



\ SCIA ENGINEER TUTORIAL STEEL FRAME AISC All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2025 SCIA nv. All rights reserved.

Table of Contents

Table of Conter	nts		3	
General inform	ation		5	
Welco	ome		5	
SCIA I	Enginee	er support	5	
Webs	site		5	
Introduction			6	
Chapter 1:	Getti	ng started	7	
1.1.	1.1. Starting SCIA Engineer			
1.2.	Starti	ng a new project	7	
Chapter 2:	Proje	ct management	9	
2.1.	Save,	Save as, Close and Open	9	
2.	1.1.	Saving a project	9	
2.	1.2.	Closing a project	9	
2.	1.3.	Opening a project	9	
2.	1.4.	Start project manager	9	
2.2.	Comp	ponents of the User Interface		
2.	2.1.	Main menu	10	
2.	2.2.	SCIA Spotlight	10	
2.	2.3.	Status Bar	10	
2.	2.4.	Process Toolbar	11	
2.	2.5.	View Bar	11	
2.	2.6.	Properties Panel	12	
2.	2.7.	Input Panel	12	
2.	2.8.	Marking menu	13	
Chapter 3:	Geon	netry input	14	
3.1.	Input	of the geometry	14	
3.	1.1.	Cross-sections	14	
3.	1.2.	Geometry	16	
3.	1.3.	Additional data	24	
3.	1.4.	Check structure		
3.	1.5.	Graphic representation of the structure	32	
Chapter 4:	Loads	s and combinations		
4.1.	Load	cases and load groups		
4.2.	Load	panels		
4.3.	Loads	5		

4.4.	Com	pinations	41
Chapter 5:	Analy	/sis	
5.1.	Linea	r analysis	42
Chapter 6:	Resul	lts	
6.1.	React	tions	44
6.2.	Inter	nal forces	45
6.3.	3D de	eformations	47
6.4.	Resul	Its-related status bar options	48
Chapter 7:	Code	checks	
7.1.	Buck	ling parameters	49
7.2.	Steel	checks	51
7	7.2.1.	Strength check	51
7	7.2.2.	Serviceability check	54
Chapter 8:	Steel	connections	
8.1.	Struc	tural model	55
8.2.	Steel	connection	56
Chapter 9:	Repo	rt	
9.1.	Engin	neering report	59
ç	9.1.1.	Adding images to the report	60
ç	9.1.2.	Displaying results	62
ç	9.1.3.	Printing or exporting the report	63

General information

Welcome

Welcome to the SCIA Engineer Tutorial – Steel Frame. 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.

System requirements to run SCIA Engineer 24:

https://www.scia.net/en/support/faq/scia-engineer/installation/system-requirements

SCIA Engineer is available in two different versions:

Licence version

The licence version of SCIA Engineer is secured with a Cloud account. For more information: https://eu-1.cdn77storage.com/scia-supportstore/support/sciaengineer/installation/24/MAN%20%5BEng%5D%20SCIA24.0%20-%20Installation%20manual%20cloud.pdf

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.

A Cloud licence is also available for "Try-out", which only limitation is in the output.

Student version

The student version has the same possibilities as the licence version for all of modules. This version is also secured by a Cloud protection.

The output contains a watermark "Student version". Projects that are stored in the student version <u>cannot be</u> <u>opened</u> in the licence version.

SCIA Engineer support

You can contact the SCIA Engineer support service

By e-mail

Send an e-mail to <u>support@scia.net</u> 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 <u>The</u> <u>contact by telephone is available only for users with Licence version or Try-out version</u>

Website

Link to Link to Manuals and Tutorials: https://www.scia.net/en/support/tutorials-manuals Link to Web help: https://help.scia.net/

Introduction

This tutorial describes the basic functions of SCIA Engineer for the input, analysis and design of a 3D steel frame.

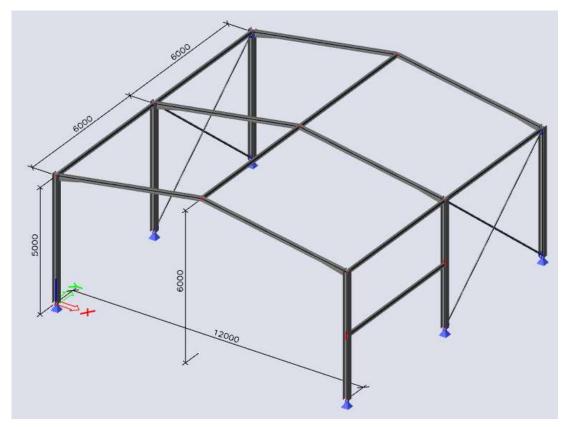
Before you start, you must be familiar with your operating system: for instance working with dialogues, menu bars, toolbars, status bars, handling the mouse, etc.

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 create a steel connection.

The tutorial ends with a brief introduction to the engineering report.

The figure below shows the calculation model of the structure that we will design.



Chapter 1: Getting started

1.1. Starting SCIA Engineer

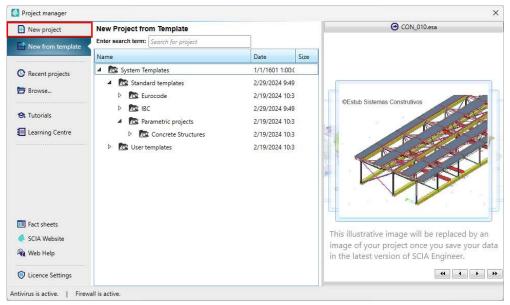
Before you can start a project, you need to start the program first.

- double-click on the SCIA Engineer shortcut in the Windows Desktop, or
- if the shortcut is not installed, click the Windows start button and choose SCIA Engineer 25.0 > SCIA Engineer 25.0.

If the program doesn't find any protection, you will see a dialogue indicating that no protection was found. You can run the Protection setup to login with your cloud licence credentials.

1.2. Starting a new project

When the Project manager dialogue appears, click New project.



You can also start new project via Main menu > File > New or with the default hotkey combination Ctrl+N.

DATA		MATERIAL	
Name:		Concrete	
		Steel	
Part		Material	
Description:	•	Masonry	
		Aluminium	
 Author:	Auteur	Timber	
Date:	31/12/2024	Other	
Structure:	🔹 General XYZ	✓ CODE	
		National Code:	
		IBC	*

Now, the **Project settings** dialogue is opened. On the **Basic data** tab, you can enter general data about the project.

In the **Data** frame, enter your preferred data. These data can be mentioned on the output (for example in the report and on drawings).

Set Structure: Frame XYZ (to limit input possibilities to 1D members in 3D plane only) or General XYZ.

In the **Material** frame, tick the **Steel** checkbox (defining a material is the only required setting to proceed). Choose **S 235** from the combo-box.

In the Code frame select National Code IBC.

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

<u>Note</u>: on the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program.

Chapter 2: Project management

2.1. 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.

2.1.1. Saving a project

Go to Main menu > File > Save or press Ctrl+S.

If a project has not yet been saved, the dialog box **Save as** appears. Choose the drive you want to save your project in. Select the (sub)folder in which you want to put the project. 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 select the drive and (sub)folder and set a name for the project file.

<u>Note</u>: the Autosave function creates a backup file every 15 minutes by default. These projects can be found in folder *C*:\Users*username*\Documents\ESA24.0\Autosave\

2.1.2. Closing a project

To close a project, choose Main menu > File > Close Project, press Ctrl+F4 or click the X button above the SCIA Spotlight in the top-middle of the application.

		Esa1.esa	× +		
Please click here a	r press Space and	type your text	It will be complete	d with lines below.	
			X		
			1		
			- 1		

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

2.1.3. **Opening a project**

Click Main menu > File > Open or press Ctrl+O 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).

2.1.4. Start project manager

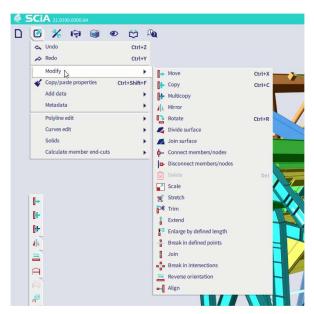
Click on **Main menu > File > Project browser** to open the project manager. Here the recently closed project can be found, as well as sample projects.

2.2. Components of the User Interface

2.2.1. Main menu

The Main menu is the component that will guide you through all the parameters of the project. The main features are:

- It contains functions, results & checks •
- Commands shown with hotkeys •
- Single letter hotkeys are allowed •
- You can populate the Process toolbar by • commands from the main menu – simply drag it



2.2.2. SCIA Spotlight

The SCIA Spolight is the command line of the interface, and it has the following features:

- Autofill function
- Hotkey "Space" to set focus •
- Insert coordinates or lengths (while using • guiding lines)
- Fulltext search, results listed in order of • priority
- Shows icons and hotkeys, same as in the • menu
- "Show me" function to show position of commands in menu
- Go to webhelp option
- Drag searched command to populate the • Process toolbar

🛝 Hinge on 2D 🔏 Hinged support in node SHOW ME 0 0 Hinges at both ends of beam 0 н Make everything in the clipping box ... A- Make everything in the workplane vi... in GeometryLengthInch Toggles between the following mod.. - Connects absolute nodes and edges .

b Hinge on 1D

O Hinge at the beginning of beam

Hinge at the end of beam

45°_Revize3_LIN_FINAL

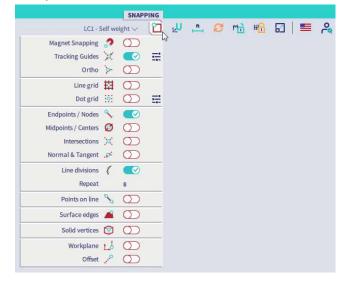
Ctrl+H

? ×

2.2.3. Status Bar

The Status bar allows the settings and toggles of most often used options

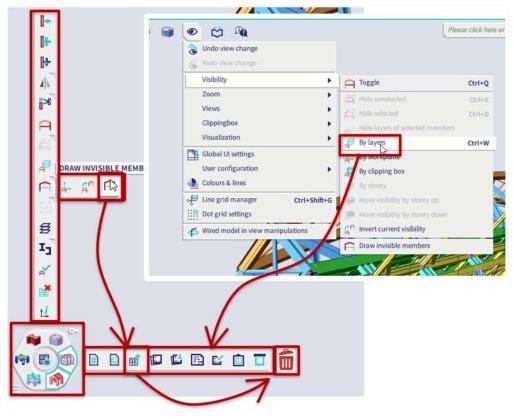
- Select the active load case
- Snapping and UCS settings •
- Result lock, Grid lock and Autorefresh •
- Codes and annexes management •



2.2.4. **Process Toolbar**

The Process toolbar bar has following features:

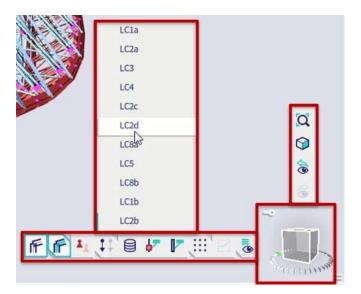
- Fully customizable panels
- Available up to 8 opened workstations selectable by you based on your project
- Have up to 2 separate toolbars open
- Additional subpanels ("stacks") may be used to save space
- Interact with the toolbars simply by drag and drop. Drag in commands from menu, SCIA Spotlight or status bar.



2.2.5. View Bar

The View bar has following features:

- View Management
 - o Navicube
 - o View toolbar
- Management of View parameters
 - o Rendering and volumes toggle
 - Draw additional data as Supports and Loads, Labels of nodes and elements, ...
 - Additional settings available on hover (pick visible Load cases)
 - Access all the settings for all entity type



2.2.6. **Properties Panel**

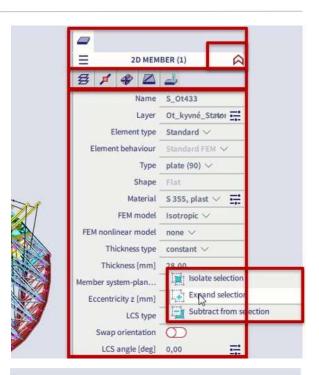
The Properties panel has following features:

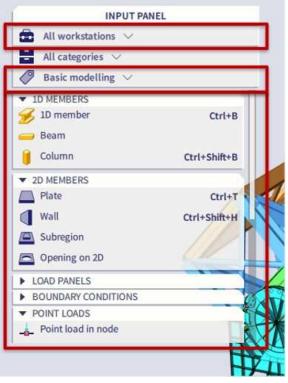
- Show associated data of the objects (show hinges when beam is selected)
- Contain all properties, illustrations, action buttons
- Possibility to use simplified view on the most important properties
- Manipulate with the selection (right click):
 - o Add elements that share some properties into the selection
 - o Isolate elements base on a property they share or based on entity type
 - o Remove some elements from selection

2.2.7. Input Panel

The Input panels contains:

- Tools to create anything
 - o Everything what may be inputted is here
 - Sorted in Workstations: Structure, Load, Dynamics,..
- Filters by Categories or tags
 - Possibility to have one workstation or multiple at the same time
 - Tags are new concept to filter based on context of the project
 - Work with all tools or only with the most common ones
- Dual mode of display
 - Mode with names and hotkeys (consistent look and feel with the Menu)
 - o Icon only mode





2.2.8. Marking menu

The Marking Menu is an invisible menu which you can call for at any location by holding ALT and clicking on the right mouse button.

By hoovering over the 4 submenus you can access SCIA's most basic functions:

- Model:
 - o 1D member
 - o Column
 - o Plate
 - o Panel with load to 1D & edges
 - o Hinge on 1D
 - o Support in node
- Modify:
 - o Move
 - о Сору
 - o Multicopy
 - o Break in defined points
 - o Connect members/nodes
 - o Disconnect members/nodes







- Load:
 - o Point load in node
 - o Point load on 1D
 - o Line load on 1D
 - o Line load on 2D edge
 - o Surface load on 2D
 - o Free surface load
- Visibility:
 - o Hide selected
 - o Hide unselected
 - o Make layers visible





Chapter 3: Geometry input

3.1. 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.

3.1.1. Cross-sections

When entering one or more 1D members, a cross-section is immediately assigned to each member. By default, the active cross-section is represented. You can open the profile library to activate another cross-section. If you want to add a member before a cross-section has been defined, cross-section IPE240 will be used automatically.

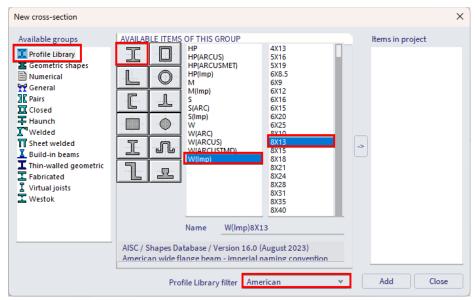
Adding a cross-section

Go to Main menu > Libraries > Cross sections (or with the icon in the workstation Structure).

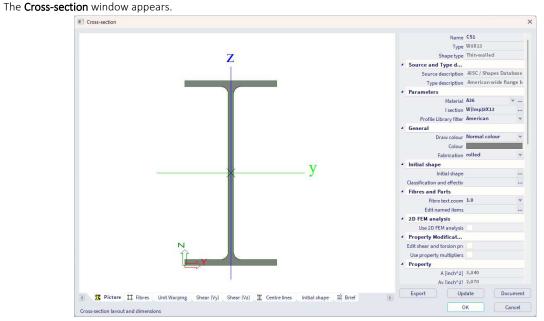
The cross-sections manager is opened. If no cross-sections have been entered in the project yet, the **New cross-section** window will be automatically opened.

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

In the Available items of this group, you can choose for an I profile. Choose W8X13 from the list.



Click [Add] or the arrow-button to add the cross-section to the project.

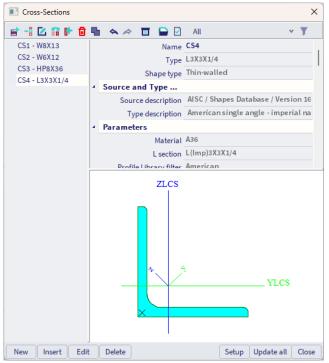


Click **[OK]** to confirm, the cross-section is now added to the cross-section list in your project. Add **W6X12** and **HP8X36** in a similar way.

In the Available items for this group, you can choose an angle section (just below the I profile group). Choose L3X3X1/4 from the list.

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

Click [Close] in the New cross-section window, the Cross-Sections manager appears.



Click [Close] to close the Cross-Sections manager and to return to the graphical scene.

3.1.2. Geometry

In the **Input panel** we can select the **workstation Structure** to choose from different structural element types 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.

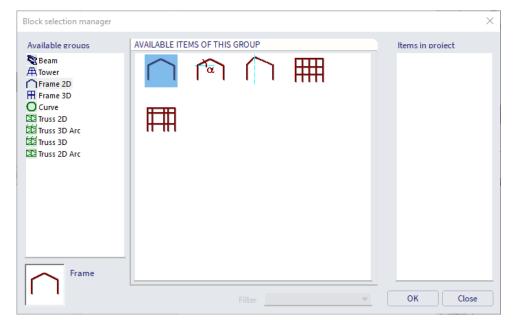
Catalogue blocks

You can use single columns and beams to enter the frame., but SCIA Engineer offers as well multiple **Catalogue blocks**, allowing for a smooth and simple input of the structure.

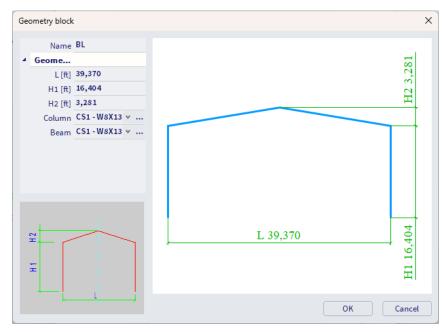
To enter a new frame, use the option Catalogue Blocks in the category Import & Blocks of the workstation Structure.

Ξ	INPUT PANEL	
	Structure	\sim
	Import & Blocks	\sim
CATA	LOGUE BLOCKS	\sim
Ę	🛱 A 📑 🖬	

The Block selection manager is opened. Under Available groups choose the first option Frame 2D. In the Available items of this group you can choose the first shape, Frame.



Confirm your choice with [OK].



The Geometry block window appears. Now, enter the frame dimensions: L = 39,37 ft, H1 = 16,404 ft and H2 = 3,281 m. In the combo-box with cross-section selector choose W8X13 for the Column and for the Beam.

Confirm your input with [OK].

The Catalogue block manager appears.

Catalogue block		×
et -: 🖸 👬 🕪 🖻 🐟 🗢 🔲 🖨 🖸		
BL Name B	8L	
4 Geometry		
L[ft] 39		
H1[ft] 10		
H2 [ft] 3,		
	S1-W8X13	
Beam C	S1-W8X13	
	,281	
	H2 3,281	
L3	39,370 +04 ⁵ 91 1H	
	1 16	
	Ξ	
New Insert Edit Delete	(ок

Click **[OK]** to return to the graphical scene. The frame is now graphically represented by thin blue lines attached to mouse cursor. You are now asked to select an insertion point.

The frame is positioned with the left column in the origin of the coordinate system. Type the coordinates **0;0** (with semicolon) **or 0 0** (with space) in the SCIA Spotlight and press **<Enter>** to confirm your input.

Block - Insert block - End point >

Finish the input with the **<Esc>** key.

Notes:

- The properties of selected elements are shown and can be modified in the **Property panel** on the right-hand side of the user interface.
- With zoom all (Main menu > View > Zoom > Zoom all), or double-click with the mouse wheel, you can visualize the entire structure.
- The use of , (comma) or . (point) to work with decimals depends on the regional settings of Windows.
- You can use the @-symbol to work with relative coordinates.

Multicopy

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

First select all (already modelled) entities to be copied. As you must copy all members, you can use the select all (Main menu > Tools > Selections > Select all) function.

This way all members and nodes are selected; this is represented by dashed violet lines:

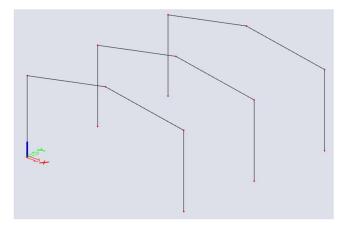


Now, you can select the **Multicopy** command via **Main menu > Edit > Modify > Multicopy** or with the icon in the **workstation Structure**. In the **Number of copies** field enter **2**. To manually set the distance between the frames, deactivate the **Define distance by cursor** option. Now, you can enter the distance 6 m in direction Y.

Multicopy			×
Number of	copies 2	+ -	Connect selected nodes with new beams Copy additional data
DISTANCE V	ECTOR		HOW TO DEFINE THE DISTANCE ?
Define dista	Define distance by cursor		 between two copies
x	0,000	ft	from original to the last copy HOW TO DEFINE THE ROTATION ?
У	19,685	ft	between two copies
z	0,000	ft	from original to the last copy
ROTATION			ROTATION AROUND
rx	0,00	deg	current UCS
ry	0,00	deg	distance vector
rz	0,00	deg	OK Cancel

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

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

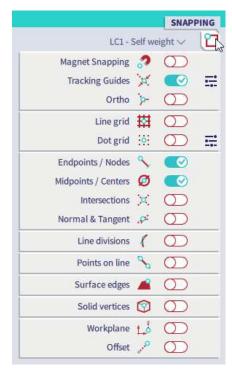


<u>Note</u>: it's possible to connect the frames automatically with beams with the multicopy command. Therefore use the option **Connect selected nodes with new beams**. After confirming with [OK] the software will ask you for the cross-section for the new beams. When you use this option for your structure, you don't need to select the lower nodes of the columns in this case.

Snapping

Connecting beams of the frames can be added after the frames are modelled. 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 snapping.

You can find Snapping in the Status bar. You can activate the options Endpoints / Nodes and Midpoints / Centers.



Entering a beam

To enter a new beam, use the Beam command in the category 1D Members of the workstation Structure.

Ξ	INPUT PANEL
	Structure 🗸
-	1D Members $$
0	BEAM gs 🖂
ø	- 1 7 9 9 0 0

In the cross-section field, choose the third cross-section, CS3 - IPE160.

For the structure type Frame XYZ or General XYZ there are two horizontal axes (i.e. X and Y), so you must indicate the right direction for the horizontal beam in the **Direct** field. Choose **axis Y**.

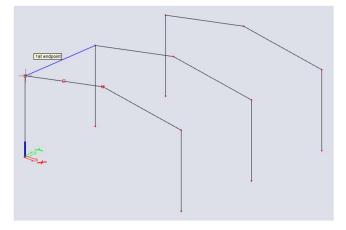
The beam **length** is **6 m**.

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

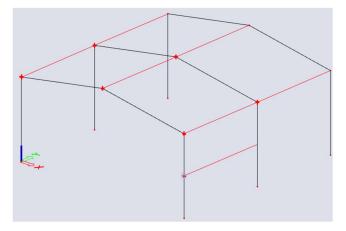
Horizontal beam				×
		Name	B13	
		Туре	beam (80)	*
α	\	Analysis model	Standard	v
		Cross-section	CS2 - W6X12	×
		Alpha [deg]	0,00	
z' J)	Member system-line at	Centre	*
		ey [inch]	0,000	
		ez [inch]	0,000	
ez		LCS	standard	*
ey (1)		LCS Rotation [deg]	0,00	
11		FEM type	standard	*
z		Layer	Layer1	۷
	Buckling			
XY		System lengths and buckling settings	Default	
		Secondary member		
	4 Geometry			
			axis Y	*
		Length [ft]	19,685	
		Insertion point	begin	Υ.
	Structural mo	del		
				OK Cancel

Confirm your input with [OK].

Now, you can enter the beams 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:



Enter the other beams of the roof in a similar way, always by clicking the top nodes of columns or inclined beams. Add a beam as well at half of the height of the first right span.



Press **<Esc>** to finish the input.

Press **<Esc>** once more to cancel the selection.

Entering diagonals

To enter a new diagonal or bracing, use the **1D member** command in the **category 1D Members** of the **workstation Structure** (or via the marking menu).

	Ξ	INPUT PANEL
		Structure 🗸
	÷	1D Members $$
1D	мемв	BER tags 🗸
	5	- 7 9 9 0 0

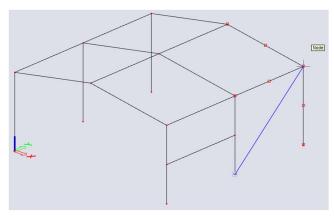
In the **Type** field, choose **Wall Bracing (0)**. This specification is considered for the structural model only and does not influence the analytical model or the results.

In the **Cross-section** field, choose the fourth section, **CS4 – L3X3X1/4**. Note that no length or direction is requested this time, as you will define the geometry afterwards.

Member		3
~	Name B20	
	Type wall bracing ((0) *
α	Analysis model Standard	*
	Cross-section CS4 - L3X3X1/	/4 ×
	Alpha [deg] 0,00	
	Member system-line at Centre	*
	ey [inch] 0,000	
	ez [inch] 0,000	
ez	LCS standard	*
	LCS Rotation [deg] 0,00	
ey	FEM type standard	*
2	Layer Layer1	*
	Buckling	
x Y	System lengths and buckling settings Default	
	Secondary member	
	Structural model	
		OK Cancel

Confirm your input with [OK].

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



Repeat selecting start nodes and end nodes to model both bracings at the right side.

Press **<Esc>** to finish the input.

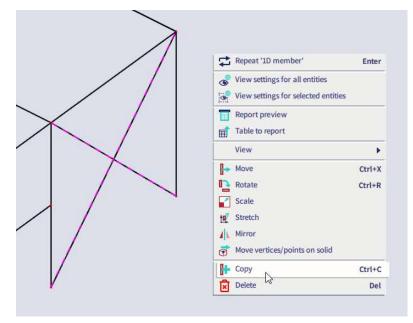
Press **<Esc>** once more to finish the selection.

Copy

We will copy the diagonals from the right side of the structure to the left side of the structure.

First select the entities to be copied. You can select both diagonals with the left mouse button of by dragging a square from right to left with the left mouse button. A violet colour indicates that the diagonals have been selected. Properties of the selected members are shown in the **property panel**.

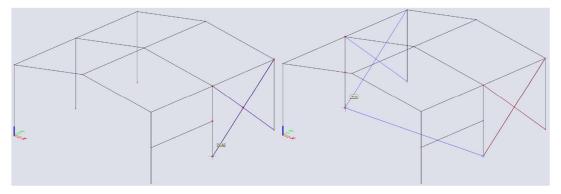
Click with the right mouse button anywhere in the graphical scene. The appearing context menu lists the available possibilities for the selected entities.



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

<u>Note</u>: instead you can also use **Ctrl+C** as hotkey or you can access the copy command via **Main menu > Edit > Modify** or as icon the in the **workstation Structure** of the **process toolbar**.

Now the program asks the start point of the copy. Click with the left mouse button on the bottom node of the column of the middle frame at the right side. Then you must enter the end point, i.e. the position where the starting point should be copied. Now you can select the bottom node of the column of the middle frame at the left side.



When the members are copied, the command remains active until you press **<Esc>**, so that you can copy as well to other positions if needed.

Press **<Esc>** to finish the input.

Press **<Esc>** once more to cancel the selection.

3.1.3. Additional data

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 a constant cross-section, until a haunch is entered. Haunches are entered on the roof beams in this project, at the column sides. A haunch is defined by the following parameters:

- a cross-section with variable height;
- a length, over which the variable height must vary up to 0.

To enter a new haunch, use Haunch on 1D command in the category 1D Members of the workstation Structure.

	INPUT PANEL
	Structure 🗸
-	1D Members $$
0	All tags HAUNCH ON 1D
ø	- 7 🤜 🐨 🖬

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.

Select in Available groups the Haunch group.

In Available items of this group, choose a I + I var cross-section.

New cross-section									×
Available groups	AVAILABLE I	TEMS OF T	THIS GROU	Р				Items in project	
Geometric shapes Numerical	Ŧ	Ţ	Ŧ	Ŧ	Ŧ	I		CS1 - W8X13 CS2 - W6X12 CS3 - HP8X36 CS4 - L3X3X1/4	
Closed Haunch Welded Sheet welded Build-in beams Thin-walled geometric Westok	포			I	T		Ą		
I+Ivar		Profile Li	ibrary filte	r Ameri	ican	*		Add	Close

Click [Add] or on the arrow-button to add the cross-section to the project.

Now the cross-section window appears. Here, the properties of the variable cross-section can be changed.

Cross-section Name CS5 Type I+Iva Detailed W8X13; 6,000 Shape type Thin-walled Parameters Material A36 v ... va [inch] 6,000 I section W(Imp)8X13 2x W(Imp)8X13 ۷ Profile Library filter All cross-secti Draw colour Normal colou Fabrication welded Initial shape Initial shape Classification and effectiv Fibres and Parts Fibre text zoom 1.0 Edit named items 4 2D FEM analysis va 6,000 Use 2D FEM analysis Property Modificat... Edit shear and torsion pre Use property multipliers Property A [inch^2] 6,220 Ay [inch^2] 3,121 Az [inch^2] 3,291 Update Export Document 🧃 🕱 Picture 🎞 Fibres UnitWarping Shear (Vy) Shear (Vz) 🌋 Centrelines Initial shape 🖹 Brief (D) Cancel OK Cross-section layout and dimensions

In the I section field, change the section in a W8X13 by clicking the button behind the section type.

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

The cross-section manager appears; click **[OK]** to close this window as well.

Now, the Haunch on beam window is opened.

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

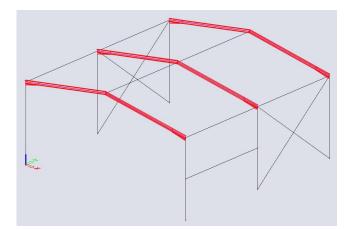
In the **Coord. definition** field, choose the option **Abso** to indicate that the length, over which the variable height must vary, can be entered in absolute units, i.e. in meter.

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 3.5 ft.

Haunch on beam	X		×
	Name	H1	
	Position		r
	Cross-section	CS5 - I + I var (W(Imp)8X13; 6,000) v	
	Use from Css	no	
	va [inch]	6,000	
	4 Geometry		
В	Coord. definition	Abso	r
i ey	Length x [ft]	3,500	
H			
-ez			
		OK Cancel	

Confirm your input with [OK].

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:

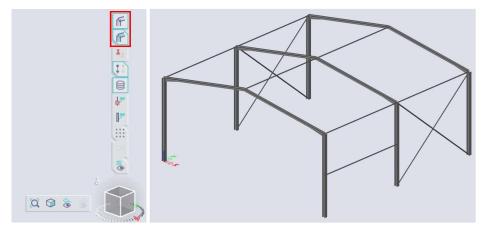


Press **<Esc>** to finish the input.

Press **<Esc>** once more to cancel the selection.

To visualize the haunches, you need to click the following buttons in the **view bar**:

- Volumes to show the surfaces of the sections;
- Shading to obtain a rendered view of the members.



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

Hinges

In SCIA Engineer, every node where two or more members connect is regarded as fixed, until a hinge is entered and some translations and/or 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.

To enter hinges, use the **Hinge on 1D** command in the **category 1D Members** of the **workstation Structure** (or via the marking menu).

Ξ	INPUT PANEL									
	Structure 🗸									
÷	Boundary conditions \checkmark									
0	All tags \lor									
4	HINGE ON 1D 🔝 🖘 🛢 👃									
4	🥺 🕨 🛝 🚹 🚠 💠 🛥 👘									
4										

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

To obtain a hinge, the rotations fiy and fiz are set to Free, the translations and the rotation fix remain Rigid.

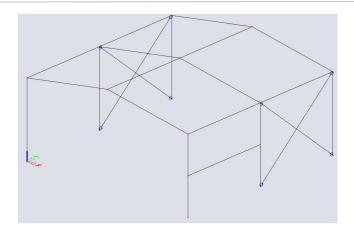
II Hinge on beam		×
Name	H1	
Å [©] Z Position	Both	*
ux	Rigid	*
(i) UZ uy	Rigid	٧
	Rigid	*
	Rigid	*
fiy	Free	*
fiz	Free	٧
	OK Ca	ncel

Confirm your input with [OK].

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

Press **<Esc>** to finish the input.

Press **<Esc>** once more to finish the selection.



<u>Notes</u>:

- Instead of adding hinges to the diagonals, you can set the property **FEM type** to **axial force only** in the **property panel**. Pay attention to not combine hinges with this property, since this could lead to issues for the solver when calculating the model.
- Hinges are always defined with regard to the local coordinate system of a member.
- Increase Scale add data in the Status bar if you feel that the hinge symbol is too small.

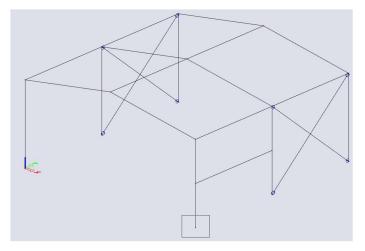
				SCALES	4	
LC1 - Self weight \lor	2 🖉 📩	. 0	mi	H		R
	Add data scal	e 1	1	0	+++	
	Results scal	e 1		0		
	Scale settings	i.				

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. You can select these nodes manually, one by one, but SCIA Engineer offers a simple method to select multiple entities based on a common property.

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.

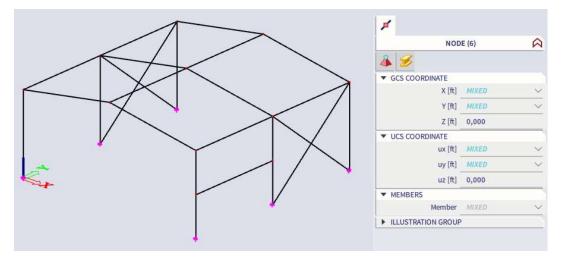


The **property panel** shows the properties of this node. Now, choose the property to be used for the selection of the entities. For this project, you want to select all bottom nodes. The common property of these nodes is their coordinate in global Z direction.

Click with the right mouse button on the Z [m] property to select the appropriate row and choose the Expand selection option.

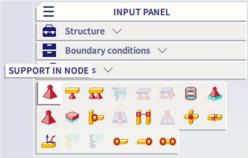
	1			
	=	NOD	E <mark>(</mark> 1)	
	8			
		Name	N5	
	▼ GCS CO	ORDINATE	-	-
		X [ft]	39,370	
		Y [ft]	0,000	
		Z [ft]	0,000	
Isolate se	lection			-
Expand s	election		2	
Subtract	from selection	n	20	
		uz [ft]	0,000	
	▼ MEMBER	RS		2
		Member	B4	

The program will search all entities with the same property. In this example, the program will select all nodes, for which the Z [m] property corresponds to 0 ft.



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

To enter supports, use the **Support in node** command in the **category 1D Members** of the **workstation Structure** (or via the marking menu).



You can easily define all 6 end conditions at once by choosing the **Constraint** as **Hinged**, so that all **translations** are **Rigid** and all **rotations** are **Free**.

Support in node			×
	Name	Sn1	
	Туре	Standard	*
	Angle [deg]		
▲Rz	Constraint	Fixed	*
	х	Rigid	~
7	Y	Rigid	*
	Z	Rigid	*
XXY	Rx	Rigid	*
Rx	Ry	Rigid	*
i		Rigid	*
	Default size [ft]	0,656	
Z	▲ Geometry		
	System	GCS	*
XY			
		ОК	Cancel

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

Press **<Esc>** to finish the selection.

<u>Notes</u>:

- If you draw the selection 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 *input panel* includes a number of predefined supports via the tag **Quick input**. For this project, you could have used the **Hinged support in node** icon.



• Similar to the selection of supports, you could have used the **Expand selection** option to select all diagonals, when the hinges had to be entered. The determinant property here would have been the cross-section.

3.1.4. Check structure

After input of the geometry, the input can be checked for errors by means of the option **Check structure**. 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.

You can find the command Check structure as icon in the workstation Structure of the process toolbar or via Ma i	n
menu > Tools > Check structure.	

+	Check of structure data				×
	CHECK OF NODES				
8+	Search nodes				
H+					0%
	Search duplicate nodes		Ignore param	eters	
4					0%
>8	CHECK OF MEMBERS				
	Check members			2	
A	Search null members	0%	Null members:	0 mbers	
	Search duplicate members			0	
		0%	Delete duplica		
#				0	
A			🖌 Delete invalid	parts	
	CHECK OF DATA REFERENCES				
	Check data references		Memory efficie	nt method	
每		0%	Fast method		
IJ	CHECK OF ADDITIONAL DATA				
	Check additional data position			0	
CHECK STRUCTURE		0%	Correct positio		
	Check free load distribution points			0	
		0%			
t <u>í</u>	CHECK OF STEEL CONNECTIONS				
	Check steel connections			0	
		0%	Delete invalid		
	Check load panels Chec	ck cross-lir	nks		
	Check additional data Check d	uplicity of	names	Check	Cancel

In the Check of structure data dialogue the different available checks are listed. Click [Check] to perform the checks.

The Data check report window appears, indicating that no problems were found.

Data check report X						
Data check finished.						
ОК						

Close the check by clicking [OK].

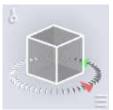
In case of any problem SCIA Engineer tries to automatically correct the structure data (delete duplicated entities, correct wrong reference, etc.).

3.1.5. Graphic representation of the structure

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 of the model;
- use Zoom commands;
- edit view parameters.

You can set the view point with the Navicube.



You can rotate/tumble, pan, zoom, set the view in the direction of one of the global axes,... All possibilities are shown on following help-page: https://help.scia.net/20.0/en/#rb/basictools/adjusting the view.htm

Notes:

- 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 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 graphical scene is filled by the structure).

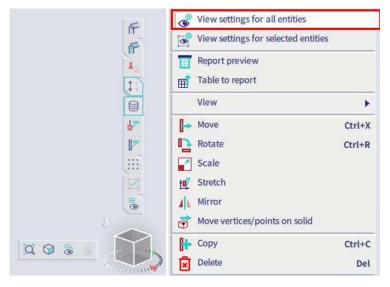
Several zoom commands are available via Main menu > View > Zoom

💌 🛱 🕰		Please click here
Undo view change Redo view change		
Visibility	•	
Zoom Views		🚱 Zoom in
Clippingbox Visualization	*	Q Zoom out Q Zoom cutout Q Zoom all
Global UI settings User configuration	•	Zoom to selection
 Colours & lines Fonts settings Structural beam types settings Dimension lines settings 		
이번 Line grid manager Ctrl+Sh 	ift+G	
General Wired model in view manipulations		

Note: Zoom to selection is only available in case that you have made a selection in the model.

You can use the view bar to show/hide volumes, shading, supports, loads, model data, labels of nodes, labels of members,...

Via **More options** you can choose for **More view settings** in order to access all view parameters. You can access this dialogue as well via right-clicking in the graphical scene and choosing for **View settings for all entities** or **View settings for selected entities** (if an entity or multiple entities are selected you can change view settings for only that selection).



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

- **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.

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 member in the label (e.g. B1)
- Cross-section name: show the name of the cross-section in the label (e.g. CS1)
- **Cross-section type**: show the cross-section type in the label (e.g. HEA200).
- Length: show the length of the member in the label (e.g. 5.000 m).

Vie	w parameters setting - Structu	re					Vie	ew parameters s	etting - Labels						
					Lock position	on 🔽	Check / Uncheck group						Lock position		
٩	Structure All Labels	A Model	Loads/masses	28 Steel	F Connections	Moc 🕨 ir	٩	T Structure	AB Labels	A Model	Loads/masses	28 Steel	F Connections	Moc 1	
	Check / Uncheck all	_		_			_	Check / Unchec		_	_	_			
	Service							Service							
	Display on opening the service								ening the servic	e 🔽					
P	Structure						E	Beam labels	,						
	Style + colour	normal				•	11-	Display label							
	Draw member system line	×						Name		V					
	Member system line style	system line				•		Cross-section	name						
	Model type	structural	model			-		Cross-section	ype						
	Display both models							Length							
	Member surface							Layer							
	Rendering	wired				-		Type and prior	ity						
	Draw cross-section						E	Nodes labels							
	Cross-section style	section				-		Display label							
	Effective width of plate ribs							Name		V					
	Draw effective width	v						X-coordinate							
	Rendering	transparen	t			-		Y-coordinate							
	Structure nodes							Z-coordinate							
	Display	v					E	System length	s						
	Mark style	Dot				-		Display label		V					
	Member parameters	_						Name		V					
	System lengths							Label							
	Member nonlinearities	V					E	Nonlinearities		_					
	FEM type	V						Display label							
	Joists						E	Labels of local	axes	_					
	Local axes	_						Nodes							
	Nodes							Members 1D							
	Members 1D						E			_					
								Display vertex	label						
	Show names in tab			ОК	Apply	Cancel		Show names in	tab			ОК	Apply	Cancel	

Chapter 4: Loads and combinations

4.1. Load cases and load groups

Each load is attributed to a **load case**. One load case can contain different load types like line loads, surface loads, ... 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.

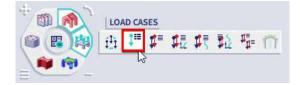
Each load case is associated with a **load group**. The load group contains information about the category of the load (service load, wind, snow...) and its appearance (default, together, exclusive). In an exclusive load group, the different load cases attributed to this load group cannot act together in a single combination when using envelope combinations or code combinations.

We recommend that you thoroughly read the chapter about loads and combinations in the reference manual or on our help pages.

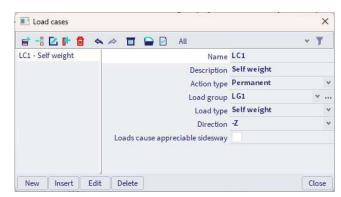
Three load cases are entered in this project:

- LC1: Permanent load case Self weight;
- LC2: permanent load case Roof dead load;
- LC2: variable load case: Wind load (side wind on the frames).

You can find the command Load cases as icon in the workstation Loads of the process toolbar, via the hotkey Ctrl+L or via Main menu > Libraries > Load cases, combinations > Load cases.



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 and by default, the load case LC1 is automatically 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.

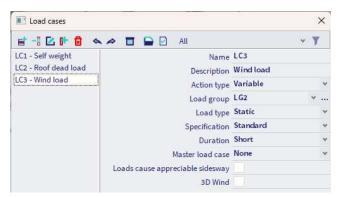


We will create a second permanent load case for the roof weight. Click on [New] and change the Description to Other permanent loads. The Load type is now set to Standard.

Load cases			×
📑 📲 🗹 🗈 🗟 🐟	🗢 🔲 🕒 All	٧	T
LC1 - Self weight	Name LC2		
LC2 - Roof dead load	Description Roof dead load		
	Action type Permanent		~
Load group LG1			×
	Load type Standard		*
	Loads cause appreciable sidesway		

Now we will create a variable load case. Click on [New], change the Description to Wind and change the Action type to Variable.

The load group LG2 is automatically created. Click on the three dots [...] to display properties of the load group.



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

Load groups			×
📑 📲 🗹 📴 📾 🐟 🗢 🛅 🖨 🖸 All			× T
LG1	Name	LG2	
LG2	Relation	Exclusive	~
	Load	Variable	~
	Load type	W - Wind load	~
New Insert Edit Delete			Close

Click [OK] to close the load group manager and to return to the load cases manager.

Click [Close] to close the load cases manager.

<u>Note</u>: each load is classified in a load 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 load group "Wind" to avoid simultaneous action.

4.2. Load panels

We will make use of load panels to distribute surface loads to line loads on the members.

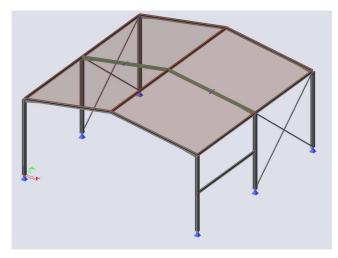
To enter load panels, use the **Panel with load to 1D & edges** command in the **category Load panels** of the **workstation Structure** (or via the marking menu).

1	INPUT PANEL		
ĺ	💼 Structure 🗸		
ĺ	🗧 Load Panels 🗸		
PANE	WITH LOAD TO 1D & EDGES		
ĺ	× = # = # A		

Set Load transfer direction to X (LCS panel) and Load transfer method to Tributary area in the properties dialogue.

Panel			×
	Name	LP1	
	Panel type	To panel edges and beams	*
	Load transfer direction	X (LCS panel)	*
	Max.angle for transfer [deg]	5,00	
	LCS type	Standard	*
	Swap orientation	no	
	LCS angle [deg]	0,00	
	Layer	Layer1	×
	Supporting members validity range (with respect to panel Z-	All	*
	Max. eccentricity of members [ft]	0,656	
	Load transfer method	Tributary area	*
	Selection of entities	Auto selection	¥
			OK Cancel

Use the snapping points in the roof to create a load panel for each pitch of the roof.



Press **<Esc>** to finish the input.

Press **<Esc>** once more to deselect the load panels.

<u>Note</u>: the first two nodes that you use to create the geometry of the load panel will determine the load transfer direction. You can change the direction if needed via the property panel.

4.3. Loads

In the **status bar** you can change the active load case. Activate load case LC2.

	ACTIVE LOAD CASE								
A T	LC2 - Roof dead load 🗸 📐	2	Ł	ft	C	mi	₩G	🖬 💻	2
1	LC1 - Self weight								
1	LC2 - Roof dead load								
Į1≡	LC3 - Wind load								
÷	Manage load cases Ctrl+L								

Select the **Surface load on 2D** command in the **category Surface loads** of the **workstation Loads** (or via the marking menu).

	INPUT PANEL			
	💼 Loads 🗸			
	Surface loads 🗸			
SURFAC	E LOAD ON 2D			
	92			

Set Value [kN/m^2] to -0.20 in the properties dialogue. The Direction is the global Z direction and the value is negative, so the load is acting vertically downwards.

Surface force			×
	Name	. SF1	
	Direction		٧
	Туре	Porce	٧
-P	Value [kip/ft^2]	-0,02	
	4 Geometry		
	System		٣
	Location	Length	٣
\checkmark			
Z			
x y			
		OK Cance	4

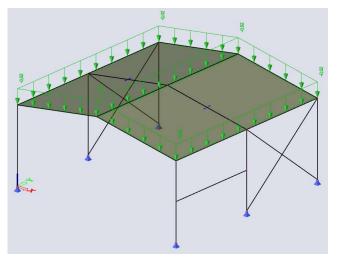
Confirm your action with [OK].

Select all load panels by clicking on them or use the select all command (Main menu > Tools > Selections > Select all).

Press **<Esc>** to finish the input.

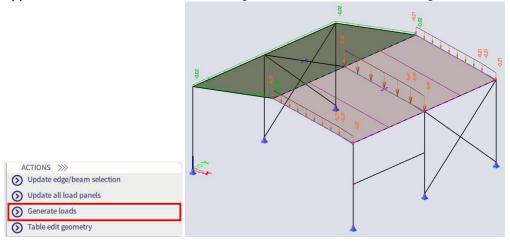
Press **<Esc>** once more to finish the selection.

The loads are coloured in green.



When you calculate the model, the loads will automatically be converted from surface loads to line loads.

If you like you can already generated them before the analysis to evaluate them. Therefore select a load panel and in the **property panel** choose **Generate loads** as **Action**. The generated loads are coloured in orange.



Go to the **view settings** and switch on the tab **Loads/masses** the **Generators** to **Generated** or **Original + Generated** if the loads remain green. Or use the toggle in the View bar.

View parameters setting			F
		Lock position 🔽	
Structure AB Labels	▲ Model 🛃 Loads/masses 🐼 Steel 🖙 Connections 🍸 Composite	Modelling/Drawin()	L (r
Check / Uncheck all			
Service			TOGGLE LOADS
Display on opening the service			Generated loads
Display loads			
Display			LC1 - Self weight
Style	Colour by action type	•	1 C2 - Roof dead load
Load case	LC2 - Other permanent loads	-	
Display eccentricity			LC3 - Wind load
Generators	Generated		
Line forces	Original		
On beam	Generated Original + Generated		
Surface loads	Uriginal + Generated		
On 2D member			
Labels of loads			
Display label			
Name			
Value			Ľ
Tot. value			
Eccentricity label			
Show names in tab		Apply Cancel	

In the **status bar** change the active load case to load case LC3.

Select the Line load on 1D command in the category Line loads of the workstation Loads (or via the marking menu).

	Ξ	INPUT PANEL		
	â	Loads \checkmark		
	📕 Line loads 🗸			
LINE	OAD	ON 10 55 🗸		
	镼	🖕 🖽 👄 🚅		
1	-			

Set Value - P [kN/m] to -2.80 in the properties dialogue. 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 elements.

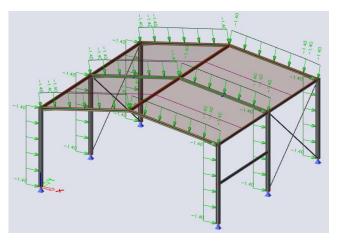
Line force on beam				>
RZ		Name	LF4	
RX RV 4-P2		Direction	Z	*
-FZ		Туре	Force	*
		Angle [deg]		
-P1		Distribution	Uniform	*
and the second		Value - P [kip/ft]	-0,03	
ey lez		Bottom flange		
	Geometry			
x1 x2		System	LCS	*
		Location	Length	
		Extent	full	*
		Coord. definition	Rela	v
		Position x1	0,000	
		Position x2	1,000	
		Origin	From start	*
	✓ Eccentricity			
		Eccentricity ey [ft]	0,000	
		Eccentricity ez [ft]		
				OK Cancel

Confirm your input with [OK].

Select the members where this load must be positioned: the roof girders and the columns.

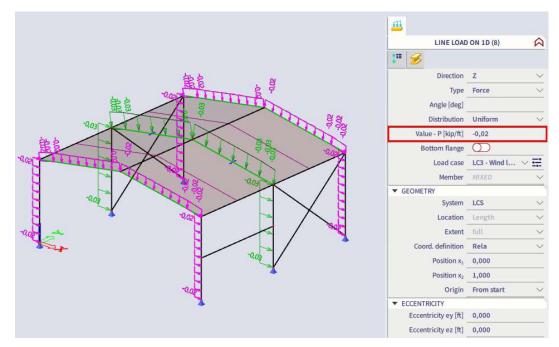
Press **<Esc>** to finish the input.

Press **<Esc>** once more to finish the selection.



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. The mentioned members must have only half of the load value because only half of the loading width, affected by wind, belongs to them.

The common properties of the 8 series are displayed in the property panel. Change the value from **-0,034 kN** to **-0,017 kN**.



Note that you can also use simple formulas in these numerical input fields:

Value - P [kip/ft]	=-0,0342906/2
Bottom flange	fx -0.0171453

4.4. Combinations

By default automatic code combinations are activated in the Actions tab of the Project data dialogue (via Main menu > File > Project Settings).

Project data		2
Basic data	Functionality Loads Unit Set Protection	
	Acceleration of gravity 32,19 ft/s^2	
	WIND LOAD	
	According to code v IBC / Basic wind speed V - 100.0 m/s	
	SNOW LOAD	
	According to code	
	CODE COMBINATIONS	
	Automatic LRFD v	
	OK Car	col

Click on the command **Combinations** in the **workstation Loads** in the **process toolbar** (or via **Main menu > Libraries > Load Cases, combinations > Combinations**) to have a look at the automatic generated combinations.

ATIONS
THE #3 \$2 TH

Two code combinations are created, one for the Ultimate Limit State and one for the Ultimate Serviceability State.

Combinations			×
📑 📲 🗹 🖬 🗟 🐟 🗢	Input combinations	*	
LRFD-Ult (auto)	Name	LRFD-Ult (auto)	
LRFD-Serv (auto)	Description		
	Туре	ASCE-7 (LRFD) - ultin	nate
	Updated automatically	~	
	Active coefficients		
4 Cc	ontents of combination		
	LC1 - Self weight [-]	1,00	
	LC2 - Roof dead load [-]	1,00	
	LC3 - Wind load [-]	1,00	
			-
Actio	ns		>>>
		Explode to envelopes	
		Explode to linear	>>>
New Insert Edit D	elete		Close

You can click on [>>>] next to the Action Explode to linear to have a look at the individual linear combinations to evaluate if the load groups are defined correctly.

Chapter 5: Analysis

5.1. Linear analysis

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

Use the **Calculate** button in the middle of the process wheel in the **process toolbar** to start the analysis (or via **Main menu > Tools > Calculation & Mesh > Calculate**).



Make sure that the option **Connect members/nodes** is turned on in the **Mesh setup**, so all entities are properly connected before the analysis. Select **Linear analysis** and press **Calculate**.

FE analysis			×
Calculations	4	Mesh setup	
✔ Linear analysis Load cases: 3		Average number of 1D mesh elements (Average size of 1D mesh element on cu Average size of 2D mesh element [ft]	0,656
Other processes	_	Connect members/nodes	
Save project after analysis		Setup for connection of structural entit	
The project version will be updated.	¢	Advanced mesh settings	
		Solver setup	
		Specify load cases for linear calculation	
		Specify combinations for linear stability	
		Specify combinations for nonlinear sta	
	⊳	Advanced solver settings	
Calculate			

The calculation process dialogue indicates the progress of the analysis. After finishing the analysis, you can consult the Results tab and view the maximum deformation and rotation for the normative load case. Click **[Close]** to close this window.

CALCULATION PROCESS		×
Calculation Results		
Calculation finished successfully	100%	Cancel
# Name	Progress	
1 Mesh calculation	🤣 Finished	
5 Linear analysis	🥑 Finished	
CALCULATION PROCESS		×
Calculation 🥝 Results		
Preparation of static load cases: OK Preparation of calculation settings: OK Mesh generation: OK Number of nodes: 55 Number of 2D elements: 0 Number of 1D elements: 55 Linear analysis: OK Maximal translation -3,376 inch In node NS [60:00,6:000] (load case LC2) Maximal translation 0,957 deg In node NS [10:00,12:000,0:000] (load case LC1) Sum of loads and reactions is OK		Close

Chapter 6: Results

When the calculation is completed, results can be viewed. The relevant commands in the **workstation Results** become active now.

6.1. Reactions

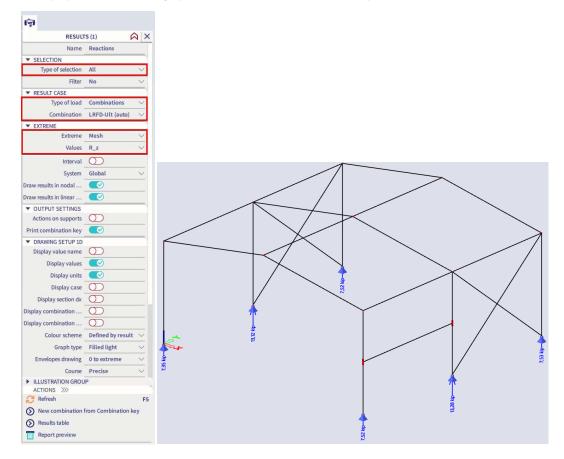
Select the workstation Results in the process toolbar and click on the command Reactions.



Go to the property panel and set:

- Type of selection to All
- Type of load to Combinations
- Combination to LRFD-Ult (auto)
- Extreme to Mesh
- Values to R_z

The action button **Refresh** has an orange highlight, i.e. the graphical scene must be refreshed. Press the action button **Refresh** to display the results in the graphical scene in accordance with the options above.



System: Gl Extreme: N Selection:	n: LRFD-Ult obal	(auto)									
System: Gl Extreme: N Selection:	obal	(auto)									
Extreme: No.											
Iodal rea	ctions										
Name	Case	Ra	Ry	Rz	Mx	My	Mz	ex	ey		
		[kip]	[kip]	[kip]	[kipft]	[kipft]	[kipft]	[inch]	[inch]		
Sn1/N5	LRFD-Ult	-1,95	0,02	5,32	0,00	0,00	0,00	0,000	0,000		
Sn1/N5	(auto)/1 LRFD-Ult	-1.52	0,02	4,83	0,00	0,00	0,00	0,000	0,000		N
DUTUAD	(auto)/2	1,52	0,02	7,05	0,00	3,00	5,00	0,000	0,000		3
Sn1/N5	LRFD-Ult	-2,37	0,04	7,52	0,00	0,00	0,00	0,000	0,000		
	(auto)/3										
Sn1/N5	LRFD-Ult	-2,46	0,03	6,93	0,00	0,00	0,00	0,000	0,000		
Sn2/N1	(auto)/4 LRFD-Ult	1,52	0,01	4,72	0,00	0,00	0,00	0,000	0,000		
5112/111	(auto)/2	1,52	0,01	4,12	0,00	0,00	0,00	0,000	0,000		
Sn2/N1	LRFD-Ult	2,37	0,02	7,35	0.00	0.00	0,00	0,000	0,000		
	(auto)/3	1000000	12200	1035573	120022	1000000	0.000		202012		
Sn2/N1	LRFD-Ult	1,35	0,01	4,93	0,00	0,00	0,00	0,000	0,000		
C-2/NO	(auto)/1	2.75	0.22	0.54	0.00	0.00	0.00	0.000	0.000		
Sn3/N9	LRFD-Ult (auto)/2	-2,75	0,23	8,54	0,00	0,00	0,00	0,000	0,000		
Sn3/N9	LRFD-Ult	-4,27	0,36	13,28	0,00	0,00	0,00	0,000	0,000		
21202040320	(auto)/3		00071062	0.000000	0.04,000	0.000.00		1000 (C)	1019020000		
Sn3/N9	LRFD-Ult	-4,41	0,33	12,24	0,00	0,00	0,00	0,000	0,000		
Sn4/N10	(auto)/4 LRFD-Ult	2,75	0,24	8,44	0,00	0.00	0,00	0,000	0.000		
5/14/110	(auto)/2	2,15	0,24	8,44	0,00	0,00	0,00	0,000	0,000		
Sn4/N10	LRFD-UIt	4,27	0,37	13,12	0,00	0,00	0,00	0,000	0,000		
27	(auto)/3		25		10.0	2.2		1.5			
Sn4/N10	LRFD-Ult	2,46	0,25	8,90	0,00	0,00	0,00	0,000	0,000		
Sn5/N14	(auto)/1 LRFD-Ult	-2.37	-0.40	7.53	0.00	0.00	0.00	0.000	0.000		
3113/1414	(auto)/3	-2,3/	0,40	1,55	0,00	0,00	0,00	0,000	0,000		
Sn5/N14	LRFD-Uit	-1,52	-0,25	4,84	0,00	0,00	0,00	0,000	0,000		
	(auto)/2			1.100000.00							
Sn5/N14	LRFD-UIt	-2,46	-0,36	6,93	0,00	0,00	0,00	0,000	0,000		
Sn6/N15	(auto)/4 LRFD-Ult	2.37	-0.39	7.52	0.00	0.00	0.00	0.000	0.000		
5110/1415	(auto)/3	2,31	0,39	1,52	3,00	0,00	0,00	0,000	0,000		
Sn6/N15	LRFD-Ult	1,52	-0,25	4,84	0,00	0,00	0,00	0,000	0,000		
10 A	(auto)/2	- 22	- 22	2	8	20		1.52	1		
Sn6/N15	LRFD-Ult	1,35	-0,27	5,05	0,00	0,00	0,00	0,000	0,000		

To display these results in a table, the **Preview** action is used. Click on **Report preview**.

6.2. Internal forces

Select the workstation Results in the process toolbar and click on the command 1D internal forces.

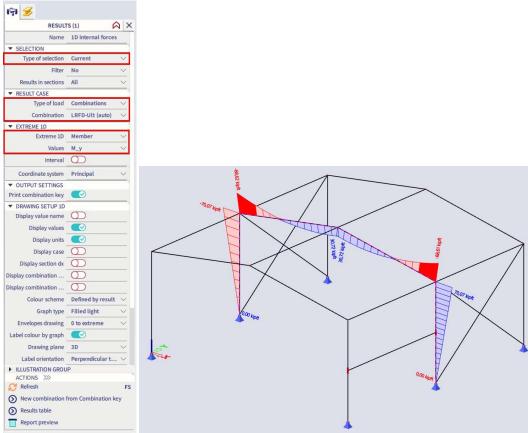


Go to the **property panel** and set:

- Type of selection to Current
- Type of load to Combinations
- Combination to LRFD-Ult (auto)
- Extreme 1D to Member
- Values to M_y

Select columns and the roof girders of the middle frame using the left mouse button.

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



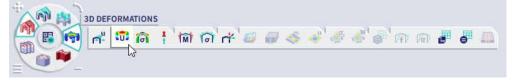
To change the display of the results, the settings of the graphical scene can be adapted via the options in the group **Drawing setup 1D** in the property panel. You can display units, change the colour scheme or graph type, ...

<u>Note</u>: to change the font size of the displayed results, you can use **Main menu > View > Colours & lines**. In this menu, the different sizes of the displayed labels can be changed. Units can be changed via the **status bar > Unit system > Manage units**.

LC1 - Self weight \smallsetminus	2	5 L	ft	Ø	Mi	# <mark>6</mark>		م
Unit system	Imperial	\sim	63					
Unit	Foot \lor							

6.3. **3D deformations**

Select the workstation Results in the process toolbar and click on the command 3D deformations.

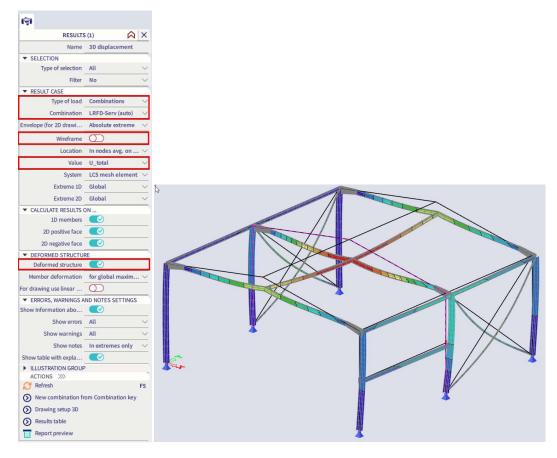


In the **property panel** you can set similar settings as we saw before, like **Type of selection**, **Type of load**, **Extreme 1D**, **Extreme 2D**, ...

For the Values you can choose between the displacements and rotations for the local axis or for the total deformation U total.

There is an option **Wireframe**. When it's turned on, only centre lines and centre planes are used for calculation and displaying the results. This option is much less demanding on performance resulting to faster visualization of the results and the deformed shape of the structure.

With the option **Deformed structure** the result values can be displayed on deformed surface of the structure.



6.4. Results-related status bar options

In the status bar you can find two results-related icons.

• You can switch from **Manual refresh** (the icon is coloured in **orange**) to **Autorefresh** (the icon is coloured in **blue**) and then the results will automatically be refreshed when a result setting in the **property panel** is changed. Pay attention with the Autorefresh setting for bigger models when the property Type of selection is set to All, because it will take longer to refresh the results (especially when you would ask for 3D results without the wireframe option turned on).



• When you activate the icon **Results lock**, results can still be requested when making changes to the model. This is clearly indicated with an exclamation mark and a red square around the graphical scene. Of course later on recalculating the model is still needed to obtain new internal forces, deformations, ...



Chapter 7: Code checks

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

- input of steel data per member;
- input and manipulation of buckling data;
- input of stiffeners, lateral-torsional buckling restraints, steel sheeting, ...;
- performing a strength unity check;
- optimization of the cross-section;
- performing a serviceability unity check;
- input, calculation and creation of drawings for connections;

We will only explain the basics of the steel design in this tutorial. For more information regarding advanced steel calculations (like 2nd order analysis, ...) we refer to the Advanced Steel Training.

7.1. Buckling parameters

Before the steel code checks can be executed, the buckling parameters of the members need to be checked. By means of the view settings, the buckling lengths of the members can be visualized. Select member B1 and activate the option **System lengths** under **Member parameters** in the tab **Structure** of the **View settings**.

Vi	ew parameters setting			
		Lock position		/
٩	🔲 Structure 斗 Labels	Modelling/Drawing 😌 Attributes 📝 Misc.		
	Check / Uncheck all			
E	Structure			
	Member surface			
	Rendering	wired		
	Draw cross-section			
	Cross-section style	section		
E	Effective width of plate ribs			
	Draw effective width			
	Rendering	transparent		-
E	Member parameters			
	System lengths			
	Member nonlinearities			
	FEM type			
	Joists			
E				
	Members 1D			
L				
	Show names in tab	🕑 🕒 ОК Арріу С	Cancel	

The figure above shows that the (default) system/reference length Ly (blue line) for buckling around the strong axis (y-y) is the 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.

To modify the buckling data of a member, select the element and click on the button **Manage** next to the option **System lengths and buckling settings** in the **property panel**.

8			
1D ME	MBER (1)	1	
≠ 🗲 I] 📇			
Name	B4		
Layer	Layer1 🗸 🕂	System lengths and buckling settings	
Layer	Layer1 V		Settings Results
Туре	column (100) 🛛 🗸		Name BG1
Analysis model	Standard V		Buckling span Deflection span
Analysis model	Stanuaru		• y-y Deflection z = y-y * z-z = z-z * Deflection y = z-z *
FEM type	standard 🗸 🗸		y-z = 2-2 * LTB = 2-2 *
Cross-section	СS3 - НР8Х36 🗸 📑		Active buckling constraints Span settings
a [deg]	0,00		Buckling length factors Settings per span
	1		ky factor <u>Calculate</u> ×
lember system-line at	Centre 🗸		Sway y-y From setup * Sway y-y
ey [inch]	0,000		
ez [inch]	0,000		
LCS	standard 🗸 🗸		
LCS Rotation [deg]	0,00		Advanced settings
BUCKLING		K.	~
System lengths and b	Default 🗸 🚍		Save Save

On the **Settings** tab, several data can be changed:

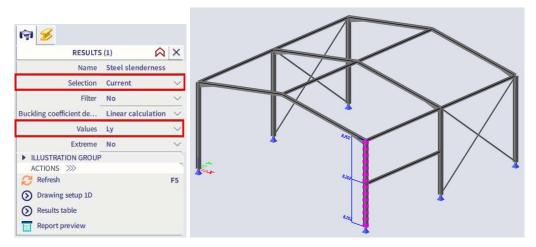
- The Name field contains the name of the buckling parameter, in this case BG1.
- For buckling around y-y and z-z you can set the **ky** and **kz factor**: you can indicate that the program should calculate the buckling factor automatically or you can choose a manual input of this factor. A third option allows for a manual input of the buckling length (instead of factor).
- You can set as well **Sway y-y** and **Sway z-z**: you can indicate if the member is "braced" or not in the direction regarded. When you choose option **From setup**, the default settings are used. <u>Note</u>: the default settings for the buckling parameters are listed in **Steel settings** in the **workstation Steel**. The structure is by default sway for buckling around the strong axis (in the plane) and non-sway for buckling around the weak axis out of plane), taking the presence of wind bracings outside of the plane into account.
- You can define as well settings for torsional buckling (y-z), lateral torsional buckling (LTB) and both deflection directions (used for the SLS check). By default settings are taken from y-y or z-z, but this can be changed with the drop-down lists.

Select the command Steel slenderness in workstation Steel of the process toolbar.



Go to the property panel and set Selection to Current and Values to Ly.

Then press **Refresh** to see the result for the reference length around the strong axis.



You can ask as well for the slenderness (Lam y or Lam z), buckling factor (ky or kz), buckling length (ly or lz),... The buckling length is determined as the reference length multiplied by the buckling factor.

7.2. Steel checks

7.2.1. Strength check

The steel strength unity check includes both a section and a stability check.

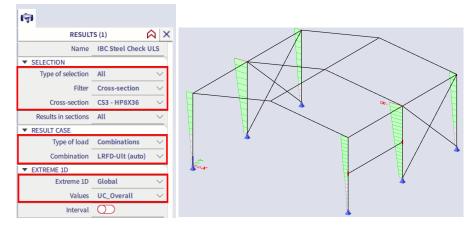
Select the command Steel strength check in workstation Steel of the process toolbar.



Go to the property panel and set:

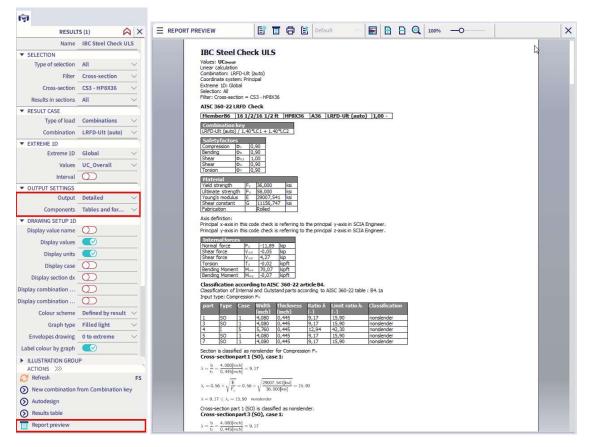
- Type of selection to All
- Filter to Cross-section
- Cross-section to CS3 HP8X36
- Type of load to Combinations
- Combination to LRFD-Ult (auto)
- Extreme 1D to Global
- Type of values to UC_Overall

Then press Refresh to see the results.



The graphical scene shows that the maximum unity check occurs for one of the middle columns. To understand the reason behind it, 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.

As output you can choose for **Brief** or **Detailed**. For the detailed output you have the option to show both **tables and formulas**. Press **Report preview** to display the output.



This detailed report reveals that the column does not comply with the **Bending and axial compression check** according to EN 1993-1-1 article 6.3.3 and formula (6.61),(6.62). In this particular project; a different cross-section with larger inertia is required.

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

In the property panel under Actions you can choose for Autodesign in order to optimize the cross-sections.

ACTIONS >>>	
🤁 Refresh	F5
New combination from Combination	key
Autodesign	
Results table	
Report preview	

<u>Note</u>: steel optimization works with cross-sections, not particular members. Therefore once the cross-section is changed, all members with that profile are changes as well. Therefore in the property panel the **Filter** should be set to **Cross-section** when you use the Autodesign action.

In the dialogue press [Search for optimal] and the software will search within the profile library for a new cross-section and stops with the first one that satisfies the defined **maximal check** (1.00 in our example).

Autodesign Maximal che Maximum ur							>
Maximal che							
	еск						
Maximum ur		1,00					
	nity check:	1,00					
Edit cons	traints	Info					
Edi	t	Change					
Next d	own	Next up					
	Search for op	timal					
Direction	Up	o&down ∨	Ν				
rameter							
	:: HP(Imp)8X36	~					
	Param.	Value	Autodesign	List		Sort by	
1 I secti	on	HP(Imp)8X36	Yes	No	~ н		v

<u>Note</u>: it is also possible to search in a list. Define lists via **Main menu > Libraries > Structure and analysis > Cross section** list.

The software finds a cross-section HP10X42 with a check of 0.68.

After confirming with **[OK]**, all members with the same cross-section will be changed.

<u>Note</u>: 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 optimization and recalculation of the structure, the concerned cross-section could possibly reveal inappropriate. In that case, you must re-execute the optimization in order to find a solution in an iterative manner.

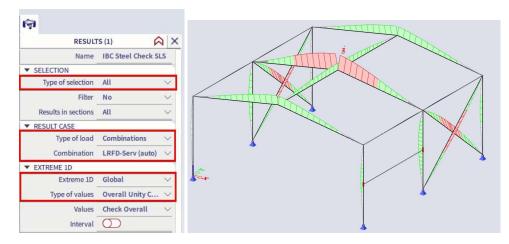
Whole structure can be optimized using the steps above again for another cross-section (CS2,...). However, the procedure was described and repeating it for all the members is not an intention of this tutorial.

7.2.2. Serviceability check

You can perform a steel SLS (Serviceability Limit State) check via **Main menu > Design > Steel members > Serviceability** check or via the process toolbar.

🞯 👁 🛱 🕰		Please
Steel members	•	Settings
Steel connections	•	βλ Slenderness
External tools	•	TT ULS check
Check manager		Fire check
Update check data		SLS check
Design forms	•	uG

The properties are set similarly as for the strength check.



With this check the relative deformations are compared with the deflection limits, as defined in the steel settings or via the system lengths and buckling settings. When the check is exceeded, you can check if the deflection limits are set properly or you need to change the cross-section of the elements. An serviceability autodesign command is not available in SCIA Engineer.

Chapter 8: Steel connections

As steel connections both frame strong-axis and frame weak-axis connections can be modelled, as well as pinned grid connections and hollow section joints.

In this tutorial, as an example a frame strong-axis connection will be inserted between a column and a roof girder.

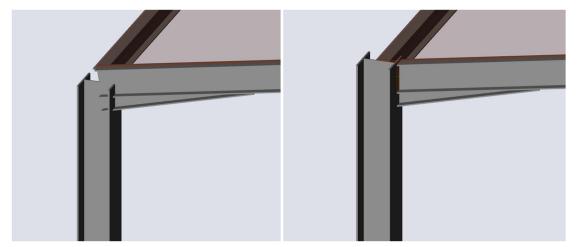
8.1. Structural model

Go to Main menu > File > Project settings and activate Structural model (and Steel Connections for adding the connection later on) in the Functionality tab.

GENERAL
Property modifiers Model modifiers Parametric input Climatic loads Mobile loads Dynamics Stability Nonlinearity Structural model IFC properties Prestressing Construction stages

In the analysis model, the members arrive in the same node. But in practice the girder is cut for instance at the column flange or vice versa. You can indicate this through the structural model.

Now go to Main menu > View > Visualization > Generate structural model.



The structural model will show you the real appearance of the structure, which is important when we want to create connections.

<u>Note</u>: 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.

If needed you can change the **Priority value** property of the members (via the property panel), what will influence which member(s) will be trimmed and which member(s) will be extended.

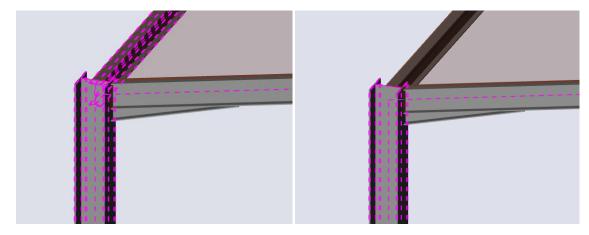
Steel connections are always based on the structural model. If the column continues, a connection with end-plate on the roof beam is obtained. If the roof beam continues, a connection with end-plate on the column is obtained.

8.2. Steel connection

In the **input panel** go to **workstation Steel** and choose **Frame strong-axis**. You can find the command as well in the **workstation Steel** of the **process toolbar**.



Select the node where you want to create the connection. Now indicate the members between which the connection should be established. The program automatically selects (and highlights) all members arriving in the node. As the connection should be inserted between the column and the roof beam, deselect the girder while pressing the **<Ctrl>** or **<Shift>** button.



Confirm with **<Esc>**.

The connection is inserted (by the mean of a flag which contains **Conn** and the **[node name]**) and the properties, including components, are displayed in the **property panel**.

In the **property panel** you can set already the **Type of loads** to **Combinations** and **Combinations** to **ULS-Set B (auto)**. Now, the components of the connection can be activated. Activate **End-plate** and click on the **Manage** icon. The endplate is added and immediately displayed in the graphical scene. In the dialogue you can change parameters of the endplate and confirm with **[OK]**.

•							
	INECTION (1)						
# 🕩							
Name	Conn		_				
Node	N2.		End-plate				
Type of loads	Combinations	\sim			Name EP Material A36		
Combinations	LRFD-Ult (auto)	~		 	Thickness[inch] 0,472		
Frame type		~			Input Top/Bottom/Left/Right		
					Top extension [inch] -0,197 Bottom extension [inch] 0,787		
Connection geometry	Knee	- 0			Left extension [inch] 2,080		
▼ SIDE ->[B2]				· · · · · · · · · · · · · · · · · · ·	Right extension [inch] 2,080		· · · · · · · · · · · · · · · · · · ·
Connection type	Frame bolted	\sim			Total width [inch] 8,160 Total height [inch] 14,365		
End-plate		Ξ		iota negric (non)	iota negacincij - 1990	iotarreign [ncii]	iotar negar pirchi
Backing plate	0						
Top haunch	\bigcirc						
Bolts	σ						
Top stiffener							
Bottom stiffener							
Diagonal stiffener	1						OK

Activate **Bolts** and click on the **Manage** icon. The bolts are automatically displayed in the graphical scene. In the dialogue you can select bolt assembly **M20 - 8.8**. A window informs you that the bolt assembly was changed. When confirming with [Yes] the bolts positions, intermediate distances, edge distances, ... are automatically adapted to the new bolt assembly. Furthermore in the dialogue you can activate or deactivate bolt-rows and set the position with the property **Location**.

Type of loads Combinations Length [inch] Combinations ULS-Set B (auto) Boit pattern Frame type braced Internal boits distance [inch] Connection geometry Single-sided 2.Row SIDE ->[B2] 4.Row 3.Row Connection type Frame bolted 5.Row End-plate ILocation [inch] 1.Location [inch]
Node N2 Type of loads Combinations ~ Combinations ULS-Set B (auto) ~ Frame type braced ~ Connection geometry Single-sided SIDE ->[B2] Frame bolted ~ Connection type Frame bolted ~ End-plate Illocation [inch]
Node N2 Type of loads Combinations ✓ Combinations ULS-Set B (auto) ✓ Frame type braced ✓ Connection geometry Single-sided ✓ SIDE →[B2] Connection type Frame bolted ✓ End-plate ♥ = =================================
Node N2 Type of loads Combinations ∨ Combinations ULS-Set B (auto) ∨ Frame type braced ∨ Connection geometry Single-sided SIDE ->[B2]
Type of loads Combinations ✓ Combinations ULS-Set B (auto) ✓ Frame type braced ✓ Connection geometry Single-sided ✓ SIDE ->[B2] Connection type Frame bolted ✓ End-plate ♥ I I Contended
Combinations ULS-Set B (auto) V Frame type braced V Connection geometry Single-sided 2.Row SIDE ->[B2] Connection type Frame bolted V End-plate V End-plate 2 Internal bolts distance [inch] SRow 1.Location [inch] 2.Location [inch]
Frame type braced ✓ Connection geometry Single-sided ✓ SIDE ->[B2] Connection type Frame bolted ✓ End-plate ♥ Internal bolts distance [Inch] 4 S.Row 1.Row 3.Row 3.Row 1.Location [Inch] 4 Connection type Frame bolted ✓ End-plate ♥ Internal bolts distance [Inch] 4 Connection type Frame bolted ✓ Connection type Frame bolted ✓ Connection type Frame bolted ✓ Connection [Inch] 4 Connection [Inch]
Connection geometry Single-sided 1.Row ▼ SIDE ->[B2] 3.Row Connection type Frame bolted ∨ 5.Row End-plate ♥ ➡ 1.Location (inch) 2.Location (inch) 2. Connection type frame bolted ∨ 5.Row 1.Location (inch) 2.Location (inch)
▼ SIDE ->[B2] Connection type Frame bolted ∨ End-plate ♥ Il.location (inch) 12 21 creation (inch) 12 21 creation (inch) 12 21 creation (inch) 12
Connection type Frame bolted V End-plate V 1.Location [Inch] 12, 2.Location [Inch] 22, 2.Location [Inch] 24, 2.Location [Inch] 24, 2
End-plate
2 Location [inch] 9.72
Backing plate 2.Location (inch) 9,744
Top haunch
Bolts 💽 📑 All distances are within the limits.
Top stiffener
Bottom stiffener
Diagonal stiffener

To complete the connection, activate the components **Top stiffener** and **Bottom stiffener**. Also for these components, you could change the properties via the **Manage** icons.

The connection is now displayed in the same colour as the model. To change this view, use the **View settings** dialogue and activate the options **Welds** and **Coloured** in the **Connections** tab.

/ Uncheck all	Lock position rels Model Cade/masses Steel Connectil
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
/ Uncheck group tructure Labe / Uncheck all ce	
tructure 🔠 Labe / Uncheck all ce	
/ Uncheck all	els 🛆 Model 🛃 Loads/masses 🐼 Steel 🖙 Connecti
ce	l
connections	
lay	
lering	rendered 🔹
is	
connections label	
lay label	
es	
symbols	
nsions	
e I sy	mbols

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

Select the connection and click on the Action Open Preview to display the results in Report Preview.

Name Node	Conn			
Connection type	N2 Frame bolted			
Connection geometry Calculation type Lc/Combi	Knee Internal forces LRFD-Ult (auto)			
Internal forces	(summary): Side [B2]			
N -2.34 kip Vz 6.36 kip My -38.04 kipft				
Design resistances Mj,y,Rd 45.69 kip Nj,Rd 0.00 kip				
Vz,Rd 25.73 kip				R
Limiting parts		t.fc.Rd)		
In tension Colu	imn Flange in bending (F imn web in compression	di di di		
Limiting parts		t.fc.Rd)		

You can set the **Output** as **Brief**, **Summary** or **Detailed**. The summary output shows the internal forces, design resistances, results of unity checks and moment-rotation diagrams.

<u>Notes</u>:

- 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.

Chapter 9: Report

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

9.1. Engineering report

You can select the Report command via Main menu > Tools > Report or with the icon in the workstation Reports.



Because no report was created before, Report_1 directly appears as a new application. This application is in a certain way independent of the SCIA Engineer application. That is visible as well in the Windows main bar by a separate icon.



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

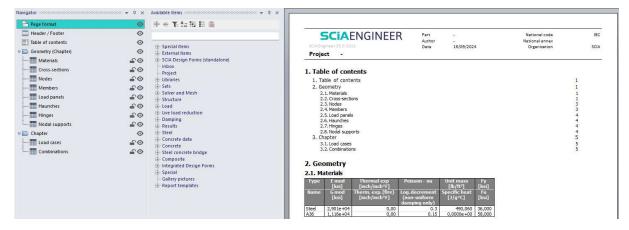
Repor		Insert	Edit	Delete	Move	Move down	Indent	Outdent
				Docume	nt item			
		lable iten		ER			▼ Ŧ ×	
	·····································	Special i External SCIA De Inbox Project Libraries Solver a Structur Load Live load Dampin Results Steel Concret Steel Concret Steel Concret Steel Concret Steel Gallery Report	items sign Forn ; nd Mesh e d reductio g e data e ncrete br ite ed Desig pictures	idge n Forms	ione)			

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

- Open the **Special items** group to add **Page Format**, **Header / Footer** and **Table of contents**. Double-click on the item or hit the **+** button to add the item to the **Navigator**.
- Create a chapters Geometry, Loads and Results with the item **Chapter** from the **Special items** group.
- Open the Libraries group and select Materials and Cross-sections.
- Open the Structure group to add Nodes, Members, Load panels, Haunches, Hinges and Nodal supports.

Drag the items with the mouse to change their order or use the icons in the **Document item** part of the ribbon.

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



9.1.1. Adding images to the report

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

Prepare any scene in the 3D modelling window, for example the analytical model with loads. You can use the icons of the **view bar** to hide surfaces, render and show loads:

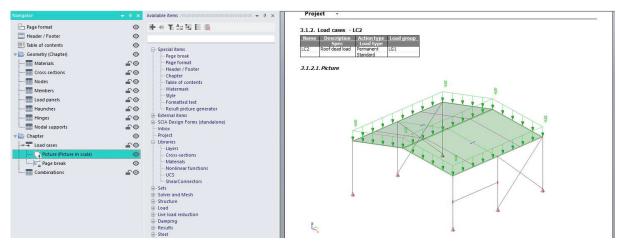
You can immediately add a live image via the Live image to report command via Main menu > Tools > Send > Live image to report or with the icon in the workstation Reports.



A properties dialogue for the image is opened. Here you can arrange the picture caption, scale, size, ... Use the button [**Two at page**] at the top ribbon, tick the option **Automatic scale to fit size** on and click on button [**Insert & Close**] from the tab **into selected report**.

Pictu	ure in scale - Insert objects to Engineering report Inbox	
Insert Insert Close Ore at Nov at Fit to page 1 2 Close Dieletted report into inbox		
	Caption Picture	
	Picture size definition Two per page	
	Automatic scale to fit size 🗹	
	Scale 1: 185,342400953381	
	Stretch mode Dark lines	,
	Rendering Standard	
	Antialiasing quality None	
	Rotation None	,
	Result information Inside picture	•
	Export to PDF as 3D	
	Position One below another	•
	Image raster	
	Scale for model data 1	
Load units in regen. (related to objects crea	Load activity in regen.	
	Draw inactive members as is in the window	
	Text scale factor 1	
	Charset of texts Western European, UK, USA (Windows-1252)	
	Line pattern length 3	
	Display GCS icon To picture corner	
Performance		
	Set as non-editable	>>>
	Settings	>>>

In the report you can indent this item under the item Load cases. When regenerating this item, an image will automatically be generated for each load case (in case that **Selection type** is set to **All** in the properties of the load cases item), based on the image that you have added before.



9.1.2. Displaying results

Go to Available items, open the Results group and click on 1D Internal forces.

In the navigator click on 1D Internal forces. The red exclamation mark both in the navigator and the preview indicates that the presented values are not up-to-date.

Nodal supports	£0	Results	
v 🔛 Loads (Chapter)	0	- Displacement of nodes - 3D displacement	4. Results
Load cases	• •	3D stress	
🏹 Picture (Picture in sc	•	Reactions Resultant of reactions	4.1. 3D displacement
Combinations	<u>•</u> •	- 1D internal forces	Regeneration is needed
v 🔤 Results (Chapter)	0	- 1D deformations	
3D displacement	£0 (3)	- 1D stresses - 2D internal forces	

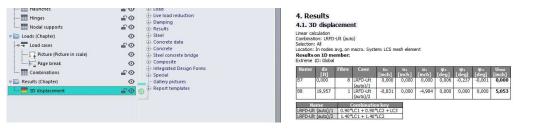
In the **properties window** the setting of this table is displayed. Parameters for displaying the results in the Engineering Report are configured in the same way as the parameters for viewing the results in the SCIA Engineer application:

- Selection type is set to All
- Type of load is set to Combinations and the Combination to LRFD-Ult (auto)
- Extreme 1D field is set to Global

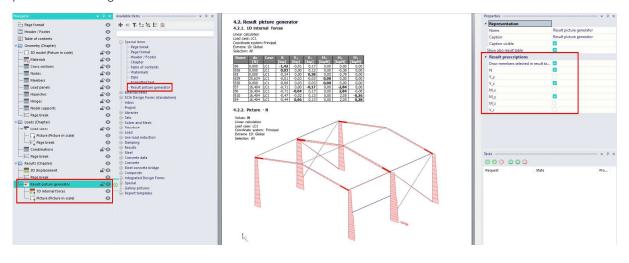
Choose **Regenerate selected** (or **Regenerate outdated** in case that you want to regenerate all outdated item in the whole report) in the top ribbon to display the table in accordance with the predefined options.



The red exclamation mark disappears.



Same as for generating multiple pictures per load case, you can automatically generate multiple pictures for a specific result. For this you need the Result picture generator (under Special items) and indent a result item (e.g. 1D internal forces) and a picture below it. In the properties of the result picture generator you can indicate for which values pictures should be generated.



A list of items for this project could look like this:

Navigator	< ×
Page format	0
📃 Header / Footer	0
Table of contents	0
🕫 🔤 Geometry (Chapter)	0
🎵 3D model (Picture in scale)	_ O
····· 🎹 Materials	₽⊙
····· Cross-sections	₽⊙
Nodes	₽⊙
Members	₽ ⊙
Ioad panels	_ O
Haunches	₽⊙
Hinges	₽ ⊙
Modal supports	^ 0
Page break	0
🗢 🗁 Loads (Chapter)	0
👓 🕶 Load cases	₽ ⊙
🔽 Picture (Picture in scale)	0
Page break	0
····· 🎹 Combinations	₽⊙
Page break	0
v 🔤 Results (Chapter)	0
🛗 3D displacement	₽⊙
	0
🔫 🐺 Result picture generator	₽ ⊙
ID internal forces	0
Picture (Picture in scale)	0
····· 🧱 IBC Steel Check ULS	• •
····· 🧱 IBC Steel Check SLS	₽ ⊙
Structural joint check	₽⊙

9.1.3. **Printing or exporting the report**

Once the report is completed you can print it or export into various formats (e.g. PDF, RTF, Html, report and Excel) by clicking the top left button in the interface:

	Report_1 [TUT [Eng] SDA25.0 - Tutorial Steef Frame ASC.esa) - Engineering report –
\odot	
Print	Print
民 Export	Print Cosles 1 🗄
😨 Export after external regeneration	Printer SCIAENGINEER M SMARTAN K
📓 Save as template	Printer Andrews at Anoral Anor
📓 Save as preview template	Table of contents
Check report data integrity	2. Genety 1 2.2.70 model 1
Engineering report info	Settings
🗘 Options	Print All Pages 3. Monkers 3. Monkers 3. Monkers 4. Mon
Ent .	From <u>1 Tot 14</u>
	Interface 1.los 1 From 1 To: 14 1 1 Ibit come: 1 1 1 1 Ibit come: 1 1 1 1 1 Ibit come: 1 1 1 1 1 1 Ibit come: 1 1 1 1 1 1
	Partial Orientation 4 Standards 6 Partial Orientation 4 Standards 6 1 Standards 6 6
	44 44 4100/00-013 12 410/00-013 12 21 cm x33 / cm 410/00-013 12 410/00-013 12 410/00-013 410/00-013 410/00-013 410/00-013 410/00-010
	2. Geometry Rendered pictures with high quality v
	4 1
	C Paret >

There is also the option Save as template, so you can reuse the items from this report for other projects.