



TUTORIAL STEEL FRAME 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 2018 SCIA nv. All rights reserved.

ole of contents	
General Information	
Welcome	
SCIA Engineer Support	
Websites	
Introduction	
Getting started	
Starting a project Project management	
Save, Save as, Close and Open	
Saving a project	
Closing a project	
Opening a project	
Start project manger	
Geometry input	
Input of the geometry	
Profiles	
Geometry	
Haunches	
Hinaes	
Supports	2
Check Structure data	2
Checking the structure	2
Connecting entities	2
Graphic representation of the structure	3
Loads and combinations	3
Load Cases and Load Groups	
Defining a Permanent Load Case	
Defining a Variable Load Case	
Loads	
Combinations	
Calculation	4
Linear Calculation	
Results	4
Viewing results	
Code check	5
Buckling parameters	5
Displaying the system lengths	
Setting the Buckling Parameters	
Steel code check	
Steel Code Check Ultimete limit state	
Optimisation of the Steel Section	
Steel connections	6
Activating the Steel Connection Input	6
Displaying the Structural model	
Inputting a Steel Connection	6
Checking the connection	7
Document	7
Engineering report	7
Epilogue	7

Welcome

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

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

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

SCIA Engineer is available in three different editions:

License version

The license version of SCIA Engineer is secured with a 'dongle', a hardlock, which you apply to the USB gate of your computer or a softwarematic license in your network.

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

In the general product overview of SCIA Engineer you will find an overview of the different modules or module editions that are available.

Viewer mode

If the program doesn't find a licence it can be used as a viewer only. That means that any project can be opened, properties of entities can be checked, if the calculation has been done also results can be seen and report can be printed.

However, no change of the model is possible, no calculation can be run, no new output can be created.

Student version

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

The output contains a watermark "Student version". Projects that are stored in the student version cannot be opened in the license version.

SCIA Engineer Support

You can contact the SCIA Engineer support service

By e-mail

Send an e-mail to support@scia.net with a description of the problem and the concerning *.esa file, and mention the number of the version you are currently working with.

By telephone

For various phone numbers to different offices visit our page https://www.scia.net/en/contact/offices Via the SCIA Customer Portal website

http://www.scia.net/en/portal

Websites

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

Introduction

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

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

First, we will explain how to create a new project and how to setup your structure. After the geometry and load input, the structure will be calculated and the results can be viewed.

Next, we will discuss the input of the buckling parameters and we will perform the steel check, the profile optimization and calculate steel connection.

The Tutorial ends with a brief introduction to the calculation report.

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



Starting a project

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

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

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

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

Starting a new project

Recent Projects	New Project from Template		
,,	Enter search term: Search for project		
) Templates	Name	Date Size	•
Tutorials	User Templates	1/1/1601 1:00:	
	A System Templates	1/1/1601 1:00:	
Blank Project	A Standard templates	5/7/2018 12:34	©Estub Sistemas Construtivos
Browse	Eurocode	5/7/2018 12:34	
	IBC	5/7/2018 12:34	
Learning Center	 Parametric projects 	5/7/2018 12:34	A CONTRACTOR OF THE OWNER OF
	Concrete Structures	5/7/2018 12:34	
	Steel Structures	5/7/2018 12:34	
	User templates	5/7/2018 12:34	
	A Drawing Templates	1/1/1601 1:00:	
	Addons	5/7/2018 2:28:	
	TeklaTemplates	5/17/2018 1:39	
Resource Centre	PredefinedShapes	5/17/2018 1:4(
SCIA Website	Concrete	5/17/2018 1:40	This illustrative image will be replaced by ar
Web Help	Shells	5/17/2018 1:4(image of your project once you save your d
•	Steel	5/17/2018 1:4(In the latest version of SCIA Engineer.
Protection Settings	Volumes	5/17/2018 1:4(

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

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

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

	Data			Material		
	Name:	-		Concrete		
FRAD				Steel	\checkmark	
AAA	Part:	-		Material	S 235 🗸	
FX				Masonry		
14	Description:	-		Aluminium		
112				Timber		
	Author:	SCIA		Steel fibre conc	. 🗆	
				Other		
	Date:	17.05.2018				
				Code		
	Structure:		environment	National Code:		_
	Frame XYZ		🚳 v17	↓ EC - EN	-	
	Model:			National annex:		
Part P	🕅 One	-		Standard E	N -	

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

Material is the only required setting to proceed

Choose S235 from the combo-box.

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

Note:

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

Save, Save as, Close and Open

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

Saving a project

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

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

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

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

Closing a project

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



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

Opening a project

Click on *to open an existing project.*

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

Start project manger

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

Geometry input

Input of the geometry

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

Profiles

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

Adding a profile

1. Click on the **Cross-Sections** ^{II} icon in the toolbar.

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

	New cros	s-section		×
Available groups	Available items of this group			Items in project
 Profile Library Geometric shapes Numerical General Pairs Closed Haunch Welded Sheet welded Build-in beams Thin-walled geometric Fabricated Virtual joists 	H(JIS) HD HD(ARC) HE HEA HEB HEC HEM HG(GOST) HHD HL HL(SZS) HM(CH) HP(ARC) HP(ARCUS) HP(GRD) HP(GRD) HP(MRCUS) HP(GRD) HP(MRCUS)	 ▲ 100 100A 120A 120A 140A 1600 160A 180A 200A 220A 220A 240A 260A 280A ✓ 280A 	~	
	Name HEA	A200		
	Profil Arbed / Structural shapes / E European wide flange beam	dition Octobre 1995		
	Profile Library filter	All cross-sections	~	Add Close

- 2. Click Profile library in the group Available groups.
- In the Available items of this group, you can choose I profile from the list.
- . Choose HEA 200
- 4. Click [Add] or to add the profile to the project.
- 5. The Cross-section window appears.

Cross-section		×
	Name CS1	^
	Type HEA200	
7	Shape type Thin-walled	
	Source and Type desc	
	Source description Profil Arbed / Stru	ict
	Type description European wide fla	ing
	Parameters	
	Material S 235	·
	I sections HEA200	
	Profile Library filter All cross-sections	-
V	General	
	Draw colour Normal colour	*
	Colour	
	Fabrication rolled	*
N	Buckling curves	
	Edit buckling curves	
	Flexural buckling y-y b	
	Flexural buckling z-z C	
	Lateral torsional buckling Default	~
4 Image: Arrow of the second secon	Export Update Docu	ument
Cross-section layout and dimensions	OK Ca	ncel

- 6. Click **[OK]** to confirm, the profile is added to the **Items in project** frame. Add **IPE 180** and **IPE 160** in a similar way.
- 7. In the **Available items for this group**, you can choose an angle section **IFLeq 70x70x7** from the list.
- 8. Click **[Add]** or to add the profile to the project. Click **[OK]** to confirm, the profile is added to the **Items in project** frame.
- 9. Click [Close] in the New cross-section window, the Cross-Sections manager appears.

a 😳 🖌 🕹 🙀) i i i i i i i i i i i i i i i i i i i	NI 🔽 🕅	
CS1 - HEA200	Name	CS4	^
CS2 - IPE180	Туре	HFLeq70x70x7	
CS3 - IPE160	Shape type	Thin-walled	
CS4 - HFLeq70x70x7	Source and Type descri.		
	Source description	Staalprofielen / deel 5 (Over)	
	Type description	Equal leg angle	
	Parameters		
	Material	S 235	
	L sections	HFLeq70x70x7	
	Profile Library filter	All cross-sections	,
	ZLCS T	YLCS	

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

Geometry

Structure menu

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



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

To model the structure you must enter the first frame. Then, this frame will be copied and the wind bracings and the horizontal beams will be added.

You can use columns and bars to enter the frame. SCIA Engineer however offers multiple Catalogue blocks, allowing for a smooth and simple input of the structure.

Entering a frame using a Catalogue Block

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



- 2. In the Available Groups group choose the first option Frame 2D
- 3. In the Available items of this group you can choose the first shape

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

		Geometry block	×
Name	BL		ol
L [m]	12,000		8
H1 [m]	5,000		- Î
H2 [m]	1,000		H
Column	CS1 - HEA20(👻		
Beam	CS2 - IPE180 🔻		
H2		L 12,000	8
Ŧ		· · · · · · · · · · · · · · · · · · ·	<u>H1 5,0</u>
Ļ	,	ОК	Cancel

- 5. Now, enter the frame dimensions: L = 12 m, H1 = 5 m and H2 = 1 m
- 6. In the combo-box with cross-section selector choose **HEA 200** for the **Column** and **IPE 180** for the **Beam**.
- 7. Confirm your input with **[OK]**. The **Catalogue block** manager appears.

		Catalogue blo	ock	
🎜 🤮 🗶 👪 📸	8 !	o e 🕘 🖻 🔒		
BL		Name	BL	
	E	Geometry		
		L [m]	12,000	
		H1 [m]	5,000	
		H2 [m]	1,000	
		Column	CS1 - HEA200	
		Beam	CS2 - IPE180	
			H2 1,000	
		I	L 12,000 H	
New Insert	Edit	Delete	0	К

- 8. Click **[OK]** to return to the modelling view. The frame is now graphically represented by thin blue lines attached to mouse cursor. You are now asked to select insert point.
- The frame is positioned with the left column in the origin of the coordinate system. Type the coordinates 0;0 in the Command line and press <Enter> to confirm your input.

Command line
₩
Block - Insert block - End point >0:0

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

Notes:

The properties of selected elements are shown and can be modified in the **Properties** window on the right hand side of the user interface.

If no cross-section has been defined in the project, the **New cross-section** window will automatically appear as soon as you try to enter a structural element (column, beam...).

With **Zoom All** button in the toolbar, or double-click with the mouse wheel, you can visualize the entire structure.

The use of , or ; to separate coordinates depends on the regional settings of Windows. You can also use spacebar to define coordinates instead of the two.

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

you can use the Multiple copy function

Create multiple copies

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

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



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

		Mul	ticopy	
Numbe	r of copies t the very last c	2	Connect selected nodes with new beams Copy additional data	
Distance Define o	e vector distance by curs	or 🗌	How to define the distance ? between two copies	
x y	0,000	m	O from original to the last o How to define the rotation ?	ору
z	0,000	m	between two copies from original to the last of	ору
Rotation	0,00	deg	Rotation around O current UCS	
ry	0,00	deg	O distance vector	
rz	0,00	deg	OK Cance	4

- 3. In the Number of copies field enter 2.
- 4. To manually set the distance between the frames, deactivate the **Define distance by cursor** option. Now, you can enter the distance **6 m** in direction Y.
- 5. Click [OK] to confirm your input. New frames are modelled.
- 6. Press **<ESC>** to cancel the selection.



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

Cursor snap settings

- Double-click on the Cursor snap settings icon in the Command line or click on the button Snap mode at the lower right corner of the application frame. The Cursor snap settings window is opened.
- 2. Activate the options a) and b) to highlight midpoints and end points of bars in this project.



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

Entering a beam

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

Name	B13
Туре	beam (80)
Analysis model	Standard
CrossSection	CS3 - IPE160 👻
Alpha [deg]	0,00
Member system-line at	Centre
ey [mm]	0
ez [mm]	0
LCS	standard
LCS Rotation [deg]	0,00
FEM type	standard
Buckling and relative lengths	Default
Layer	Layer1
Geometry	
Direct	axis Y
Length [m]	6,000
Insertion point	begin

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

- 4. The beam length is **6 m**.
- 5. The insertion point is (as default) set to **begin** so that the left point determines the position of the beam.
- 6. Confirm your input with [OK].
- 7. Now, you can enter the beam by clicking with your mouse on the top node of the left-hand side column of the first frame and similar node on the middle frame:



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



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

Note:

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

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

Copying entities

- First select the entity to be copied. As this is a horizontal beam, you can select one of the entered beams with the left mouse button. Violet colour indicates that the bar has been selected. Properties of the bar are shown in the **Properties** window.
- 2. Click with the right mouse button anywhere in the user environment. Context menu lists the available possibilities for the selected entity:



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

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

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



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



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

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

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

Entering bracings

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



- 2. In the **Type** field, choose **Wall Bracing (0)**. This specification is considered for the STRUCTURAL model only and does not influence the calculation model or the results.
- 3. In the **CrossSection** field, choose the fourth section, **CS4 HFLeq 70x70x7**. Note that no length or direction is requested this time, as you will define the geometry afterwards.
- 4. Confirm your input with [OK].
- 5. Now, the bracings can be entered between the second and third frame. Click on the start and end nodes of the columns to draw diagonal:



- 6. Repeat selecting start nodes and end nodes until all bracings are modelled.
- 7. Press **<ESC>** to finish the input.
- 8. Press <ESC> once more to finish the selection

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



Haunches

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

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

Entering Haunches

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



 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 crosssection window automatically appears.

_

3. Select in the Available groups the Haunch group.

			New cro	oss-sect	ion			
Available groups Geometric shapes Numerical Pairs Closed Haunch Welded Sheet welded Build-in beams Thin-walled geometric	Available it	ems of thi	s group	I	H	I	->	Items in project CS1 - HEA200 CS2 - IPE180 CS3 - IPE160 CS4 - HFLeq70x70x7
I+Ivar		Profil	e Librany filt	All or				Add

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

Cross	-section	
	Name	CS5 ^
z	Туре	l + I var
	Detailed	IPE180; 150
	Shape type	Thick-walled
	Parameters	
> TPE 1 00	Material	S 235 •
AITELOU	va [mm]	150
	I sections	IPE180
	Profile Library filter	All cross-sections
	🖻 General	
у	Draw colour	Normal colour
	Colour	
	Fabrication	welded 💌
	Buckling curves	
120	Edit buckling curves	E
8	Flexural buckling y-y	b
N	Flexural buckling z-z	c
	Lateral torsional buckling	Default
	Fibres and Parts	
4 Z Picture Initial shape	b Epport Upd	ate Document
Cross-section layout and dimensions	0	K Cancel

- 6. In the **I sections** field, change the section in an **IPE 180** by clicking the <u>understand button</u> behind the section type.
- 7. When the correct I-section is set, the variable height va (mm) is set to 150mm
- 8. Confirm your input with **[OK]** and use button **[Close]** to close the **New cross-section** dialogue.
- 9. The Cross-Section manager appears; click [OK] to close this window as well.
- 10. Now, the Haunch on beam window is opened.

•	Haunch on beam		×
	Name	H1	
	Position	Begin	-
	Cross-section	CS5 - I + I var (IPE180; 150)	·
	Use from Css	no 🖂	
	va [mm]	150,0	
	Geometry		
. 🗡 В	Coord. definition	Abso	-
i-ev	Length x [m]	1,000	
		ОК	Cancel

- 11. In the **Position** field, choose **Begin** to position the haunch at the start node of the member.
- 12. In the **Coord. definition** field, choose the option **Abso** to indicate that the length, over which the variable height must vary, can be entered in absolute units, i.e. in meter.

- 13. When the Coordinate Definition is adapted, the length of the haunch can be entered in the **Length x [m]** field. For this project, enter length **1 m**.
- 14. Confirm your input with [OK]
- 15. Now, the program asks to indicate the members on which a haunch must be entered. Select the 6 roof beams with the left mouse button:



- 16. Press **<ESC>** to finish the input.
- 17. Press **<ESC>** once more to cancel the selection.

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

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



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

Note:

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

Hinges

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

In this project, the diagonals should be connected with the other members in a hinged way. Therefore, you must enter hinges manually.

Entering hinges

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



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

	Hinge on be	am	×
100 March 100 Ma	Name	H1	
1 ^{φz}	Position	Both	*
	ux	Rigid	*
(i) ^{LUZ}	uy	Rigid	*
	uz	Rigid	-
ΦΧ ΦΥ	fix	Rigid	-
*	fiy	Free	-
	fiz	Rigid	*
		C	K Cancel

- 3. To obtain a hinge, the rotation **fiy** is set to **Free**, the translations and the other rotations remain **Rigid**. In this way, the diagonals will be exclusively hinged in the planes of the sidewalls.
- 4. Confirm your input with [OK].
- 5. The hinges are added when you click the diagonals with the left mouse button.
- 6. Press **<ESC>** to finish the input.
- 7. Press **<ESC>** once more to finish the selection.



Note:

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

Increase the scale for input data in toolbars

Supports

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

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

Selecting elements by property

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



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

Pr	operties			Ψ×
N	ode (1)		🔁 Va	V/ 🖉
				* 3
1	Vame	N5		
	GCS coordinate			
	Coord X [m]	12,000		
	Coord Y [m]	0,000		
	Coord Z [m]	0,000		
	UCS coordinate			
	Coord ux [m]	12,000		
	Coord uy [m]	0,000		
	Coord uz [m]	0,000		
	Members			
	Member	B4		

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

Click with the left mouse button on the **Coord Z (m)** property to select appropriate row. The table cell is highlighted by blue colour.

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



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

Entering supports

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



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

•	Support in node		×
	Name	Sn1	
	Туре	Standard	
	Angle [deg]		
. P7	Constraint	Hinged	
Anz	x	Rigid	-
7	Y	Rigid	-
	z	Rigid	-
XXX	Rx	Free	-
Rx	Ry	Free	
(i) *	Rz	Free	-
\smile	Default size [m]	0,200	
17	Geometry		
	System	GCS	*
x Y			
		0	K Cancel

- 3. Confirm your input with [OK]. The supports are automatically attributed to the selected nodes.
- 4. Press **<ESC>** to finish the selection.



Notes:

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

The **Command line** includes a number of predefined supports. For this project, you could have used the **Hinged support** $\leq 1 + 3 = 3 = 3$ icon.

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

Check Structure data

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

Checking the structure

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

- P	or click on the	: <i>5</i> I) fi @ #	86 🛛 🕄	icon in the
toolbar.				

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

Check of st	ructure data	×
Check of nodes		
Search nodes		
Search duplicate nodes	Ignore parame	eters
Check of members		
Check members Search null members	Null members:	0 embers
Search duplicate members	Duplicate	0 ate members
	Invalid parts:	0 parts
Check of data references		
Check data references	O Memory efficie	nt method
	Fast method	
Check of additional data		
Check additional data position	Invalid position	0
	Correct position	on
Check free load distribution points	Invalid loads	0
Check of steel connections		
Check steel connections	Invalid ✓ Delete invalid	0 connections
Check load panels Check cross- Check additional data Check duplicity (links of names	Check Cancel

- 3. Click [Check] to perform the checks.
- 4. The Data Check Report window appears, indicating that no problems were found.

Basic Concept Training -

	Check of structure data
Check of nodes Search nodes	
Search duplicate nodes	Ignore parameters
Check of members Check members Search null members	Null members: 0
Search duplicate members	Duplicate 0 Duplicate duplicate members Data check report X lid parts
Check of data references Check data references	Data check finished.
Check of additional data Check additional data position	OK n 0
Check free load distribution points	Invalid loads
Check of steel connections Check steel connections	Invalid 0
Check load panels Check additional data	Check cross-links heck duplicity of names Continue Cancel

- 5. Close the check by clicking [OK].
- 6. In case of any problem SCIA Engineer can automatically correct the structure data (delete duplicated entities, correct wrong reference, etc.)

Connecting entities

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

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

Activating node labels

Node labels are activated by means of the *O* **O L L E E E E E i** con at bottom of the modelling window.

Activating member labels



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



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

Connecting entities

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

4. Click <Yes>.

- Double-click on the Model data > Connect members/nodes option in the Structure service
 Connect members/nodes or click the icon in the toolbar.
- 3. A dialogue asks if all nodes must be connected to bars:



5. The Setup for connection of structural

S S	etup for connection of structural entiti	ies
	 Align structural entities to planes Align Geometrical tolerance Min. distance of two nodes, node to cur Max. distance of node to 2D member pl Connect (generate linked nodes, i Connect Connect 1D members as ribs Connect 1D members with rigid arms Max. length of rigid arm [m] Create new linked node for master node Check structure data Check (merge duplicate nodes, erase inv 	0,001 0,000
		OK Cancel

- 6. Confirm the settings by clicking **<OK>**.
- 7. A window appears to indicate the number of connected nodes:



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

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

118

ᇳ

	^	Properties	4 ×
NIZ BIO		Member (1)	🖃 🖬 🏹 🖉
BLA NIJ		a	8 ×
		Name	B21
		Туре	beam (80) 🔹
		Analysis model	Standard 🔹
N18 00 00 00 00 00 00 00 00 00 00 00 00 00		CrossSection	CS3 - IPE160 🔹
		Alpha [deg]	0,00
		Member system-lin	Centre 🔹
		ey [mm]	0
		ez [mm]	0
		LCS	standard 🔹
		LCS Rotation [deg]	0,00
		FEM type	standard 🔹
X 116		Buckling and relativ	Default 🔹
		Layer	Layer1 🔹
		Geometry	
		Length [m]	6,000
e a tradição de la companya de 🕶 🖉 de la companya de la comp		Shape	Line
		Beg. node	N18
		End node	N19
		Nodes	
		N18	to B1
	~	N19	to B5

Note:

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

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

Graphic representation of the structure

Edit view

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

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

Editing the view point on the model

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

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

OR

Set the view point by combining the buttons and mouse:

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

Remark:

If the structure is being turned while a node is selected, the structure will turn around the selected node.

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

Setting a view direction with regard to the global coordinate system

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

Remark:

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

The magnifier

- Use ¹/₁ to enlarge.
- Use to decrease.
- Use 🔼 to zoom in on a window.

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

Editing view parameters through the menu View parameters

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

22	Zoom all
R	Zoom by cut out
8	Set view parameters for all
R	Cursor snap setting
D.	Print/ Preview table
	Table to Engineering report
Ŕ	Print picture
暂	Picture to gallery
F	Save picture to file
6	Copy picture to clipboard
[2]	Screenshot into Engineering report
	Live picture into Engineering report
0,	Wired model in view manipulations
8	Advanced graphic setup
[]?	Coordinates info
*	Picture wizard

Remark:

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

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

View parameters – Structure

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

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

View	parameters –	Labels

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

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

	View parameter	rs setting
Check / U	Incheck group	Lock position
1/2	😐 🖾 🔛 🎯 🌌	
Check / l	Uncheck all	
Service		
Display	on opening the service	v
Structu	re	1
Style +	colour	normal 🗾
Membe	r system line style	rystem line -
Model 1	type	analysis model
Display	both models	
Membe	r surface	
Render	ing	wired 👻
Draw cr	ross-section	
Cross-s	ection style	section 💌
Effectiv	e width of plate ribs	
Draw er	ing	transparant -
- Structu	re nodes	transparent _
Display		v
Mark st	yle	Dot 🔹
Membe	er parameters	
System	lengths	
Membe	r nonlinearities	
FEM typ	De	
	voc	
Nodes	AC5	
Membe	rs 1D	
Design	groups	
	View parameters s	setting - Labels
	view parameters :	Labels
	< / Uncheck group	Lock position
	k / Uncheck group	Lock position
	Incheck group Image: State of the state of	Lock position
 ↓ / ▲ ■ Chec ■ Ser Dis 	C Uncheck group Image: Second secon	Lock position
Check Check Ser Dis Bea	c / Uncheck group Image: Control of the service total opening the service millabels	
Check Check Check Ser Dis Bea Dis	c / Uncheck group Image: Control of the service ick / Uncheck all vice play on opening the service imabels play label	Lock position
Check Check Ser Dis Bea Dis Nar	c / Uncheck group (a) (b) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c) (Lock position
Chee Ser Dis Bea Dis Nar Cro	c / Uncheck group All A P P P P P P P P P P P P P P P P P	
Chee Ser Dis Bea Dis Nar Cro	c / Uncheck group Image: Control of the service	Lock position
Check Check Ser Check Ser Check Beau Check Chec	<pre><!-- Uncheck group</td--><td></td></pre>	
Check Check Ser Dis Bea Dis Nar Croo Cro Len Lay Typ	c/ Uncheck group Image: Control of the service Image: Control of the service <td>Lock position</td>	Lock position
Crook	c/ Uncheck group Image: Control of the service Image: Control of the service <td>Lock position</td>	Lock position
Crowners of the second	c / Uncheck group Image: Control of the service Image: Control of the service <td></td>	
Cree See Dis Dis Cree C	<pre><!-- Uncheck group</td--><td></td></pre>	
A Part Check Chec	C / Uncheck group Image: Control of the service Image: Control of the service <td>Lock position</td>	Lock position
Crowner of the second	C / Uncheck group Image: Contract of the service	Lock position
Cree Ser Dis Bea Dis Bea Dis No Cro C	c/ Uncheck group Image: Ima	Lock position
 Check Seer Dis Beaz <li< td=""><td>c/ Uncheck group Image: Control of the service play on opening the service play on opening the service mabels play label me sss-section name sss-section type gth end priority des labels play label me oordinate poordinate poordinate tem lengths play label</td><td>Lock position</td></li<>	c/ Uncheck group Image: Control of the service play on opening the service play on opening the service mabels play label me sss-section name sss-section type gth end priority des labels play label me oordinate poordinate poordinate tem lengths play label	Lock position
 Check Seer Dis Beaz Dis Beaz Dis Beaz Dis Narr Croor Croor Len Lay Typ No Dis Narr Arrow Second Second Construction Constructio Construction Constructio	C/ Uncheck group Image: Control of the service m labels play on opening the service m labels play label me e and priority des labels play label ordinate oordinate	Lock position
 Creating Series Series Beau Dis Beau Dis Beau Dis Dis Beau Creating Creat	C/ Uncheck group Image: Control of the service play on opening the service play abol makeds play label me ss-section name ss-section type reget and priority des labels play label me oordinate oordinate tem lengths play label me oordinate oordinate tem lengths play label me	Lock position
 Cree Ser Bese Dision 	C / Uncheck group Image: Control of the service play on opening the service m labels play label me ss-section name sss-section name solution me solution solution me etal ninearities nointame solution	Lock position
4 (fee Ser Dis Bee Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo Croo	c/ Uncheck group Image: Control of the service play on opening the service play abel me sss-section name sss-section name sss-section name order labels play label e and priority desi labels play label me oordinate oordinate oordinate oordinate oordinate oordinate oordinate oelel ninearities play label me eel ninearities play label	Lock position
 Crown of the second seco	C/ Uncheck group Image: Control of the service m labels play on opening the service m labels play label me e and priority des labels play label oprimate oordinate oordinate oordinate oordinate nemethils me	
 Crowner of the second se	C/ Uncheck group Image: Control of the service play on opening the service play on opening the service makeds play abel me ss-section name sss-section type igth e and priority des labels play label me oordinate oordinate oordinate me ninearties play label me dot labels play label me dot labels play label me dot labels play label me des me des me des me des	Lock position
 Cree Ser Bes Bes Dis Bes Dis Bis Dis Dis Na Cro No No Lat No Mo Ger 	C/ Uncheck group Image: Control of the service play on opening the service play abol mature sss-section name sold stabels play label me section me section minearities play label me section flocal axes des	
 A () Check Ser Beac Dis Dis Dis Dis Dis Dis Dis Nar Croor Croor Croor Croor Dis Nar Nar Accord Ser Ser Ser Nar Nar<td>C/ Uncheck group Image: Control of the service play on opening the service play abel play label me ss-section name ordinate oordinate oordinate oordinate oordinate sector play label me eel nlinearities play label me eels of local axes des merait structural shape play vertex label</td><td>Lock position</td>	C/ Uncheck group Image: Control of the service play on opening the service play abel play label me ss-section name ordinate oordinate oordinate oordinate oordinate sector play label me eel nlinearities play label me eels of local axes des merait structural shape play vertex label	Lock position
 Crowner of the second second	C/ Uncheck group Image: Control of the service s	
 Cree Ser Bea Bea Bea Bea Bea Bis Bea Bis Na Cro Cro	(-/ Uncheck group) (a) (a) (b) (c) (c) (c) (c) (c) (c) (c) (c) (c) (c	
 Crowner of the second se	C/ Uncheck group Image: Control of the service mail abels play on opening the service mail abels play abel me ss-section name sss-section type right er e and priority des labels play label me oordinate oordinate oordinate oordinate ne ninearities play label me des mbers 1D metars 1D metars 1D play vertex label	Lock position

View parameters – shortcuts

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

- Show/hide surfaces 🖉 to show the surfaces of the cross-sections.
- **Render geometry 1** to view the rendered members.
- Show/hide supports <a>Image: to show supports and hinges.
- Show/hide load 4 to show the load case.
- Show/hide other model data 🖾 to show other model data (like hinges, internal nodes, ...).
- Show/hide node labels to view the label of the nodes.
- Show/hide member labels 🕮 to view the label of members.
- Set load case for view 🕮 to edit the active load case.
• Fast adjustment of view parameters on the whole construction 📴 to quickly access to the options from the menu View parameters.



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

Loads and combinations

Load Cases and Load Groups

Each load is attributed to a **load case**. One load case can contain different load types. To each load case, properties are attributed which are determinant for the generation of combinations. The action type of a load case can be permanent or variable.

Each variable load case is associated with a **load group**. The group contains information about the category of the load (service load, wind, snow...) and its appearance (default, together, exclusive). In an exclusive group, the different loads attributed to the group cannot act together in a single combination. For default **combinations**, on the other hand, the combination generator allows the simultaneous action of the loads of a same group.

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

Two load cases are entered in this project:

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

Defining a Permanent Load Case

- 1. Double-click on Load in the Main tree.
- 2. Before you can define loads, you must enter load cases first. Since this project does not contain any load cases yet, the **Load Cases** manager will automatically appear.
- By default, the load case named LC1 is created. This load is a permanent load of the Self Weight load type. The self weight of the structure is automatically calculated, although not graphically displayed.
- 4. Since you will also manually enter loads in the first load case of this project (Roof Weight), you must change the Load Type to **Standard**.
- 5. In the Description field, you can describe the content of this load case. For this project, enter the description **"Self Weight Structure"**.



Defining a Variable Load Case

- 1. Click even or to create a second load case.
- 2. Enter the description "Wind".
- 3. As this is a variable load, change the Action type to Variable.

- 	Load cases		- 87	
LC1 - Self Weight Stru LC2 - Wind	Name Description	LC2 Wind		
	Action type	Variable		Ŧ
	LoadGroup	LG2	۰.	
	Load type	Static		÷
	Specification	Standard		*
	Duration	Short		*
	Master load case	None		•
	Actions			1
	Delete all loads		>>>	1
	Copy all loads to another loadc	ase	>>>	
New Insert Edit	Delete		Close	

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

Load groups			х
🎾 🤮 🧶 📸 💽 🗠	2 4 6		
LG2	Name	LG2	
	Relation	Standard	Ψ.
	Load	Variable	+
	Structure	Building	
	Load type	Wind	-
New Insert Edit	Delete		ок

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

- 5. Click [OK] to close the Load group manager and to return to the Load cases manager.
- 6. Click [Close] to close the Load cases manager.

Remark:

Load groups

Each load is classified in a group. These groups influence the combinations that are generated as well as the code-dependent coefficients to be applied. The following logic is adopted.

Variable load cases that are independent from each other are associated to different variable groups. For each group, you set the load category (see EN 1991). The combination factors from the Eurocode are generated from the available load groups. When a generated combination contains two load cases belonging to different groups, reduction factors will be applied for the transient loads. If the load is divisible, its different components are entered as individual load cases. As long as the load combination does not contain any variable load belonging to another group, no reduction factors may be applied. The different load cases of a divisible load are therefore associated to one variable group. Load cases of the same type that may not act together, are put into one group, which is made exclusive, e.g. "Wind X" and "Wind -X" are associated to one exclusive group "Wind" to avoid simultaneous action.

Loads

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

The first load case (LC1) includes two loads:

- Self weight of the bars
- Roof weight

Switching between load cases

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

Load 4 X
LC1 - Self Weight Structure
LC1 - Self Weight Structure 😽 📖
LC2 - Wind
📲 Line force - on beam
🚟 Thermal load - on beam
🖶 🖞 Moment
🖶 🦸 Point displacement
🖶 🚟 Line displacement
Plane generator
Not calculated internal forces

Entering the self weight as linear load

- 1. Cancel any possibly active selection by pressing **<ESC>**.
- 2. Click on Line Force on beam in the Loads menu. The dialogue Line Force on beam appears.
- 3. In the field **Type**, choose **Self Weight**. The Direction is the global Z-direction and the Gravity coefficient is set to -1, so that the load is acting vertically downwards.

Load \mp X	•	Line force on beam		×
Load LC1 - Self Weight Structure Point force Line force - on beam Point displacement Point displacement Point displacement Point displacement Point displacement Point displacement Point olad - water accumulation Not calculated internal forces	$q_{max} = q_{min}$	Line force on beam Name Direction Type Gravity coef. Distribution Bottom flange Load above joint Geometry System Location Extent Coord. definition Position x1 Position x2 Ociai	LF1 Z • • · · · · · · · · · · · · · · · · ·	
			OK Canc	sel

4. Confirm your input with [OK].

- 5. Select all the bars by means of the **Select all** icon in the toolbar.
- 6. Press **<ESC>** to finish the input.
- 7. Press **<ESC>** once more to finish the selection.

The self weight load is represented in brown:



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

Entering the roof load as a surface load

In order to enter a **Surface load on beams**, an option called **Plane generator** will be used. It enables us to insert plane load in kN/m² even though there is no plate (2D) member. And program redistributes this load into linear load in kN/m.

- 1. Click on Plane generator in the Load menu. The Plane geometry dialogue appears
- 2. The load **Direction** is **Z** and the **System** is the global coordinate system **GCS**. In this way, the load acting vertically downwards.
- 3. Change the Value to -1,5 kN/m² and press [OK].

Basic Concept Training -

Load # ×	Plane geometry		×
LC1 - Self Weight Structure	Name	PG1	
Point force	Loaded beams :	All	
Thermal load on beam	Direction	Z	•
International - On Deani	System	GCS	· · · · · ·
	g [kN/m^2]	-1,50	
🛊 – 🌮 Point displacement			
Ine displacement			
Plane generator Plane do load - water accumulation Not calculated internal forces		2	OK Cancel

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



- The load stays in violet colour which means it is still in active selection. Click on the combo-box in the Properties window > Loaded beams and change the option from All to Advanced.
- In the Actions buttons choose the option Update beams selection to indicate that only the girders and not the longitudinal beams will be loaded directly by this surface load.
- 7. Click on beams **B2**, **B6** and **B10**.

Properties		×
Plane geometry (1)	- 14	V/ /
		6 🔺
Name	PG1	
Loaded beams :	Advanced	Ψ.
Direction	Z	*
System	GCS	
q [kN/m^2]	-1,50	
Load case	LC1 - Self Weight	Stri 👻
Actions		
Update beams selectio	n	>>>
Refresh		>>>
Edit plane load geometry		>>>
Table edit geometry		>>>



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

	F	ecalculation of pla	ne load to members		×
Calcula	te Plane load	System UCS Icoba	-1.5 kN/m^2	ОК Са	ncel

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

		Recalculation of pla	ne load to members		
Calcula	te Plane load	System UCS Icloba	I -1.5 kN/m^2	ОК Са	ancel

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

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



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



Entering roof load as second load case

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

Load	φ×
LC2 - Wind	R
LC1 - Self Weight Structure	4
LC2 - Wind	
Line force - on beam	

Entering a linear load

- 1. Click on Line Force on beam in the Load Menu. The dialogue Line Force on beam appears.
- 2. Change the **Type** to **Force**.
- 3. The load **Direction** is **Z** and the **System** is the local coordinate system **LCS**. The linear loads are acting in accordance with the local Z-axes of bars.
- 4. Change the Value to -4,8 kN/m.



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



Adapting a load

- 1. Select the linear loads on the roof girders and the columns of the first and last frame by clicking with the left mouse button on these loads. Mentioned members must have only half of the load value because only half of the loading width, affected by wind, belongs to them.
- 2. The common properties of the 8 series are displayed in the Properties window.
- 3. Change the Value from -4,8 kN to -2,4 kN in the Properties window.



- 4. Confirm the modification with **<ENTER>**.
- 5. Press **<ESC>** to finish the selection.

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

Note:

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

Combinations

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

Defining Combinations

- 1. Double-click on Combinations in the Main tree.
- 2. Since no combination has been entered yet, the window to create a new combination automatically appears.

🚊 📲 Load cases, Combinations
Result classes

	Combina	Combination - CO1		
Contents of combination		List of load cases Load case LC1 - Self Weight Structure LC2 - Wind		
Name :	C01	Delete Add		
Coeff :	1 Correct	Delete All Add All		
Type :	EN-ULS (STR/GEO) Set B			
Structure:	Building V			
Description :]		
Nonlinear combination :	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	OK Cancel		

- 3. The Type of the combination is changed to **EN ULS (STR/GEO) Set B**. With this envelope combination type SCIA Engineer will automatically generate linear combinations in accordance with the complex composition rules of the Eurocode.
- 4. A warning message that controls the content of code combinations with respect to load type may appear. Close it with **[Yes]**



- 5. With the button **[Add all]**, all load cases can be added to the combination. Otherwise you can manually drag&drop load cases from the list of load cases (right frame) to the contents of combinations (left frame).
- 6. Type "ULS" into Description row to distinguish the combination from the second one.
- 7. Confirm your input with [OK]. The Combinations manager is opened.
- 8. Click or it to create a second combination.

- 9. Change the **Type** of the combination to **EN-SLS Characteristic**. Type "SLS" into Description row to distinguish the combination from the first one.
- 10. Confirm your input with **[OK]**.
- 11. Click [Close] to close the Combination manager.

	Combinations			
🎜 🤮 🗶 🞼 👢	🚚 💱 🖋 👪 🗠 😂 🕴 Input combinations 🔹			
CO1 - ULS	Name	C02		
CO2 - SLS	Description	SLS		
	Туре	EN-SLS Characteristic		
	Structure	Building		
	Active coefficients			
	Contents of combination	Contents of combination		
	LC1 - Self Weight Structure [-]	1,00		
	LC2 - Wind [-]	1,00		
	Actions			
	Explode to envelopes	>>>		
	Explode to linear	->>>		
	Show Decomposed EN combinati	ons		
New Insert	New Insert Edit Delete Close			

Calculation

Linear Calculation

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

Executing the Linear Calculation

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

🚊 📲 Calculation, mesh
🛅 Check structure data
🔤 📥 Connect members/nodes
- 🛗 Calculation
- 🛱 Hidden calculation
Autodesign

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

	FE analysis	×
	Single analysis Batch analysis	
	Linear calculation	
	O Nonlinear calculation	
	O Modal analysis	
2.	C Linear stability	
11	Concrete - Code Dependent Deflections (CDD)	
133	Construction stage analysis	
	Nonlinear stage analysis	
	O Nonlinear stability	
	○ Test of input data	
	Number of load cases: 2	
and the second second		
	Solver setup Mesh setup	
all and the		
	OK Cancel	

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

Scia Engineer: End of analysis	<u>ہ</u>
Linear calculation: - Maximal translation 233.958 mm, in node gen 32 [11.014,6.000,5.164] (loadcase LC2) - Maximal rotation 60.257 mrad, in node N10 [12.000,6.000,0.000] (loadcase LC2) Sum of loads and reactions is OK	
ОК	

Results

Viewing results

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

Viewing the Reaction Forces

1. Double-click on

in the Main tree. The Results menu appears.

2. Below Supports, click Reactions.



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

Results

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

Properties		×
Reactions (1)	🖂 Va V/ -	1
	e 2	6
Name	Reactions	
Selection	All	-
Type of loads	Combinations	+
Combinations	CO1 - ULS	+
Filter	No	-
Values	Rz	+
Extreme	Node	+
Drawing setup 1D		
Rotated supports		
Actions		
Refresh	>>>	>
Table results	>>>	>
Preview	>>>	> (

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

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



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

Results		4 ×				<hr/>					
Displacem Deformed 3D displace 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4	ent of nodes Structure ement ons ation fractions ation table space support resu erial n protocol	litant	Report preview	2/906/12	34,06	75,06					
			the state of border								
New Close				Reacti	ons						
<u>%</u>				Linear calcu	lation Ev	treme · No	de				
Properties		4 ×		Selection :	All	ueme . No	ue				
Reactions (1)	- 10	80		Combination	s : CO1						
		10		Support	Case	Dv	Pv	D7	My	Mv	Mz
Name	Reactions			Support	Case	[kN]	[kN]	[kN]	[kNm]	[kNm]	[kNm]
Selection	All	*		Sn1/N5	CO1/1	-42,11	0,14	74,91	0.00	0.00	0,00
Type of loads	Combinations		· · · · · · · · · · · · · · ·	Sn1/N5	CO1/2	-10,97	0,10	32,63	0,00	0,00	0,00
Combinations	CO1 - ULS	-		Sn1/N5	CO1/3	-14,81	0,14	44,05	0,00	0,00	0,00
Filter	No	-		Sn2/N1	CO1/4	-0,07	0,12	46,35	0,00	0,00	0,00
Values	Rz	*		Sn2/N1	CO1/3	14,84	0,14	44,06	0,00	0,00	0,00
Extreme	Node	-		Sn2/N1	CO1/2	10,99	0,11	32,64	0,00	0,00	0,00
Drawing setup 1D				Sn2/N1	CO1/1	3,78	0,16	57,77	0,00	0,00	0,00
Rotated supports	10		Taske								
Actions			Commending								
Refresh		>>>	M Vy Vy My My			1 m e l					
Table results		>>>		555A.	5 2 7	4) 1 3					
Preview		>>>	Command >								
			m Place XY Ready								

Note:

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

Viewing internal forces on beam

- 1. In the **Results** menu, open the **Beams** group and select **Internal forces on beams.**
- 2. The options in the **Property Window** are configured in the following way:
 - The Selection field is set to Current.
 - The Load type is set to **Combinations** and Combination to **CO1 ULS**
 - The Values are wanted for My.
 - The Extreme field is changed to **Global**.
- 3. Select columns and the roof girders of the centre (middle) frame using the left mouse button.
- 4. Click on the **>>>** button next to **Refresh** to display the results on the graphical screen in accordance with the set options.

Properties		ų.	×
Internal forces on	memb🔁 🔏	V/ (7
		e 2	6
Name	Internal forces	on me	
Selection	Current		+
Type of loads	Combinations		Ŧ
Combinations	CO1 - ULS		Ŧ
Filter	No		+
Values	My		+
Extreme	Member		*
Drawing setup			
Section	All		*
Actions			
Refresh		>>>	•
Detailed		>>>	
			•
Table results		>>>	•



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

Configuring the Graphical Screen

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

	Dra	awing setup	
Representation : Limits :		Filled	v
🗆 My			
Maximum [kNm]	0		
Minimum [kNm]	0		
Description Values			Units
Draw section in la	abels		
Draw load case of	r combinati	on in labels	
Angle of text		0	
O deg		O User defin	ned
) 90 deg		0,00	deg
Setup for more comp	onents		
O Same scale		Space betwe	en diagrams
• Same height		1	0
		Shift of the fi	rst diagram
		0	0
			•

- 2. In the **Representation** field, choose **Filled**.
- 3. The Angle of text is set to 0°.
- 4. Click **[OK]** to confirm your input.
- 5. In the **Property** window, click the button next to **Refresh** to display the results in the graphical screen in accordance with the set options.



- 6. Then click [Close] to leave the Results Menu.
- 7. Press **<ESC>** to cancel the selection.

Note:

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

Code check

SCIA Engineer steel modules include a number of powerful tools to perform the steel calculations in accordance with the chosen design code.

The possibilities are as following:

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

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

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

Buckling parameters

Displaying the system lengths

- 1. Select with the left mouse button column **B1**, the left column of the first frame.
- 2. Click the right mouse button at an arbitrary position in the workspace. Context menu lists the possibilities for the selected entity.
- 3. In this menu select the Set view parameters for selected option. Limited View parameter settings window appears.

View parameters	setting
Check / Uncheck group	Lock position
4 🖭 🔛 🚱 🌌	Þ
Check / Uncheck all	
Structure	
Member surface	
Rendering	wired 🔹
Draw cross-section	
Cross-section style	section 🗾
Effective width of plate ribs	
Draw effective width	
Rendering	transparent 🗾 🛨
Member parameters	
System lengths	<u> </u>
Member nonlinearities	
FEM type	
	<u> </u>
Members 1D	
Show names in tab	Apply Cancel

- 4. Activate the **System lengths** and **Draw cross-section** options to display the reference lengths and the section of the bar.
- 5. Activate the Local axes Members 1D option to display the local coordinate system of the bar.
- 6. Confirm your input with [OK].
- 7. Press **<ESC>** to cancel the selection.



The figure shows that (default) system length Ly (blue line) for buckling around the strong axis (y-y) is total height of the column and Lz (green line) for buckling around the weak axis (z-z) is half of the height. The girder in the middle of the column therefore supports the column for buckling around the weak axis, i.e. for bending in the Y direction.

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

Setting the Buckling Parameters

- 1. Select both columns of the first frame with the left mouse button.
- 2. The **Properties window** shows the common properties of both entities. The **Buckling and** relative lengths are set to **Default**.

4 ×
🖃 Va V/ 🖉
6 x
column (100) 🔹 🔹
Standard 🔹
CS1 - HEA200 🔹
0,00
Centre 🔹
0
0
standard 🔹
0,00
standard 🔹
Default 🔹
Layer1 🔹
5,000
Line

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



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

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

		Name	BC1		Number	of parts	2	
		Buckling system	ns relatio	n				
	\mathbf{x}	ZZ =	zz	~	k	factor	Calculate	
	X	yz =	zz	~	k	z factor	Calculate	
		lt =	ZZ	Ý	S	way yy	acc. to Steel>Beams>Setup	~
Ly D2 Lz					S	way zz	acc. to Steel>Beams>Setup	~
	Lyz				Point of load app	lication	In shear center	
						Mcr	Calculated	v
					Bow imperfection	1		
	\sim				eo dy	no bow	imperfection	~
					eo dz	no bow	imperfection	~
		Relative deform	nation sys	tems relatio	n			
		def z =	уу	*		def y =	zz ↔	
		Warping ch	neck					
		Buckling syste	m	Chandre	and mother of			

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

Note:

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

- **Buckling systems relation**: in these fields you can define the system length to be used for, amongst other things, torsional buckling and lateral-torsional buckling.
- Relative deformation systems relation: in these fields you can define the system length to be used for the relative deformations (SLS check).
- 6. On the **Buckling data** tab you can edit the parameters in detail. The column consists of 2 components, i.e. 3 positions are available: (1) at the start, (2) in the middle at the horizontal girder and (3) at the end, at the roof girders.
- For instance, by modifying the **Free** option on position (2) for yy to **Fixed**, buckling of the column in the middle around the strong axis would be influenced as well. This would mean the system length around this axes would become also half of the total length (= 2,5 m). For this tutorial, the default options are kept.

	уу	ky	Sway yy	eo dy [mm]	ZZ	kz	Sway zz	eo dz [mm]	kyz	klt	k		kw
1	Fixed		acc. to B 👻		V Fixed		acc. to B 🔹		1,00	1,00	1,00	1,00	
2	Free Free				Fixed		acc. to B 🔻		1,00	1,00	1,00	1,00	
3	Fixed				V Fixed								

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

Remark:

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



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

Steel code check

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

Displaying the Slenderness and the Buckling Lengths

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



- If this option is not available, you must restart the linear analysis using for example Hidden calculation in the Project toolbar.
- 3. The options in the **Properties window** are configured in the following way:
 - Selection field is set to Current.
 - Values are wanted for Lam y, i.e. the slenderness around yy axis.
 - Extreme field is modified to No.
- 4. Select column **B1**, the left column of the first frame and hit **Refresh** button in **Actions**.

Properties			×
Steel slenderness (1)	- 74	7/ /	9
		e 2	6
Name	Steel slenderness		
Selection	Current		*
Filter	No		*
Buckling coefficient	Linear calculation		*
Values	Lam y		*
Extreme	No		*
Drawing setup 1D			
Actions			
Refresh		>>>	>
Table results		>>>	>
Preview		>>>	>



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



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

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



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

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

Steel Code Check – Ultimate limit state

- 1. Click ULS Checks Check in the Steel menu
- 2. The options in the **Properties** window are configured in the following way:

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

Properties			×	
Check of steel (1)	- Va	7/	9	
		6 🤞	6	
Name	EC 3			
Selection	All		*	
Type of loads	Combinations		Ψ.	
Combinations	CO1 - ULS		Ψ.	
Filter	Cross-section		*	
Cross-section	CS1 - HEA200		*	
Values	un.check		Ψ.	
Extreme	Global		Ψ.	
Output	Brief		*	
Drawing setup 1D				
Section	All		*	
Actions				
Refresh		>>:	>	
Single Check		>>:	>	
Autodesign >>			>>>	
Split CSS >>			>>>	
Unify CSS >>			>	
Table results >>			>	
Preview		>>:	>	

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



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

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



Check of steel

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

Combined bending, axial force and shear force check

According to EN 1993-1-1 article 6.2.9.1 and formula (6.41)

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

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

Note: Since the shear forces are less than half the plastic shear resistances their effect on the moment resistances is neglected.

Note: Since the axial force satisfies both criteria (6.33) and (6.34) of EN 1993-1-1 article 6.2.9.1(4) its effect on the moment resistance about the y-y axis is neglected.

Note: Since the axial force satisfies criteria (6.35) of EN 1993-1-1 article 6.2.9.1(4) its effect on the moment resistance about the z-z axis is neglected.

The member does NOT satisfy the section check!

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

Optimisation of the Steel Section

1. In the **Properties** window, click the **Properties** window are maintained, so that HEA200 will be optimised.

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

1 ic info Change Next up or Up & down v Value Autodesign List Sort by HEA200 Yes			Autode	esign of the cross-s	ection	>
sk: 7.946 Info Change Next up for optimal Up & down v Value Value HEA200 Ves No H v	Aut	todesign ximal check	1			
Info Change Next up of or optimal Up & down v Value Autodesign List Sort by HEA200 Ves No • H •	Ма	ximum unity check:	7.946		Y	
Change Next up If or optimal Up & down v Value Autodesign List Sort by HEA200 Ves No v H v		Edit constraints	Info			
Next up of or optimal Up & down v Value Autodesign List Sort by HEA200 Ves No v H v		Edit	Change			
Value Autodesign List Sort by HEA200 Ves No VH V		Next down	Next up			
Up & down		Search fo	or optimal			
Value Autodesign List Sort by HEA200 Ves No VH V	Dir	rection	Up & down ♥	N		
Value Autodesign List Sort by HEA200 Ves No H	Parar	neter		Î		
Value Autodesign List Sort by HEA200 Image: Yes No Image: Head to be addressed on the second	1 - c	atalogue: HEA200	~			
HEA200 Ves No H		Param.	Value	Autodesign	List	Sort by
	1	l sections	HEA200	Ves Ves	No	• H •
	1	Param.	Value HEA200	Autodesign	List No	Sort by
		Set value	Select/Deselect A	II Test relat	ions	OK Cancel

The Autodesign of the cross-section dialogue looks like below

- This window again displays the maximum unity check for all members with CS1 HEA200 which is 7,946. Just above this value there is a maximum unity check to be reached (but not exceeded, which is by default 1.
- 3. Click the **Search for optimal** button. The program will search within the profile library for a new cross-section and stops with the first one that satisfies the unity check.

		Autodes	sign of the cross-se	ection		×
Auto	odesign					
Max	imal check	1				
Max	imum unity check:	0.916				
E	dit constraints	Info				
	Edit	Change				
	Next down	Next up				
	Search for	r optimal				
Dire	ection	Up & down ∨	Ν			
			Î.			
Param 1 - ca	ieter italogue: HEA340	~	u <u>⊢</u> ⇒ĭ			
	Daram	Value	Autodesign	liet	Sorth	
1	I sections	HEA340	V Yes	No	▼ H	y -
	_					
	Set value	Select/Deselect All	Test relati	ons	ОК С	ancel

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

4. Confirm the optimisation with [OK].

Note:

The project must be recalculated after the optimization. The changed cross-section modifies the self weight of the structure as well as the stiffness of the whole model, which will lead to a different distribution of the internal forces.

This specifically means that, after optimisation and recalculation of the structure, the profile concerned could possibly reveal inappropriate. In that case, you must re-execute the optimisation in order to find a solution in an iterative manner.

- To quickly restart the calculation after an optimisation, use the hidden calculation command. Click on the Hidden Calculation in the Project toolbar.
- 6. Click [Close] to quit the Steel service.

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

Steel connections

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

Activating the Steel Connection Input

- To obtain access to the steel connections, appropriate functionality needs to be activated first. Double-click on project in the Main tree to open the Project data and click on the Functionality tab.
- In the Steel part, activate the Frame rigid connections functionality. The functionality Structural model is automatically activated as well, as this one is required for the definition of the connection.

Project data							
Basic data Functionality Actions Protection							
	Dynamics		~	E	Steel		
	Initial stress				Fire resistance		
	Subsoil				Connection modeller		
	Nonlinearity				Frame rigid connections		
	Stability				Frame pinned connections		
	Climatic loads				Grid pinned connections		
	Prestressing				Bolted diagonal connections		
AL	Pipelines				Expert system		
	Structural model	V			Connection monodrawings		
	BIM properties				Scaffolding		
	Parameters				LTB 2nd Order		
	Mobile loads				ArcelorMittal		
	Automated GA drawings				Girders with sinusoidal webs		
STATE OF LAND	LTA - load cases						
A DECEMBER OF THE OWNER OWNE	External application checks						
	Property modifiers						
at the	Bridge design		~				
	L=	[man					
					OK Sto	mo	

3. Confirm your choice with [OK] button.

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

Displaying the Structural model

- 1. Zoom in on node N2, i.e. the node that connects column B1 and roof girder B2.
- 2. In the **Command line** click on the **Show/hide surfaces** and the **Render geometry** for icons. The program displays the calculation model like this:



- 3. In this analysis model, the bars are arriving in the same node, i.e. node **N2**. But in practice the girder is cut for instance at the column flange or vice versa. In SCIA Engineer, you can indicate this through the structural model.
- 4. Activate the structural model by clicking on view \rightarrow Set view parameters \rightarrow Generate



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

Note:

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

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

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

Inputting a Steel Connection

- 1. Double-click on Steel in the Main window to open the Steel service.
- Double-click on Frame bolted/welded-strong axis under Connections to enter a new rigid frame connection.
- 3. The program asks for a point of connection now, select node **N2**.



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



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



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



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

	End-plate	
Name	EP	
Material	S 235	·
Thickness[mm]	20	
Input	Top/Bottom/Left/Right	
Top extension [mm]	-5	
Bottom extension [mm]	20	
Left extension [mm]	105	
Right extension [mm]	105	
Total width [mm]	300	
Total height [mm]	328	
		OK H Cancel
	Name Material Thickness[mm] Input Top extension [mm] Bottom extension [mm] Left extension [mm] Total width [mm] Total height [mm]	Name EP Material \$ 235 Thickness[mm] 20 Input Top/Bottom/Left/Right Top extension [mm] -5 Bottom extension [mm] 20 Left extension [mm] 105 Right extension [mm] 105 Total width [mm] 300 Total height [mm] 328

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



The bolts are automatically displayed in the graphical screen.

- 10. To change the properties of bolts, click the icon next to **Bolts** option in the **Properties** window.
- 11. For the **Selected bolt position**, an **M20 8.8** is chosen. A window informs you that the bolt position has changed.



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

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

Bolts		
Selected bolt assembly	M20 - 8.8	·
Length [mm]	65	
 Bolt pattern	2 bolts/row	· ·
Reference	Bottom of the beam	· ·
Internal bolts distance [mm]	91	
Use last bolt-row for shear capacity only		
1.Row		
2.Row		
3.Row	V	
1.Location [mm]	265	
2.Location [mm]	178	
3.Location [mm]	69	
Actions		
Update location		>>>
All distances are within the limits.		^
		\sim
	ОК	Cancel

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


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



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

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



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

View parameters setting			
Check / Uncheck group	Lock position		
	⊳		
Check / Uncheck all			
Steel connections			
Display			
Rendering	rendered with edges 🔹		
Welds			
Coloured			
Steel connections label			
Display label			
Name			
Weld symbols			
Dimensions			
Show names in tab	Cancel		
Close this menu with [OI	<].		

18. In the **Properties window**, click the **icon** next to **Refresh** to regenerate the input in the graphical screen. The connection is displayed in a rendered style:



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

Checking the connection

- 1. Select the connection by mouse cursor and in the Properties window set the following:
 - Type of load is set to Combinations and Combinations to CO1 ULS.
 - For the Frame type, choose braced.
 - Output is set to Summary.
- 2. In the **Property** window, click the button next to **Open Preview** to display the results in **Report Preview** below the modelling screen.



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

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

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

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

Notes:

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

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

Document

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

Engineering report

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



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

pe	rties up do Document item
	New items 7 ×
	₩ < T 🏤 🖽 🗮 🔍
	I ∓- Special items
	E SCIA Design Forms (standalone)
	Inbox
	Project
	🕀 Libraries
	🕀 Sets
	E Solver and Mesh
	. Structure
	⊞- Load
	Construction stages
	• Results
	H- Special
	± Steel
	E Custom check
	E Pipeline
	Timber
	E Concrete data
	+ Concrete
	E Concrete 15
	🕀 Steel concrete bridge
	E Geotechnics
	E Composite Beam
	E Composite Column
	E Composite
	Mobile loads
	Influence lines
	⊞ Special
	Gallery pictures
	H Report templates

- 3. Using this window, various data can be added to the report.
 - Open the Libraries group and select Materials. Double-click in this item or hit button to add this item to the document navigator.
 - Add also **Cross-Sections** one row above.
 - Open the Structure group and double-click Members.
 - Open the **Results** group and click **Internal forces on beam.**
- 4. You can directly see these items in the Navigator and on the paper preview as well:

Navigator	4 ×	New items 4 ×	
Materials	₽ 0	计全下结话 医原	
Cross-sections			1. Materials
Members	-0		Steel FC3
Peartions		Construction stages	Name p E _{not} µ Lowerlimit Upperlimit F ₂ F ₃ Colour
E O REACTIONS	• •	- Results	[kg/m³] [MPa] [mm] [MPa] [MPa]
		- Deformed Structure	
		- 3D stress	S 235 7850,0 2,1000e+05 0.3 0 40 235,0 360,0
		- Internal forces on beam	8,0769e+04 0,00 40 80 215,0 360,0
		- Deformation on beam	
		 Displacement of nodes 	2. Cross-sections
		 Acceleration of nodes 	CS1 Picture
		- Reactions	Type HEA340 Z
		- Resultant of reactions	Formcode 1 - 1 sections
		Nodal space support result	Item material 5 235
N		- Intensity Member Stress	Fabrication rolled
3		- Shear in joint	Colour Colour
		- Relative deformation	Fiexural bucking y-y, b cy
		- Bill of material	A [m ²] 1,3400e-02
		- Connection Forces	A, [m], A, [m] 9,5495e-03 3,3201e-03
		- Foundation table	Ac [m 'm] Ao [m 'm] 1,8000±400 1,7944±400
		- Displacement of nodes - m	a [dea] 0.00
		- Member2D - Internal Force:	I _v [m ⁻¹], I _z [m ⁻¹] 2,7700e-04 7,4400e-05
		Mambar3D Stractor	li [mm] i [mm] 144 75 75 75

Drag the items with the mouse to change their order.

Displaying results in the document

- In the Navigator click Internal forces on beam. The red exclamation mark both in Navigator and preview indicates that the values presented are not up-to-date. In the Properties window the setting of this table is displayed. Parameters for displaying the results in the Engineering Report are configured in the same way as the parameters for viewing the results in the Results Menu of the SCIA Engineer application.
 - Selection type is set to All.
 - Type of load is set to Combinations and the Combination to CO1 ULS.
 - Values are set to vertical reaction Rz.
 - Extreme field is changed to Global.



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

text term text term text term text term ss-sections	iastor	п ×	New items II ×
tetrials ↑ ◆ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↑ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲ ↓ ▲	igutor	1.4	Trew reems
sizections Implete mbers Implete ictions Implete Implete Implete <t< th=""><th>Materials</th><th>•••</th><th>半≪玉台田目●</th></t<>	Materials	• ••	半≪玉台田目 ●
mbers •••••••••••••••••••••••••••••	Cross-sections	.	
et: Construction stages A b: Construction stages A b: Results - Deformed Structure -30 displacement -30 stress -internal forces on beam - Deformation on beam - Displacement of nodes - Resultant of reactions - Resultant of reactions - Nodal space support result - Internal forces - Nodal space support result - Internal forces - Stress - Nodal space support result - Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000 NS Bit GC3: - PE160 S 235 6,000	Members	<u>•</u> ••	13
Cloops E+ Cell - Fe340 5 235 5,000 NS B+ Cell - Fe4340 5 235 5,000 NS B- D displacement B- Sci - Fe4340 5 235 5,000 NS B- D displacement B- Sci - Fe4340 5 235 5,000 NS B- Deformation on beam Deformation on des B- Sci - Fe4340 5 235 5,000 NI B- Acceleration of nodes - Acceleration of nodes Resultant of reactions Resultant of reactions Sci - Fe4340 5 235 6,003 NI Nodal space support result Bit Sci - Fe14340 5 235 6,000 NI Bit Sci - Fe14340 S 235 6,000 NI Bit Sci - Fe14340 S 235 6,000 NI Bit Sci - Fe14340 S 235 6,000 NI Bit Sci - Fe14340 S 235 6,000 NI Bit Sci - Fe14340 S 235 6,000 NI Bit Sci - Fe14340 S 235 6,000 NI Bit Sci - Fe14340		00	Construction stages
→ Deformed Structure 55 CS1 - FEA340 5235 5.000 N6 → 30 displacement 56 CS5 - T+ 1 var (PE180); 150 5235 6.083 N7 → Deformation on beam → Deformation on beam 58 CS1 - FEA340 5235 5.000 N10 → Deformation on beam → Deformation on beam 99 CS1 - FEA340 5235 5.000 N10 → Displacement of nodes 11 CS5 - T+ 1 var (PE180); 150 5235 6.003 N12 → Acceleration of nodes 11 CS5 - T+ 1 var (PE180); 150 5235 6.000 N12 → Resultant of reactions Nodal space support result 115 CS3 - PE160 5235 6.000 N14 B14 CS3 - PE160 5235 6.000 N12 116 CS3 - PE160 5235 6.000 N14 B15 CS3 - PE160 5235 6.000 N14 116 121 CS3 - PE160 5235 6.000 N14 B20 CS3 - PE160 5235 6.000 N18 116 122 CS4 - HELeq70x70x7 5235 7.810 N10	Reactions	- O	⊡ Results
-30 displacement -30 displacement -30 tress -31 displacement -32 otress -33 displacement -30 tress -31 displacement -32 otress -33 displacement -34 displacement -35 displacement -35 displacement -36 displacement -36 displacement -36 displacement -36 displacement -36 displacement -36 displacement -37 displacement -38 displacement -39 displacement -30 displacement -			Deformed Structure
→ 3D stress → 3D stress → Internal forces on beam → 67 CSS 1 + 14 var (PE180; 150) 5 235 5,000 N10 → Deformation on beam → 65 1 + 4EA340 5 235 5,000 N10 → Displacement of nodes ⇒ 11 CSS 1 + 1 var (PE180; 150) 5 235 6,083 N12 → Acceleration of nodes ⇒ 11 CSS 1 + 1 var (PE180; 150) 5 235 6,083 N12 → Acceleration of nodes ⇒ 11 CSS 1 + 1 var (PE180; 150) 5 235 6,000 N15 → Reactions → 10 stress ⇒ 15 CS3 - PE160 5 235 6,000 N2 → Nodal space support result → 11 stress > 235 6,000 N2 → 11 intensity → 11 stress > 235 6,000 N2 → 11 intensity → 11 stress > 235 6,000 N8 → 11 intensity → 11 stress > 235 6,000 N8 → 11 intensit → 11 stress > 235 6,000 N8 → 21 is CS3 - PE160 > 2235 6,000 N8 → 12 is CS3 - PE160 > 2235 6,000 N8 → 12 is CS3 - PE160 > 2235 5,000 N18 → 12 is CS3 - PE160 > 2235 7,810 N0 Displacement of nodes			3D displacement
- internal forces on beam - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - <td></td> <td></td> <td>3D stress</td>			3D stress
Internation on beam Deformation on beam Displacement of nodes Acceleration of nodes Acceleration of nodes Reactions Nodal space support result Intensity Member Stress Shear in joint Relative deformation Bit of naterial Connection Forces Foundation fable Displacement of nodes Bit of naterial Connection Forces Foundation fable Displacement of nodes Bit of naterial Connection Forces Foundation fable Displacement of nodes Bit of naterial Connection Forces Member2D - Contact stress Member2D			Internal forcer on heam
→ Deformation on beam ⇒ Deformation on beam ⇒ 235 6,083 N14 → Acceleration of nodes ⇒ Cascing ≥ 235 6,083 N14 → Acceleration of nodes ⇒ Cascing ≥ 235 6,003 N14 → Resultant of reactions ⇒ B13 CS3 - IPE 160 > 2235 6,000 N2 → Nodal space support result ⇒ IIA S3 - IPE 160 > 2235 6,000 N3 → Intensity → Member Stress ⇒ B18 CS3 - IPE 160 > 2235 6,000 N4 B18 CS3 - IPE 160 > 2235 6,000 N4 B18 CS3 - IPE 160 > 2235 6,000 N4 B18 CS3 - IPE 160 > 2235 6,000 N4 B18 CS3 - IPE 160 > 2235 6,000 N4 B19 CS3 - IPE 160 > 2235 6,000 N4 B10 CR3 - IPE 160 > 2235 6,000 N4 B20 CS3 - IPE 160 > 2235 6,000 N4 B21 CS3 - IPE 160 > 2235 7,200 N7 N04 N04 <td></td> <td></td> <td>internal forces on beam</td>			internal forces on beam
→ Displacement of nodes → Acceleration of nodes → Acceleration of nodes Reactions → Reactions Nodal space support result − Intensity Member Stress → Biil of material → Connection Forces → Readiant forces → Biil of material → Connection Forces → Member2D - Stresses → Member2D - Contact stress → Member2D - Stresses → Member2D - Contact stress → Member2D - Contact stress → Member2D - Stresses → Member2D - Contact stress → Member2D - Contact stress → Displacement ef nodes - material ⊕ Displacement of nodes → Subsoil - C parameters → Subsoil - C parameters <t< td=""><td></td><td></td><td> Deformation on beam</td></t<>			Deformation on beam
Acceleration of nodes Resultant of reactions B13 CS 1: #FE43400 S 235 5,000 N2 Resultant of reactions Nodal space support result B13 CS 3: #FE160 S 235 6,000 N2 Intensity Member Stress B16 CS 3: #FE160 S 235 6,000 N2 Schemin joint B16 CS 3: #FE160 S 235 6,000 N2 B18 CS 3: #FE160 S 235 6,000 N4 B18 CS 3: #FE160 S 235 6,000 N4 B18 CS 3: #FE160 S 235 6,000 N4 B19 CS 3: #FE160 S 235 6,000 N4 B10 CS 3: #FE160 S 235 6,000 N4 B10 CS 3: #FE160 S 235 6,000 N4 B20 CS 3: #FE160 S 235 6,000 N4 B21 CS 3: #FE160 S 235 6,000 N4 B22 CS 3: #FE160 S 235 6,000 N4 B22 CS 3: #FE160 S 235 6,000 N4 B22			Displacement of nodes
Reactions Reactions Resultant of reactions B14 CS3. JPE160 5.235 6.000 1/2 Intensity B14 CS3. JPE160 5.235 6.000 1/2 Intensity Member Stress 5.235 6.000 1/2 1/2 6.000 1/8 Shear in joint B16 CS3. JPE160 5.235 6.000 1/8 1/8 6.000 1/8 B16 CS3. JPE160 S.235 6.000 1/8 1/8 6.000 1/8 1/8 6.000 1/8 1/8 6.000 1/8 1/8 6.000 1/8 1/8 6.000 1/8 1/8 6.000 1/8 1/8 6.000 1/8 1/8 1/8 5.235 6.000 1/8 1/8 1/8 1/8 5.235 6.000 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8 1/8			- Acceleration of nodes
Resultant of reactions B14 CSS - IPE180 S 225 6,000 N3 Intensity B14 CSS - IPE180 S 225 6,000 N3 Intensity Member Stress B18 CS3 - IPE180 S 225 6,000 N4 Intensity Member Stress B18 CS3 - IPE180 S 225 6,000 N4 B17 CS3 - IPE180 S 225 6,000 N4 B17 CS3 - IPE180 S 225 6,000 N4 B18 CS3 - IPE180 S 225 6,000 N4 B17 CS3 - IPE180 S 225 6,000 N4 B17 CS3 - IPE180 S 225 6,000 N4 B17 CS3 - IPE180 S 225 6,000 N4 B18 CS3 - IPE180 S 225 6,000 N4 B17 S 225 6,000 N4 B18 CS3 - IPE180 S 225 6,000 N4 B17 S 225 6,000 N4 B18 CS3 - IPE180 S 225 6,000 N4 B17 S 225 7,810 N6 B24 CS4 - I			Reactions
- Nodal space support result			Resultant of reactions
Intensity 2.33 1/2 2.33 6,000 1/4 Member Stress 317 2.33 1/2 6,000 1/4 Member Stress 312 2.33 1/2 6,000 1/4 Stress 312 2.33 1/2 6,000 1/4 B12 C33 1/2 1/2 6,000 1/4 B13 C33 1/2 1/2 6,000 1/4 B13 C33 1/2 1/2 6,000 1/4 B14 C34 1/2 1/2 6,000 1/4 B12 C35 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2 1/2			
Internation Implement of nodes - m			Intensity
Member stress 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 253 104 102 253 253 253 253 104 102 253 253 104 105 104 105 105 106 106 106 106 106 106 106 106 106 106 106 106 106 106 106 106 106 106 106 106 106 <			Marchan Change
- Shear in joint 223 223 223 200 110 - Relative deformation 223 223 6,000 110 - Bill of material 222 234 - FEEd200,700/7 2.235 7,010 110 - Connection Forces - Foundation table 224 254 - FEEd200,700/7 5.235 7,010 110 - Displacement of nodes - m - Member20 - Internal Force - B22 C54 - FEEd200,700/7 5.235 7,810 N7 - Displacement of nodes - m - Member20 - Stresses - Linear calculation, Extreme : Global - Selection : All - Combinations : C01 - Subsoli - Costati stress - Support <u>Case</u> Rx Ry Ry R/x My - Subsoli - Costati stress - Subsoli - Costati stress - Subsoli - Costati stress - Support <u>Case</u> Rx Ry R/x My - Subsoli - Costati stress - Subsoli - Costati stress <td< td=""><td></td><td></td><td>Member stress</td></td<>			Member stress
Relative deformation Bit 22 C43 FE42070:70x7 2:35 7.810 140 Bit 10 material C53 FE42070:70x7 2:35 7.810 140 Bit 22 C54 FE42070:70x7 2:35 7.810 140 Bit 24 C54 FE42070:70x7 2:35 7.810 140 Bit 24 C54 FE42070:70x7 2:35 7.810 147 Bit 25 C54 FE42070:70x7 2:35 7.810 147 Member20 - Internal Force: Member20 - Contact stress Feaculation, Extreme : Global Selection : All Selection : All Subsoil - C parameters Subsoil - C parameters FC01 Support Case Rx Ry Rz Rx/m (ktm) Bit 10 results (beta) Bit 20 Csubs (Lea) Coll Support Case Rx Ry Rx Rt/m (ktm)			Shear in joint
Bill of material Bill of material - Bill of material Bill of material - Connection Forces Bill of material - Foundation table Bill of material - Displacement of nodes - m Bill of material - Member20 - Internal Force: - Member20 - Stresses - Member20 - Stresses - Member20 - Stresses - Subsoil - Contract stress - Subsoil - Contract stress - Subsoil - Other data Selection : All B: D results (beta) Support Case Rx B: 2D results (beta) Support Case Rx Subsoil - Cold(-1) - 79,312 1,93 B: 2D results (beta) Support Case Rx B: 2D results (beta) Support Case Rx B: 2D results (beta) Support Case Rx Support Case Rx Rty Support Case Rty Rty			Relative deformation
- Connection Forces B24 C54.+Ffteg270x70x7 B235 7,810 M6 - Dusplacement of nodes - mi - Member20 - Internal Force - Member20 - Internal Force - Member20 - Stresses			- Bill of material
- Foundation table - Displacement of nodes - m - Member20 - Internal Force - Member20 - Stresses - Member20 - Stresses - Member20 - Stresses - Member20 - Stresses - Member20 - Stresses - Subsoil - Contact stress - Subsoil - Contact stress - Subsoil - Other data - Subsoil - Contact stress - Disults (beta) - Support Case B: 20 results (beta) - Support			Connection Forces
Displacement of nodes - m Member2D - Internal Force: Member2D - Contact stress Subjoil - C parameters Subjoil - C par			Foundation table
Member2D - Stresses 4. RedCUOIDS Member2D - Contat stress Linear calculation, Extreme : Global Selection : All Combinations : CO1 Subsoli - C parameters Combinations : CO1 Subsoli - C parameters Combinations : CO1 B: 10 results (beta) Support Case Rix Ry (Leit) [Leit]			···· Displacement of nodes - mi
Member20 - Contact stress Linear calculation, Extreme : Global Subsoil - Other data Selection : All Combinations : CO1 Combinations : CO1 B: DD results (beta) Support Case Rx Ry Rz My B: 2D results (beta) Sn4/N10 Col1/1 -79,17 1,93 139,80 0,00 0,00			Member2D - Stresses
Immunol 2013 Selection : All - Subsiol - C parameters Combinations : CO1 - Subsiol - C parameters Combinations : CO1 - Subsiol - C parameters Combinations : CO1 B: D2 results (beta) Support Case Rx Ry Rz Mx My B: 2D results (beta) Sn={N10 CO1/1 -79,27 1,93 139,80 0,00 0,00			Member2D Contact strass
			Subacily Contact stress
- Subsoli - Other data Support Case R;x Ry H;x Hy ⊕: 10 results (beta) B: 20 results (beta) Sn4/N10 CO1/1 [Ar3], [Ar3] Sa9,80 0,00 0,00			Subsoil - C parameters
			Subsoil - Other data
B: 2D results (beta) Sn4/N10 CO1/1 -79,77 1,93 139,80 0,00 0,00 0,00			ID results (beta)
			2D results (beta)
Calculation protocol Sn3/N6 CO1/4 28,91 1.19 84.22 0.00 0.00			Calculation protocol
Eigen Frequencies Sn6/N15 CO1/1 -43,27 -2,09 78,45 0,00 0,00			Eigen Frequencies
Critical load coefficients Sn1/N5 CO1/2 -11,63 0,12 35,46 0,00 0,00			Critical load coefficients
Seismic detailed Sn1/N5 CO1/4 -15,70 0,16 47,88 0,00 0,00			Seismic detailed

Adding an image to the document

- 1. Any picture from SCIA Engineer application can be set to Engineering Report. Either as printscreen (that is unchanged for ever) 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 analytic model with loads. You can use the icons above the **Command line** to hide surfaces and rendering and show loads:

 Image: Ima
- 3. Click on button Print Picture in toolbars and select Live picture into Engineering Report



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

Document picture - Insert objects to Engineering report Inbox			
Insert			
Wert Insert Insert Close One at Two at page width into selected report into inbox	Image: Solution of the second seco		
Caption	LC1 / Tot. value / Name		
Picture size definition	Two at page	•	
Automatic scale to fit size			
Scale 1:	104,596767804259		
Stretch mode	Dark lines	•	
Rendering	Standard	-	
Antialiasing quality	None	-	
Rotation	None	-	
Result legend	Right		
Export to PDF as 3D			
Position	One below another		
Load units in regen. (related to objects created in picture editor	V		
Load activity in regen.			
Draw inactive members	as is in the window		
Settings of activity	>>>	•	
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	>>>	•	

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



Printing Engineering Report

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

Basic Concept Training -

	5	Report_1 [Tutorial Frame Steel.esa] - Engineering report	- 0
Joe Joe	Print Deer Printer Printer Printer Printer Printer Australia Printer Australia Pr	<section-header><complex-block></complex-block></section-header>	

Epilogue

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

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

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