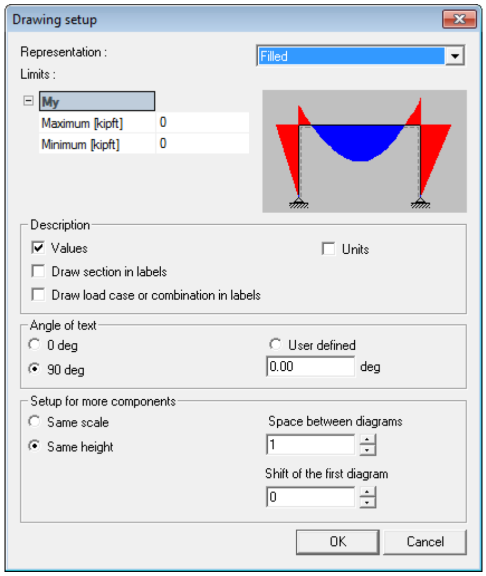
4. Click on the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options. The results for My displayed on the frame are in kip-ft.



To change the display of the results, the settings of the Graphical Screen can be adapted.

Configuring the Graphical Screen

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

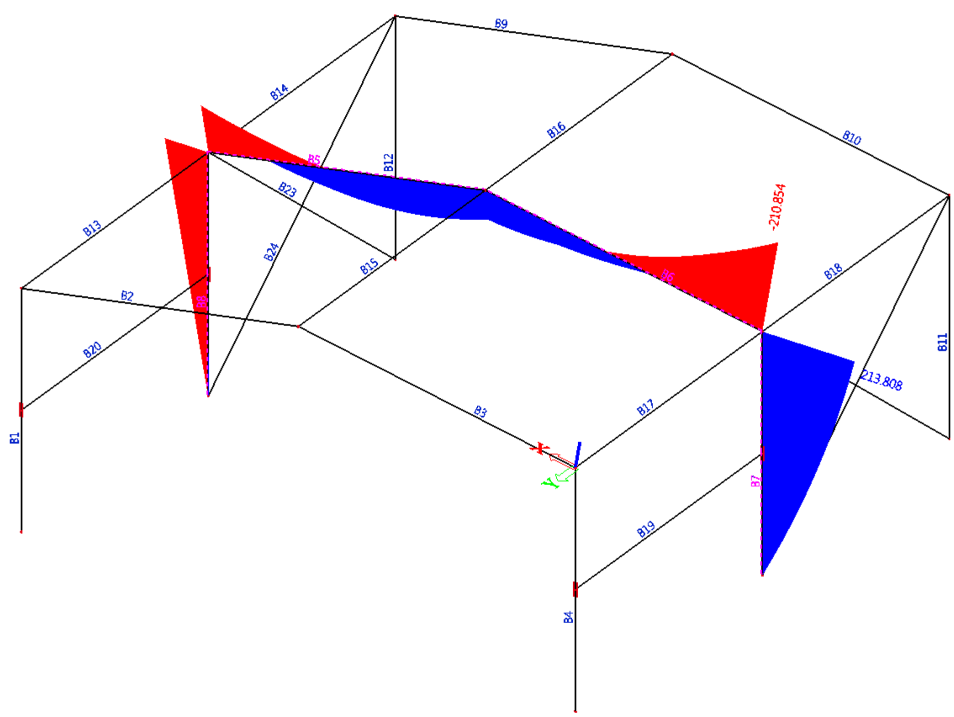


2. In the **Representation** field, choose **Filled**.

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

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

5. In the **Property Window**, click the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options. Again the results for My displayed on the frame are in kip-ft.



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

6. 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 as well as the sizes of any text associated with a specific layer.*

Code check

The structural steel modules of Scia Engineer contain a number of powerful tools to perform the steel calculations in accordance with the chosen standard, which in this case is AISC 360-10. Other specifications for steel design can also be used including AISC 360-05, AISC LRFD 2001 3rd Edition and AISC ASD 1989 9th Edition. The specification can be changed by selecting Setup within the Steel menu of the Main Service Tree.

The possibilities at a glance:

- Input of advanced steel data
- Simple addition, input and modification of member buckling data
- Input of restraints against lateral-torsional buckling, web/flange stiffeners
- Unity check of the profile section per AISC 360
- Optimization of the profile section in relation to unity check
- Input of frame connections
- Input of diagonal connections
- Automatic generation of sectional drawings
- Automatic generation of assembly drawings and anchorage plans
- Relative deformation check and web crippling data

In this Tutorial, we will only review the basics of the steel structural analysis. For more information regarding advanced steel calculations, we refer to the Advanced Training Curriculum.

Before you can start the steel calculations, you first must check the buckling parameters of the members. By means of the view parameters, the buckling lengths of the members can be displayed and modified.

Buckling parameters

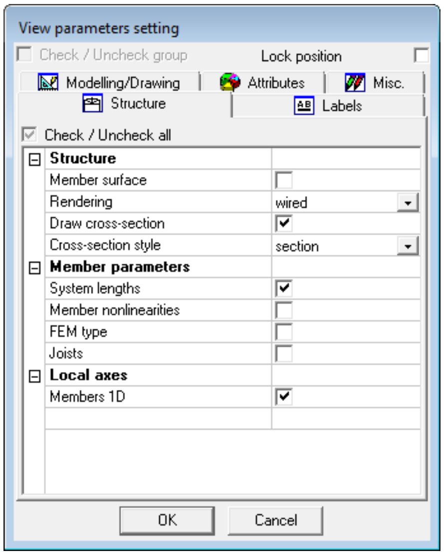
Displaying the system lengths

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

2. Click the right mouse button at any arbitrary position in the workspace. By doing this, a menu of the possible actions for the selected entity appears.

3. In this menu, select the 

option. The **View parameter settings** window appears.

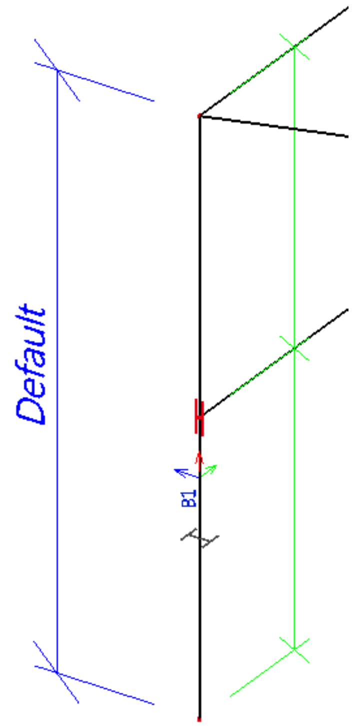


4. Activate the **System lengths** and **Draw cross-section** options to display the reference lengths and the section of the member.

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

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

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



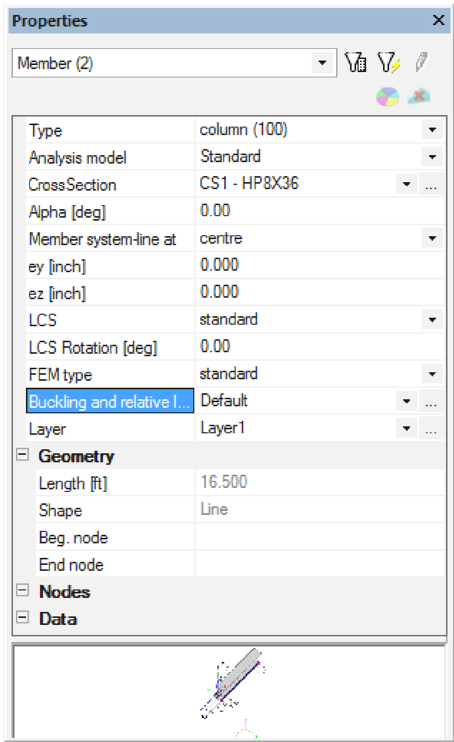
The figure shows that system length Ly for buckling around the strong axis (y-y) is 16.5’ and Lz for buckling around the weak axis (z-z) 8.25’ (2 times). The beam in the middle of the column therefore supports the column for buckling around the weak axis, i.e. for bending in the Y direction.

To modify the buckling data of a member, use the option **Buckling and relative lengths** in the **Property window** of the selected beam.

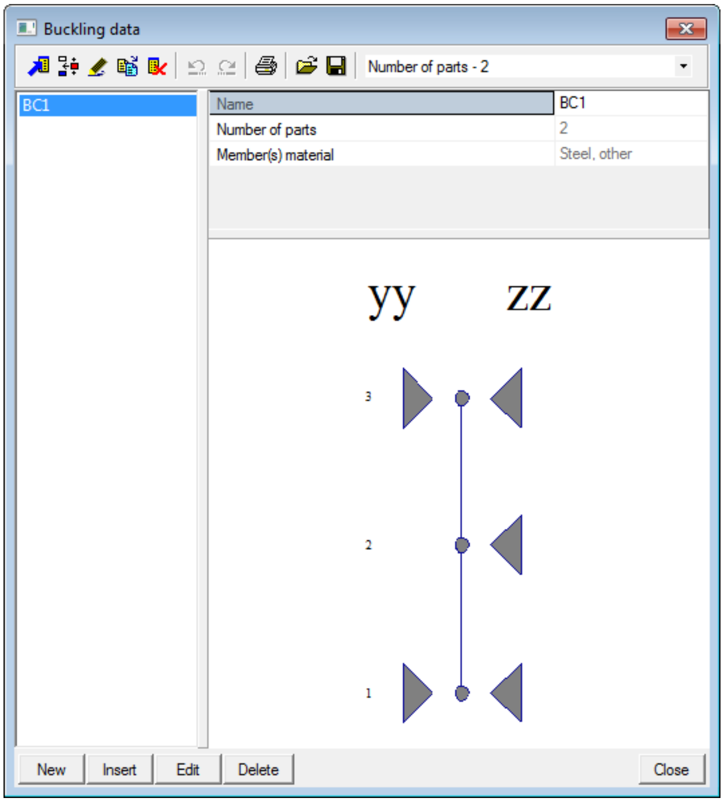
Setting the Buckling Parameters

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**.

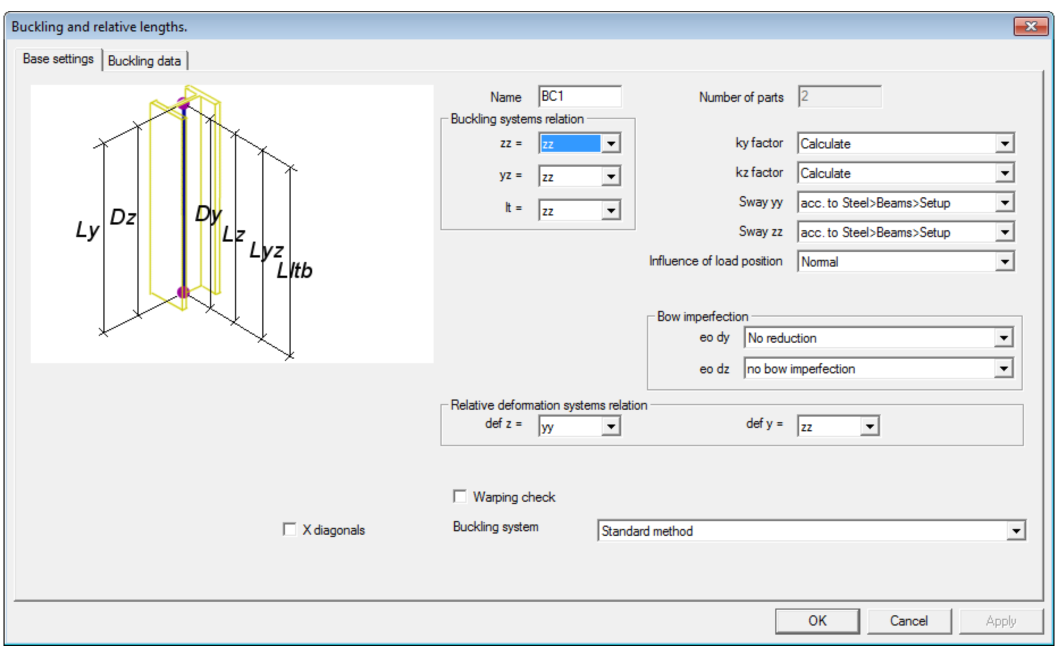


3. Click the  icon behind **Buckling and relative lengths**. The **Buckling data** window appears.



This window shows that at the midpoint, the column is supported for buckling around the weak axis (zz) but not for buckling around the strong axis (yy).

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



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

   - The **Name** field contains the name of the buckling parameter, in this case **BC1**.

- **kyy** and **kzz**: in these fields, you can indicate if the program must calculate the buckling factor around the axis regarded or if you prefer entering this factor manually. A third option allows for a manual input of the buckling length. The Support option can be used to determine the buckling factor in accordance with the applicable cross section and applicable material code.

- **Sway yy** and **Sway zz**: in these fields, you can indicate if the member is braced or not in the direction regarded. When you choose the Settings option, the default settings are used.

Note:

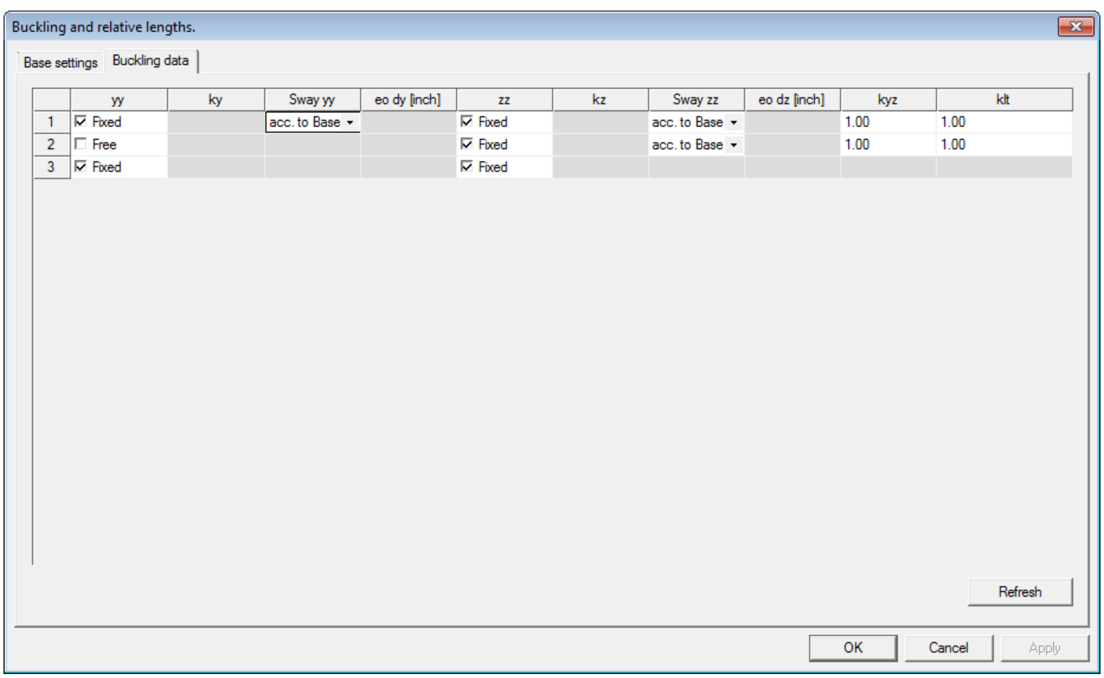
*The default settings for the buckling parameters are displayed in **Steel > Beams > Setup > Setup for check of steel members**. For a steel calculation, the structure is by default unbraced for buckling around the strong axis and braced for buckling around the weak axis. Therefore, a frame is unbraced in 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 deformations systems relation**: in these fields, you can define the system length to be used for the relative deformations.

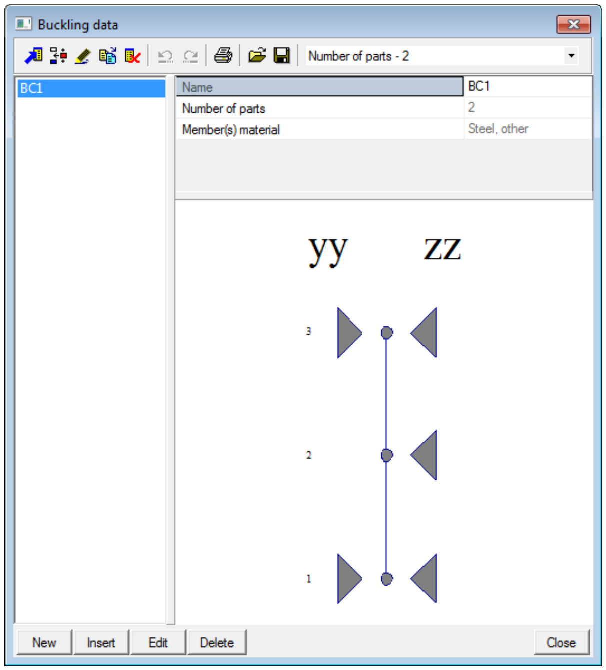
6. On the **Buckling data** tab, you can edit each of the parameters in detail. As the column consists of 2 components, i.e. 3 positions are available for modification: (1) at the start, (2) in the middle at the horizontal beam and (3) at the end, where the roof girders are connected

   For instance, by modifying the **Free** option on position (2) for yy to **Fixed**, buckling of the column at the midpoint around the strong axis would be impacted. This would mean the relative length around this axes would also become half of the total length (= 8.25'). For this Tutorial, the default options are maintained.



7. Click **[OK]** to close this window.

8. The **Buckling data** window re-appears. Click **[Close]** to close this window.



9. The **Properties window** indicates that the buckling parameter **BC1** is used for the columns of the first frame.

10. Press **<Esc>** to cancel the selection.

When the buckling parameters are set, you can continue with the steel check. Before proceeding, deactivate the **Member parameters** and **Local axes** representation by means of the **Fast adjustment of view flags on whole model** option.

Steel code check

Double-click on Steel in the **Main window** to open the **Steel menu**.

Displaying the Slenderness and the Buckling Lengths

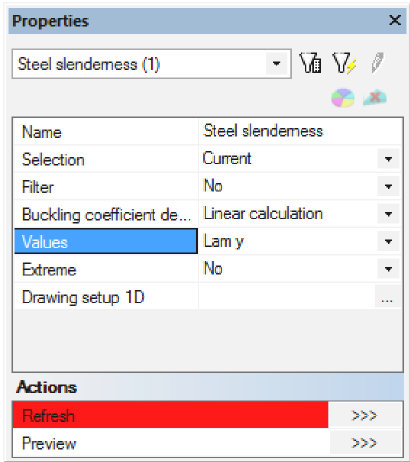
1. Click the Steel slenderness icon in the **Steel menu**

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

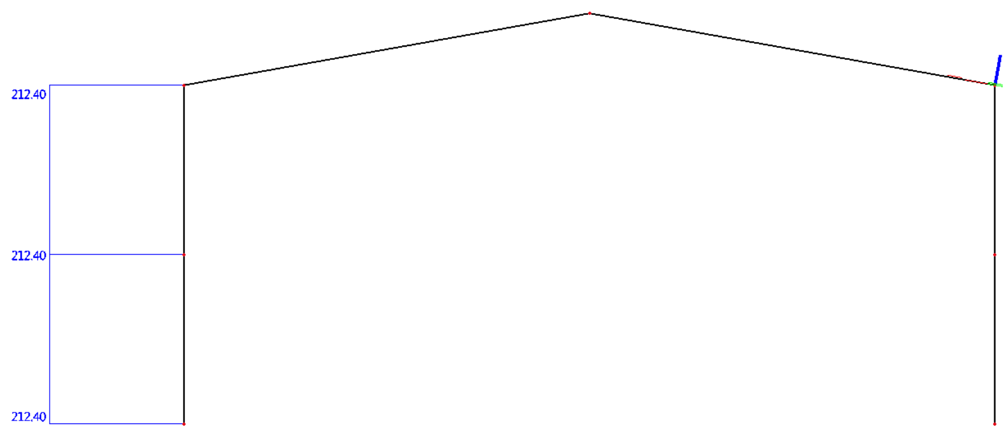
3. The options in the **Properties window** are configured in the following way:

   - The **Selection** field is set to **Current**.
   - The **Values** are wanted for **Lam y**, i.e. the slenderness around the yy axis.
   - The **Extreme** field is modified to **No**.

4. Select column **B1**, the left column of the first frame.

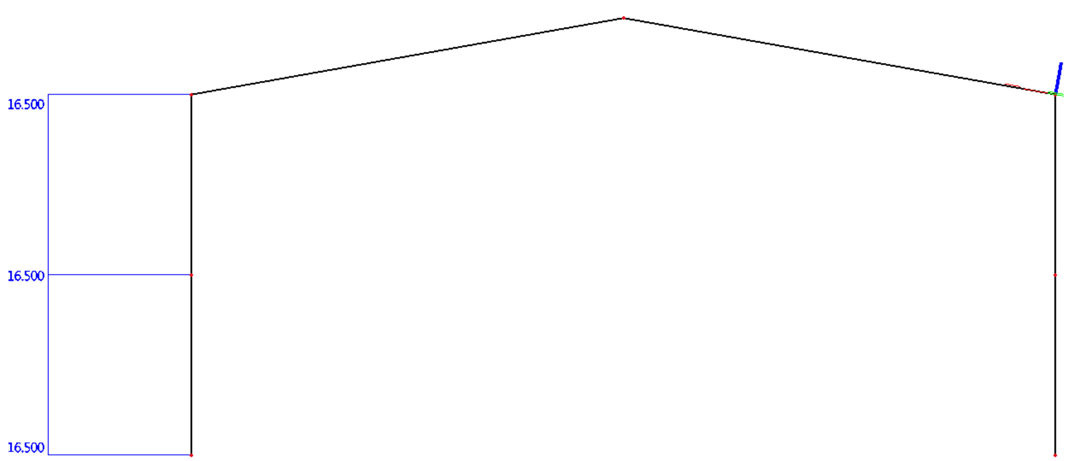


5. In the **Property Window**, click the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options.



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

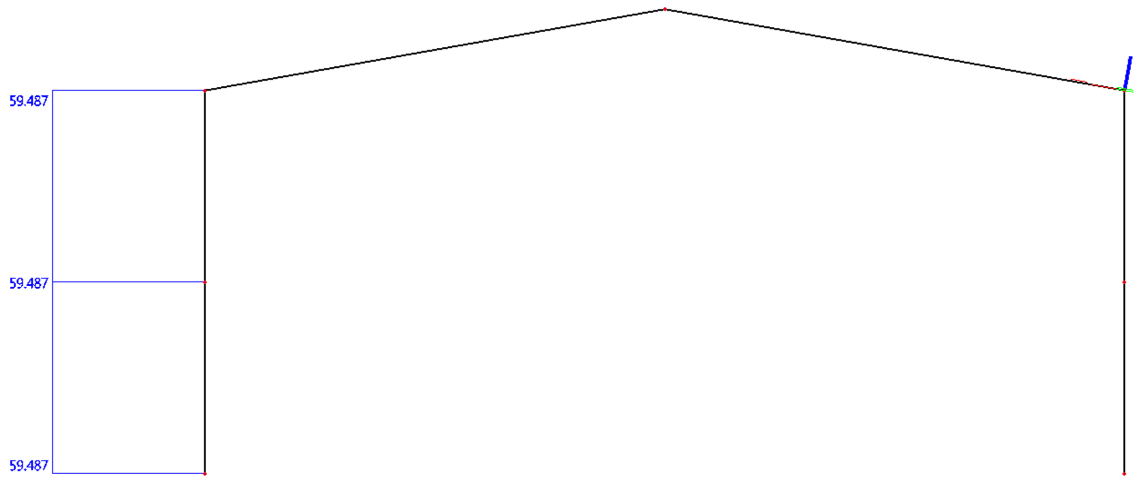
7. In the **Property Window**, click the >>> button behind **Refresh** to display the new set values.



As already indicated in the buckling parameters, the reference length is 16.5ft.

8. Change the **Values** field to **Iy** to display the buckling length for buckling around the strong axis.

9. In the **Property Window**, click the >>> button behind **Refresh** to display the new set value.



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

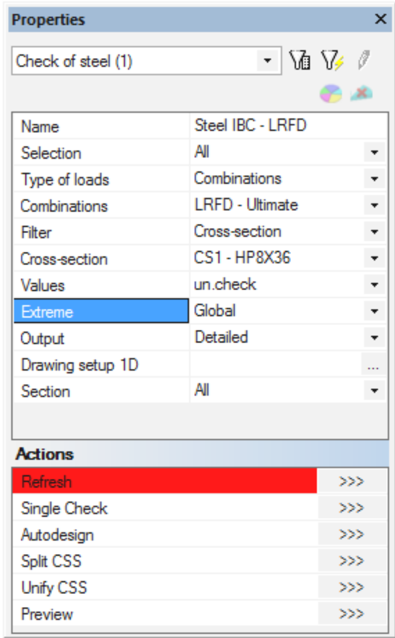
After viewing the steel data, you can perform the steel check for the columns. A unity check is carried out in accordance with AISC 360-05. This unity check includes both a capacity and a stability check.

Steel Code Check

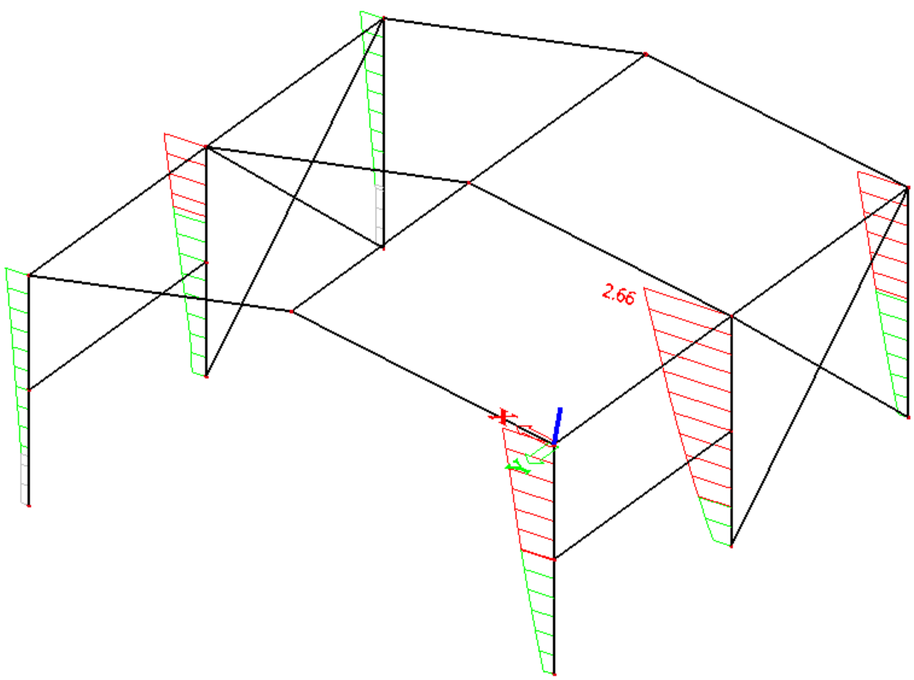
1. Click **ULS Checks > Check LRFD** in the **Steel menu**

2. The options in the **Properties 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 **LRFD - Ultimate**.
   - The **Filter** is changed to **Cross-Section.**
   - For the **Cross-Section**, choose **HP8X36** to ensure that only the results for the columns are displayed.
   - For the **Values**, choose a **un. check.**
   - The **Extreme** field is changed to **Global**.



3. In the **Property Window**, click the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options.



The graphical screen shows that the maximum of unity checks occurs for one of the columns in the middle frame. To better understand and analyze the cause of the problem, you can open the **Preview** with a detailed representation of the performed steel check.

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

Steel Design Check - Preview

This report reveals that the column does not comply with the combined bending, axial force and shear force check according to the AISC-LRFD checks found in AISC 360-10. As a result, the member does not satisfy the stability check and therefore a section with a larger inertia is required. 1

Check of steel

Linear calculation, Extreme : Global
Selection : All
Combinations : LRFD - Ultimate
Cross-section : CS1 - HP8X36

AISC 360-10 LRFD Check

Member B7	**HP8X36**	**A36**	**LRFD - Ultimate**	**2.66**

Material data		
Yield stress Fy	35.97	ksi
Tensile stress Fu	58.02	ksi
fabrication	rolled	

The critical check is on position 16.50 ft

Classification for local buckling
according to article B4 and Table B4.1

Compression N	ratio	Lambda,p (Compact)	Lambda,r (Non-compact)
Webs	12.92	-	42.31
Outstanding flanges	9.16	-	15.90

Section is classified as non-compact section.

Bending Mx	ratio	Lambda,p (Compact)	Lambda,r (Non-compact)
Webs	12.92	106.78	161.87
Outstanding flanges	9.16	10.79	28.40

Section is classified as compact section.

Bending My	ratio	Lambda,p (Compact)	Lambda,r (Non-compact)
Outstanding flanges	9.16	10.79	28.40

Section is classified as compact section.
Axis definition :
- local x- axis in this code check is referring to the local y axis in Scia Engineer
- local y- axis in this code check is referring to the local z axis in Scia Engineer

Internal forces		
Pu	-28.26	kip
Vux	-0.03	kip
Vuy	8.35	kip
Mut	-0.09	kipft
Mux	213.81	kipft
Muy	-0.00	kipft

Buckling parameters	xx	yy	
type	sway	non-sway	
Slenderness	213.23	31.11	
Reduced slenderness	2.39	0.35	
Length	16.50	8.25	ft
Buckling factor	3.60	0.61	
Buckling length	59.36	5.07	ft

Warning: slenderness 213.23 is larger then 200.00 !

Buckling check
according to article E3 and formula (E3-1)

Table of values		
Pn	58.38	kip
Pu	28.26	kip
Fcr	5.52	ksi
Resistance factor	0.90	
unity check	0.54	

Torsional buckling check
according to article E4 and formula (E4-1)

Table of values		
Pn	346.26	kip
Pu	28.26	kip
Fe	160.84	ksi
Fez	160.84	ksi
Resistance factor	0.90	
unity check	0.09	

LTB data		
Lb	8.25	ft
Cb	1.18	

according to article F2 and formula (F2-1)

Table of values		
Lr	36.72	ft
Lp	8.14	ft
Mn	103.80	kipft
Mu	213.81	kipft
Resistance factor	0.90	
unity check	2.29	

Weak axis bending check
according to article F6 and formula (F6-1)

Table of values		
Mn	45.83	kipft
Mu	0.00	kipft
Resistance factor	0.90	
unity check	0.00	

Shear stress check
according to article G2 and formula (G2-1)
in buckling field 1

Table of values		
a	16.50	ft
h	0.48	ft
tw	0.44	inch
kv	5.00	
Vn	77.11	kip
Vu	8.35	kip
Resistance factor	1.00	
unity check	0.11	

Combined stresses check
according to article H1.2 and formula (H1-1a)

Table of values		
Pr	28.26	kip
Mrx	213.81	kipft
Mry	0.00	kipft
Pc	52.54	kip
Mcx	93.42	kipft
Mcy	41.24	kipft
Res. factor compression	0.90	
Res. factor flexure	0.90	

unity check = 0.54+8/9(2.29+0.00) =2.57 (H1-1a)
Members under torsion and combined torsion. Flexure,shear and/or Axial force as per H3.
according to article H3 and formula (H3-7)

Table of values		
Mux	213.81	kipft
Muy	0.00	kipft
fun	86.16	ksi
Fny	35.97	ksi
Res. factor torsion	0.90	

unity check = 86.16/32.37=2.66 (H3-7)
according to article H3 and formula (H3-8)
Critical fibre position 7

Table of values		
Vux	0.03	kip
Vuy	8.35	kip
Mut	0.09	kipft
fsx	0.000	kip/ft^2
fsy	45.508	kip/ft^2
fT	996.824	kip/ft^2
Stress	3.25	ksi
Fn	21.58	ksi
Res. factor torsion	0.90	

unity check = 3.25/19.42=0.17 (H3-8)
according to article H3 and formula (H3-9)

Table of values		
Pux	28.26	kip
fua	2.67	ksi
Fnb	5.52	ksi
Res. factor torsion	0.90	

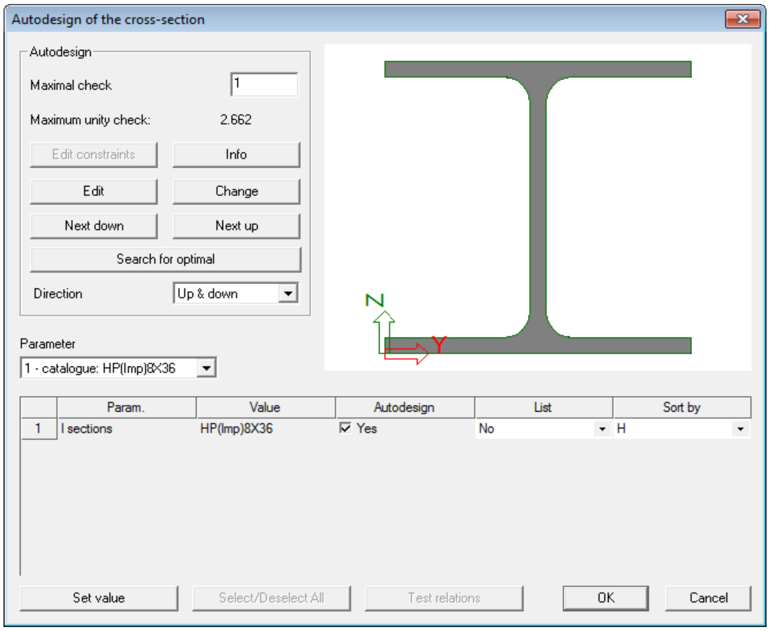
unity check = 2.67/4.97=0.54 (H3-9)
The member does NOT satisfy the check !

After reviewing the results of the steel design check, Scia Engineer will allow the user to complete a simple and smooth optimization of the steel section. The program will automatically propose a profile section, which complies with the unity check.

Optimization of the Steel Section

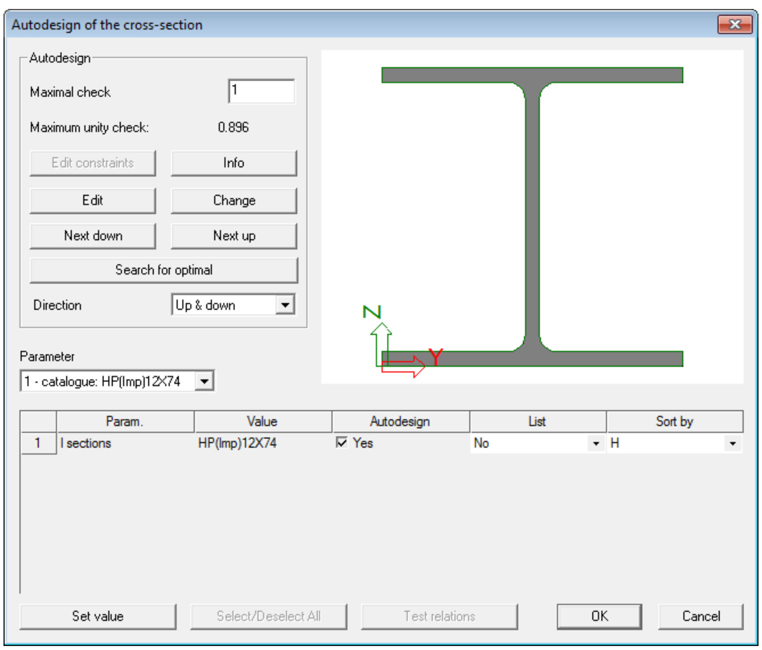
1. In the **Properties window**, click the >>> icon behind **Autodesign**. The settings of the **Properties window** are maintained, so that the **HP8x36** can be optimised.

The **Optimization of the cross-section** for the maximum loading conditions is as follows:



2. This window once more displays the maximum unity check for HP8x36 as **2.662.**

3. Click the **Search for Optimal** button. The program will search in the profile library for a profile complying with the unit check.



It appears that **HP12X74** complies with the requirements: maximum unit check **0.896.**

4. Confirm the optimization of the HP8x36 by clicking **[OK]**.

<u>Note:</u>

After an optimization, the project must be recalculated. The change in cross-section ultimately modifies the total self weight of the structure as well as the structures stiffness. This modification in stiffness will lead to a different distribution of the forces throughout the frame. This means that after the optimization and recalculation of the structure, the profile chosen could possibly be overstressed. In that case, you must re-execute the optimization in order to find a cross-section that has a unity below 1.0.

An optimization is always performed for a section, i.e. the optimized section is always attributed to all members with that particular section. In this Tutorial, the filter was already set to section. If not, the program would automatically switch to this filter selection.

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

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

Engineering Report

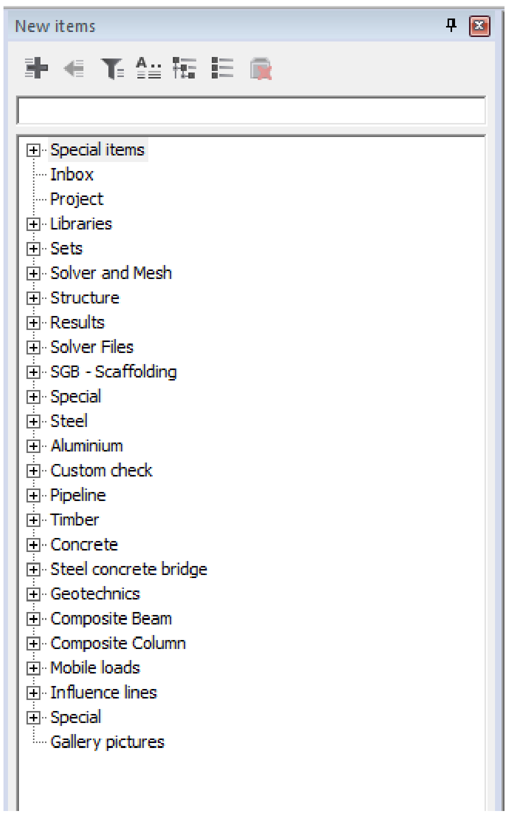
In this final part of the tutorial, we will explain how the Engineering Report can be used to easily create a calculation package that includes input data, analysis results, design tables and graphical representations of the model. In addition, external document items can be inserted into the Engineering Report within Scia Engineer so that the user can manage their entire calculation package in one integrated location.

Formatting the Document

1. Double-click Engineering report in the **Main Window** or click in the button bar. The **Engineering Report** appears in a separate window.

   The Engineering Report is divided into four main elements; the **Navigator**, the **New Items** list, the Document and the **Properties** palette.

2. To add **New items** to the **Navigator**, which serves as the documents Table of Contents, select an item from the New Items list and click the **[+]** button or simply double click on the selected item.



3. By means of this window, many different data types can be added to the document:

   - Open the **Special items** group and double click on **Header/Footer** and **Table of Contents.**
   - Open the **Libraries** group and double click on **Materials** and **Cross Sections.**
   - Open the **Structure** group and double click **Members** and **Haunches**.
   - Open the **Results** group and double click **Internal forces on beam** and **Reactions.**

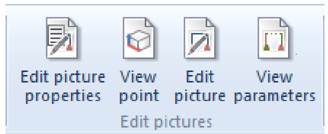
The items that were added to the document are displayed in the **Navigator**. Within the Navigator it is possible to drag the items with the mouse to change their order or even add the **Special Item, Chapters** to further organize and indent specific calculation elements. Additionally, it is possible to change the properties of any item within the list by selection the item and modifying the properties found in the **Properties Palette.**

Additional Document Functions

1. Add picture to document

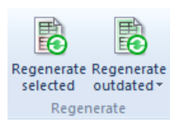
- Directly via Right Click and Screen Shot of Scia Engineer workspace into Engineering Report or Live Picture into Engineering Report
- To make a Live Picture 3D PDF capable, select picture and check "Export to PDF as 3D"

2. Make changes to Live Pictures
   - Select Live Picture in Navigator and now the Picture Editing buttons become active.



   - Select "Edit Picture Properties" to change size, scale, view point, regeneration settings and result type.

3. Adapt header template
   - Insert "Header/Footer" to Navigator
   - Modify Header information, add company logo and page numbers
4. Adapt Style of Document
   - Via adding "Style" to Navigator from New Items list
5. Modification of Tables
   - Select specific table in Navigator and click "Edit" button.  This enables the user to select and modify specific items in the table, change the table layout and also the size and spacing of the table items.
6. Refresh of Engineering Report
   - After adaptations to data or (content of) tables: Red Exclamation Point means item regeneration needed.
   - Regenerate Selected Items
   - Regenerate Outdated Items (Pictures, Tables)



7. Properties of the different components
   - After selection of a component in the Engineering report Navigator, its properties can be consulted and adapted in the Properties menu.

Note: The Engineering Manager gives you the option to have multiple Engineering Reports in the same Scia file. You can also have the Engineering Report open in the separate window while working in the Scia file. Any changes made to the Scia file can then be updated by regenerating the outdated items (marked by a red exclamation point) in the Engineering Report.

Epilogue

In this tutorial, the basic functionalities of Scia Engineer for the input of a steel structure, including the steel member design check, member optimization and modelling of rigid connections were introduced by means of a steel frame building example.

After reading the text and completing the example, the user should be able to model and calculate simple steel structures that include; steel members, surface loads, wind loads, load combinations, hinges, connections and the basic calculations shown in a document.

For more detailed information about steel calculations, please refer to the Advanced Steel Training tutorials.