



TUTORIAL CONCRETE

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 2024 SCIA nv. All rights reserved.

Table of Contents

Table o	f Con	tents		3
Genera	l info	rmatio	n	5
	Wel	come.		5
	SCI	A Engi	neer support	5
	Web	site		5
Introdu	ction			6
Chapte	r 1:	Gett	ing started	7
	1.1	Star	ting SCIA Engineer	7
	1.2	Star	ting a new project	7
Chapte	r 2:	Proj	ect management	9
	2.1	Save	e, Save as, Close and Open	9
	2	.1.1	Saving a project	9
	2	.1.2	Closing a project	9
	2	.1.3	Opening a project	9
	2	.1.4	Start project manager	9
	2.2	Com	ponents of the User Interface	10
	2	.2.1	Main menu	10
	2	.2.2	SCIA Spotlight	10
	2	.2.3	Status Bar	10
	2	.2.4	Process Toolbar	11
	2	.2.5	View Bar	11
	2	.2.6	Properties Panel	12
	2	.2.7	Input Panel	12
	2	.2.8	Marking menu	13
Chapte	r 3:	Geo	metry input	.14
	3.1	Inpu	t of the geometry	14
	3	.1.1	Cross-sections	14
	3	.1.2	Geometry	16
	3	.1.3	Additional data	23
	3	.1.4	Check structure	
	3	.1.5	Graphic representation of the structure	27
Chapte	r 4:	Load	ds and combinations	.29
	4.1	Load	d cases and load groups	29
	4.2	Load	ds	31
	4.3	Com	ibinations	34
Chapte	r 5:	Ana	lysis	.35
	5.1	Line	ar analysis	35

Chapter	6:	Resu	ults	38
	6.1	Read	ctions	38
	6.2	Inter	nal forces	39
	6.3	3D d	eformations	42
	6.4	Resu	ults-related status bar options	43
Chapter	7:	Rein	forcement design	44
	7.1	Cond	crete settings	44
	7.2	Buck	kling parameters	44
	7.3	Inter	nal forces	47
	7.4	1D re	einforcement design	47
	7.5	2D re	einforcement design	49
	7.6	Cond	crete checks	51
Chapter	8:	Repo	ort	52
	8.1	Engi	neering report	52
	8	.1.1	Adding images to the report	53
	8	.1.2	Displaying results	55
	8	.1.3	Printing or exporting the report	57

General information

Welcome

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

The program treats the calculation of 2D/3D frameworks, design and check of reinforcement included. Besides frames, it is also possible to dimension plate structures, inclusive of advanced concrete calculations. The complete process of calculation and design has been integrated in one program: input of the geometry, input of the calculation model (loads, supports ...), linear and nonlinear calculation, output of results, reinforcement design and checks according to various codes, generating the calculation report, etc.

System requirements to run SCIA Engineer 24:

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

SCIA Engineer is available in two different versions:

Licence version

The licence version of SCIA Engineer is secured with a Cloud account. For more information: https://eu-1.cdn77-storage.com/scia-

supportstore/support/sciaengineer/installation/24/MAN%20%5BEng%5D%20SCIA24.0%20-%20Installation%20manual%20cloud.pdf

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

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

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

• Student version

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

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

SCIA Engineer support

You can contact the SCIA Engineer support service

• By e-mail

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

By telephone
 For various phone numbers to different offices visit our page https://www.scia.net/en/contact/offices
 <u>The contact by telephone is available only for users with Licence version or Try-out version</u>

Website

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

Introduction

This tutorial describes the basic functions of SCIA Engineer for the input, analysis and design of a 3D concrete structure with 1D and 2D elements.

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 concrete settings and we will perform the concrete design for 1D and 2D elements.

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

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



Chapter 1: Getting started

1.1 Starting SCIA Engineer

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

- double-click on the SCIA Engineer shortcut in the Windows Desktop, or
- if the shortcut is not installed, click the Windows start button and choose SCIA Engineer 24.0 > SCIA Engineer 24.0.

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

1.2 Starting a new project

When the Project manager dialogue appears, click New project.



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

Now, the	Project settin	gs dialogue is	opened.	On the Ba	isic data tab,	you can enter	general d	ata about the
project.								

Basic data Fi	unctionality Action	as Unit Set Protection			
Dasie data 11	DATA	is one see Trotection	MATERIAL		
	Name: Part: Description: Author: Date:	- - - 25/07/2024	Concrete Material Reinforcement mai Steel Masonry Aluminium Timber Steel fibre concrete	C25/30 B 500B	¥
	Structure:	🚀 General XYZ 🛛 👻	CODE National Code:		·
	'Thick-walled' Con	crete cross-sections: the advanced 2	National annex: Standard EN		•)

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

Set Structure: General XYZ.

In the **Material** frame, tick the **Concrete** checkbox (defining a material is the only required setting to proceed). Choose **C25/30** from the combo-box and **B 500B** as reinforcement material.

In the Code frame select National Code EC-EN and National annex: Standard EN.

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

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

Chapter 2: Project management

2.1 Save, Save as, Close and Open

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

2.1.1 Saving a project

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

If a project has not yet been saved, the dialog box **Save as** appears. Choose the drive you want to save your project in. Select the (sub)folder in which you want to put the project. Enter the file name in **File name** and click on **[Save]** to save the project.

If you choose **File > Save as** in the main menu, you can select the drive and (sub)folder and set a name for the project file.

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

2.1.2 Closing a project

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



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

2.1.3 Opening a project

Click Main menu > File > Open or press Ctrl+O to open an existing project.

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

2.1.4 Start project manager

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

2.2 Components of the User Interface

2.2.1 Main menu

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

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



2.2.2 SCIA Spotlight

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

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

08 b- Hinge on 1D Ctrl+H O Hinge at the beginning of beam -O Hinge at the end of beam 🔔 Hinge on 2D 🔏 Hinged support in node SHOW ME 0 0 Hinges at both ends of beam н 0 Make everything in the clipping box ... ____ Make everything in the workplane vi... Hinge typ in GeometryLengthInch Toggles between the following mod. +- Connects absolute nodes and edges .

45°_Revize3_LIN_FINAL

2.2.3 Status Bar

The Status bar contains settings and toggles of most often used options:

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



2.2.4 Process Toolbar

The Process toolbar bar has following features:

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



2.2.5 View Bar

The View bar has following features:

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



2.2.6 Properties Panel

The Properties panel has following features:

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



2.2.7 Input Panel

The Input panels contains:

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



2.2.8 Marking menu

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

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

- Model:
 - o 1D member
 - o Column
 - o Plate
 - o Panel with load to 1D & edges
 - o Hinge on 1D
 - o Support in node
- Modify:
 - o Move
 - о Сору
 - o Multicopy
 - o Break in defined points
 - o Connect members/nodes
 - o Disconnect members/nodes
- Load:
 - o Point load in node
 - Point load on 1D
 - Line load on 1D
 - o Line load on 2D edge
 - Surface load on 2D
 - o Free surface load
- Visibility:
 - Hide selected
 - o Hide unselected
 - o Make layers visible











Chapter 3: Geometry input

3.1 Input of the geometry

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

3.1.1 Cross-sections

When entering one or more 1D members, a cross-section is immediately assigned to each member. By default, the active cross-section is represented. You can open the profile library to activate another cross-section. If you want to add a member before a cross-section has been defined, a rectangular cross-section with height 400 mm and width 300 mm will be used automatically.

Adding a cross-section

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

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

Click Concrete in the group Available groups.

In the Available items of this group, you can choose for a Rectangle.

New cross-section		\times
Available groups	AVAILABLE ITEMS OF THIS GROUP	
Rectangle	Profile Library filter	•

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

×

The Cross-section window appears. Cross-section Name CS1 Type Rectangle Detailed 500; 500 Ζ Shape type Thick-walled Parameters C25/3 n] 500 mm] 500 Gener Draw colour Normal colour Colour AutoDesign constraints Fabrication concrete Concrete Curve dividing 36 Edit joints 500 Edit cuts Fibres and Parts Fibre text zoom 1.0 H Edit named items 2D FEM analysis Use 2D FEM analysis 🛃 Calculation type Full FEM Mesh size [mm] 0 Min. point distance [mm] 0 Property Modificat... Edit shear and torsion pro Use property multipliers B 500 Export Update Document 4 2 Picture Þ Cancel ок Cross-section layout and dimensions

Set H as 500 mm and B as 500 mm and activate the option 'Use 2D FEM analysis' (in order to calculate the torsion and warping properties correctly). Click [OK] to confirm, the cross-section is now added to the cross-section list in your project. Add another rectangular cross-section with H = 700 mm and B = 450 mm in a similar way.

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



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

3.1.2 Geometry

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



Columns

To enter a new column, use the **Column** command in the **category 1D Members** of the **workstation Structure**.

Ξ	INPUT PANEL						
Ô	All workstations $$						
	1D Members $$						
Ø	All COLUMN						
B	- 17 - 20						

Via **ALT and right-mouse click** you can also use the **Marking menu** to quickly launch the Column command in the Model tab.

In the cross-section field, choose the first cross-section, CS1 - Rectangle (500; 500).

The column height is 7 m.

The **insertion point** is (as default) set to **Bottom** so that the bottom point determines the position of the column.

Column			>
. 107	Name B1		
	Type colo	umn (100)	۷
ey	Analysis model Sta	indard	v
	Cross-section CS1	1 - Rectangle (500; 500) 🛛 🗸 🗸	
(j)	Alpha [deg] 0.00	0	
<u> </u>	Member system-line at Cer	ntre	۷
	ey [mm] 0		
	ez [mm] 0		
	LCS sta	ndard	۷
	LCS Rotation [deg] 0.00	0	
	FEM type star	ndard	¥
	Layer Lay	yer1 v	
(i)	4 Buckling		
	System lengths and buckling settings. Def	fault	
	Secondary member		
	4 Geometry		
	Length [m] 7.00	00	- +
	Insertion point Bot	ttom	*
	Structural model		
		OK Canc	el

Confirm your input with [OK].

Now, you can enter the columns by activating the SCIA Spotlight (by clicking in it or using the Space bar) and entering as coordinates 0 0 0 (with space) or 0;0;0 (with semicolon) and confirming with <Enter>. Enter a second column with coordinates 6 0 0.



Press <**Esc**> to finish the input and press <**Esc**> once more to cancel the selection.

Notes:

- Instead of Column elements you can also work with the 1D Member command. A 1D Member is defined by a start and endpoint instead of a length and direction. Pay attention to the Type of the element (e.g. Column(100), Beam(80),... when you work with 1D members.
- The properties of selected elements are shown and can be modified in the **Property panel** on the right-hand side of the user interface.
- With zoom all (Main menu > View > Zoom > Zoom all), or double-click with the mouse wheel, you can visualize the entire structure.
- The use of , (comma) or . (point) to work with decimals depends on the regional settings of Windows.
- You can use the @-symbol to work with relative coordinates.
- You can also make use of Catalogue blocks (Input panel > workstation Structure > category Import & Blocks) to quickly model a frame.

Multicopy

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

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

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



Now, you can select the **Multicopy** command via **Main menu > Edit > Modify > Multicopy** or with the icon in the **workstation Structure** or with the icon in the Modify tab of the **Marking menu**. In the **Number of copies** field enter **2**. To manually set the distance between the frames, deactivate the **Define distance by cursor** option. Now, you can enter the distance 5 m in direction Y.

lulticopy			:
Number o	of copies	2 +	Connect selected nodes with new beams Copy additional data
DISTANCE	VECTOR		HOW TO DEFINE THE DISTANCE ?
Define dis x y z ROTATION	tance by cur 0.000 5 0.000	sor m m m	between two copies from original to the last copy HOW TO DEFINE THE ROTATION ? between two copies from original to the last copy ROTATION AROUND
rx ry rz	0.00 0.00 0.00	deg deg deg	current UCS distance vector

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

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



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

Snapping

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

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

		SNAP	PING	
LC1 - 1	Self we	ight \checkmark	P)	
Magnet Snapping	2	\bigcirc		
Tracking Guides	, pr		Ŧ	
Ortho	7	\odot		
Line grid	₩	\bigcirc		
Dot grid	: :	\bigcirc	E	
Endpoints / Nodes	3			
Midpoints / Centers	Ø			
Intersections	Ø	\bigcirc		
Normal & Tangent	.9:	\bigcirc		
Line divisions	1	σ		
Points on line	S	\bigcirc		
Surface edges		\bigcirc		
Solid vertices		\bigcirc		
Workplane	Lå	σ		
Offset	,. P	\bigcirc		
he				

Beams

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



In the cross-section field, choose the second cross-section, CS2 - Rectangle (700; 450).

There are two horizontal axes (i.e. X and Y), so you must indicate the right direction for the horizontal beam in the **Direct** field. Choose **axis X**.

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

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

	Name B7		
	Type bea	im (80)	*
α	Analysis model Sta	ndard	Y
	Cross-section CS2	! - Rectangle (700; 450)	×
	Alpha [deg] 0.00	D	
z' (J)	Member system-line at Cen	ıtre	Y
	ey [mm] 0		
	ez [mm] 0		
ez	LCS star	ndard	*
\times (1)	LCS Rotation [deg] 0.00	D	
	FEM type star	ndard	~
z	Layer Lay	er1	۰. ۲
A Buckling			
Y	System lengths and buckling settings Defa	ault	
	Secondary member		
▲ Geometry			
	Direct axis	s X	*
	Length [m] 6.00	00	
	Insertion point beg	jin	*
Structural mod	el		

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

Now, you can enter the beams by clicking with your mouse on the top node and the middle node of the lefthand side columns:



Press **<Esc>** to finish the input and press **<Esc>** once more to cancel the selection.

Enter the other beams with a length of 5 m in the Y direction in a similar way.



Press <**Esc**> to finish the input and press <**Esc**> once more to cancel the selection.

Plate

To enter a new plate, use the Plate command in the category 2D Members of the workstation Structure .

â	All workstations $$
-	2D Members $$
PLATE	All tags \vee

Via **ALT and right-mouse click** you can also use the **Marking menu** to quickly launch the Plate command in the Model tab.

You can specify the Material and define the thickness of the plate. We will use a thickness of 200 mm.

📧 2D member			×
	Name	\$1	
	Element type	Standard	*
	Element behaviour	Standard FEM	
	Туре	plate (111)	×
	Material	C25/30	¥
	FEM model	Isotropic	*
	FEM nonlinear model	none	*
ez	Thickness type	constant	
·∠	Thickness [mm]	200	
-	Member system-plane at	Centre	*
	Eccentricity z [mm]	0	
Z	LCS type	Standard	*
	Swap orientation	no	
x y	LCS angle [deg]	0.00	
	Layer	Layer1	v
Þ	Structural model		
			UN Cancel

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

Draw the plate at the middle height and on top of the structure.



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

Notes:

• When entering a 2D element in the graphical scene, below the SCIA Spotlight icons which can help you with inputting the element will appear. The icon **Select line** can be useful if you want to make use of lines which are already available in your model.

New 2D member - New polygon - Start point >



• Various 2D member components such as **subregions** or **openings**, ... can be added to 2D members.



3.1.3 Additional data

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

Supports

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

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

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



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

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

	1			
	≡	NOD	E (1)	2
	3			
		Name	N3	
	▼ GCS COORDI	NATE		
		X [m]	6.000	
		Y [m]	0.000	
		Z [m]	0.000	
	Isolate selection			_
1	Expand selection			
E	Subtract from sele	ction		
		uz [m]	0.000	
	▼ MEMBERS			-
		Member	82	

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



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

To enter supports, you can use the **Support in node** command in the **category Boundary Conditions** of the **workstation Structure** (or via the marking menu). Afterwards you can choose one of the predefined Constraints (Fixed, Hinged, Sliding) or you can define all 6 end conditions one by one. You can find the predefined constraints as well as icon in the Input panel. In our project we will use the **Fixed support in node**:

=	INPUT PANE	L
	All workstations \smallsetminus	
-	Boundary conditions $$	
0	All tags 🗸	FIXED SUPPORT IN NOT
*	7 7 7 7 8	A A 9 b

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

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

Hinges

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

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

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



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

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

Hinge on beam				×
	Name	H1		
\$ ^φ ^z	Position	Both		~
T	ux	Rigid		~
(i) UZ	uy	Rigid		*
UX UV	uz	Rigid		*
φχ	fix	Rigid		۷
	fiy	Free		¥
	fiz	Free		~
			ОК	Cancel

Confirm your input with [OK].

The hinges are added when you select the beams. You can click on front view of the Navicube and select the beams by drawing a frame from the left to the right with the mouse cursor.



Press < Esc> to finish the input and press < Esc> once more to finish the selection.

Notes:

- Hinges are always defined with regard to the local coordinate system of a member.
- Increase Scale add data in the Status bar if you feel that the hinge symbol is too small.



• As alternative you can use the quick input icon Hinges at both ends of beam.

3.1.4 Check structure

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

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

R+	Check of structure data					×
	CHECK OF NODES					
6 *	Search nodes					
₽	Search duplicate poder			Ignore parame	ators	0%
AN I	Jearen dupicate nodes			- Ignore parame		0%
18	CHECK OF MEMBERS					
	Check members				0	
H_	Search null members		0%	Null members:	mbers	
	Search duplicate members				0	
			0%	Delete duplicat	te members	
~				Delete invalid p	parts	
	CHECK OF DATA REFERENCES					
	Check data references			Memory efficient	nt method	
每			0%	 Fast method 		
I	CHECK OF ADDITIONAL DATA				_	
	Check additional data positi	on	0%		0	
			0,0	le l'ille de	0	
		points	0%			
t <u>i</u>	CHECK OF STEEL CONNECTION	S				
	Check steel connections			Invalid	0	
			0%	I Delete invalid	connections	
	Check load panels	Check	cross-lin	iks		
	Check additional data	Check dup	licity of	names	Check	Cancel

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

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



Close the check by clicking [OK].

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

3.1.5 Graphic representation of the structure

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

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

You can set the view point with the Navicube.



You can rotate/tumble, pan, zoom, set the view in the direction of one of the global axes,... All possibilities are shown on following help-page:

https://help.scia.net/webhelplatest/en/#modelling/basics/basictools/adjusting_the_view.htm

Notes:

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

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

۲	🛱 🕰			Please click here
00	Undo view change Redo view change			
	Visibility			
	Zoom		Zoom in	
	Views		Zoom out	
	Clippingbox	0	Zoom cutout	
	Visualization	1	Zoom all	
	Global UI settings	3	Zoom to selection	
	User configuration			
	Colours & lines			
با	Line grid manager Ctrl+Shift+C	5		
	Dot grid settings			
ø	Wired model in view manipulations			

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

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

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



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

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

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

- Name: show the name of the member in the label (e.g. B1)
- Cross-section name: show the name of the cross-section in the label (e.g. CS1)
- Cross-section type: show the cross-section type in the label (e.g. Rectangle (500; 500)).
- Length: show the length of the member in the label (e.g. 5.000 m).

View parameters setting - Stru	ucture					1	View pa	arameters se	tting - Labe	ls				
				Lock positio	n								Lock position	n (
d TT Structure AP Label		onde/masses	Concrete	Composito	MARI	1	a m	Structure	AP Labele	Model	Londs/masses	Concrete	Composito	Ne Ma b
Charle (Uncharle all		iouus, musses	(in concrete	T composite	enter inter inter			di Alteria		(iii) model	(<u>v</u>) couds) masses	(in concrete	T composite	ACCESS 11100
Check / Uncheck all	1					114	Chee	CK / Uncheck	all					
	-						Ser	rvice	the second second					
Display on opening the se	v						Dis	splay on oper	ning the se	v				
Structure							Bea	am labels						
Style + colour	normal				-		Dis	splay label		-				
Draw member system line	V						Nar	ime		•				
Member system line style	system line				-		Cro	oss-section n	ame					
Model type	analysis model				<u>•</u>		Cro	oss-section ty	pe					
Display both models							Len	ngth						
Member surface							Lay	yer		_				
Rendering	wired				-		Тур	pe and priorit	y					
Draw cross-section							- No	odes labels		_				
Cross-section style	section				<u>•</u>		Dis	splay label						
Effective width of plate ril	_						Nar	ime		~				
Draw effective width	~						X-0	coordinate						
Rendering	transparent				•		Y-co	coordinate						
Panel							Z-0	coordinate						
Member surface							🖃 Slai	ab members						
Rendering	wired				•		Dis	splay label						
Highlight supporting edg							Nar	ime		~				
Load distribution symbol	▼						Lay	yer						
Display linked members	✓						Тур	pe and priorit	У					
Structure nodes							Edg	iges						
Display	~						FEN	M model						
Mark style	Dot				•		Ma	aterial						
Member parameters							Pro	ofiled sheetin	g					
System lengths							- Sys	stem lengths						
Member nonlinearities	v						Dis	splay label		~				
FEM type	~						Nar	ime		~				
Joists							Lab	bel						
E Local axes								onlinearities						
Nodes							Die	splay label						
Members 1D								hels of local a	ixes					
Members 2D	-						No	dec						
memoers ab							Me	embers 1D						
	1						Me	embers 2D						
							Ger	meral structu	ral chane					
							Gei Die	colay vertex la	an anape	_				
								opidy remex to	in set					
_						115								
Show names in tab			ок	Apply	Cancel		Sho	ow names in	tab			ок	Apply	Cancel

Chapter 4: Loads and combinations

4.1 Load cases and load groups

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

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

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

Three load cases are entered in this project:

- LC1: permanent load case Self weight;
- LC2: permanent load case Permanent (for other permanent loads);
- LC2: variable load case: Variable (for variable loads such as Wind, Snow,...).

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



Before you can define loads, you must enter load cases first. Since this project does not contain any load cases yet, the **Load Cases** manager will automatically appear and by default, the load case LC1 is automatically created. This load is a permanent load of the **Self weight** load type. The self weight of the structure is automatically calculated, although not graphically displayed.

Load cases			2	×
et -: 🖸 🕩 🗟 🖷	s 🖉 🗖 🕒 🖸	All	× T	
LC1 - Self weight	Name	LC1		
	Description	Self weight		
	Action type	Permanent		٧
	Load group	LG1	*	
	Load type	Self weight		٧
	Direction	-Z		٧
New Insert Edit	Delete		Close	e

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

Load cases						×
et -: 🖸 🕩 🛢 🖷	۵		All	٧	T	
LC1 - Self weight		Name	LC2			
LC2 - Permanent		Description	Permanent			
		Action type	Permanent			٧
		Load group	LG1		٧	
		Load type	Standard			٧

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

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



The **Load type** determines the composition factors (psi-factors) that are attributed to the load cases in this load group. There are different categories available, as well as Snow, Wind, ... In this project choose **Cat A: Domestic**.

Load groups			×
📑 📲 🗹 🕩 🗟 🐟 /	ar 🗖 📄 🖸		
LG2	Nam	ie LG2	
	Relatio	n Standard	*
	Loa	d Variable	
	Structu	e Building	
	Load typ	e Cat A : Domestic	*

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

Click [Close] to close the load cases manager.

<u>Note</u>: each load is classified in a load group. These groups influence the combinations that are generated as well as the code-dependant coefficients to be applied. The following logic is adopted. Variable load cases that are independent from each other are associated to different variable groups.

For each group, you set the load category (see EN 1991). The combination factors from the Eurocode are generated from the available load groups. When a generated combination contains two load cases belonging to different groups, reduction factors will be applied for the transient loads.

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

4.2 Loads

In the **status bar** you can change the active load case. Loads cannot be added if the load case Self weight is the active one. Activate load case LC2.

	ACTIVE LOAD CASE										
*	LC2 -	Permanent V	2	2U	, m .	C	Ma	# <mark>6</mark>	(O)	0	പ്പ
1 LC1 - S	elf weight	63									
LC2 - P	ermanent										
1 LC3 - Va	riable										
🚍 Manag	e load cases	Ctrl+L									

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

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

Set Value [kN/m^2] to -2.50 in the properties dialogue. The Direction is the global Z direction and the value is negative, so the load is acting vertically downwards.

Surface force				×
	Name	SF1		
	Direction	Z		٣
	Туре	Force		٣
	Value [kN/m^2]	-2.50		
	4 Geometry			
	System	LCS		*
	Location	Length		
IZ				
· · ·				
			ОК	Cancel

Confirm your action with [OK].

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

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

The loads are coloured in green.



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

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

ΠΞ	INPUT PANEL
6	Loads 🗸
	Line loads $$
	ON 1D 55 🗸
册	4 🖽 🍅 🛋 🖽

Set Value - P [kN/m] to -1.00 in the properties dialogue. The load Direction is Z and the System is the local coordinate system LCS. The linear loads are acting in accordance with the local Z-axes of the elements.

Line force on beam				×
R7)		Name	LF1	
		Direction	Z	*
Try I-FZ		Туре	Force	*
		Angle [deg]		
-P1 (j)		Distribution	Uniform	*
Gen Street		Value - P [kN/m]	-1.00	
ey ez		Load above joint	no	
	Geometry			
x1 x2		System	LCS	*
	-	Location	Length	
		Extent	full	*
		Coord. definition	Rela	*
		Position x1	0.000	
		Position x2	1.000	
		Origin	From start	*
	 Eccentricity 			
		Eccentricity ey [m]	0.000	
		Eccentricity ez [m]	0.000	
				OK Cancel

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

Select the beams at the first level.

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



Notes:

- Loads are considered as additional data (as well as supports, hinges, ...), so they can be copied or move via Main menu > Edit > Add data or by selecting the entity and using the context menu (right-clicking in the graphical scene).
- If you want to load only a part of a 2D member with a surface load or if you want to use non-uniform surface loads, you can make use of the Free surface load command. If you want to add a line load somewhere on the 2D member, you can make use of the Free line load command. You can go through a tutorial about this in SCIA Engineer via Main menu > Help > Tutorials > Free loads.

4.3 Combinations

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

Basic data	Functionality Actions Unit Set Protection	
	Acceleration of gravity9.810m/s^2	
	WIND LOAD	
	According to code	
	SNOW LOAD	
	According to code	
	SEISMIC COMBINATIONS	
	Factor for concomitant components 0,30	
	CODE COMBINATIONS	
	Automatic	

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



Three code combinations are created, one for the Ultimate Limit State and two for the Serviceability Limit State (characteristic and quasi-permanent).

Combinations			×
📑 📲 🗹 🕩 🗟 🔶	🗙 🗢 🔲 Input combina	ations 👻	
ULS-Set B (auto)	Name	ULS-Set B (auto)	
SLS-Char (auto)	Description		
SLS-Quasi (auto)	Туре	EN-ULS (STR/GEO) Set B	
	Updated automatically	2	
	Structure	Building	
	Active coefficients		
	Contents of comb		
	LC1 - Self weight [-]	1.000	
	LC2 - Permanent [-]	1.000	
	LC3 - Variable [-]	1.000	
	Actions		
		Explode to envelopes	>>>
		Explode to linear	>>>
	Sho	w Decomposed EN combinations	>>>
New Insert Edi	Delete		Close

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

Chapter 5: Analysis

5.1 Linear analysis

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

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



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

FE analysis	>
Calculations	 Mesh setup
Calculations Linear analysis Load cases: 3 Other processes Save project after analysis	 Mesh setup Average number of 1D mesh elements 1 Average size of 1D mesh element on cu 0,200 Average size of 2D mesh element [m] 0,500 Connect members/nodes ✓ Setup for connection of structural entit Advanced mesh settings Solver setup Specify load cases for linear calculatior
Calculate	

Click on the three dots next to **Setup for connection of structural entities** to open the dialogue with the settings. Activate the option **Connect 1D members as ribs**. This option will connect the plate and the beams as a rib.

	 Align structural entities to planes (moving nodes) 	
	Align	
	Geometrical tolerance	
Long	Min. distance of two nodes, node to curve [m] 0.00	1
	Max. distance of node to 2D member plane [m] 0.00	D
	🖌 🤞 Connect (generate linked nodes, intersections, internal	
	Connect 🛃	
	Autorestore buckling group when reconnecting 🔽	
	Connect 1D members as ribs 🔽	
	Connect 1D members with rigid links	
2	Max. length of rigid link [m] 0.10	D
Ky	Create new linked node for master node 📝	
	4 Check structure data	
	Check (merge duplicate nodes, erase invalid entities)	

Confirm with the [OK] button to close the dialogue.

The mesh size of 2D mesh elements is an important parameter. In the mesh settings you will find an option to set the average size of 2D mesh element. As a first estimation you could set the value for the mesh size as 1-1.5 times the thickness of the 2D elements, so we could start with a value of 300 mm.

FE analysis		×
Calculations	 Mesh setup 	
	Average number of 1D mesh elements (1	
Linear analysis	Average size of 1D mesh element on cu 0,200	
Load cases. 5	Average size of 2D mesh element [m] 0,300	
Other processes	Connect members/nodes 🔽	
Save project after analysis	Setup for connection of structural entit	
	Advanced mesh settings	
	 Solver setup 	
	Specify load cases for linear calculation	
	Specify combinations for linear stability	
	Specify combinations for nonlinear sta	
	Advanced solver settings	

Select Linear analysis in the analysis dialogue and press Calculate.

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

CALCULATION PROCESS	
Calculation Results	
Calculation finished successfully	100% 🧭 Cancel
# Name	Progress
 Mesh calculation Linear analysis 	 Finished Finished
	Close
CALCULATION PROCESS	
Messages Preparation for mesh generation: OK Preparation of calculation settings: OK Mesh generation: OK Mumber of nodes: 1504 Number of 1D elements: 1360 Number of 1D elements: 296 Linear analysis: OK Maximal translation -0,7 mm in node 925 [3.000,2.647,7.000] (load case LC1) Maximal translation -0,7 mrad in node 798 [5.100,2.353,7.000] (load case LC1) Sum of loads and reactions is OK	
	Close

Notes:

• You can adjust the mesh locally by using local mesh refinements such as **Node mesh refinement** (refines the around a single node), **Edge mesh refinement** (refines the mesh along a particular edge or internal line of a 2D member) and **Surface mesh refinement** (refines the mesh over the whole surface of a 2D member).

	INPUT PANEL
â	Calculation & Results $$
÷	Local mesh refinement $$
0	All tags 🗸
	×

• The mesh can be displayed via Draw mesh checkbox in the view settings dialog.

Vie	w parameters set	ting			
C	heck / Uncheck g	roup		Lock pos	ition
٩	T Structure	😬 Labels	📥 Model	🛃 Loads/mass	es 🛛 📴 Co 🕩 e
	Check / Uncheck a	all			
Ξ	Mesh				~
	Draw mesh	V			
1.1	Free edges				
	Display mode	wired			•
	Local axes				
	Nodes				
	Members 1D				
	Members 2D				
	Mesh elements				~
	Show names		ОК	Apply	Cancel

Chapter 6: Results

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

6.1 Reactions

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



Go to the property panel and set:

- Type of selection to All
- Type of load to Combinations
- Combination to ULS-Set B (auto)
- Extreme to Mesh
- Values to R_z

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



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

System: C	on: ULS-Set B	(auto)							
Extreme: I	Mesh								
Nodal rea	actions								
Name	Case	R _x [kN]	R _y [kN]	R₂ [kN]	Mx [kNm]	My [kNm]	Mz [kNm]	e _x [mm]	e _y [mm]
Sn1/N18	ULS-Set B (auto)/1	-4.55	3.49	228.70	-3.92	-4.92	-0.61	21.5	-17.1
Sn1/N18	ULS-Set B	-6.44	4.88	316.79	-5,48	-6.96	-0.84	22.0	-17.3
Sn2/N1	ULS-Set B	6.44	4.88	316.79	- <mark>5.4</mark> 8	6.96	0.84	-22.0	-17.3
Sn2/N1	ULS-Set B	4.55	3.49	228.70	-3.92	4.92	0.61	-21.5	-17 <mark>.</mark> 1
Sn3/N3	ULS-Set B	10.58	0.00	564.87	0.00	11.76	0.00	-20.8	0.0
Sn3/N3	ULS-Set B	7.63	0.00	409.22	0.00	8.47	0.00	-20.7	0.0
Sn4/N5	ULS-Set B	4.55	-3.49	228.70	3.92	4.92	-0.61	-21.5	17.1
Sn4/N5	ULS-Set B	6.44	-4.88	316,79	5.48	6.96	-0.84	-22.0	17.3
Sn5/N16	ULS-Set B	-6.44	-4.88	316.79	5,48	-6.96	0.84	22.0	17.3
Sn5/N16	(auto)/2 ULS-Set B	- 4 .55	-3.49	228.70	3.92	- <mark>4.9</mark> 2	0.61	21.5	17.1
Sn6/N17	ULS-Set B	-7.63	0.00	409.22	0.00	-8,47	0.00	20.7	0.0
Sn6/N17	ULS-Set B	-10.58	0.00	564.87	0.00	-11.76	0.00	20.8	0.0
	(auto)/2				_		8		
N P	ime	COL	ndinatio	n key					

6.2 Internal forces

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



Go to the property panel and set:

- Type of selection to All
- Filter to Type of beam
- Type of beam to Column
- Type of load to Combinations
- Combination to ULS-Set B (auto)
- Extreme 1D to Global
- Values to N

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



You can play with the **Selection** settings in order to show results on all 1D elements or only on certain elements that you have selected, ...

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

▼ DRAWING SETUP 1D	
Display value name	\bigcirc
Display values	
Display units	0
Display case	0
Display section dx	\bigcirc
Display combination	0
Display combination	0
Color scheme	Defined by result \vee
Graph type	Filled transparent 🗸
Envelopes drawing	0 to extreme $ \smallsetminus $
Label colour by graph	
Drawing plane	3D 🗸
Label orientation	Perpendicular to axis

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

UN	IIT SYSTEM				V		
LC1 - Self weight 🗸 📋	2U [m	Ø	Mì	₿	0	0	പ
Unit system Metric	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~						
Unit Metre	\sim						
Manage units							

It's also possible to show results on the 2D elements via the command **2D internal forces** in the **workstation Results** in the **process toolbar**.



Below you can find the m_x result for the 2D elements for combination ULS-Set B (option Rib is activated).



Note that in the properties panel you have the option '**Rib**', which can be activated or deactivated. We recommend to use the same setting for both 1D and 2D results, otherwise you are neglecting forces or counting them twice. More information on ribs (e.g. information on the effective width, ...) can be find in our FAQs or in our Concrete manual.

To change the display of the results, the settings of the graphical scene can be adapted via the options in the group **Drawing setup 2D** action button at the bottom of the property panel. You can display isobands (and define the legend via Advanced settings), set minimum and maximum values, ... The font size (e.g. when numbers are shown) can be changed via Main menu > View > **Fonts settings**.

6.3 3D deformations

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



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

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

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

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



6.4 Results-related status bar options

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

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



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



Chapter 7: Reinforcement design

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

- input of concrete data per member;
- input and manipulation of buckling data and calculation of the slenderness;
- design of theoretically needed amount of reinforcement;
- input of (practical) reinforcement;
- performing ULS unity checks (capacity-response, ...);
- performing SLS unity checks (stress limitation, crack width, ...);

We will only explain the basics of the concrete design in this tutorial. For more information regarding concrete calculations, we refer to the Advanced Concrete Training and to the tutorials **Concrete 1D** reinforcement and **Concrete 2D reinforcement** via **Main menu > Help > Tutorials**.

7.1 Concrete settings

It's very important to know that all design assumptions are taken from the Concrete Settings (Main menu > Design > Concrete settings > Settings).

7.2 Buckling parameters

Before you can start the concrete design you must first check the buckling parameters of the 1D elements. By means of the view settings, the buckling lengths of the members can be visualized. Select member B1 and activate the option **System lengths** under **Member parameters** in the tab **Structure** of the **View settings**.

Vie	w parameters s	etting										
							Lock	position		7		1
1	Structure	AB Labels	Moe Moe	delling/Drawing	🚱 Attr	ibutes	Misc.		Þ			
	Check / Uncheck	all										
Ξ	Structure											4 6
	Member surfac	e										1 8
	Rendering			wired					-			4 B
	Draw cross-sec	tion										1 8
	Cross-section s	tyle		section					-			
Ξ	Effective width	of plate ribs										1 1
	Draw effective	width		✓								1 1
	Rendering			transparent					-			
Ξ	Member paran	neters								N		X
	System lengths										·	
	Member nonlin	earities										
	FEM type									-		1. 6
	Joists											
Ξ	Local axes									2		4 B
	Members 1D									0		
												10
										o o		
										X	-	X
	Show names in	tab				ок	Apply	/ Ca	ncel		4	

The figure above shows that the (default) system/reference length Ly (blue line) for buckling around the strong axis (y-y) is half of the height of the column and Lz (green line) for buckling around the weak axis (z-z) is half of the height. The beams and plate in the middle of the column therefore supports the column for buckling around the strong and weak axis.

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



On the Settings tab, several data can be changed:

- The Name field contains the name of the buckling parameter, in this case BG1.
- For buckling around y-y and z-z you can set the **Beta yy** and **Beta zz factor**: you can indicate that the program should calculate the buckling factor automatically or you can choose a manual input of this factor. A third option allows for a manual input of the buckling length (instead of factor). With the option Support the buckling factor is obtained based on the inputted supports.
- You can set as well Sway y-y and Sway z-z: you can indicate if the member is "braced" or not in the direction regarded. When you choose option From setup, the default settings are used.
 <u>Note</u>: the default settings for the buckling parameters are listed in Concrete settings > Solver setting > General > Default sway type. The structure is by default sway for buckling around the strong axis (in the plane) and sway for buckling around the weak axis out of plane).



• You can define as well settings for both deflection directions (used for the SLS check). By default settings are taken from y-y or z-z, but this can be changed with the drop-down lists.

Select the command Slenderness for design via Main menu > Design > Concrete 1D.

🞯 👁 🛱 🕰		Please clic
Concrete settings	+	
Concrete 1D	•	m Internal forces for design
Concrete 2D	•	λ Slenderness for design
ILS punching		D minforcement design

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

Then press **Refresh** to see the result for the reference lengths.

RESULTS	(1)	×		
Name	Slenderness(Desig	gn)		
SELECTION				
Type of selection	Current	\sim		
Filter	No	\sim		
Results in sections	All	\sim		
RESULT CASE				
Type of load	Combinations	\sim		
Combination	ULS-Set B (auto)	\sim		
Rib				
EXTREME 1D		_		
Extreme 1D	Global	\sim		
Values	L	\sim	3.500 m	
Interval	0			Q

You can ask as well for the slenderness (λ), buckling factor (β), buckling length (I), ... The buckling length is determined as the reference length multiplied by the buckling factor.

As output you can choose for Brief, Standard or Detailed. Press Report preview to display the output.

▼ OUTPUT SETTINGS		
Output	Detailed	Y
DRAWING SETUP 1D		
ERRORS, WARNINGS A	ND NOTES SETTINGS	
Run using Model Data	0	
► ILLUSTRATION GROUP	ŝ.	
ACTIONS >>>		
C Refresh		F5
New combination fro	om Combination key	
Results table		
Report preview		

The slenderness is important for reinforcement calculation in columns since it will determine whether a second order moment has to be taken into account or not.

7.3 Internal forces

Internal forces in the Results workstation are purely values obtained by finite element analysis. For concrete design we need to recalculate these forces into design values, using various code regulations. Those internal forces can be shown via Main menu > Design > Concrete 1D > Internal forces for design and Main menu > Design > Concrete 2D > Internal forces.

Finite element analysis often gives peak values in places of singularities (e.g. local node support, plate edge, ...). Averaging strips can help handle that and lower the unrealistic peak values.



7.4 1D reinforcement design

Select the command **1D reinforcement design** in workstation Concrete of the process toolbar:



First we can ask for the theoretically required reinforcement.

Go to the property panel and set:

- Type of selection to All
- Filter to Type of beam
- Type of beam to Column
- Type of load to Combinations
- Combination to ULS-Set B (auto)
- Extreme 1D to Global
- Type of values to Required
- Values to As,req

Then press **Refresh** to see the results.



It is also possible to show the provided reinforcement (Type of values = Provided). This will give you a proposal based on the **Design defaults** you have set for the column in the **Concrete settings**.



In the property panel you can set the **Output settings** as **Detailed** to get a detailed output of the result in the **Report preview**.

You can convert provided reinforcement directly into real bars (= practical or user reinforcement) via the action **Conversion for real bars** at the bottom of the property panel:

F5
- *

Or you can add the practical reinforcement manually via 1D reinforcement in workstation Concrete of the process toolbar (or via the input panel).



7.5 2D reinforcement design

You can check the required and provided reinforcement for plates using the same logic as we saw previously.

Select the command ULS & SLS 2D reinforcement design in workstation Concrete of the process toolbar:



With this command you can perform ULS and/or SLS design for 2D elements.

First we can ask again for the theoretically required reinforcement.

- Go to the property panel and set:
 - Type of selection to All
 - Type of load to Combinations
 - Combination to ULS-Set B (auto)
 - Type of values to Required
 - Values to As,req,1- (to show the bottom reinforcement in the first direction = parallel to the local x axis of the 2D member)



In the view settings you can activate the Drawing directions for design, so it's clear what is direction 1+, 1-, 2+ and 2-:

Vi	ew parameters setting		
	Check / Uncheck group		
٩	🔲 Structure 斗 🛤 Labels 🔺 Mode	l 🛃 Loads/masses 🛛 📅 Concrete	
	Check / Uncheck all		
E	Service		
	Display on opening the service		
E	Concrete + reinforcement		
	Display	V	
	Member data		
	SaT detail data	V	
	Drawing directions for design		
E	Concrete labels		
	Display label	V	
	Name	V	
	User defined reinforcement		
	Provided reinforcement		è
	Diameter and number of bars/spacing		
	Material		
	Cover		*
	Environment class		l and the second se
~	Show names in tab		क्षू

If you want to perform a ULS and SLS design together, you'll need to select a class containing at least one ULS and one SLS combination.

The provided reinforcement will be shown based on the design defaults for a plate in the concrete settings.



You can add practical or user reinforcement to the 2D member(s) via 2D reinforcement in workstation Concrete of the process toolbar (or via the input panel).



You can execute the reinforcement design including the user reinforcement with the option Consider user reinforcement in the property panel.

Type of values	Provided & User \vee
Values	As,prov,1+ V
Consider user reinforcement	

SCIA Engineer provides also an option to execute the punching design.

7.6 Concrete checks

There are multiple concrete checks available.

For checks on 1D members user reinforcement is needed on the elements. Multiple ULS (capacity-response, capacity-diagram and shear+torsion) and SLS (stress limitation, crack width and deflection) checks are available, as well as a check for the detailing provisions. Via main menu > Design > Concrete 1D > **Overall check** you can evaluate all checks at once. For a detailed output of each individual check you can execute the checks one by one as well.



For 2D members you can execute a bending capacity check and a crack width check. It can be executed based on required reinforcement, provided reinforcement or user reinforcement.

Concrete settings		
Concrete 1D		
Concrete 2D	•	Internal forces for design
Explode reinforcement to free bars		Reinforcement design
Code dependent deflection		Internal forces for checks
Bill of reinforcement		ULS bending capacity check
External tools	•	TULS punching shear check
Check manager		SLS crack width check
Update check data		

Finally, you can execute the code dependent deflection check with the required reinforcement, provided reinforcement or user reinforcement. This check will take into account long-term effects such as creep and shrinkage.



Chapter 8: Report

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

8.1 Engineering report

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



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



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

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

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

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

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



8.1.1 Adding images to the report

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

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

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



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

Picture in scale - Insert objects	to Engineering report Inbox	
Insert Insert	3 ,4 ,Save = erties	
Caption	1 LC2 / Tot. value	
Picture size definition	1 Two per page	~
Automatic scale to fit siz	e 🗹	
Scale 1	: 158.207031914881	
Stretch mod	e Dark lines	*
Renderin	s Standard	~
Antialiasing qualit	/ None	Y
Rotatio	n None	*
Result information	n Inside picture	¥
Export to PDF as 31		
Positio	0 One below another	Y
Image raste	r 📃	
Scale for model data	1	
Load units in regen. (related to objects created in picture editor only) 🛃	
Load activity in reger	4 🔲	
Draw inactive member	s as is in the window	*
Text scale facto	r 1	
Charset of text	5 Western European, UK, USA (Windows-1252)	~
Line pattern lengt	n 3	*
Display GCS ico	1 To picture corner	*
4 Performance		
Set as non-editabl	e	>>>
Setting	5	>>>

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



8.1.2 Displaying results

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

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

Combinations	0 •	Timber Concrete Steel concrete bridge	4. Results	
🔛 Results (Chapter)	0	Geotechnics	4.1. 3D displacement	
🌄 3D displacement	@	Composite	Regeneration is needed	
🧮 Reactions	£0	Integrated Design Forms		

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

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

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



The red exclamation mark disappears.



4. Results 4.1. 3D displacement

4.1. 3D displacement Unear calculation Class: All SLS Selection: All Location: In nodes avg. on macro. System: LCS mesh element Results on 1D member: Extrem 10: Global

Name	dx [m]	Fibre	Case	u _x [mm]	Uy [mm]	uz [mm]	φx (mrad)	φ _γ [mrad]	φz [mrad]	U _{total} [mm]
B1	0.000	1	SLS-Char (auto)/1	0.0	0.0	0.0	0.0	0.0	0.0	0.0
87	3.000-	1	SLS-Char (auto)/2	0.0	0.0	-0.6	0.0	0.0	0.0	0.6

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

Navigator	Ψ×
Page format	0
Header / Footer	0
Table of contents	0
v 📄 Geometry (Chapter)	0
∏ Analysis model (Picture in scale)	^ 0
III Materials	^ 0
Cross-sections	^ 0
The second	^ 0
III Members	^ 0
📅 2D members	0
🔠 Hinges	0
🎹 Nodal supports	0
📖 🧮 Page break	0
🕫 🔤 Loads (Chapter)	0
	0
🔽 Image (Picture in scale)	0
Page break	0
Combinations	0
v 🔤 Results (Chapter)	0
📰 3D displacement	0
Reactions	0
🕫 🗁 Concrete design (Chapter)	0
🖙 🗁 1D members (Chapter)	0
Internal forces (Design) (Internal forces)	• •
🛗 Overall Design (ULS) (Reinforcement design)	0
🛗 Check capacity-response (Capacity-response (ULS))	• •
Check capacity-interaction diagram (Capacity-diagram (ULS))	• •
····· 🗮 Check shear+torsion (ULS) (Shear + Torsion (ULS)))	• •
🛗 Check Stress limitation (Stress limitation (SLS))	• 0
Check crack width (SLS) (Crack width (SLS))	• •
····· 🗮 Check deflection (Deflection (SLS))	• •
🦾 🛗 Check Detailing provisions (Detailing provisions)	^ 0
	0
Internal forces 2D (Internal forces)	• •
Reinforcement design (ULS+SLS)	0
Punching design	0
Crack width (SLS)	0
🦾 🛅 Code dependent deflection (CDD deflection)	•••

8.1.3 Printing or exporting the report

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

$\overline{\mathbf{e}}$	Report_1 (TUT (Eng) SCN21.1 - Tutorial Concrete.esa) - Engineering report – Ø	×
Print	Print Codes 1 [2]	
	Printer	
	Continue Control Contro Control Control Control Control Control Control Contr	
	PointIngs Common Com	
	From 1 To: 15 13.1 torizon:10 B3.1 torizon:10 13.1 torizon:10 13.1 torizon:10 B1.1 torizon:10 13.1 torizon:10 14.1 torizon:10 B1.1 torizon:10 13.1 torizon:10 14.1 torizon:10 B1.1 torizon:10 14.1 torizon:10 14.1 torizon:10 B1.1 torizon:10 14.1 torizon:10 14.1 torizon:10	
	Pertual Orientation Pertual Orientation Letter	
	2 1.99 mx 27.94 cm 11 0.04 the state state Endered picturer with high quality 11 0.00 the state state	
	132 Goodward 60 9 N. To magnetic whole 9	
	1 Parei 3	

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