



TUTORIAL CONCRETE 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.

le of contents	
General Information	1
Welcome	1
SCIA Engineer Support	1
Websites	1
Introduction	2
Getting started	3
Starting a project	3
Project management	5
Save, Save as, Close and Open	5
Saving a project	5
Closing a project	5
Opening a project	5
Start project manger	5
Geometry input	6
Input of the geometry	6
Profiles	6
Geometry	7
Hinges	11
Supports	12
Check Structure data	
Checking the structure	14
Connecting entities	
Graphic representation of the structure	
Loads and combinations	23
Load Cases and Load Groups	
LOADS	
Calculation	
Linear Calculation	32
Results	
Viewing results	33
Code check	
Buckling parameters	38
Displaying the system lengths	30 ຊຂ
Setting the Buckling Parameters	30 20
Concrete design	
Displaying the Slenderness and the Buckling Lengths	
Internal forces for reinforcement design	45
Theoretically required reinforcement.	45
Reinforcement check (ULS)	
Practical reinforcement	50
Reinforcement check (SLS)	
Detailing provisions	60
Document	63
Engineering Report	63

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 2D concrete frame.

Before you start, you need to be acquainted with the use of your operating system, e.g. working with dialogs (windows), menu bars, toolbars, status lines, the mouse, etc.

First, we will explain how to create a new project and how to model the structure. After the input of the geometry and the loads, the structure will be calculated and you will be able to view the results. Next, the input of the buckling parameters is discussed and then we will proceed with the concrete design and checks. Finally, we will teach you how to make a nice documentation.

The figure below shows the analysis model of the structure that will be modelled:



Starting a project

Starting the program

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

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

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

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

Starting a new project



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

2. You can also start new project with an icon 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.

Project data							×
Basic data Fun	ctionality Actio	ns Unit Set	Protection				
	Data				Material		
	Name:				Concrete		וור
ARK D					Steel		
HAR.	Part:	-			Masonry		
7 HT					Aluminium		
THE	Description:	-			Timber		
TTTT -					Steel fibre conc		
	Author:	SCIA			Other		
Since P	Date:	17. 05. 201	8				
	Structure:		Post processing environment		Code National Code:	*	
	💭 Frame XZ	*	🚳 v17	Ŧ	LO PEN		·
	Model:				National annex:		
	🛛 One	Ŧ			Standard EN	l ▼	
					0	(Can	cel

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

Below the item **Concrete** you can choose your preferred concrete grade (select C25/30 here) and reinforcement material (select B 500B here).

Material is the only required setting to proceed

- 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 📕 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 cross-section	×
Available groups Concrete Geometric shapes Numerical General Precast Bridge	Available items of this group	
Rectangle	Profile Library filter V Add Close	

- 2. Click **Concrete** in the left frame **Available groups**.
- 3. In the Available items of this group, you can choose a rectangular section [
- 4. Click **[Add]** or it to add the profile to the project. The **Cross-section** window appears.

Cross-	section	>
	Name CS1	^
7	Type Rectangle	
Z	Detailed 500; 300	
	Shape type Thick-walled	
	Parameters	
	Material C12/15	
	H [mm] 500	
	B [mm] 300	
	🗆 General	
	Draw colour Normal colour	
	Colour	
	AutoDesign constraints	
	Fabrication concrete 💌	
N		
	Curve dividing 36	
	Edit joints	
	Edit cuts	
D 200	Fibres and Parts	
B 300	Fibre text zoom 1.0	~
d	Export Update Document	
Cross-section layout and dimensions	OK	

- 5. In this window, you can change the properties of the rectangular section. Enter **500 mm** for height **H** and **500 mm** for width **B**.
- 6. Click **[OK]** to confirm, the profile is added to the **Items in project** frame.
- 7. A second rectangular section with height **H** = 700 mm and width **B** = 450 mm is added in a similar way.
- 8. Click [Close] in the New Cross-Section window, the Cross-Sections Manager appears.
- 9. Click [Close] to close the Cross-Section 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 frame you first must enter the columns. Next, the beams on top and in the middle of the columns are added.

Entering a column

1. Use the **Column** command in the **Structure** service to enter a new column.

	Column		×
1	Name	B5	
ez	Туре	column (100)	*
	Analysis model	Standard	-
~	CrossSection	CS1 - Rectangle (500; 500)	÷
	Alpha	0	*
	Member system-line at	Centre	
(\mathbf{j})	ez [mm]	0	
	LCS	standard	*
	FEM type	standard	-
	Buckling and relative lengths	Default	
	Layer	Layer1	×
	E Geometry		
	Length [m]	7,000	
	Insertion point	Bottom	
(i) -			
		ОК	Cancel

- 2. In the CrossSection row choose the first entered section CS1 Rectangle (500, 500).
- 3. The column length is 7 m.
- 4. The insertion position is by default set to **Bottom** so that the bottommost point determines the position of the column.
- 5. Confirm your input with the [OK] button.
- The column is positioned in the origin of the coordinate system. To this end, you must enter the coordinates 0;0 in the Command line and then press <Enter> to confirm your input.

Com	mand line
ß	D
Ne	w column - Enter point >0:0

- 7. A second column is entered in a similar way at position 6;0
- 8. End the input using the **<Esc>** key.
- 9. After input of an entity in SCIA Engineer, the entity is always selected. The columns for instance obtain a violet colour.



To cancel the selection, press the **<Esc>** key once more.

Notes:

The properties of selected elements are shown and can be modified in the Properties window.

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

You can end your input by pressing either the **<Esc>** key or the right mouse button and pressing **End**.

₽	End+Continue
\checkmark	End
×	Cancers

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

To define coordinates in command line you can use semicolon or just leave one space.

When both columns are entered, you can start entering the beams. The start and end points of the beams are already known, i.e. the centre and the end point of a column. Therefore, the beams must not be entered through coordinates, but 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 <u>Snap mode</u> at the lower right corner of the screen. The Cursor snap settings window is opened:



- 2. Activate the options a) Midpoints and b) Endpoints/Nodes to pick the desired locations on members in this project.
- 3. Click **[OK]** to confirm.

Now, you can enter the beams.

Entering a beam

- 1. To enter a new beam, you use the Beam option in the Structure service.
- 2. In the CrossSection row choose the second section CS2 Rectangular (700, 450).

Analysis model Standard CrossSection CS2 - Rectangle (700; 450) • Alpha 0 • Member system-line at Centre • ez [mm] 0 • LCS standard Buckling and relative lengths Default Layer Layer1 • Geometry Length [m] 6,000 Insertion point begin				
α Type beam (80) Analysis model Standard CrossSection CS2 - Rectangle (700; 450) ◄ Alpha 0 Member system-line at Centre ez [mm] 0 LCS standard Buckling and relative lengths Default Layer Layer1 ◄ Geometry Length [m] 6,000 Insertion point begin		Name	B3	
α Analysis model Standard CrossSection CS2 - Rectangle (700; 450) ▼ Alpha 0 ✓ Alpha 0 ✓ Member system-line at Centre ✓ ez [mm] 0 ✓ LCS standard ✓ Buckling and relative lengths Default ✓ Layer Layer1 ▼ Geometry Length [m] 6,000 Insertion point begin ✓		Туре	beam (80)	
CrossSection CS2 - Rectangle (700; 450) Alpha 0 Alpha 0 Member system-line at Centre ez [mm] 0 LCS standard LCS standard Buckling and relative lengths Default Layer Layer1 Construct Co	α	Analysis model	Standard	
Alpha 0 Member system-line at Centre ez [mm] 0 LCS standard FEM type standard Buckling and relative lengths Default Layer Layer1 * Geometry Length [m] 6,000 Insertion point begin		CrossSection	CS2 - Rectangle (700; 450)	÷.,
Z' Member system-line at Centre ez [mm] 0 LCS standard FEM type standard Buckling and relative lengths Default Layer1 * Geometry Length [m] Insertion point begin		Alpha	0	
	Z'	Member system-line at	Centre	
LCS standard FEM type standard Buckling and relative lengths Default Layer Layer1 Length [m] 6,000 Insertion point begin		ez [mm]	0	
FEM type standard Buckling and relative lengths Default Layer Layer1 Geometry Length [m] 6,000 Insertion point begin		LCS	standard	
ez Buckling and relative lengths Default Layer Layer1 ✓ Geometry Length [m] 6,000 Insertion point begin		FEM type	standard	2
Layer Layer1 □ Geometry Length [m] 6,000 Insertion point begin	🕵 ez	Buckling and relative lengths	Default	
Geometry Length [m] 6,000 Insertion point begin	(i) \	Layer	Layer1	τ.
Length [m] 6,000		Geometry		
Insertion point begin	>	Length [m]	6,000	
		Insertion point	begin	

- 3. The beam length is 6 m.
- 4. The insertion position is by default set to **Begin** so that the left point determines the position of the beam.
- 5. Confirm your input with **[OK]**.
- 6. Now click with your mouse on the midpoint of the left-hand side column to enter the beam:

Ĩ				I	
	Midpoint				
Ņ					
Ţ	⇒x			ļ	

- 7. The upper beam is entered in a similar way by clicking the top node of the left-hand side column.
- 8. Press **<Esc>** to terminate the input.
- 9. Press **<Esc>** once more to cancel the selection.

5	7				
ļ	Ŀ	> X			

Hinges

In this project, the beams are connected to the columns in a hinged way. As you have chosen the Frame XZ construction type, this means that by default the members will be connected to each other with fixed ends. Therefore, hinges must be manually entered. Note that hinges are not entered to frame corner nodes but to beam end/ends.

Entering hinges

- 1. To enter hinges, use the **Model data > Hinge on beam** option in the **Structure menu**.
- 2. The dialogue with hinge properties appear:



- 3. The hinges are entered on both sides of the beams, so that you have to choose **Both** for **Position**.
- 4. To obtain a hinge that does not transfer bending moment from one member to another, the rotation **fiy** is taken **Free**, the translations remain **Rigid**.
- 5. Confirm your input with **[OK]**.
- 6. Hinges are added by clicking with the left mouse button on both beams.
- 7. Press **<Esc>** to terminate the input.

0	_		 	0	
Υ.				9	
Γ				٦	
	\mathbf{v}				

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

Supports

The geometry input can be completed with supports. Both column bases are modelled as fixed supports.

Entering supports

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

🚊 🖗 Model data
🚊 🖾 Support
🖳 📥 in node
→ 🚡 point on beam
🔤 🔤 line on beam

2. To model the fixed ends, both the translations and the rotation are taken **Rigid**. You can also easily select **Contraint** to **Fixed** which makes the definition of below end conditions.

8	Support in node		×
	Name		
	Туре	Standard	+
	Angle [deg]		
	Constraint	Fixed	
	x	Rigid	*
	Z	Rigid	
12	Ry	Rigid	*
X RV	Default size [m]	0,200	
	Geometry		
(i)	System	GCS	+
Î			
		0	Cancel

- 3. Confirm your input with [OK].
- 4. You can simply select both bottom nodes by drawing a box with the mouse from the left-hand side to the right-hand side:



- 5. Press <Esc> to terminate the input.
- 6. Press **<Esc>** once more to cancel the selection.



Notes:

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

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

Check structure data	or click on the	- <i>5</i> Ijfi 🞯 🛱	8 A 🗐 🗊	icon in the
toolbar.				

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

Check of	of structure data	
Check of nodes		
Search duplicate nodes	🗌 Ignore parame	eters
Check of members		
Search null members	Null members: ✓ Delete null me	0 embers
Search duplicate members	Duplicate	0 ate members
	Invalid parts: Delete invalid	0 parts
Check of data references		nt method
	Fast method	ni nicerioù
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 c	ross-links	
Check additional data Check dupl	icity of names	Check Cancel

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

	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
	✓ Delete null members
Search duplicate members	Duplicate 0
	✓ Delete duplicate members
	Data check report
	Id parts
Check of data references	
Check data references	Data check finished. cient method
	j.
Check of additional data	
Check additional data position	
	Correct position
Check free load distribution point	s Invalid loads 0
Check of steel connections	
Check steel connections	Invalid 0
	✓ Delete invalid connections
Check load panels	Check cross-links
Check additional data C	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

Begin and end node of the top beam is also an end node of a column. Therefore, this beam is automatically connected to the columns.

The beam in the middle of the columns is not modelled in end nodes of other entities, but in midpoints. The end nodes of the beam are located in the internal of the column and therefore are not yet connected to the columns. In this paragraph, we will explain how to connect the members to each other in case they don't have common end nodes. 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	01	"	741 P	HBC	НВС	<u>k</u>	87	B) 🖽	icon	at
bottom of the modelling window.										

Activating member labels



When you select for example column **B1** with the left mouse button, it becomes highlighted by dashed violet line and the properties are displayed in the **Properties** window:



This window indicates that the Begin node is **N1** and the End node **N2**. Node **N5** is not part of the column. To connect beam **B3** to the columns, you must use the command **Connect members/nodes**.

Connecting entities

- 1. Press <ESC> or click the Cancel selection icon to deactivate any selection of entities.
- 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:		be connected	SCIA Engineer 16.0.1071
			1 D	o you want to proceed with all entities?
4. 5.	Click <yes></yes> . The Setup for connection of s entities dialogue box now appe	str i ear	uctural s.	<u>Ano N</u> e Storno
	E Se	tup	for connection of structural entiti	es 🛛 🗙
			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 1D members as ribs 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:

If you select for instance beam **B3**, the **Properties** window shows that node **N5** connects the beam to column **B1** and that node **N6** connects the beam to column **B2**.





Note:

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

It is also possible to perform 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] at the bottom of the Structure menu.

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

Remark:

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

The magnifier

- Use Ω 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
æ	Cursor snap setting
Q.	Print/ Preview table
	Table to Engineering report
Ŕ	Print picture
1	Picture to gallery
H	Save picture to file
Ē	Copy picture to clipboard
	Screenshot into Engineering report
	Live picture into Engineering report
0 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 – I	Labels
1.011	purumeters i	Luncio

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).

	g
Check / Uncheck group	Lock position
4 🖻 🔛 🔛 🧐 🌌 🔍	Þ
Check / Uncheck all	
Service	
Display on opening the service	
Style + colour	-
Draw member system line	
Member system line style system lin	ne 🗾 🗾
Model type analysis r	nodel 🗾
Display both models	
Rendering wired	-
Draw cross-section	
Cross-section style section	•
Effective width of plate ribs	
Draw effective width	
Rendering transpare	ent <u>-</u>
Display	
Mark style Dot	-
Member parameters	
System lengths	
Member nonlinearities	
FEM type	
Nodes	
Members 1D	
View parameters setting -	Labels
Check / Uncheck aroun	Lock position
Check / Uncheck all	
Service	
Service Display on opening the service	
Service Display on opening the service Beam labels	
Service Display on opening the service Beam labels Display label Name V	
Service Display on opening the service Beam labels Display label Name Cross-section name	
Service Display on opening the service Beam labels Display label Name Cross-section name Cross-section type	
Service Display on opening the service Beam labels Display label V Cross-section name Cross-section type Length	
Service Service Seam labels Display label V Name Cross-section name Cross-section name Length Layer Une and priority	
Service Service Seam label Uisplay an opening the service Beam label Uisplay label V Name Cross-section name Cross-section type V Length Layer Type and priority N Note slabels	
□ Service □ □ Display on opening the service ▼ □ Beam labels □ □ Display label ▼ Name ▼ Cross-section name ▼ Length ▼ Layer □ Type and priority □ ■ Nodes labels □ Display label ▼	
□ Service □ □ Display on opening the service ▼ □ Beam labels □ □ Display label ▼ □ Cross-section name ▼ □ Cross-section name ▼ □ Length ▼ □ Layer □ □ Type and priority □ □ Display label ▼ □ Nodes labels □ □ Name ▼	
■ Service ■ Display on opening the service ● ■ Beam labels ■ Display label ● Name ● Cross-section name ● Cross-section name ● Length ● Layer ● Type and priority ● Display label ● Nodes labels ● Display label ● Name ● X-coordinate ● X-coordinate ●	
Service Image: Construct of the service Display an opening the service Image: Construct of the service Display label Image: Construct of the service Cross-section name Image: Construct of the service Cross-section name Image: Construct of the service Cross-section name Image: Construct of the service Layer Image: Construct of the service Type and priority Image: Construct of the service Display label Image: Construct of the service Name Image: Construct of the service V-coordinate Image: Construct of the service Z-coordinate Image: Construct of the service	
■ Service ■ Display on opening the service ▼ Display label ▼ Name ▼ Cross-section name ▼ Cross-section type ▼ Length ▼ Layer □ Type and priority □ Display label ▼ Name ▼ V-coordinate □ Z-coordinate □ Z-coordinate □ System lengths ■	
□ Service □ Display on opening the service □ Display label □ Name □ Cross-section name □ Cross-section type □ Layer Type and priority □ Bode slabels □ Display label ∨ Name ∨ Layer □ Stock slabels □ Display label ∨ ×-coordinate ∠-coordinate □ System lengths □ Display label	
■ Service Display on opening the service V Beam labels Display label V Name Cross-section name Cross-section type Layer Type and priority Nodes labels Display label V Name X-coordinate Z-coordinate System lengths Display label Vame	
■ Service Display on opening the service ♥ Beam labels Display label ♥ Cross-section name Cross-section type ♥ Length ♥ Nodes labels Display label ♥ Name ¥-coordinate Z-coordinate Display label ♥ System lengths Display label ♥ Name ♥ Nonlinearities	
Service Image: Service Display on opening the service Image: Service Display label Image: Service Name Image: Service Cross-section name Image: Service Cross-section name Image: Service Cross-section name Image: Service Layer Image: Service Type and priority Image: Service Display label Image: Service Y-coordinate Image: Service Z-coordinate Image: Service Display label Image: Service Name Image: Service Label Image: Service Nonlinearities Image: Service	
Service Display on opening the service Display on opening the service Display label V Name Cross-section name Cross-section type V Layer Type and priority Name Display label V-coordinate Z-coordinate Z-coordinate Display label Name Display label Name Display label	
Service Display on opening the service Display abel Vame Cross-section name Cross-section type V Layer Type and priority Display label Vodes labels Display label V-coordinate Z-coordinate Z-coordinate Display label Name V-coordinate Display label Name Usbel Display label Name Value Name Nollinearities Display label Name Nollinearities Display label	
Service Display on opening the service V Name Cross-section name Cross-section type V Layer Type and priority Nodes tabels Display label V Name V.Coordinate Z-coordinate Z-coordinate System lengths Display label Name Ublabel Display label Name Z-coordinate Display label Name Label Ibliplay label Name Label Nonlinearities Display label Display label Labels of local axes Nodes Members 1D	
Service Display on opening the service Display label V Name Cross-section name Cross-section name Cross-section name Layer Type and priority Nodes labels Display label V-coordinate V-coordinate V-coordinate System lengths Display label Name V-coordinate Display label Name V-coordinate Display label Name Name Display label Name Mame Mame Objective Name Objective Display label Name Members 1D General structural shape Ditolay wertex label	
Service Display on opening the service Display label V Name Cross-section name Cross-section type Length V Nodes labels Display label V-Coordinate V-coordinate V-coordinate V-coordinate V-coordinate Display label Name V-coordinate Ibel Name Uabel Display label Name Valabel Display label Name Display label Display users Display vertex label Display vertex label	
Service Display on opening the service Display iabel Vame Cross-section name Cross-section type Cross-section type Layer Type and priority Name Display label V-Coordinate Z-coordinate Z-coordinate Display label Name Display label Display label Display label Display label Display label Display label Display vertex label Display vertex label	
Service Display on opening the service Visplay label Name Cross-section name Cross-section type V Layer Type and priority Node slabels Display label V.coordinate Z-coordinate Z-coordinate Z-coordinate Display label Name User lengths Display label Name Display label Name Label Display label Name Label Display label Nonlinearities Display label Nodes Members 1D General structural shape Display vertex label	

View parameters – shortcuts

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

- Show/hide surfaces 🖉 to show the surfaces of the cross-sections.
- Render geometry 💋 to view the rendered members.
- Show/hide supports to show supports and hinges.
- Show/hide load 🌆 to show the load case.
- Show/hide other model data 🖾 to show other model data (like hinges, internal nodes, ...).
- Show/hide node labels to view the label of the nodes.
- Show/hide member labels ²⁵/₂ 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.

In this project, two load cases are entered:

LC1: Permanent load case: Self weight of the members + weight of the floor and weight of the roof **LC2**: Variable load case: Service load on the floor

Defining a Permanent Load Case

- 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 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 type the description **"Self Weight Construction"**.

	Load cases		×
🔎 💱 🗶 📸 💺 📴	<u>의 역 🚭 🖻 📕 🛛</u>		• 7
LC1 - Self Weight Con	Name	LC1	
	Description	Self Weight Constructio	n
	Action type	Permanent	Ψ.
	LoadGroup LG1		×
	Load type	Standard	÷.
	Actions		
	Delete all loads		>>>
	Copy all loads to another loadcase		>>>
New Insert Edit	Delete		Close

Defining a Variable Load Case

New

1. Click

or 📕 to create a second load case.

- 2. Enter the description "Service load".
- 3. As this is a variable load, change the Action type to Variable.

	Load cases		×
🏓 🕃 🖋 🖍 📴	의 요리 🚭 😂 🖶 🛯 Al		• 7
LC1 - Self Weight Con	Name	LC2	
LC2 - Service load	Description	Service load	
	Action type	Variable	· •
	LoadGroup	LG2	×
	Load type	Static	Ψ.
	Specification	Standard	Ψ.
	Duration	Short	Ψ.
	Master load case	None	Ψ.
	Actions		
	Delete all loads		>>>
	Copy all loads to another loadcase		>>>
New Insert Edit	Delete		Close

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

•	Load groups		×
🔎 🎼 🖉 📑 💽 🗅	. 🗠 🖨 🖨 🔒		
LG2	Name	LG2	
	Relation	Standard	*
	Load	Variable	+
	Structure	Building	
	Load type	Cat A : Domestic	*
New Insert Edit	Delete	0	к

The Load type determines the composition factors that are attributed to the load cases in this load group when Eurocode combination is requested. In this project, choose **Cat A: Domestic.**

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

Remark:

Load groups

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

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

Loads

After input of Load cases, the Load service will automatically appear:

The first load case (LC1) includes two loads:

- Self weight of the members
- Roof weight + Floor weight

Switching between load cases

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



Entering the self weight as linear load

- 1. Cancel any possibly active selection of entities by pressing <ESC>.
- 2. Click on Line Force on beam in the Loads Menu. The dialogue Line Force on beam appears.

Load # X		Line force on bea	m	×
Point force Point force Ine force - on beam Thermal load - on beam Point displacement Ine displacement Plane generator Pond load - water accumulation Not calculated internal forces	-P1 -P1 i	Name Direction Type Gravity coef. Distribution Load above joint Geometry System Location Extent Coord. definition Position x1 Position x2 Origin Fecentricity	LF1 Z Self weight -1 Uniform no GCS Length full Rela 0,000 1,000 From start	Cancel

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

- 4. Confirm your input with [OK].
- 5. Select all the bars by means of the **Select all** icon in the toolbar.
- 6. Press **<ESC>** to finish the input.
- 7. Press **<ESC>** once more to finish the selection.

The self weight load is represented in brown:



Now, the floor and roof weight is entered as a series of concentrated loads:

Entering a series of concentrated loads

1. Click on **Point force on beam** in the **Loads Menu**. The dialogue **Point force on beam** appears.

Load 4 ×		Point force on beam		×
LC1 - Self Weight Construction	$ \begin{array}{c} $	Name Direction Type Angle [deg] Value - F [KN] Geometry Extent System Coord. definition Position x [m] Origin Repeat (n) Delta x [m] Eccentricity Fccentricity er [m]	F1 Z • Force • -25,00 full • GCS • Abso • 0,500 From start • 6 1,000 0K Cance	

- 2. The floor beam is charged with 6 concentrated loads of **25 kN** with a distance of **1 m**. The first concentrated load of the series is at **0.5 m** from the starting node of the beam.
- 3. The Value of the point load is changed to -25 kN.
- 4. The coordinate definition is set to Abso (meaning absolute).
- 5. The starting position "Position x" is changed to **0.5 m**.
- 6. The series consists of 6 concentrated loads so that the Repeat (n) field is set to 6.

- 7. The clearance Delta x between the concentrated loads is **1 m**.
- 8. Confirm your input with [OK].
- 9. Select beam B3.
- 10. Press <Esc> to terminate the input.
- 11. Press **<Esc>** once more to cancel the selection.

The roof beam is similarly loaded with concentrated loads of **12.5 kN**. Therefore, the load of the floor beam can be copied to the roof beam and adapted according to the steps below.

Note:

Loads, supports, hinges etc. are considered as additional data, i.e. data that are additionally added to entities such as nodes, bars...

Copying loads

- 1. Select one of the point loads on the floor beam with the left mouse button. As this point load is part of a series, the entire series is automatically selected.
- 2. Press the right mouse button, a popup menu appears:



- 3. Choose the option Copy add data F1.
- 4. Select the bar where this load should be copied the roof beam B4.
- 5. Press **<Esc>** to terminate the input.
- 6. Press **<Esc>** once more to cancel the selection.

Now, the value of the roof load can be changed.

Adapting a load

- 1. Select one of the point loads on the roof beam. As this point load is part of a series, the entire series is automatically selected.
- 2. The properties of the series are displayed in the **Property Window**.
- 3. Change the value from -25 kN to -12.5 kN.
- 4. Confirm the change by pressing **<Enter>**. Below is the resulting view:







After input of the loads in the first load case you can enter the service load into the second one. The floor beam is loaded with a service load of 10 kN/m.

Switching between load cases

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



Entering a linear load

- 1. Click on Line force on beam in the Loads Menu. The dialogue Line force on beam appears.
- 2. Change the type to Force and the value to -10 kN/m.

Load 🛛 🖓 🗙		Line force on beam		×
LC2 - Service load	P2 -P1 -P2 i x	Name Direction Type Angle [deg] Distribution Value - P [kN/m] Load above joint Geometry System Location Extent Coord. definition Position x1 Position x2 Orinin	LF5 Z Force Uniform -10,00 Ino GCS Length full Rela 0,000 1,000 From start	Cancel

- 3. Confirm your input with [OK].
- 4. Select the bar on which this load must be positioned: floor beam **B3**.
- 5. Press **<Esc>** to terminate the input.
- 6. Press **<Esc>** once more to cancel the selection.



7. Click [Close] to leave the Loads service and to return to the Main tree.

Note:

The **Command line** includes a number of predefined loads **t u u u u**, which enable a fast and simple input of usually used type 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.

	Comb	ination - CO1
Contents of co	mbination	List of load cases
	ase 1 - Self Weight Construction 2 - Service Ioad	Lc2 - Service load
Name :	C01	Delete Add
Coeff :	1 Correct	t Delete All Add All
Type :	EN-ULS (STR/GEO) Set B	v
Structure:	Building	✓
Description :	ULS	
Nonlinear combination :		V OK Cancel

Result classes

- 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 zero to create a second combination.

- 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	٣	
CO1 - ULS	Name	CO2	
CO2 - SLS	Description	SLS	
	Туре	EN-SLS Characteristic	
	Structure	Building	
	Active coefficients		
	Contents of combination		
	LC1 - Self Weight Construction	1,00	
	LC2 - Service load [-]	1,00	
	Actions		
	Explode to envelopes		>>>
	Explode to linear		>>>
	Show Decomposed EN combinatio	ns	>>>
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
🔤 🏄 Connect members/nodes
- 🔯 Local mesh refinement
- 🔘 Mesh generation
- 🛱 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.	Linear stability	
	Concrete - Code Dependent Deflections (CDD)	
133	Construction stage analysis	
	Nonlinear stage analysis	
	O Nonlinear stability	
	○ Test of input data	
	Number of load cases: 2	
	Solver setup Mesh setup	
also for		
	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.


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.



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.

4. The action button **Refresh** has a red highlight, i.e. the <u>graphical</u> 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.

Properties		×
Reactions (1)	<mark>-</mark> Va V/	0
	6	8
Name	Reactions	
Selection	All	Ψ.
Type of loads	Combinations	Ψ.
Combinations	CO1 - ULS	Ψ.
Filter	No	-
Values	Rz	-
Extreme	Node	-
Drawing setup 1D		
Rotated supports		
Actions		
Refresh	>>	>
Table results	>>	>
Preview	>>	·>
N2 B4		N4



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



Note:

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

Viewing internal forces on beam

- 1. In the **Results** menu, open the **Beams** group and select **Internal forces on beams.**
- 2. The options in the **Property Window** are configured in the following way:
 - The Selection field is set to Current.
 - The Load type is set to **Combinations** and Combination to **CO1 ULS**
 - The Values are wanted for My.
 - The Extreme field is changed to Member.
- 3. Select the two beams **B3** and **B4** 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		ą	×
Internal forces on	memb🔁 🌆	₩/	0
		8	8
Name	Internal forces	on m	e
Selection	Current		*
Type of loads	Combinations		*
Combinations	CO1 - ULS		*
Filter	No		•
Values	My		*
Extreme	Member		*
Drawing setup			
Section	All		+
Actions			
Petrech			_
Detailed			5
Table results			<u> </u>
nubic results			

next to



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

Configuring the Graphical Screen

1. In the **Properties window**, click the icon next to **Drawing setup 1D**. Possibilities for various graphical representations are opened.

	Drawing	setup	×
Representation : Limits :		Filled	<
My Maximum [kNm] Minimum [kNm]	0		•
Description Values Draw section in la Draw load case or	bels combination in la	Units	
Angle of text O deg 90 deg		User defined	
Setup for more comp Same scale Same height	onents	Space between diagrams 1 \$ Shift of the first diagram \$	
		OK Cancel	

- 2. In the Representation combo-box choose Filled.
- 3. The Angle of text is set to 0 deg.
- 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 above options.



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

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

Concrete modules of SCIA Engineer contain a number of powerful tools to execute the concrete calculations in accordance with the chosen design code. The possibilities are:

- Input of advanced concrete data
- Calculation of the slenderness
- Reduction of bending moment and shear force above supports
- Design of theoretically needed reinforcement amount
- Input of the Practical reinforcement
- Capacity-response check
- Stress limitation
- · Crack width analysis
- Calculation of physical non-linear (PNL) deformation
- and more ...

In this tutorial, we will only explain the basics of the concrete design. For more information regarding the advanced concrete calculations we refer to the Concrete course (training) document. Before you can start the concrete design you must first check the buckling parameters of the bars. By means of the view parameters, the buckling lengths of the bars can be displayed on the structure.

Buckling parameters

Displaying the system lengths

- 1. Select column **B1** with the left mouse button, 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.
- 📴 Set view parameters for selected 3. In this menu, hit the option. The View parameter settings window appears. View parameters setting Lock position 4 🕿 🔤 🔛 6 1 Check / Uncheck all Structure Member surface Г Rendering rendered with edges **V** Draw cross-section Cross-section style section • Effective width of plate ribs • Draw effective width Rendering transparent Member parameters System lengths ~ Member nonlinearities ~ FEM type ~ Joists Local axes 7 Show names in tab ок 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 system length (Ly) for buckling around the strong axis (y-y) is 3,5 m and Lz for buckling around the weak axis (z-z) is 7 m. The beam in the middle of the column therefore supports the column for buckling around the strong axis.

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

Setting the Buckling Parameters

- 1. Select both columns 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**.

Properties P ×				
Ν	lember (2)	🔁 🖬 🌾 i	1	
1	6	e 2	\$	
-	Гуре	column (100)	•	
/	Analysis model	Standard	*	
0	CrossSection	CS1 - Rectangle (500; 💌		
/	Alpha	0	*	
I	Member system-lin	Centre	*	
	ez [mm]	0		
I	LCS	standard	*	
ł	FEM type	standard	*	
	Buckling and relativ	Default 🔹		
l	ayer	Layer1 🔹		
-	Geometry			
	Length [m]	7,000		
	Shape	Line		
	Beg. node			
	End node			

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

•	Buckling data	
🎾 🤮 🏒 📸 💽 🗠	2. 🖂 🎒 🖨 🔚 Number of pa	arts - 2 🔹
BC1	Name	BC1
	Number of parts	2
	Member(s) material	Concrete
	V.V.	77
	уу	LL
	3	•
	2	•
	1	•
New Insert Edit	Delete	Close

This window shows that the column is supported in the middle for buckling around the strong axis (Y-Y) but not for buckling around the weak axis (Z-Z).

	Name BCT	Number of parts	2	
y y	Buckling systems relation	Beta w	Calculate	
	22 = 22 🗸	Deta yy	Calculate	•
		Beta zz	Factor	· · ·
		Sway yy	Settings	~
Ly Lz		Sway zz	Settings	~
	Relative deformation systems rel	ation		
	Relative deformation systems rel def z = yy ~ ~	ation def y =	zz v	
	Relative deformation systems rel def z = yy v	ation def y =	22 V	
	Relative deformation systems rel def z = yy v	ation def y =	2Z ¥	
	Relative deformation systems rel def z = yy →	ation def y =	22 V	

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

- 5. On the **Base Settings** tab several data can be changed.
 - The Name field contains the name of the buckling parameter, in this case BC1.
 - Beta yy and Beta zz: in these combo-boxes you can indicate if the program should calculate the buckling factor round the particular axis or if you prefer entering this factor manually. A third option allows for a manual input of the buckling length, instead of buckling factor. The Support option can be used to determine the buckling factor in accordance with the model column from the Eurocode.

- Buckling system relation zz: in this field, you can indicate the system length to be used for the weak axis, whether it is dependent on the system for strong axis (zz=yy) or if it is independent (zz=zz).
- Sway yy and Sway zz: in these fields, you can indicate if the member is braced or not in the particular direction. When you choose the Settings option, the default settings (from the general concrete setting) are used.
 The default settings for the buckling parameters are displayed in Concrete 15 > Concrete setting (structure) > Design defaults > Default sway type. By default, both directions are unbraced for a concrete calculation. Thus for concrete calculation, no wind bracings are assumed.
- **def z** and **def y**: in these fields, you can indicate the buckling system to be used for the relative deformations the length to be used to detect relative deformation check.
- 6. On the **Buckling data** tab you can change the parameters in detail. As the columns consist of 2 components, 3 positions are available: (1) at the beginning, (2) in the centre of the floor beam and (3) at the end.
 - For this project, we assume that the columns are not braced in both directions. Therefore, the **Non-braced** property can be set to **Yes** both for the strong and the weak axis. This could also be left on settings, since default settings are sway for concrete calculation.
 - For this project, we also assume that the column is supported in the middle for buckling around the weak axis. The **zz** property at position (2) therefore can be set to **Fixed**.

					Buck	ing and relati	ive	e lengths.					
Base s	ettings Buck	ling data											
	уу	Sway	уу	ZZ	Beta zz	Sway zz		Tot. heigt	h	Tot. heigth [m]	my	n	nz
1	Fixed	Yes	+	V Fixed	1,00	Yes	*	Calculate	*	20,00	1,00	1,00	
2	Fixed	Yes	*	Fixed	1,00	Yes	*						
3	Fixed			Fixed									
													Refresh
											ОК	Stomo	Použiţ

- 7. Click [OK] to close this window.
- 8. The **Buckling data** window re-appears and displays the changed buckling data. Click **[Close]** to close this window.

	Buckling data	×
🖈 🤮 🗶 🖬 🔛 🖆	2. 🗠 🎒 😂 🔚 Number of pa	rts - 2
BC1	Name	BC1
	Number of parts	2
	Member(s) material	Concrete
	1/1/	77
	уу	LL
	3	•
	,	
	-	
	1	•
New Insert Edit	Delete	Close

- 9. The Properties window indicates that the buckling parameter BC1 is used for both columns.
- 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 concrete design. Before proceeding, deactivate the display of system lengths through Fast adjustment of view parameters on whole model button.

Structure	-	Effective width of plate ribs
Labels	•	Structure nodes
Model	- • [Member parameters
Loads/masses	•	Mesh
Modelling/Drawing	•	✓ Local axes
Attributes	•	Design groups
Misc.	÷Г	
Detailed on/off		
Setup dialog		
	_	

When the buckling data are adapted, you must recalculate the project. See chapter Linear calculation.

Concrete design

Double-click on Concrete 15 in the Main tree to open the Concrete menu.

	Π.	Concrete	

Concrete 15

There are two services for concrete design in version SCIA Engineer 15 and 16. The old one called simply Concrete won't be discussed in this tutorial because the new one called Concrete 15 replaces it.

Displaying the Slenderness and the Buckling Lengths

 Click the Slenderness icon in the Concrete 15 menu under chapter Reinforcement design – 1D members



- 2. The options in the **Properties window** are configured in the following way:
 - Type of selection field is set to Current.
 - Type of load is set to Combinations and Combination to CO1 ULS.
 - Values are set to λ , i.e. the slenderness lambda in both directions.
 - Extreme 1D field is modified to Section.
- 3. Select column **B1**, the left column of the frame and hit the button **Refresh**



4. Change the **Values** field to I to display the buckling length for buckling around both axes. Hit again the **Refresh** action button to regenerate the view.



As already indicated in the buckling parameters, the buckling length is 3,5 m.

Notes:

The buckling length Iy/Iz is determined from system length Ly/Lz multiplied with buckling factor $\beta y/\beta z$. The calculation of slenderness is important for reinforcement calculation in columns. It will determine whether a second order moment has to be taken into account. For a detailed explanation reference is made to the Concrete Course Document.

Detailed calculation including formulas and code references can be however displayed in Report preview if you change the output type in Properties window to **Detailed** and click on Preview action button:

л×

		1919 m 191		Slenderness(Design) (1)) 🛛 🔽 🔞	V/ /
		and a second	::::			e 🐣
				Name	Slenderness(Design	1)
		$\sim \chi_{1}$	1941 🔳 🔳	Selection		
2 4 12	🏽 🎬 😹 🌆 📾 🗰 <	>		Type of selection	Current	*
				Filter	No	*
A D D-4			τ	Result case		
	Parameters for calculation limit slenderness:			Type of load	Combinations	*
	1 1			Combination	CO1 - ULS	*
	$A = \frac{1}{1 + 0.2 \cdot \varphi_{ef}} = \frac{1}{1 + 0.2 \cdot 2.34} = 0.681$	(\$5.8.3.1(1))		Extreme 1D	Global	*
				Output	Detailed	*
	$B = \max(\sqrt{1 + 2 \cdot \omega}; 1.1) = \max(\sqrt{1 + 2 \cdot 0.522}; 1.1) = 1.43$	(\$5.8.3.1(1))		Run using Model D		
	$C_y = 1.7 - r_{my} = 1.7 - 1 = 0.7$	(§5.8.3.1(1))		Values	I.	-
	$C_z = 1.7 - r_{mz} = 1.7 - 1 = 0.7$	(§5.8.3.1(1))		Drawing Setup		
	Limit value of slenderness around y-axis:			Display value name		
	$A = \frac{20 \cdot A \cdot B \cdot C}{20 \cdot 0.681 \cdot 1.43 \cdot 0.7} = 112$	(5.1250)		Display values	V	
	∧im	(3.1314)		Display units	V	
	Limit value of slenderness around z-axis:			Display case		
	20-A-B-C 20-0.681-1.43-0.7			Display section dx		
	$\lambda_{im} = \frac{1}{\sqrt{n}} = \frac{1}{\sqrt{n}} = \frac{1}{\sqrt{n}} = \frac{1}{\sqrt{n}} = 113$	(5.13N)		Display combinati		
	4 H 4 0.0140			Color scheme	Defined by result	-
C	neck of slenderness			Graph type	Filled transparent	*
	$\lambda_y = 126 > \lambda_{iimy} = 113 = >$ second order effect will be taken into accour	nt around y-axis		Envelopes drawing	0 to extrem	Ψ.
	Warning: Slenderness around viavis is higger than limit slenderness. Second	order effect has to be taken into		Label colour by gr	V	
	account.			Drawing plane	3D	*
	$\lambda_z = 24.2 \le \lambda_{limz} = 113 => second order effect will be ignored around a$	z-axis				
	Note: Slenderness around z axis is lesser than limit slenderness, it follows see	cond order effect will be janored.				
			~			
			>_	Actions		_
		116%	(=)(+)	Refeesh	5	
f line			4 ×	New combination from	n Combination key	>>>
H H H		ヽ ヽ ♀ × ∧ * ▽ ~ 濖 囲口 × ヽ	·[[]	Table results	Combined of Key	>>>
and a				Draview		~
mu <				THE TEN		

Internal forces for reinforcement design

Internal forces in service Results are purely values obtained by finite element analysis. For concrete design we need to recalculate these forces into design values, using various code regulations. Although this command is identically named in service Results and Concrete 15, here we show both basic and recalculated values that will be used to get required reinforcement areas.

 Click the Internal forces in the concrete 15 menu under chapter Reinforcement design – 1D members

🖶 📊 Reinforcement design - 1D members
Internal forces
Slenderness
🐨 🖫 Reinforcement design

- 5. The options in the **Properties window** are configured in the following way:
 - Type of selection field is set to Current.
 - Type of load is set to Combinations and Combination to CO1 ULS.
 - Values are set to M-MEd, i.e. comparison of basic and design bending moment diagram.
 - Extreme 1D field is modified to Global.



7. The brief output in **Report preview** shows in two tables the difference between internal forces from Fem analysis and recalculated internal forces.

Theoretically required reinforcement

The reinforcement design will only be illustrated for the beam B3. Obviously, the reinforcement calculation for all the other members is identical. For the background of this calculation we refer to the Concrete Course Document.

Longitudinal reinforcement As

 In the Concrete 15 menu, go to Reinforcement design under chapter Reinforcement design – 1D members.

Find Reinforcement design - 1D members
 Find Reinforces
 Slenderness
 Reinforcement design

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

- Type of selection field is set to Current.
- Type of loads is set to Combinations and Combination to CO1 ULS.
- Values are wanted for As,req.
- Extreme field is changed to Global.
- 3. Select beam B3 with the left mouse button Click the button **Refresh** to display the results on the graphical screen in accordance with the options above.



4. Click the button next to **Preview** to display the results on the table form in accordance with the options above.

Over Linear ca Combina Extreme Selection Longitu	all De loculation tion: CO: 1D: Glob : B3 dinal rei	esign 1 al	n (ULS)						10			
Name	dx [m]	Case	Member	A _{sz_mq+} [mm ²] A _{sz_mw+} [mm ²]	A _{52,00} [mm ²] A _{52,00} - [mm ²]	A _{sy_req+} [mm ²] A _{sy_prov+} [mm ²]	A _{sy_m} [mm ²] A _{sy_prov} - [mm ²]	A _{12,000} [mm ²] A _{12,000} [mm ²]	A _{sy_req} [mm ²] A _{sy_prov} [mm ²]	A _{1,701} [mm ²] A _{1,200} [mm ²]	Reinf]
B3	3,000	C01	Beam	0	951 1005	0	0	951 1005	0	951 1005	[z-]5ф16	1
Shear n	einforce	ment							-			
Name	dx [m]	Case	Member	A swm_req [mm²/m	A	/m]	ShearRein	nf				
83	2,000	CO1	Beam	30	60	366 \$8/	275mm, (n	s=2)				
83	0,000	C01	Beam	60	04	670 \$8 (ns	(150mm, =2)					

Note:

- Detailed analysis, pictures and explanation of all calculation steps can be seen if the Output type is changed to **Detailed**. Instead of a table with a few rows (**Brief** output) this can be extended to many pages.
- Alternatively compromise solution is to display Standard output which summarizes the design on one page A4 approximately.

Shear reinforcement Aswm,req

- 1. Now change the **Values** field to **Aswm,req** to change the diagram in graphical screen so that it displays required shear reinforcement.
- 2. Set Extreme 1D to Section.
- 3. Click the button **Refresh** to display the results on the graphical screen in accordance with the options above. Optionally, click the Preview action button to show text output too.



Reinforcement check (ULS)

Once you know the required reinforcement amount you can start modelling so called practical reinforcement. This means that every beam and column gets graphically clear representation of longitudinal bars and stirrups along whole length of the member. This procedure will be described later in this tutorial.

In case that knowing reinforcement layout in cross-section at particular sections along the member is sufficient, there is no need to use practical reinforcement. Instead, Section check tool can be used to design and immediate check of cross-section in particular position of the beam or column.

In this tutorial we will demonstrate this tool by checking capacity-response diagram on one beam and checking interaction diagram on one column.

Section check – response check

- Select Capacity-response (ULS) command in the Concrete 15 menu under chapter Reinforcement check (ULS+SLS).
- 2. Adapt the setting in Properties window:
 - Type of load change to Combinations
 - Combination set to CO1 ULS
- 3. Click on button next to Section check in





4. A new dialogue **Section check** opens. Here you can define reinforcement template in that particular section, edit diameter and number of bars and directly see the unity check value. Number 3,00 with the red cross means that the current reinforcement does not satisfy.

	Section	check	- D X
	Section	check	
Home			
	Export	Any check hasn't been exported yet.	
Longitudinal reinforcement	Stimups	Report	
Section properties	Report		Check manager
ToolTip Grid Bars Stirr Cover Axis Grid size: 100 mm / 🗠	Standard Check name: Capacity-response Combination name: CO0 [ULS]	Check value: 3.00 💥	Check table
25		<u> </u>	Check Value Status
	Member P2 section no. 0.	hy = 2 m Poom	Capacity-response 3.00
	Member B5, Section no. 9, d	Materials	Capacity-interaction diagram
	Buckling length y Ly = 6 m	Concrete C25/30	Shear+torsion
- 25-3	Buckling length z Lz = 6 m	Reinforcement B 500B	Detailing provisions
	Longitudinal bars	Coefficients	Combination table
	2020	Concrete parameters γr 1.5, σrc 1 = Reinforcement parameters γr 1.15	Combination Value Status
		Coefficient for effective height $Coeff_d = 0.9$	1991 CO0 [ULS] 3.00
Basic reinforcement (layout) Additional reinforcement	+Z	Coefficient for inner lever arm Coeff ₂ = 0.9	
	× 8	Reinforcement	■ CO2 [ULS]
Layer Position Bars Diameter Mater =		Stirrups: $\phi = 8 \text{ mm}$, $A_{ow} = 101 \text{ mm}^2$, $A_{ows} = 503 \text{ mm}^2/\text{m}$ Cover of stirrups:	
L1 top 2 20 B 500E	2020	Bottom 25 mm	4
L2 bottom 2 20 8 5008	Stimups \$8/200 mm, ns = 2	Left 25 mm Right 25 mm	
4	450		
😞 Shear	Summary of check	-	4
Stirrup Legs count Ø [mm] s [mm] Mater		E =	The calculation of cross-section not satisfied !
Připraven			



5. Change number of bars for longitudinal reinforcement at bottom layer to 3, press **<Enter>** and you immediately see the recalculated value of unity check. With this new reinforcement the check satisfies for all linear combinations from the code envelope CO1.



6. Close the Section check dialogue with red cross icon at top right corner.

Section check – capacity diagram

- 7. Select **Capacity-diagram (ULS)** command in the **Concrete 15** menu under chapter Reinforcement check (ULS+SLS).
- 8. Adapt the setting in Properties window:
 - Type of load change to Combinations
 - Combination set to CO1 ULS



Click on button next to Section check in Actions and select column B1 with mouse cursor. At this moment a set of green crosses appear on the selected member, representing various sections along the beam where the check can be performed. Click on the middle one, Section: 0.



- 10. The dialogue **Section check** opens. It clearly shows that the column satisfies with the current reinforcement.
- 11. Switch the report type from **Standard** to **Detailed** and scroll-down the page in the middle of the dialogue to see the N-M interaction diagram.



12. Close the Section check dialogue with red cross icon at top right corner.

Practical reinforcement

Previous chapter can give you an insight on the reinforcement amount you need to input in particular beam of column. You may need to represent it graphically to make bill of reinforcement or detailing drawings.

But you can also input practical reinforcement without previous design and do the checks directly, to see if the provided reinforcement area is sufficient.

- 1. Press **<Esc>** to cancel any the selection.
- To enter practical reinforcement you can open the group Reinforcement input+edit in Concrete 15 menu and select the item New reinforcement.



3. Select beam **B3** with the left mouse button. You will be directly asked (as can be seen in the Command line) to select the first point of the reinforcement zone and the last point. Click on node N5 and node N6 consecutively.

4. The **Longitudinal reinforcement** manager will appear. Choose the first template (LR_B_R1) from the list and click OK.

•		Longit	tudinal reinforcement	×
🚚 💱 🖉	🖬 🛃 🖸	2 ₿	i 🛱 🖬	
LR_B_R1 LR_B_R2 LR_C_R1 LR_B_R3 LR_B_R4 LR_B_R5 LR_B_R6 LR_C_R2 LR_B_R7 LR_B_R8 LR_B_R9 LR_C_R3 LR_B_R10 LR_B_R11				
Name	LR B R1			
Description	Long. re			
Stirrup na	StirrupR9			
Number o	2			
Diameter [16,0			
Area [mm	804			
Type of be	beams a			
New	sert Edit	Delete		ок

5. **Reinforcement parameters** dialogue now appears, just before the reinforcement is inputted. Select the middle option "from the Concrete setup (Design default)" to specify the diameters of bars and stirrups according to concrete design defaults and hit **OK** button.

Reinforcement parameters	
Do you want to use parameters of reinforcement (diamater of long.reinforcement, stirrup and concrete cover) from the Concrete member data	
from the Concrete setup (Design default) from the defined template	
ОК	



6. We have calculated that some additional reinforcement was required in the middle of this beam. Therefore we will add a new bar of reinforcement in the zone from 1 m to 5 m.

To enter a layer of reinforcement on a selected interval you could either use command New longitudinal bars which can be found in the **Concrete 15** menu under or the

shortcut in the command line to add longitudinal reinforcement on selected interval

When you start the command to add longitudinal reinforcement on a selected interval, you first have to select the beam B3.

After the beam is selected you have to define the interval.

To define the interval it can be helpful to use snap points. Therefore select the cursor snap setting in the command line

In the cursor snap setting you activate option h) to create six snap points on each beam. On beam B3 you will have now one snap point every one meter. Now select the interval from 1 m until 5 m on this member B3.



7. The next screen shows us the reinforcement layers in the zone from 1 m to 5 m of member B3.

	Member B	3, Zone from	,000 m to 5,000 m(0.167 - 0.	833)	
	•	2		Filter All L1-S1E4 L2-S1E2	
				Delete	Delete all
				Name	L2-S1E2 ^
				Position num	3
	3		1	Material	B 500B 👻
				Diameter [mm]	16,0
				Number of b	2
				Area [mm^2]	402
				Layer type	Uniform 💌
				Cover type	Surface to 💌
				Cover [mm]	0,0
			0	Left bar	Before the 💌
	<u></u>	4		Right bar	Before the 💌
				Stirrup name	S1 👻 🗸
				Analysis model	Automatic design
Longitunidal reinforcement	New reinforcemer	t parameters	Type of beam	Reinforcement la	ayers area
New layer	Number of bars	2	beams and ribs 🛛 👻	Selected layers	402 mm^
Add bars to corners	Diameter [mm]	8,0		All layers	804 mm^
	Stirrup name	S1 N	Stirrups	Picture propertie	es
Parr positions	Edge index	2	Edit stirrups	Draw dimens	sions
Collision of bars		1	Edit cover	lexts scale	0.5
Collision	Between existin Move layer	ng bars	Save to template	Re	draw

Here you can create new layers by using the icons in the left bottom corner.

Create a new layer of 1 bar of 16 mm in the middle of the lower edge.

To do this you set the new reinforcement parameters:

- Number of bars:
- Diameter (mm): 16
- Stirrup name S1
- Edge index : 4

If these new reinforcement parameters are set, click on New layer button.

1

	Member B	3, Zone from 1,00	0 m to 5,000 m(0.167 - 0.8	833)		
	0	2		Filter All L1-51E4 L2-51E2 L3-51E4		
				Delete	Delet	e all
				Name	L3-S1E4	^
				Position num	4	
3			1	Material	B 500B 💌	
				Diameter [mm]	16,0	
				Number of b	1	
				Area [mm^2]	201	
				Layer type	No corner	*
				Cover type	Surface to	
				Cover [mm]	0,0	
	6			Stirrup name	S1	*
	<u></u>	4		Edge index	4	-
_				Detailing	🔲 no	~
	/			Analysis model	Automatic	design
ngitunidal reinforcement	New reinforcemen	t parameters	Type of beam	Reinforcement la	ayers area	
New layer	Number of bars	1 v	beams and ribs 🗸 🗸	Selected layers	201	mm^
Add bars to corners	Diameter [mm]	16,0 🗸		All layers	1005	mm^
2	itirrup name	51 ¥	Stirrups	Picture propertie	es	
	dge index	4 ~	Edit stirrups	Draw dimens	sions	
Bars positions	en e to/2010000		Edit cover	Texts scale	0.5	
comsion of pars	Retween evictin	a barr	Save to template	Re	draw	
(ollicion						

8. Confirm the input by **OK** button. Confirm reinforcement parameters "from the Concrete setup (Design defaults)" in the next dialogue by **OK** button again.

You can choose axonometric view by button 🙆 in View toolbar to see the new reinforcement bar



After the input of practical reinforcement layers, these layers are additional data on beam. They can be copied easily to other beams.

Properties of a reinforcement layer can be viewed and adapted in the Properties window.

- 9. We will now adapt the layer which contains the stirrups to change the stirrup distances.
 - a) Select the Reinforcement layer of the stirrups by clicking with the left mouse button on the

circled digit 1 _____ above grey dimension line below the beam.

b) Activate the properties of this **Reinforcement layer** in the **Properties Window** by selecting this entity with the mouse pointer in the combo-box:

P	roperties	4	×
S	itirrups layer (1)	🖃 Va Vy 🖉	
		6 ×	
	Name	SL	
	Type of zone	stirrups	
	Detailing	🔲 no	
	Position number	1	
	Material	B 500B	
	Diameter [mm]	8,0	
	Stirrups covers [mm]	30,0	
	Calculation of cuts	Automatic	•
	Type stirrup	single	•
	Stirrups distances [0,300	
	Real distance [m]	0,295	
	Diameter of mandr	4	
ŧ	Anchorage		
ŧ	Geometry		
ŧ	Description po		

c) Click the Action button Edit stirrup distances at bottom right corner

Actions	
Edit stirrup shape	>>>
Edit covers	>>>
Edit stirrups distances	>>>

d) The window Stirrup zones appears



- e) Click on the [New Part] button to enter a new part.
- f) Change the Input type to **distance + total distance**.

g) Change the **Distance [m]** field for this part from **0.300** into **0.100** and take **1** as **Total distance [m]**.

Note that the stirrups are put closer because the parts are automatically defined symmetrically from both ends.



h) Click [OK] to confirm your input.



- 10. The graphical representation of the practical reinforcement can be changed so that a 3D view of the reinforcement for so far can be obtained. To do this, follow the steps described below:
 - a) Right-click on an arbitrary position in the workspace.

b) In the context menu that is opened, select the **Set view parameters for all** option. The window **View parameter setting** appears.

On the **Concrete** tab, the reinforcement data are displayed under the heading **Concrete + Reinforcement**. Change the options in the following way:

• Set the option **Style of stirrups** to **All**.

• The option **Reinforcement** drawing type is set to 3D

• The option **Rounded bends** is set ticked on.

c) Confirm your input by [OK].

Below is shown a picture of a part of the practical reinforcement.

		View parameters	setting - Concrete
		Check / Uncheck group	Lock position
ne	⊲		😔 🌌 🔍 🕨
13	E	Check / Uncheck all	
	E	Service	^
		Display on opening the service	
n	E	Concrete + reinforcement	
0		Display	
		Main reinforcement	
		Style of main reinforcement	all 🔹
		Stirrups	
		Style of stirrups	all 🔹
		Number of stirrups	all
st.		Color of reinforcement	normal 🔹
		Scheme of reinforcement	
		Reinforcement drawing type	3D 🔹
		Rounded bends	
	E	Reinforcement labels	
		Display label	
		Type position number	
		Name	
		Diameter	
		Materials	
		Reinforcement area	
		Reinforcement position	
		Style of reinforcement position	positions on member 🔹
		Labels plane	local beam plane xz 🔍
f		Stirrup label	dimension 🔹
1		Type position number	local 🗸 🗸
		Show names in tab	OK Apply Cancel



11. It is also possible to show a bill of reinforcement which contains a list of all practical reinforcement in the project. The command can be found in the **Concrete 15** menu in chapter **Reinforcement input+edit**.

>>>



Report preview, opened by action button

is shown below.

SCIAENGINEER

Bill of reinforcement Selection : All

The lengths of longitudinal reinforcement bars and stirrups are calculated without rounded bends. Type of position number : Global

Member	#	Ø [mm]	Material	Length [m]	Number of bars	B 500B length [m]	B 500B weight [kg]
B3	1	8	B 500B	2,900	34	98,600	38,9
B3	2	16	B 500B	6,000	2	12,000	18,9
B3	3	16	B 500B	6,000	2	12,000	18,9
B3	4	16	B 500B	4,000	1	4,000	6,3
		8	Total for diameter			98,600	38,9
		16	Total for diameter			28,000	44,2
			Total for material			126,600	83,1
			Total			126,600	83,1

Note

Automatic picture generator creates individual picture with reinforcement scheme for every single beam or column. User right-mouse-click in the modelling window, click option Picture wizard at the bottom of

context menu
Picture Gallery Wizard:
Generate reinforcement schemes
<u>OK</u> <u>C</u> ancel

and then click [OK] to generate reinforcement schemes.

Reinforcement check (SLS)

There are several more checks in SCIA Engineer that controls also the serviceability limit state:

- Stress limitation
- Crack width
- Deflection



Crack control

- 1. In the Concrete 15 menu select Crack width (SLS) check function.
- 2. Press **<Esc>** to cancel the selection.
- 3. Options in **Properties window** are configured in the following way:
 - Type of selection is set to Current.

Type of loads is changed to Combinations and second combination named CO2 - SLS is chosen. Program automatically filters only combinations for serviceability limit state here.
 Values are set to UC (meaning Unity Check).

- 4. Select horizontal beam **B3** with the left mouse button.
- 5. Click the >>> next to **Refresh**, highlighted by red background. The crack control is carried out and the unity check is then displayed on the screen.



Note

In Report preview you can open Brief, Standard or Detailed output to see the calculation process that gives this unity check value and stress and strain of uncracked and cracked section pictures.

 It is clear from this result that the currently provided reinforcement does not satisfy the crack control check. Therefore we will make some adaptations to the practical reinforcement. Select reinforcement layer 4 and change the number of bars of diameter 16 mm into 3 bars.



7. If we repeat the steps 1 to 5 we can check again if crack control is satisfied.



It is clear that now the crack control is satisfied because we have reinforcement bars with shorter distance between them.

Detailing provisions

- 1. In Concrete 15 menu, choose the command Detailing provisions.
 - Reinforcement check (ULS+SLS)
 Internal forces
 Slenderness
 Stiffnesses
 Capacity-response (ULS)
 Capacity-diagram (ULS)
 Shear + Torsion (ULS)
 Stress limitation (SLS)
 Crack width (SLS)
 Deflection (SLS)
- 2. Properties window is configured in the following way:
 - Selection field is set to Current.
 - **Type of load** is irrelevant for the check of detailing provisions.
 - Output define as Brief.
 - Values set to UC (meaning Unity Check).
- 3. Select floor beam B3 with the left mouse button if not selected from previous check.
- 4. Click the next to **Refresh** and afterwards **Preview** in Actions.

			1								1			0	Properties		4 ×
					6										Check Detailing provis	ions (1) 📑 🔏 \	7 0
					1	0										4	10.15
				_	L	0			-		_				Name	Check Detailing pro	visions
														100	Selection		
					$\Upsilon/1$		_			Y I				2.4	Type of selection	Current	-
															Filter	No	-
					111										Result case		
			1	·····	111-1										Type of load	Load cases	*
				\	1									· ·	Load case	LC1 - Self Weight Co	onstr -
				1	- (1)	- 0	6	2							Extreme 1D	Global	
					-		C	5			-			- 14	Output	Brief	-
	I RECIRCIT	S. Mila		4								>			Run using Model D	23	
		A DESCRIPTION OF THE OWNER OWNER OF THE OWNER															
	+0;	co (der)	a. 113. M.											a x	Values	UC	*
view	- 0 P		a. 103. No.			10								ā ×	Values Contracting Setup	UC	
view D C De	iant			•) # (E) () () () () () () () () () () () () () (U								۶×	Values Contracting Setup Display value name	UC	
view D C De	int		1 23 4	*) R (A) m i nai	U								• ×	Values Drawing Setup Display value name Display values		
New New D	fait			-) R (E) m i mi	U		_						• ×	Values Drawing Setup Display value name Display values Display units		
view	fait			* / H (E) m i mai	۵					5CiA	ENGIN	IEER	• ×	Values Drawing Setup Display value name Display values Display units Display case		
	fait			•) # (F)n (m	U				5	5 C iA(ENGIN	IEER	• ×	Values Drawing Setup Display value name Display values Display units Display case Display section dx		
	****	ailing	g pro	vision)n m	۲				5	5 C iA(ENGIN	IEER	⊽ ×	Values Drawing Setup Display value name Display values Display units Display case Display section dx Display combinati		
Chec	fault	ailing	g pro	vision	s	Ð				5	5CiA	ENGIN	IEER	۰× ۲	Values Display value name Display values Display values Display units Display case Display section dx Display combinati Color scheme	UC	
Chec	fault	ailing	g pro	vision)n ma	U				5	5 C iA(engin	IEER	⇒ ×	Values Values Values Values Visplay value name Visplay values Visplay values Visplay case Visplay combinati Color scheme Graph type	UC	•
Chec Linear ci Load cas Extreme	fault	ailing	g pro	vision)n ma	U				5	5 C iA(ENGIN	IEER	a ×	Values Drawing Setup Display value name Display values Display units Display case Display section dx Display section dx Display combinati Color scheme Graph type Envelopes drawing	UC	•
Chec Linear ca Load cas Extreme Selection	fault	ailing	g pro	vision	S	6				9	5 C iA(ENGIN	IEER	a ×	Values Drawing Setup Display value name Display values Display units Display case Display section dx Display combinati Color scheme Graph type Envelopes drawing Label colour by gr	UC V Defined by result Filled transparent O to extrem V	•
Chec Linear ci Load cas Extreme Selection	fault fault ck Det alculation se: LC1 1D: Globa : 83 dx fm1	ailing	g pro	vision	S Mta	UC kong	UC shear [-1	UC	Check	9	5 C iA	ENGIN	IEER	× ×	Values Drawing Setup Display value name Display values Display values Display case Display section dx Display case Display section dx Display combinati Color scheme Graph type Envelopes drawing Label color by gr Drawing plane	UC V V Defined by result Filled transparent O to extrem V 3D	•
Chec Linear ci Load cass Extreme Selection	fault fault the Det alculation se: LC1 1D: Globa : 83 dx [m] 0.000	Case	g pro	vision	S M tat [ktm]	UC tors [-]	UC store [-]	UC [-]	Check	9	5 C iA(ENGIN	IEER	• ×	Values Drawing Setup Display value name Display values Display values Display case Display section dx Display combinati Color scheme Graph type Envelopes drawing Label colour by gr Drawing plane	UC V V Defined by result Filled transparent 0 to extrem V 3D	•

- 5. Shear reinforcement check shows value higher than 1,00 at particular span. We need to shorten the distance of stirrups to satisfy the detailing provisions.
 - a) Select the Reinforcement layer of the stirrups by clicking with the left mouse button on the

circled digit 1 _____ above grey dimension line below the beam.

- b) Activate the properties of this **Reinforcement layer** in the **Properties Window** by selecting this entity with the mouse pointer in the combo-box:
- c) Click the Action button Edit stirrup distances at bottom right corner
- d) The window **Stirrup zones** appears. Change **Distance (m)** value of the zone 1 from 0,3 m to 0,25 m and hit **[OK]** button.



6. If you repeat the Detailing provisions check you can see al values green with maximum UC = 0,99.



Check Detailing provisions

Linear calculation Load case: LC1 Extreme 1D: Global Selection: B3

Name	dx [m]	Case	N _{Ed} [kN]	M _{Edy} [kNm]	M _{Edz} [kNm]	UC long [-]	UC _{shear} [-]	UC [-]	Check
B3	1,500-	LC1	0,00	125,28	0,00	0,55	0,87	0,87	OK
B3	0,000	LC1	0,00	32,52	0,00	0,99	0,36	0,99	ОК

Note

The detailing provisions are checked according to Eurocode and its thorough setting can be viewed in the Concrete setting (structure).

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:

Document item						
New items P ×						
⊕ Special items						
E SCIA Design Forms (standalone)						
Inbox						
Project						
Elbraries						
E Sets						
H Solver and Mesh						
H- Structure						
H. Pasults						
E. Special						
E Custom check						
+ Pipeline						
E Concrete data						
🕀 Concrete						
🗄 Concrete 15						
🗄 Steel concrete bridge						
🗄 Geotechnics						
🗄 Composite Beam						
🗄 Composite Column						
🕀 Composite						
🕀 Mobile loads						
🗄 Influence lines						
🗄 Special						
Gallery pictures						
I						

- 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:

Tutorial Frame concrete

Navigator $- + \times$	New items $$	
📰 Materials 🔐 🕜 💽	▶ ← 飞船 田田 📦	
Cross-sections ••••		1. Materials
Members 🕰 🔿		Concrete EC3
	- Hinges 🔨 🔨	Culture2
mointernal forces on mem 🖆 🕐	Nodal supports	Name Type ρ Ε _{mod} μ α f _{ck.20} Colour
	E-Load	Ikg/m ² IMPa Im/mK IMPa
	H- Construction stages	C25/30 Concrete 2500,0 3,1500e+04 0.2 0,00 25,00
	D. Parultr	ReinforcementEC2
	Defended for the	
	Deformed structure	realine rype p Emod Omod U ryx [field] [MDpa] [MDpa] [MDpa]
	3D displacement	
	3D stress	B 500B Reinforcement stee /850,0 2,0000e+05 8,3333e+04 0,00 500,0
	- Internal forces on beam	
	Deformation on beam	2. Cross-sections
	Displacement of nodes	
	Acceleration of nodes	051
	Beatings	Type Rectangle
	Reactions	Detailed 500; 500
	Resultant of reactions	Shape type I hick-walled
	Nodal space support result	Item material C25/30
	- Intensity	Fabrication concrete
	Member Stress	Colour
	Shear in joint	A [m ⁴] 2,5000e-01

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 bending moment My.
 - 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.



Adding an image to the Report

- 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					
Insert Insert Close One at page Two at page Tit to page into selected report into inbox Picture size Picture size	Image: Solution of the soluti				
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					
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.

	Report_1 [Tuto	rial Frame Concrete.esa] - Engineering report	- 0 ×
 Form Export Export after external regeneration Save as template Save as preview template Check report data integrity Engineering report into Options Exit 	Print Derive: Printer Printer Printer Prooedies Settings Print All Pages Print All Pages Print Nurcolated 1,1,1,2,2,2,3,3,3 Portrat Orientation Portrat Orientation Portrat Orientation Portrat Orientation Rendered pictures with high quality	<complex-block><complex-block></complex-block></complex-block>	Î

Note:

Any output, standard or deatiled, from the Section check in Concrete 15 can be easily send to Engineering Report by the button at the top of the dialogue:



Epilogue

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

After reading the text and executing the example the user should be able to model and calculate simple structures consisting of concrete beams and columns.

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