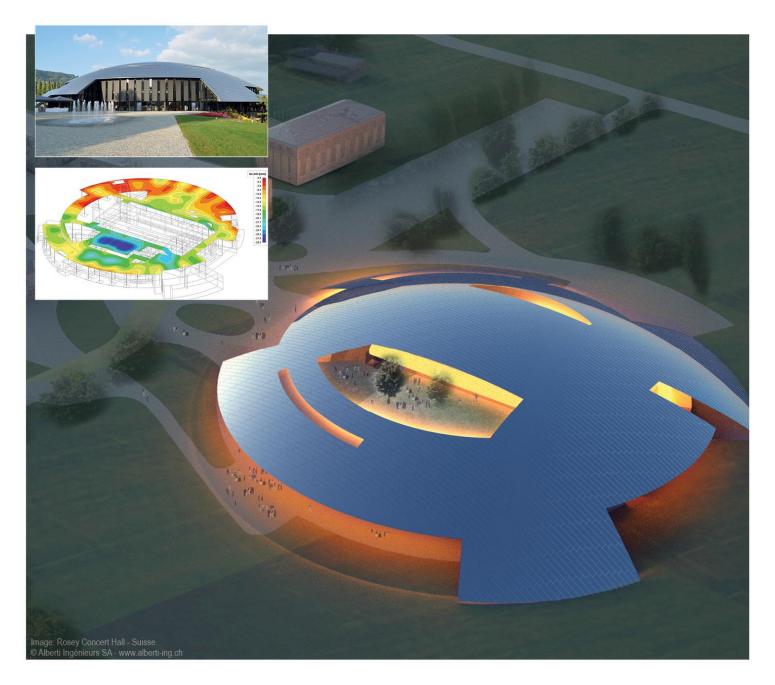
SCIAENGINEER



Tutorial Concrete Slab (ACI 318-08)

All information in this document is subject to modification without prior notice. No part or 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 2016 SCIA. All rights reserved.

Table of Contents

General Information	1
Welcome	
Scia Engineer Support	1
Website	1
Introduction	2
Getting started	3
Starting a project	
Project management	5
Save, Save as, Close and Open	5
Saving a project	
Closing a project	
Opening a project	
Geometry input	6
Input of the geometry	
Supports	
Check Structure data	
Checking the structure	
Use of labels	
Graphic representation of the structure	
Input of the Calculation Data	20
Load Cases and Load Groups	
Defining a Permanent Load Case	
Defining a Variable Load Case	
Loads	
Combinations	
Calculation and Mesh generation	27
Mesh generation	
Linear Calculation	
Results	29
Viewing results	
Post processing	34
Automatic mesh refinement	
Reinforcement design	37
Pre-processing	
Required areas	
Input user reinforcement	41
Engineering Report	
Engineering Report Saving Model Views or Results to Engineering Report	44

General Information

Welcome

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

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

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

- Concept
- Professional
- Expert

License version

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

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

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

Demo version

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

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

Scia Engineer Support

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

By e-mail

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

By telephone

From USA: 443-542-0637

Via the Scia Support website

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

Website

Link to Tutorials

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

Link to eLearning

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

Link to Demo version

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

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

Introduction

The example of this Tutorial can be designed with the **Licensed** or **Student** Versions. Before you proceed, you must be familiar with your operating system: for instance working with dialogues, menu bars, toolbars, status bars, handling the mouse, etc.

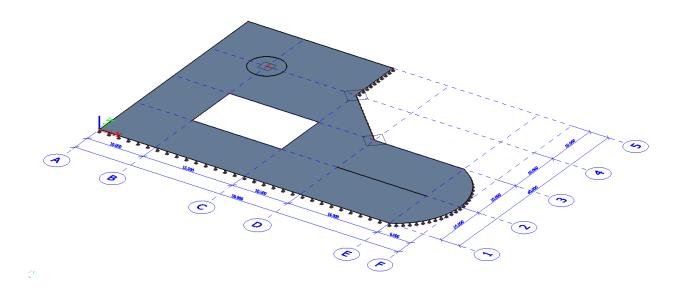
This Tutorial describes the main functions of Scia Engineer for the input and calculation of a plate.

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

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

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

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



Getting started

Starting a project

Starting the program

Before you can start a project, you need to start the program first. To do this, Double-click on the Scia Engineer shortcut in the Windows Desktop or if the shortcut is not installed, click [Start] and choose Programs > Scia Engineer 14 > Scia Engineer 14.

If the program does not find any protection, you will obtain a dialogue indicating that no protection was found. A second dialogue will list the restrictions of the demo version. Click **[OK]** in both windows.

For this Tutorial, you must start a new project.

Starting a new project

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





In the Select New Project dialogue, choose for the Analysis environment by double clicking on the corresponding icon.

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

Project data	nctionality Load	ds Code Setup Protection			X
Scia	Data		Material]	
Engineer	Name:	Plate	Concrete	C4000	
	Part:	-	Material Steel	C4000 💌	
			Timber		
	Description:	Tutorial Plate Concrete	Other Aluminium		
	Author:	RY			
	Date:	23. 12. 2014			
	Structure:		Code National Code:		
	General XYZ		IBC		
	Project Level:	Model:			
	Advanced	▼ One ▼			
	'Thick-walled' Cor	ncrete cross-sections: the advanced 2D FE	M method is off!		
				OK Cancel	

- 3. In the **Data** group, enter your preferred name, part, description, author and data. These data can be mentioned on the output, e.g. in the document and on the drawings.
- 4. Select **General XYZ** in the **Structure** field. The structure type (Frame XZ, Frame XYZ, Plate XY, General XYZ...) will restrict the input possibilities during the calculation.

- 5. Choose the Project level: Advanced and Model: One.
- 6. In the Material group, select Concrete.

Below the item **Concrete**, a new item **Material** will appear.

- 7. Choose C4000 from the menu.
- 8. Click on the rectangular button below **National Code** to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the project of this tutorial, choose **IBC** which appears as default. The window **Codes in project** shows different available codes. To access a separate code click **[Add]**.
- 9. The dialogue Available national codes is opened and other codes can be chosen and confirmed by clicking [OK].



- Other codes can be chosen and confirmed by clicking [OK].
- 10. Confirm your code input with [OK].

Notes:

On the **Basic data** tab, you can set a project level. If you choose "standard", the program will only show the most frequently used basic functions. If you choose "advanced", all basic functions will be shown.

On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program.

On the Combinations tab, you will find the values for the partial safety factors. In this Tutorial, we will use the default settings.

Project management

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.

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 press while twice, the project is automatically stored with the same name. If you choose **File > Save as** in the main menu, you can enter a new/other drive, folder and name for the project file.

Closing a project

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

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

Opening a project

Click on

to open an existing project.

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

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.

Line grids and storeys

1. The 2D line grids can be inserted through the first option in the Main window, Line grid and Storeys. Once in the menu, select the Rectangular grid in the 2D Line grid menu.

Line grid and storeys	×
2D Line grid Free lines Circular grid 3D Line grid Storeys	
New Close	

2. Generate the following grid lines and confirm by clicking [OK].

Dir X -			Туре	Sp	an	•	Dir Y			Туре	Sp	an	•
	Name	X [ft]	dx [ft]	Rep	SL			Name	Y [ft]	dy [ft]	Rep	SL	
1	Α	0.000			no	-	1	1	0.000			no	+
2	В	10.000	10.000	1	no	Ψ.	2	2	10.000	10.000	1	no	-
3	С	26.000	16.000	1	no	+	3	3	20.000	10.000	1	no	+
4	D	36.000	10.000	1	no	-	4	4	30.000	10.000	1	no	+
5	E	52.000	16.000	1	no	-	5	5	40.000	10.000	1	no	+
6	F	58.000	6.000	1	no	-	*		0.000	0.000	0		+
*		0.000	0.000	0		-		-					
V Ge	enerate name	e automatica	ally				V Ge	nerate name	automatica	ily			
									0	ĸ	Cancel		pply

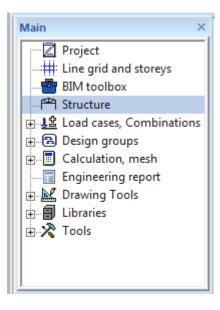
3. Accept the default values in the 2D line grid window.

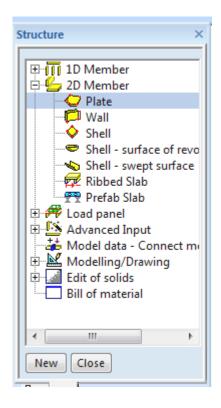
	(so) Name	LG1
	туре	Span
2600	Co-ordinate	
;;;;;;;;;;;;-	v x	1*10.000;1*16.000;1*10.000;1*16.000;1*6.000;
2000	Y	1*10.000;1*10.000;1*10.000;1*10.000;
	(m) Rotation [deg]	0.00
0000	Label	
	X (N)	A;B;C;D;E;F;
2000	V Y	1;2;3;4;5;
Ň	Direction X	Begin
2000 2000 2000	Label shift X [ft]	3.281
6000	Direction Y	Begin
	Label shift Y [ft]	3.281
A) (B) (C) (D)	Draw label in	Circle
	Dimension	
	Direction X	Begin
	Direction Y	Begin
	Grids dimensions	V yes
	Overall dimensions	V yes
	Style name	LG

4. In the comment line insert 0;0;0 in order to locate the A1 point on the origin: Rectangular grid - Enter point > 0;0;0

Structure menu

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





2. In the **Structure menu** different branches will appear, in accordance with the already input items, i.e. support branch will appear if a structure is physically available.

We will input the structure as a **2D member > Plate**. We will also use the advanced input options, like definition of an opening in the slab and drawing of a plate rib.

Input of a plane 2D member

- 1. In the structure menu double click on the Plate in the 2D member branch
- 2. The window **2D member** will be opened.

Name	S1	
Туре	plate (90)	
Analysis model	Standard	
Material	C4000	· .
γ FEM model	Isotropic	
FEM nonlinear model	none	
Thickness type	constant	
Thickness [inch]	10.000	
Member system-plane at	Centre	
Eccentricity z [inch]	0.000	
LCS type	Standard	
Swap orientation	🔲 no	
LCS angle [deg]	0.00	
Layer	Layer1	· .

- 3. Define the following plate properties: name=S1, type=plate (90), Material=C4000, Thickness = 10in.
- 4. After accepting with [OK] the program asks in the command line for the starting point of the polyline.
- 5. The buttons in the **command line** allow one to build up the polygonal edges using different line types, or to directly choose a circular or rectangular surface.

- 1

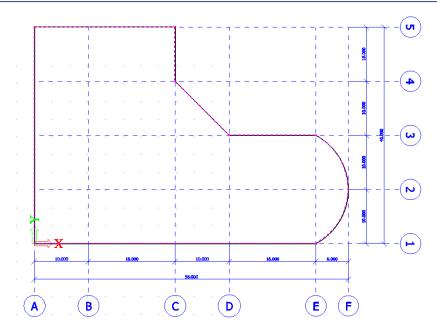
Command line						
R D D O D D C V X	s 7 R	\land \land a	· X 🔨 🐔	7. 🖂 🗶		なびた 10
New 2D member - New polygon - Start point	•					

6. The geometry can be input with help of a dot or line grid or with use of the mouse or direct input of coordinates in the command line which the former is used here:

Starting point:	A1 <click></click>
	E1 <click></click>
	Choose the New circular Arc symbol on the Command line
New polygon – Circul	ar Arc – Intermediate point:
	F2 <click></click>
	E3 <click></click>
	D3 <click></click>
	C4 <click></click>
	C5 <click></click>
	A5 <click></click>
	A1 <click></click>

7. To End the command, right mouse click or press <ESC> in order to close the input of the polygon.

The following picture is now depicted in the screen:

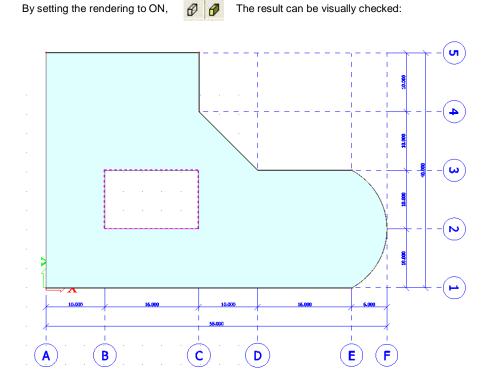


Definition of an opening

- 1. In the Structure menu under 2D member components create an Opening.
- 2. In the command line, quick access buttons are available for quick definition of geometrical outlines like circular or rectangular shapes. By default, definition by drawing of a closed polygon is started, in this example we will define a rectangular opening.
- 3. The two nodes of its diagonal define a **rectangle**. This is also depicted by the two red dots on the icon.

New rectangle > Starting Point: 10;10;0 End Point: 26;20;0

The program displays the proposed rectangle. Accept the New Rectangle by clicking the right mouse button.



Definition of a subregion

1. Click on the Subregion in the 2D member components.

Subregion			
	Name	R1	
	Material	C4000	Υ.
	Thickness type	constant	
	Thickness [inch]	16.000	
	Member system-plane at	Bottom	
	Eccentricity z [inch]	0.000	
	2D member	S1	
x			
	-		OK Cance

- 2. Define the following properties: name=R1, Material=C4000, Thickness = 16in, Member system plate at=Bottom.
- 3. Define a circular subregion on the slab. First, click on the circle symbol on the command line to define the circle extents.

New circle > Center Point: B4 Point on circle: 7;30;0

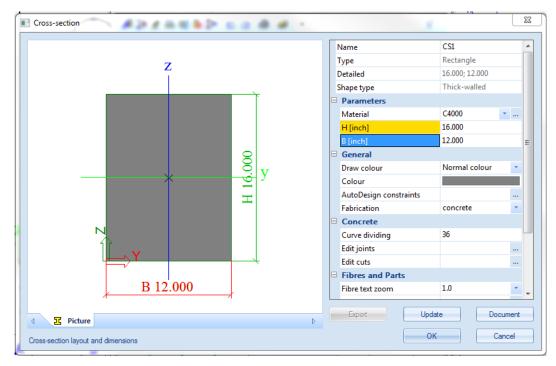
Input of plate rib

- 1. In the Structure menu, under 2D member components choose Rib.
- 2. If no cross-section has been defined in the project, the dialogue New cross-section will open.

Here we will be able to select and define from the group Concrete a rectangular cross section as the new rib.

New cross-section				×
Available groups	Available items of this group		•	Items in project
	Profile Library filt	er	v	Add Close

3. Click [Add]. This will take us to a new dialogue **Cross-section**. For this Tutorial we will use a rectangular concrete cross-section with height 16in and width 12in and a concrete grade of C4000.



- 4. Accept the cross-section by pressing the [OK] button. After this we will [Close] both dialogues.
- 5. In the **Plate rib** window we can define the parameters of the ribs as shown below:

Plate rib			×
	Name	B1	
	Type rib	plate rib (92)	
	Analysis model	Standard	-
	CrossSection	CS1 - Rectangle (16.000; 12.000)	·
	Alignment	Bottom	-
	Shape of rib	T symmetric	-
	Effective width	width	-
	for int. forces [inch]	36.000	
	for check [inch]	36.000	
	FEM type	standard	-
	Buckling and relative lengths	Default	
w	Layer	Layer1	×
X	,		
		ОК	Cancel

6. After clicking **<OK>** we will have to define the starting and end points of the ribs. After the input in complete, right click or press **<ESC>** to end the command.

Starting point: E2 End point: D2

Notes:

Various properties of a plate rib can be manually defined in order to modify the plate ribs effect on the structure.

roperties			>
Member (1)	🖃 Va V	4 1	1
a	•	6	6
Name	B1		*
Type rib	plate rib (92)		
Analysis model	Standard	*	Ε
CrossSection	CS1 - Rectangle (16.000; 12.(🔻		
Alignment	Bottom	*	
Shape of rib	T symmetric	+	
Effective width	width	+	
for int. forces [inch]	36.000		
for check [inch]	36.000		
FEM type	standard	+	Ŧ

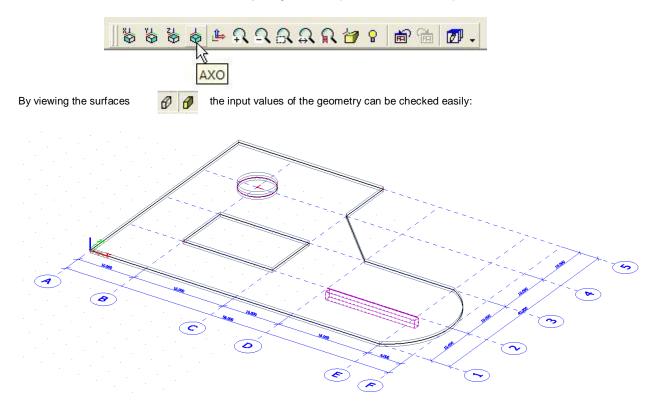
Effective Width:

Width: The user can input the width for the internal forces (FE analysis) or the checks (Design As) by hand.

Number of thickness: The width of the slab for the rib is defined as a factor times the plate thickness. The user enters the factor by hand.

Default: The width of the slab for the rib is defined as a factor times the plate thickness. The factor is set in **Setup > Solver > Number of thicknesses of rib plate**

The user can also ask for a 3D view on the slab by using the button [view in direction AXO]



Supports

The input of the geometry can be finalized by the definition of the support conditions. We assume that the whole edge of the slab is supported in global z-direction, thus simulating a wall or strip foundation support.

Definition of a support on an edge

- 1. Select in the Structure menu Model data > Support > line on 2D member edge
- 2. The window Line support on 2D member edge will pop up.

Line support on 2D member edge			×
	Name	Sle1	
	Constraint	Hinged	-
	x	Rigid	*
	Y	Rigid	*
<mark>t</mark> Rz	z	Rigid	*
7	Rx	Free	*
	Ry	Free	*
Dy X	Rz	Free	*
Rx	Geometry		
x1 x2	System	GCS	*
¥ ** 💉	Position x1	0.000	
	Position x2	1.000	
z	Coord. definition	Rela	+
1	Origin	From start	*
x Y			
			OK Cancel
			Canoci

- 3. Constrain the translational degrees of freedom along the edges of the slab, creating a Hinged Support.
- 4. Select the edges around the slab one by one: edge1, edge2, edge3, edge4, edge5, edge6, edge7.
- 5. Press <ESC> to finish the input command

Input of Nodal supports

1. In order to input the nodal support for the subregion, we will use the option **Model data > Support > in Node** in the **Structure menu.**

Support in node		×
	Name	Sn1
	Туре	Standard 💌
	Angle [deg]	
	Constraint	Hinged 💌
↓ ^{Rz}	x	Rigid 💌
	Y	Rigid 💌
	Z	Rigid 💌
XXX	Rx	Free
Rx Ry	Ry	Free
(i) ~	Rz	Free
\smile	Default size [ft]	0.656
IZ	Geometry	
	System	GCS 💌
x γ		
		OK Cancel

- 2. Constrain the translational degrees of freedom of the centroid of the subregion, creating a **hinged** support. This support may for instance account for a column under the slab.
- 3. Apply the nodal support at B4 point.

Notes:

If required a flexible support can be defined in order to model the behavior of the columns more adequately. Also only can model the supports as "column", then the stiffness is directly derived for the entered column data. A set of shortcuts of supports is defined in the **Command line**. In this project the button **Hinged Support** could have been used.

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, null members, duplicate members and other incorrect data references.

Checking the structure

- 1. Press on **<Esc>** or click on the button **Cancel Selection** in order to ensure that no entities are selected.
- 2. Double-click on the Check Structure data option in the Structure Menu or click on the ¹/₁ icon in the toolbar.
- 3. The window Check of structure data will pop up for which a different set of checks is depicted.

Check of structure data	×
Check of nodes	
Search nodes	
Search duplicate nodes	☐ Ignore parameters
Check of members	
Check members Search null members	Null members: 0
Search duplicate members	Duplicate 0
	Invalid parts: 0 Delete invalid parts
Check of data references	
Check data references	C Memory efficient method
	Fast method
Check of additional data	
Check additional data position	Invalid position
	Correct position
Check free load distribution points	Invalid loads 0
Check of steel connections	
Check steel connections	Invalid 0 Delete invalid connections
Check load panels Check cross	links
Check additional data Check duplicity	of names Check Cancel

- 4. In the bottom four checking options for load panels, cross-links, additional data and duplicity of names are available to be selected individually. For this example, click [Check].
- 5. The message box Data Check Report opens.

Data check report
Data check finished.
ОК

6. Close the check by clicking [OK].

Use of labels

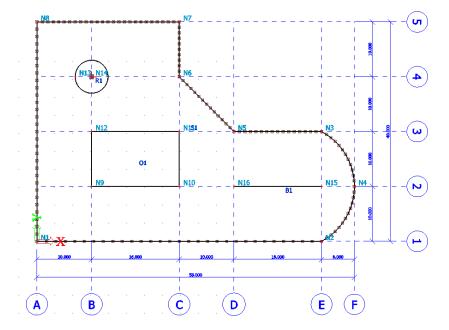
In order to display the names of the members, nodes or support symbols the labels of each item can be turned ON / OFF by the shortcut buttons in the lower left corner of the graphical screen $\mathcal{O} \mathcal{O} \mathcal{O} \stackrel{\texttt{L}}{=} \mathcal{O} \stackrel{\texttt{W}}{=} \mathcal{O}$

The 3rd button can visualize supports.

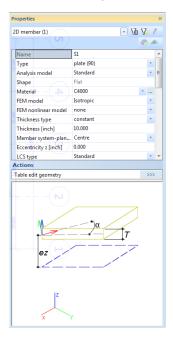
Labels of nodes can be activated by the button Labels of nodes can be activated by the button

Labels of bars can be activated by the button 🕮 located above the **Command line**.

A view in direction Z 😸 shows the following:



If the slab is selected by single clicking with the left mouse button on the 2D member edge, the properties of the slab can be reviewed in the **Properties window**:



The properties contain for additional information including, attached line supports, internal nodes, openings and ribs.

Graphic representation of the structure

Edit view

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

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

Editing the view point on the construction

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 structure, 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.

Note:

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

Setting a view direction with regard to the global coordinate system

- 1. Click on the button View in direction- X 🕌 for a view in the X-direction.
 - Click on the button View in direction-Y by for a view in the Y-direction.
- 3. Click on the button View in direction- Z 🎇 for a view in the Z-direction.

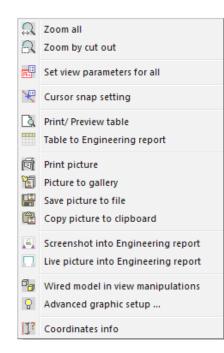
The magnifier

2.

- Use to enlarge.
- Use Sto decrease.
- Use to zoom in on a window.
- Use to view the whole structure.
- Use Is to zoom in on the selection.

Editing view parameters through the menu View parameters

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



Note:

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

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

View parameters – Entities

Using the tab page **Structure** graphical representation of various entities can be adjusted. From this tab page the following items are of importance:

Style and Color: Changing the colors by layer, by material, by cross-section, or by structural type.

Draw Cross-section: With this tick box a graphical view of the cross-section is shown in the reference line of a member.

Local Axes: Using this tool the local axes can be shown and modified for nodes, 1D and 2D members.

]0	heck / Uncheck group				Lock po	sition	
	🖴 Structure 🛛 🔠 Labels 🛛 👗 Mo	odel	Modelling/Drawing	🔗 Attributes	Misc.	View	
7	Check / Uncheck all						
Ξ	Service						1
_	Display on opening the service	V					Π
Ξ	Structure						
	Style + colour	norma	I			•	Ī
	Draw member system line	V					۱
	Member system line style	system	line			•	Ī
	Model type	analysi	is model			•	Ì
	Display both models						
	Member surface						
	Rendering	wired				-	Ī
	Draw cross-section						
	Cross-section style	section	ı			-	Ī
	Profiled sheeting orientation	V					
	Draw cross-section	~					1
Ξ	Panel						
	Member surface						
	Rendering	wired				-	Ī
	Highlight supporting edges/nodes						
	Load distribution symbol	~					
	Display linked members	~					
Ξ	Structure nodes						
	Display	~					
	Mark style	Dot				-	
Ξ	Member parameters						
	System lengths						
	Member nonlinearities	V					
	FEM type	V					
	Joists						
	Substitution boundary	✓					
Ξ	Local axes						
	Nodes						-

View parameters – Labels and description

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:

- Cross-section Name: The name of the cross-section (ex. CS1) is visible in the label.
- Cross-section type: Show the cross-section type (ex. W14x22) in visible in the label.

B1/CS1-Rectangle (16.000; 12.000)/4876800.000 mlm/Layer1/plate rlb (92)	

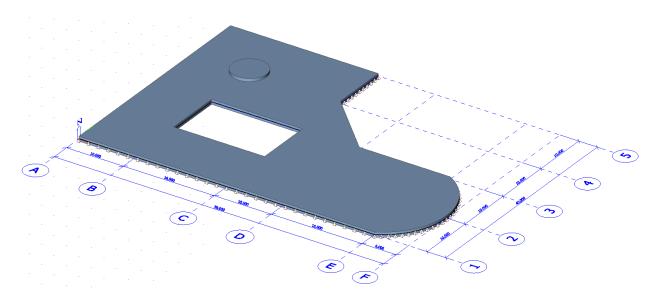
- Length: show the length of the beam in the label.
- Display labels: Only when this tick box is turned ON, will the labels be visible on the graphical screen.

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 elements.
- Render geometry 🗗 to view the rendered members.
- Show/hide supports to show supports and hinges.
- Show/hide load ¹ to show the load case.
- Show/hide node labels to view the label of the nodes.
- Show/hide member labels ^{BEC} 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 it o quickly access to the options from the menu View parameters.

After rendering the following structure is obtained:



Input of the Calculation Data

Load Cases and Load Groups

Each load applied to the structure is attributed to a specific load case and additionally, a load case may contain different load types. Each load case has properties that are attributed to it, which are used to determine how the loads within the case are utilized in combinations. Additionally, load cases include action types (permanent or variable) which guide the load case to specific load groups.

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

The way in which load cases are defined, is essential for the correct creation of load combinations by the generator. For more information concerning load cases, groups and combinations please review the chapter about loads and combinations in the reference manual.

In this project, two load cases are entered:

- LC1: Dead load
- LC2: Live Load Case

Defining a Permanent Load Case

- 1. Double-click on Load in the Main window.
- 2. Before you can define loads, you first must enter load cases. Since this project does not contain any load cases yet, the Load Cases Manager will automatically appear.
- 3. By default, 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 by means of this type.
- 4. In the Description field, you can describe the content of this load case. For this project, enter the description "Self Weight".
- 5. Click New and enter Dead load in the Name field. This load has Permanent Action type and a Standard load type.

Load cases			×
🏓 🤮 🗶 🛍 🔛 🔁	🖸 🗠 🕼 🕼 🚰 🛛 🗛		• 7
Self weight	Name	Dead	
Dead	Description		
Live	Action type	Permanent	Ψ.
	LoadGroup	LG1	×
	Load type	Standard	Ψ.
	Loads cause appreciable sidesway		
	Actions		
	Delete all loads		>>>
	Copy all loads to another loadcase		>>>
New Insert Edit	Delete		Close

Defining a Variable Load Case

- 1. Click New or to create a second load case.
- 2. Enter the description "Live".
- 3. As this is a variable load, change the Action type to Variable.
- 4. The Load Group LG2 is automatically created. Click ... to display the properties of the Load Group.

Load groups			٢
🔎 🤮 🥵 💕	k 🗅 🗠 🖨 🕞		
LG2	Name	LG2	
	Relation	Standard	-
	Load	Variable	-
	IBC - load type	L - Live load	-
	Combination factor f1	Public assembly and garages	-

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

🔎 🤮 🖋 🕏	📴 🗠 🗠 🚭 🖓 🗛		• 7
Self weight	Name	Live	
Dead	Description		
Live	Action type	Variable	
	LoadGroup	LG2	×
	Load type	Static	
	Specification	Standard	
	Duration	Short	-
	Master load case	None	-
	Loads cause appreciable sid	esway 🔲	
	Actions		
	Delete all loads		>>>
	Copy all loads to another loa	adcase	>>>

6. Click [Close] to close the Load cases manager.

Notes:

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

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".

Loads

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

- Self weight
- Dead
- Live

Switching between load cases

Different load cases can be selected through the tab in the Load window:

Load ×
Live 🔤
Self weight
Live
🗄 🏄 Surface load
🗄 🛲 Thermal
⊞…) Moment
🚟 Line moment on beam
🛛 🖓 Line moment on 2D member ed
🗄 🦸 Point displacement
🕀 🚟 Line displacement
2D member displacement, curva
Plane generator
Pond load - water accumulation
Not calculated internal forces
🗄 🐝 Traffic loads

Entering the Self weight

The self-weight loads are automatically generated. The user is not required to calculate these action loads.

Input of dead loads

- 1. Choose **Dead** in the load window tab.
- 2. Select on 2D member edge in the Line force option.
- 3. Insert a -0.25 kip/ft line load as the load value.

	Name	LFS1	
	Direction	Z	
52	Туре	Force	
-P2	Distribution	Uniform	
	Value - P [kip/ft]	-0.25	
-P1	Geometry		
	System	LCS	
	Location	Length	
	Position x1	0.000	
x1	Position x2	1.000	
x2	Coord. definition	Rela	
· · · · · · · · · · · · · · · · · · ·	Origin	From start	
Z			
\checkmark			
Υ Y			

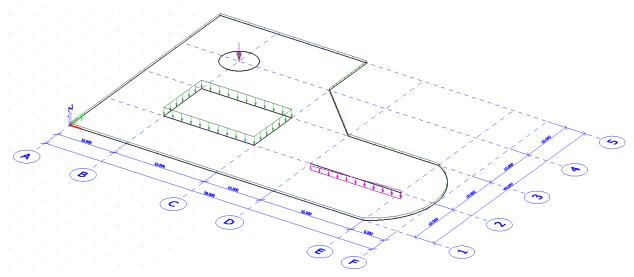
4. Select the outer slab edges of the plate and when finished press <ESC>.

- 5. Similarly insert a -1.0 kip/ft line load on the edges of the opening accordingly.
- 6. Next, apply a trapezoidal load on the beam varying from -1 kip/ft to -0.5 kip/ft. Select on beam in the Line force option and choose Trapez in the Distribution field and insert P1 and P2 as shown below:

P-7)	Name	LF1	
	Direction	Z	*
RX RY 1-P2	Туре	Force	· · · · · · · · · · · · · · · · · · ·
	Angle [deg]		
-P1	Distribution	Trapez	*
-PI	Value - P1 [kip/ft]	-1.00	
ey tez	Value - P2 [kip/ft]	-0.50	
	Geometry		
x1 x2	System	LCS	-
(i) × ···	Location	Length	
	Extent	full	-
	Coord. definition	Rela	-
q _{max} q _{min} q _{max} -q _{min}	Position x1	0.000	
q_{max} q_{min} $Q = \frac{q_{\text{max}} - q_{\text{min}}}{q_{\text{max}} + q_{\text{min}}}$	Position x2	1.000	
The second secon	Origin	From start	-
	Eccentricity		
	Eccentricity ey [mim]	0.000	
	Eccentricity ez [mim]	0.000	

- 7. Finally, input a point load by selecting on **in node** under the **Point Force** heading and apply **-5 kip** load at location B4 on the subregion.
- 8. Press [ESC] to confirm your selection.

The layouts of the Dead loads are as following

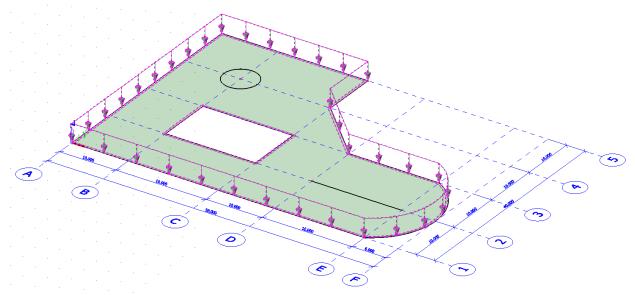


Input of live surface loads

- 1. Select the Live load from the Load menu.
- 2. Click on Surface load on 2D member in the Load menu and apply -0.04 kip/ft² for Value.

Surface force	,		×
	Name	SF1	
	Direction	Z	-
	Туре	Force	-
• •	Value [kip/ft^2]	-0.04	
-P	Geometry		
The work	System	LCS	*
	Location	Length	
MILLITER.			
x y			
		0	Cancel

- 3. Confirm your input with **[OK]**.
- 4. The program will then ask the user to define the surface of the free surface load. As the slab has been defined as a closed polygon, selecting any point on the outer edges of the slab would select the surface.
- 5. Confirm your selection with **<ESC>**.



Adapting a load

- 1. Select **Dead** load in the **Load menu**.
- 2. Select the line loads around the opening by clicking them with the left mouse button.
- 3. The common properties of the 4 loads are displayed in the **Properties window**.
- 4. Change the Value from -1.0 kip/ft to -0.75 kip/ft in the Properties window.
- 5. Confirm the modification with **<ENTER>**.
- 6. Press **<ESC>** to finish the selection.

Properties				>
Line force on 2D m	ember edge (1)	- 74	74	Ø
			6	8
Name	LFS9			4
Direction	Z		-	
Туре	Force		*	
Distribution	Uniform		-	
Value - P [kip/ft]	-0.75			
Opening/Panel	01			
Load case	Dead		•	Ξ
Geometry				
System	LCS		*	
Location	Length			
Edge	2		•	
Position x1	0.000			
Position x2	1.000			
Coord. definition	n Rela			-

Combinations

After input of the load cases is complete, it is possible for them to be grouped in combinations. In this project, one combination is created in order to evaluate the self-weight, dead and live loads together.

Defining Combinations

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

Combination - D+L	<u> </u>
Contents of combination	List of load cases
Dead Live	◆ Dead ◆ Live
Name : D+L	Delete Add
Coeff: 1 Correct Type: Linear - ultimate	Delete All Add All
Description	
Nonlinear combination	OK Cancel

- 3. By means of the button [Add all], all load cases can be added to the combination.
- 4. Change the Type to Linear ultimate.
- 5. Confirm your input with [OK]. The Combination Manager is opened. Name the combination as D+L.
- 6. Click [Close] to close the Combination manager.

Notes:

There are 3 main combination types available for the IBC code:

Envelope - user defined base for automatic generation of combinations (both ultimate and serviceability)

Linear - user defined combination using user defined combination coefficients

Code Dependant - According to the active code set for the project and the applicable load groups

Combinations		×
🥕 🤮 🏂 👪 🔽 🖂 e	Input combinations	
D+L	Name	D+L
	Description	
	Туре	Linear - ultimate
6	Contents of combination	
	Self Weight [-]	1.00
	Dead [-]	1.00
	Live [-]	1.00
New Insert Edit Delete		Close

Calculation and Mesh generation

The calculation of the slab will be done according to the finite element method. According to the calculation method a mesh will be generated on the slab and the results will be calculated at each finite element created by the mesh. The result in the middle of a finite element is determined as the average value of the results in the four nodes of the typical quadratic element.

Mesh generation

Mesh setup

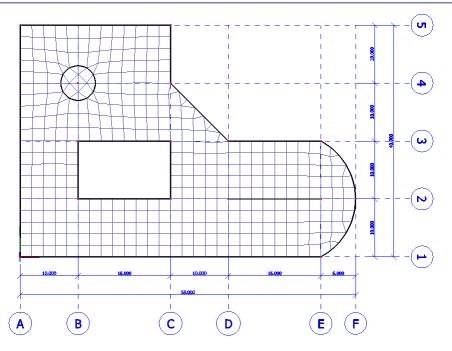
- 1. In order to generate the mesh, select **Mesh setup** in the **Calculation, mesh** option from the **Main menu**.
- 2. In the Mesh setup dialogue box insert 24" for the Average size of 2D element/curved element/nonlinear soil spring.

Mesh setup	X
Name	
General mesh settings	
Minimal distance between two points [m]	0.001
Average number of tiles of 1d element	1
Average size of 2d element/curved element [inch]	24.000
Definition of mesh element size for panels	Manual 👻
Average size of panel element [inch]	39.370
Elastic mesh	
Use automatic mesh refinement	
1D elements	
Minimal length of beam element [inch]	3.937
Maximal length of beam element [inch]	39370.079
Average size of cables, tendons, elements on subsoil, nonlinear soil spring [inch]	39.370
Generation of nodes in connections of beam elements	
Generation of nodes under concentrated loads on beam elements	
Generation of eccentric elements on members with variable height	
Division on haunches and arbitrary members	5
Division for 2D-1D upgrade	50
Mesh refinement following the beam type	None 👻
2D elements	
To generate predefined mesh	
To smooth the border of predefined mesh	
Maximal out of plane angle of a quadrilateral [deg]	1.719
Predefined mesh ratio	1.5
Hanging nodes for prestressing	
	OK Cancel

- 3. Confirm by selecting [OK].
- 4. Choose the Mesh generation i option to create the mesh based on the selected finite element size.

Mesh successfully created Number of nodes: 442 Number of 2D elements: 397 Number of 1D elements: 8		neer 14.0.1043- student vers
ОК		Number of 2D elements: 397
		ОК

5. The mesh can be viewed by selecting Mesh in Structure within the Fast adjustment of view parameters on whole model symbol above the Command line.



Linear Calculation

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

Solver settings

- 1. Double-click on below in the Main window.
- 2. The FE Analysis window appears. Click [OK] to start the calculation.

FE analysis		×
Scia	Single analysis Batch analysis	
Engineer	• Linear calculation	
	C Nonlinear calculation	
	C Modal analysis	
	C Linear stability	
	C Concrete - Code Dependent Deflections (CDD)	
	C Construction stage analysis	
	C Nonlinear stage analysis	
	C Nonlinear stability	
	C Test of input data	
	Number of load cases: 3	
	Solver setup Mesh setup	
	OK Cancel	

3. After the calculation, a window pops out and provides the maximum translation and rotation in the FE mesh in the simulation.

Results

Viewing results

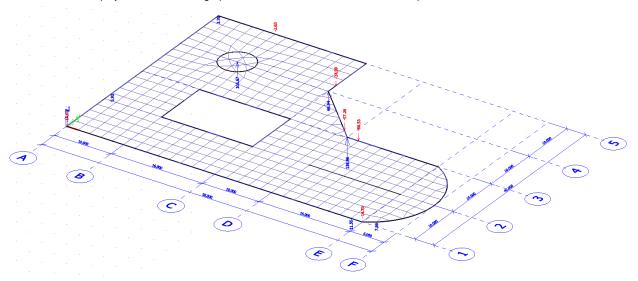
After the calculation is executed, the results can be viewed.

Viewing the Reaction Forces

- 1. Open the **Results** service from the **Main Window.**
- 2. Below Supports in the Results service, click Reactions.
- 3. The options in the **Property Window** are configured in the following way:
 - The Selection field is set to All.
 - The Load type is set to **Combinations** and the Combination to **D+L**.
 - The Values are wanted for Rz.
 - The Extreme field is changed to Node.

Properties		×
Reactions (1)	- 1	V/ /
		8
Name	Reactions	
Selection	All	*
Type of loads	Combinations	*
Combinations	D+L	*
Filter	No	*
Values	Rz	*
Extreme	Node	*
Drawing setup 1D		
Rotated supports		
Actions		
Refresh		>>>
Preview		>>>

4. The action **Refresh** has a red background, i.e. the graphical screen must be refreshed. Click on the **Series** button behind **Refresh** to display the results in the graphical screen in accordance with the set options.



- 5. To display these results in a table, the **Preview** action is used. Click on the **Preview** to open the Preview.
- 6. Additionally, the results can be accessed in a dynamic table format by using the Table Results.

	Support	Case	dx [inch]	Rx [kip]	Ry [kip]	Rz [kip]	Mx [kipft]	My [kipft]	Mz [kipft]
	Sn1/N14	D+L/1		4.27	-10.67	88.49	0.00	0.00	0.00
	Sn1/N14	D+L/2		5.11	-12.52	105.67	0.00	0.00	0.00
	Sle1/S1	D+L/2	432.000	-0.21	-0.39	4.10	0.00	0.00	0.00
	Sle1/S1 mented	D+L/2	168.000	0.07	-0.20	4.85	0.00	0.00	0.00
	Sle1/S1	D+L/2	72.000	-0.06	-0.48	3.91	0.00	0.00	0.00
	Sle1/S1	D+L/2	576.000	-0.16	0.47	3.47	0.00	0.00	0.00
	Sle1/S1 C 1	D+L/2 1 -	624.000	-0.16	0.26	-14.70	0.00	0.00	0.00
	Sle1/S1	D+L/2	600.000	-0.19	0.41	11.55	0.00	0.00	0.00
	Sle1/S1	D+L/1	0.000	-0.06	-0.07	-11.78	0.00	0.00	0.00 election
5	Sle2/S1	D+L/2	269.489	-0.22	-0.19	7.54	0.00	0.00	0.00 ype of
	Sle2/S1	D+L/2	146.994	1.22	0.00	3.08	0.00	0.00	0.00
	Sle2/S1	D+L/2	97.996	0.73	-0.45	2.77	0.00	0.00	0.00
C	Sle2/S1	D+L/2	195.992	0.67	0.45	2.76	0.00	0.00	0.00
	Sle2/S1	D+L/2	0.000	-0.16	0.26	-14.70	0.00	0.00	0.00 ^{Values}
	Sle2/S1	D+L/2	24.499	-0.21	0.16	7.66	0.00	0.00	0.00 streme
	Sle2/S1	D+L/1	0.000	-0.13	0.22	-12.08	0.00	0.00	0.00
	Sle3/S1	D+L/2	24.000	-0.19	-0.42	11.43	0.00	0.00	0.00
	Sle3/S1	D+L/2	192.000	0.26	0.34	138.96	0.00	0.00	0.00
	Sle3/S1	D+L/2	48.000	-0.16	-0.47	3.45	0.00	0.00	0.00
	Sle3/S1	D+L/2	168.000	-0.11	0.37	-56.53	0.00	0.00	0.00
	Sle3/S1	D+L/1	0.000	-0.14	-0.23	-11.88	0.00	0.00	0.00 Sefresh
	Sle4/S1	D+L/1	0.000	0.24	0.29	116.31	0.00	0.00	0.00 [°] review
1	Sle4/S1	D+L/2	96.975	1.27	0.23	4.60	0.00	0.00	0.00
	Sle4/S1	D+L/2	169.706	1.14	-1.03	68.94	0.00	0.00	0.00
	Sle4/S1	D+L/2	0.000	0.26	0.34	138.96	0.00	0.00	0.00
	Sle4/S1	D+L/2	24.244	0.67	0.03	-57.28	0.00	0.00	0.00
	Sle5/S1	D+L/1	96.000	-0.04	-0.09	0.94	0.00	0.00	0.00
	Sle5/S1	D+L/2	0.000	1.14	-1.03	68.94	0.00	0.00	0.00
	Sle5/S1	D+L/1	120.000	-0.03	-0.05	-0.95	0.00	0.00	0.00
	Sle5/S1	D+L/2	24.000	0.53	-0.43	-34.99	0.00	0.00	0.00
	Sle5/S1	D+L/1	0.000	0.93	-0.43	57.75	0.00	0.00	0.00
Ż	Sle6/S1	D+L/2	264.000	-1.67	3.50	-0.44	0.00	0.00	0.00
	Sle6/S1	D+L/2 D+L/2	144.000	1.50	1.94	-0.44	0.00	0.00	0.00
	Sle6/S1	D+L/2 D+L/2	72.000	0.32	-0.43	1.84	0.00	0.00	0.00
				-0.89	5.53	-2.45	0.00		0.00
	Sle6/S1	D+L/2	216.000					0.00	
	Sle6/S1	D+L/2	192.000	0.28	5.28	-2.63	0.00	0.00	0.00
	Sle6/S1	D+L/2	312.000	-0.63	0.63	3.78	0.00	0.00	0.00
	Sle6/S1	D+L/1	0.000	-0.03	-0.05	-0.95	0.00	0.00	0.00
	Sle7/S1	D+L/2	48.000	-2.80	0.97	-0.85	0.00	0.00	0.00
	Sle7/S1	D+L/2	168.000	0.71	-1.06	-0.30	0.00	0.00	0.00
	Sle7/S1	D+L/2	144.000	0.21	-1.07	-1.46	0.00	0.00	0.00
	Sle7/S1 🔿 🔛 🖁	D+L/2	480.000	-0.07	N-0.08 × 1 × 1	-14.02	0.00	0.00 2 2 2 2	0.00
	Sle7/S1	D+L/2	336.000	-0.76	-0.75	5.93	0.00	0.00	0.00
	Sle7/S1	D+L/1	0.000	-0.54	0.54	3.37	0.00	0.00	0.00

<u>Note:</u>

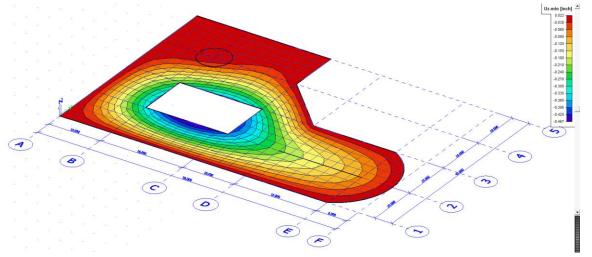
The Preview appears between the Graphical Screen and the Command line. This screen can be maximized to display more data at one time.

Viewing Results on 2D elements

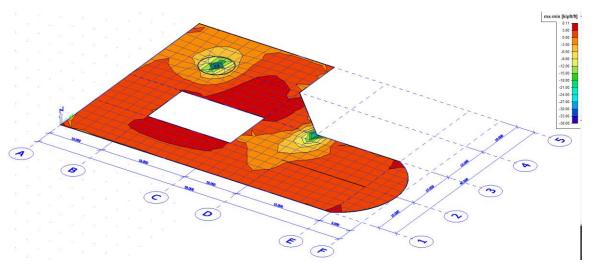
- 1. Below 2D Members in the Results service, click Displacement of Nodes.
- 2. The options in the **Property Window** are configured in the following way:
 - The Selection field is set to All
 - The Load type is set to Combinations and Combination to D+L
 - The Values will be reviewed for Uz.
 - The Extreme field is changed to Global.

Properties		×
Displacement of nodes (1) - Va	V/ /
		8 🐣
Name	Displacement of nodes	
Selection	All	-
Type of loads	Combinations	-
Combinations	D+L	-
Filter	No	-
Structure	Initial	Ψ.
Envelope	Minimum	Ψ.
Standard	V	
Section		
Edge		
Values	Uz	-
Extreme	Global	-
Drawing setup 2D		(40
Actions		
Refresh		>>>
Detailed results in mesh	node 💦 🛼	>>>
Preview		>>>

3. Click on the >>> button behind **Refresh** to display the results in the graphical screen for the chosen settings.

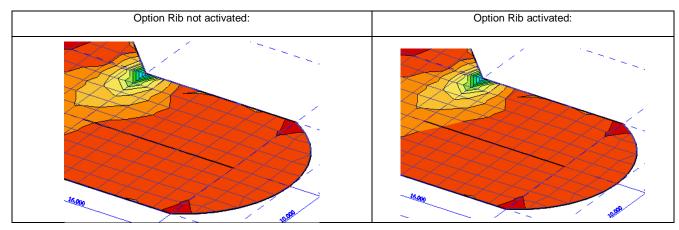


Internal forces can be configured in the **Member 2D – Internal forces** result option. For instance the moment about the x axis (mx) has the following graphical contour for the combination D+L.



Results for rib

1. By clicking on the tick box '**Rib**' in the Properties window, the results will be adjusted in order to take into account the stiffness of the ribs that are included within the slab



2. Note the difference for the two cases that are modelled. It is clear that the forces in the plate are reduced, because the joint stiffness of the slab and the ribs are now considered.

Configuring the Graphical Screen

1. In the **Properties window**, click the icon behind **Drawing Setup.** The different options for the graphical screen will appear.

	Minimum and maximum settings .
Isobands	
Display mesh	Ground value
🗌 Lighting	Use value 0
	Draw isoline
	Local extrems
	Local extrems
Advanced settings	None
	None Style Text with cross
Advanced settings Automatic palette values - rounded	None Style Text with cross Description colour
	None Style Text with cross

- 2. For the **Display** the option **Isobands** will be chosen.
- 3. The button Advanced settings allows for the setting of the legend for the graphical screen.

lumber of isobands 16 👻	Palette colours
Style	8.11
Filled	3.00
	0.00
	-3.00
Isoband contours	-6.00
✓ Display	-9.00
Labels	-12.00
Predefined palette colours	-15.00
Light rainbow	-18.00
Dark rainbow	-21.00
	-24.00
Greyscale	-27.00
- Palette values	-30.00
Normal	-33.00
Rounded	-38.08
From parameters	

- 4. Click **[OK]** to accept the settings or **[Cancel]** to ignore the selected settings.
- 5. Click in the **Property Window**, on the button **>>>** behind **Refresh** in order to display the results in the graphical screen in accordance with the set options.
- 6. Click [Close] to leave the Results Menu.

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.

Post processing

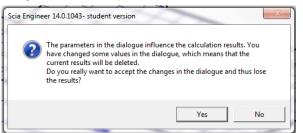
Automatic mesh refinement

This feature enables the user to obtain a precise FE simulation near the locations with high stress concentration e.g. near the corners of the openings, the effect of point loads and edges of the slab. The following discusses the procedures to obtain an accurate mesh and the mathematical background that is utilized.

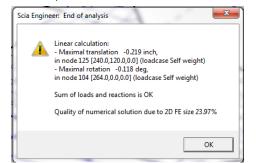
- 1. Select the Mesh setup in the Calculation, mesh from the Main menu.
- 2. Choose 10% for Target error for mesh refinement and select Dead for Load case for mesh refinement.

Name	
General mesh settings	
Minimal distance between two points [m]	0.001
Average number of tiles of 1d element	1
Average size of 2d element/curved element [inch]	24.000
Definition of mesh element size for panels	Manual 👻
Average size of panel element [inch]	39.370
Elastic mesh	
Use automatic mesh refinement	
Target error for mesh refinement [%]	10
Load case for mesh refinement	Dead 👻
1D elements	
Minimal length of beam element [inch]	3.937
Maximal length of beam element [inch]	39370.079
Average size of cables, tendons, elements on subsoil, nonlinear soil spring [inch]	39.370
Generation of nodes in connections of beam elements	
Generation of nodes under concentrated loads on beam elements	
Generation of eccentric elements on members with variable height	
Division on haunches and arbitrary members	5
Division for 2D-1D upgrade	50
Mesh refinement following the beam type	None 👻
2D elements	
To generate predefined mesh	
To smooth the border of predefined mesh	
Maximal out of plane angle of a quadrilateral [deg]	1.719
Predefined mesh ratio	1.5
Hanging nodes for prestressing	7

3. Select [Yes] in the dialogue box to lose the previous results.



4. Click on Mesh generation. As the previous results are deleted, the same mesh will be utilized. Run the linear analysis.



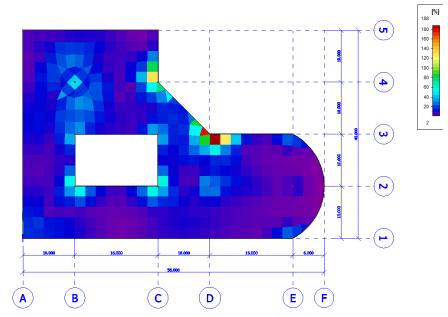
5. The quality of numerical solution is demonstrated in the analysis window. This is the relative % error, as calculated:

 $\% = \frac{\text{Energetic Norm of Estimated Error}}{\text{Energetic Norm of Displacements}}$ and is numerically acceptable if it is below 30%.

- 6. The sensitivity of the numerical solution can be visualized through the **Num, error, Mesh refinement** in the **2D members** section of the **Results menu**.
- 7. In the Properties window select Load cases for Type of loads, Dead for Load cases, Global for Extreme and Global mesh error for Magnitude.

Pro	perties		×
N	um. error, Mesh refinen	nent (1) 🔄 🚺	7/ /
			8
1	Vame	Num. error, Mesh refine	ment
	Result case		
	Type of load	Load cases	*
	Load case	Dead	*
	Selection		
	Type of selection	All	-
	Filter	No	-
	Location		
	Location	In nodes avg.	-
	Extreme		
	Extreme	Global	-
	/lagnitude	Global mesh error	-
	User defined iso		
	Use user scale isolines		
	Prawing setup 2D		
A	tions		
R	efresh		>>>
P	review		>>>

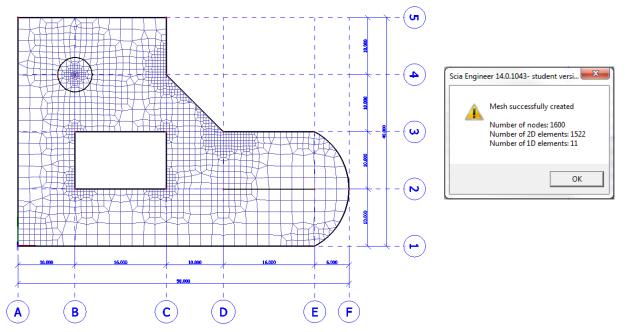
8. The error contour is as follows:



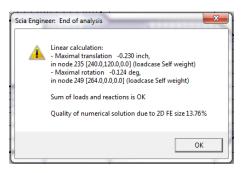
9. Other options for the Magnitude are available as well:

Extreme	Global	
Magnitude	Mesh refinement	
User defined iso	<u></u>	
Use user scale isolines	Local mesh error Numerical error	
Drawing setup 2D	Mesh refinement	

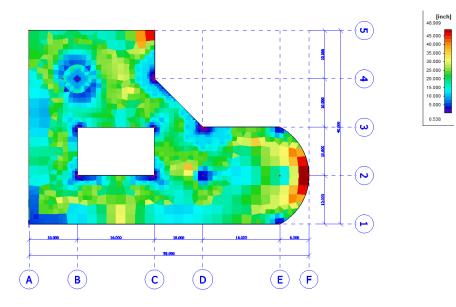
10. The optimal mesh size contours can be viewed through the Mesh refinement option above. In order to apply the error analysis to the previous mesh, simply regenerate the mesh using **Mesh generation** in the **Calculation**, **mesh** from the **Main menu**. The new mesh will then be created based on the refinement criteria.



11. Rerun the linear analysis. After doing this, the numerical error has been reduced to 13.76%.



12. Just as was previously done, the user can review the suggested mesh refinement as a graphical result by choosing **Mesh** refinement from the available **Magnitude** in the **Property window**. In this example, no further refinement is needed.

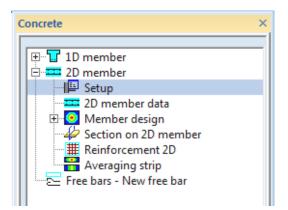


Reinforcement design

The reinforcement design can be viewed in the concrete menu.

Pre-processing

1. Select Setup in the Concrete menu frim the Main Window.



Choose S60 in the Reinforcement from the Design Defaults tab. 2.

Design defaults	Design defaults		
General Calculation Design strength	General Concrete cover	Use min concrete cover 🔹	
Shear Detailing provisions	2D structures and slab		
		· · · ·	
	Default reinforceme	0.75 inch	
	Reinforcement	S60 ▼	
	Size nu	umber Diameter	
	Upper 5	0.625 inch Angle 0 deg	
	Lower 5	0.625 inch Angle 0 deg	
	Lower cover	0.75 inch	
		1	

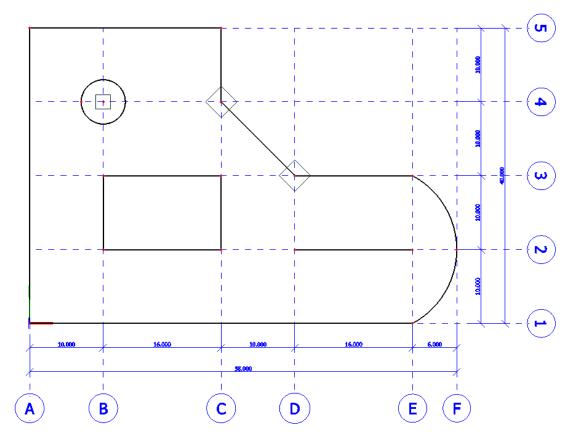
- 3. Select [OK] to confirm.
- Define **Averaging strips** through the icon **the Concrete window** at the locations with high potential stress concentrations such as points **D3**, **C4** and **B4**. 4.

lame	RS3	
ype	Point	
Vidth [inch]	36.000	
ength [inch]	36.000	
ingle [deg]	45.00	
Direction	longitudinal	

- 5. Choose 36in for the Width and Length and 45° for the Angle. Confirm by clicking [OK].
- 6. Select points D3 and then C4 by clicking on these points accordingly. Finalize the selection by [ESC].

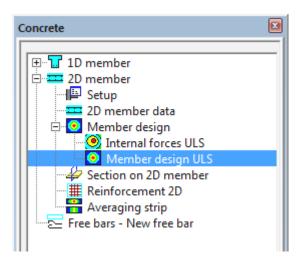
Enter point: D3 <click> C4 <click>

- 7. Similarly define a 24in x 24in point at B4 with a 0° angle.
- 8. The layout of the slab will be as follows while showing the averaging strips.



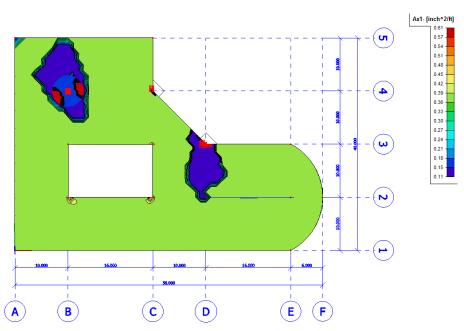
Required areas

1. Select Member design ULS form Member design in the 2D Concrete menu.



2. Choose Combinations for the Types of loads, D+L for Combinations, uncheck the Show warnings, Check the Averaging of peak in order to smooth out unrealistic peak values, set the Location to In nodes, avg. on macro, Required Areas for Type values, Required reinforcement for Reinforcement, check the Standard, As1- for Values for the required lower level reinforcements along x-direction. Finally, click Refresh in the Actions to view the graphical results.

Descention			×
Properties			X
Member 2D - design - rec	quired areas (1) 🛛 🔽 🚺	√∕	1
		8 🖉	8
Name	Member 2D - design - require	d area	is
Selection	All		Ŧ
Type of loads	Combinations		÷
Combinations	D+L		-
Filter	No		-
System	Local		
Output	Brief		+
Show errors	\checkmark		
Show warnings			
Print explanation of er			
Use user scale isolines			
Averaging of peak	V		
Location	In nodes, avg. on macro		+
Type values	Required areas		*
Reinforcement	Required reinforcement		*
Standard			
Section			
Edge			
Rib / Integration strip			
Values	As1-		*
Extreme	Global		*
Drawing setup 2D			
Actions			
Refresh		>>:	>
Calculation info		>>:	
Concrete setup		>>>	
New reinforcement		>>>	
Export reinf. area to CAD	program	>>:	
Preview	F - 2	>>:	>
			_



Different options in values represent various reinforcement in various levels and directions of the slab:

As-1 = Lower reinforcement in x-direction

As-2 = Lower reinforcement in y-direction

As+1 = Upper reinforcement in x-direction

As+2 = Upper reinforcement in y-direction

Asw = Shear reinforcement

3. The warnings and errors for the design are located in the **Calculation info** in the **Actions** from the **Property window**.

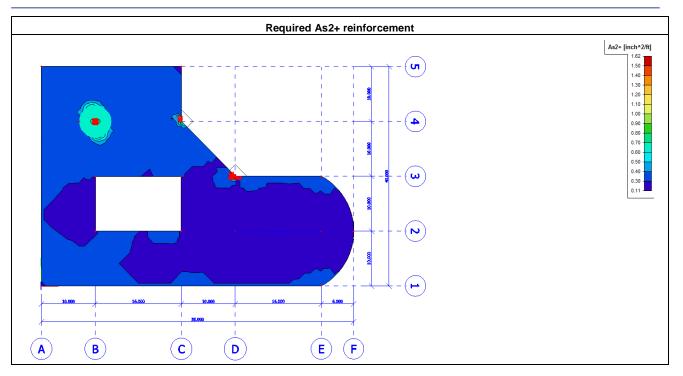
Calculation	n Info			23
Þa 🛄 🚆) 6 🔲	H 🗌 💻 20	0 % 🔽 🗹 🚾 🗌 🗖 default 🔍 🖳 🖉 default 🔍 🗉 📲	
warnings	and errors	;	for slabs	-
Slab	No.	Туре	Description	
<mark>S1</mark>	1	Warning	Calculation successful. There are no error messages.	
<mark>S1</mark>	101	Warning	Tension reinforcement.	
<mark>S1</mark>	103	Warning	Minimum constructive reinforcement superposing statically required tension reinforcement	
S1	152	Warning	Minimum pressure reinforcement.	
S1	153	Warning	Minimum constructive reinforcement superposing statically required pressure reinforcement.	
S1	200	Warning	Shear reinforcement is not required.	
<mark>S1</mark>	201	Warning	Shear reinforcement required.	
Ready [en]			•	<u>ا ا</u>
Show List for			nings and errors slabs Cite	ose

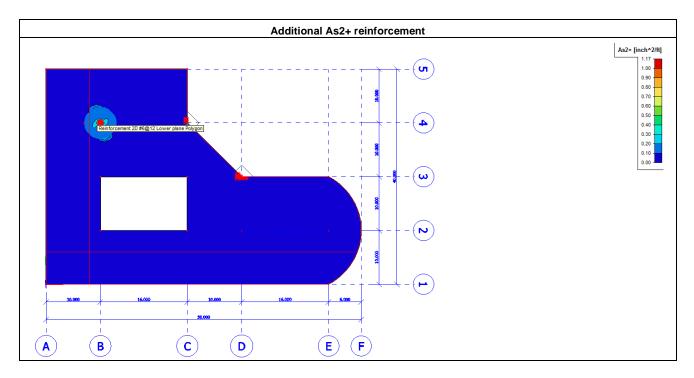
Input user reinforcement

1. Select **H** Reinforcement 2D in the Concrete menu.

	Name	#6@12 Lower plane	
	2D member	S1	
1 //	Reinforcement		
	Туре	Bars	*
	Material	S60	·
	Surface	Lower	-
	Number of directions	2	*
	Direction closest to surface	1	-
	Angle of first direction [deg]	0.00	
	. 🗆 1		
	Size number	6	
	Diameter (dl) [inch]	0.750	
	Concrete cover (cl,cu) [inch]	0.750	
	Bar distance (sl) [inch]	12.000	
	Offset [inch]	0.000	
	Reinf. area [inch^2/ft]	0.44	
	□ 2		
	Size number	6	
	Diameter (dl) [inch]	0.750	
	Concrete cover (cl,cu) [inch]	1.500	
	Bar distance (sl) [inch]	12.000	
	Offset [inch]	0.000	
	Reinf. area [inch^2/ft]	0.44	
	Total weight [kips]	0.000	
	Geometry		
	Geometry defined by	Polygon	*
	Actions		
	Load from setup		>>>

- Choose a name for reinforcement, e.g. #6@12 Lower for defining reinforcement with size #6 at a center to center distance of 12" at the lower level of the slab. Choose Lower for Surface. For directions 1 & 2 choose 6 for Size number and 12 for the Bar distance and then click [OK].
- 3. Select 🔯 Select existing polygon/polyline above the Command line and then click on one edge of the concrete slab
- 4. In a similar fashion, create reinforcement for the upper level. Name this reinforcement e.g. **#6@12 Upper** and choose **Upper** for **Surface** in Reinforcement 2D. Use the **select existing polygon** to apply the reinforcement to the concrete slab.
- 5. Choose **Member design ULS** keep the previously configured input options, except for changing Reinforcement to **Additional reinforcements.** This will allow the user to graphically view the additional required reinforcement at each finite element.

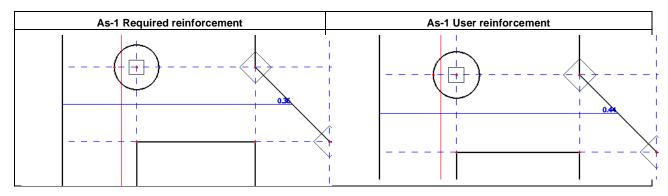




6. In order to see the exact amount of reinforcement along a section, select the Section on 2D member In the Concrete menu and choose Z direction for Draw and click [OK].

Section on 2D member	
lame	SEI
Draw	Z direction
Direction of cut [ft]	0.000 / 0.000 / 3.281
	OK Cancel

7. The **command line** will ask for the coordinates of the section cut. Here, it is possible to draw a linear line between edges **3** and **4** by clicking on the screen with left mouse and then confirming with **[ESC]**. The reinforcements can be visualized by checking **Section** in the **Properties window** and clicking **Refresh**.



Engineering Report

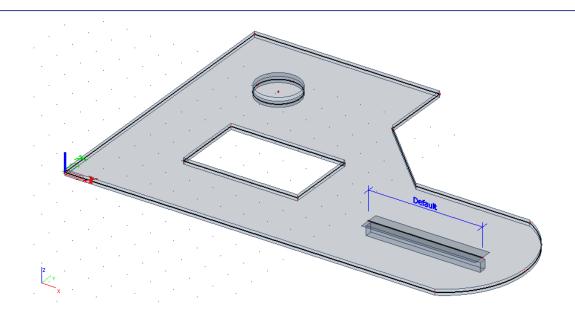
The final section of the tutorial will focus on how to utilize model information (input, output and graphics) in Scia Engineer's Engineering

Report interface. The Engineering Report is accessed from the Main Service Tree or by the shortcut button

Saving Model Views or Results to Engineering Report

Following actions are accessible via Main menu > Drawing tools, or 'Project' toolbar, or right mouse click in screen

- 1. Table to Engineering Report
 - o Send table Result items or other output directly to Engineering Report
- 2. Print picture
 - o Print the picture on the display, after choice of printer, choice of template file and possible editing
- 3. Picture to document
 - o Send the picture on the display directly to the document
- 4. Save Picture to File
 - o Allows user to save image (2D/3D) as vector image file in AutoCAD for use in further documentation.
- 5. Picture to gallery
 - o Send the picture on the display to the Picture gallery, where it can be edited before saving it or adding it to the document
- 6. Picture gallery
 - o Edit pictures by means of the Gallery editor; e.g. text and dimension lines can be added
- 7. Paperspace gallery
 - o Choose/make a template file for printing + input and arrange the picture(s) to be printed
- 8. Screenshot into Engineering Report
 - Send the picture on the display directly to the Engineering Report as an uneditable screenshot.
- 9. Live Picture into Engineering Report
 - Send the picture on the display directly to the Engineering Report. Live pictures can be modified and are dynamic, meaning any change to the display model will be made in the Engineering Report.



Adding New Items to Engineering Report

- 1. Main menu > Engineering Report, or 'Project' toolbar
- 2. Content of Engineering Report
 - Via "New Items" List and Insert Button
 - o Prepare the Engineering Report as shown in the report navigator

Navigator	Ψ×
Header / Footer	0
Style	0
Page format	0
Table of contents	0
🔲 Analysis model (Snapshot picture)	£ 0
Page break	0
v 🗁 Input	0
Materials	£ 0
Cross-sections	_ O
Load cases	~ 0
····· 🔛 Line force	@ O
Surface load	_ O
Page break	0
v 🗁 Results	0
Displacement of nodes	@ O
III Reactions	^ 0
🇮 Num. error, Mesh refinement	^ 0
Displacement of nodes; Uz (Document	_ O
Page break	0
🛛 🗁 Design	0
····· 🔠 Reinforcement 2D	₽ 0
Internal forces ULS	^ 0
····· 🎹 Member design ULS	~ 0
Displacement of nodes; Uy (Document	• ••

Insert

• After adding items to the Engineering Report, individual results can be manipulated

46

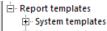
Representation Name Internal forces on beam Caption Internal forces on beam Caption visible Image: Caption visible Selection Image: Caption visible Selection Selection Selection type All Result properties Image: Caption visible Type of loads Load cases Load cases D Filter No Structure Initial Values My	X
Caption Internal forces on beam Caption visible Image: Caption visible Selection Image: Caption visible Selection type All Selection type All Parallel properties Image: Caption visible Load cases D Filter No Structure Initial	
Capiton visible Image: Constraint of the section	
Selection Selection type All Selection type Load cases Image: Type of loads Load cases Load cases D Filter No Structure Initial	
Selection type All Result properties Load cases Type of loads Load cases Load cases D Filter No Structure Initial	
Result properties Type of loads Load cases Load cases D Filter No Structure Initial	
Type of loads Load cases Load cases D Filter No Structure Initial	•
Load cases D Filter No Structure Initial	
Filter No Structure Initial	-
Structure Initial	•
	-
Valuer Mv	-
Values	-
System Principal	-
Extreme Global	-
Section All	-
Table Template	
Template name Default (embedded)	-

3. Add picture to document

- o Directly via Right Click and Screen Shot into Engineering Report or Live Picture into Engineering Report
- To make a Live Picture 3D PDF capable, select picture and check "Export to PDF as 3D"
- 4. Make Changes to Live Pictures
 - \circ \quad Select Live Picture in Navigator and now the Picture Editing buttons become active.
 - Select "Edit Picture Properties" to change size, scale, view point, regeneration settings and result type.
- 5. Adapt Style of Document
 - Via adding "Style" to Navigator from New Items list
- 6. Modification of Tables
 - Select specific table in Navigator and click "Edit" button. Edit This enables the user to select and modify specific items in the table, change the table layout and also the size and spacing of the table items.

1

- 7. Refresh of Engineering Report
 - After adaptations to data or (content of) tables: Red Exclamation Point means item regeneration needed.
 - o Regenerate Selected Items
 - Regenerate Outdated Items (Pictures, Tables)
- 8. The Engineering Report also allows for any information in the "Navigator" to be saved as a report template. These templates are found in the "Report Templates" item in the List of Item.



🗄 User templates



