

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 2008 Scia. All rights reserved.

#### **Table of Contents**

General Information	1
Welcome	1
Scia Engineer Support	1
Website	
Requirements	2
Project management	
Save, Save as, Close and open	
Saving a project	
Closing a project	
Opening a project	3
Introduction	4
Getting Started	5
Starting a project	5
Starting the program	5
Creating a new project	5
Input geometry	7
Inputting the geometry	
Geometry	
Line grid	
Cursor snap settings	
Entering a plane 2D element	
Entering a shell element	
Importing the wiring	
Intersection and cut outs	
Visualizing the mesh	
Visualizing the rendered structure	
Supports	
Check structure	25
Checking the structure	
Connecting members/nodes	
Connect members/nodes	
Graphic representation of the structure	27
Input calculation data	32
Load cases and load groups	32
Defining a permanent load case	32
Defining a second load case	
Loads	34
Free load	35
Calculation	40
Linear Calculation	40
Results	41
2D data viewer	41
View deformations of nodes on 2D element	42
Display stresses	46
Document	47
Drawing up document	47
Showing results in the document	
Adding a picture to the document	
Epilogue	

3

**Tutorial Shell** 

# **General Information**

# Welcome

Welcome to the Scia Engineer Tutorial Shell. Scia Engineer is a design program under Windows with a broad application field: from checking simple frames to the advanced design of complex projects in steel, concrete, timber,...

The program treats the calculation of 2D/3D frameworks, profile check and check of connections for steel structures 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, member check and optimization according to various codes, generating the calculation note, ...

Scia Engineer is available in three different editions:

#### License version

The license version of Scia Engineer is secured with a 'dongle', a code lock, which you apply to the parallel or USB gate of your computer or a softwarematic license on 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 that are available.

#### Demo version

If the program doesn't find a protection, it will automatically start the demo version. The properties of the demo version are: All projects can be inserted;

The calculation is restricted to projects with 25 elements, 3 plates/shells and two load cases;

The output contains a watermark "Unlicensed software";

The projects that are stored in the demo version cannot be opened in a license version.

#### Student version

The student version has the same possibilities as the license version for all modules. This version is also secured by a 'dongle' or 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.be 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 Belgium : +32 13 350310

From the Netherlands : +31 26 3201230

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

# Requirements

Release: Scia ENGINEER 2009.0

 Req. Module:
 ESA.00
 Base Modeller

 ESA.01
 2D surfaces (plates and walls)

 ESA.02
 Curved surfaces

 ESA.04
 Intersections of surfaces

 ESA.05
 Linear Statics 3D

 Manual: Scia Engineer Tutorial Shell

 Revision: 06/2009

# 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 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).

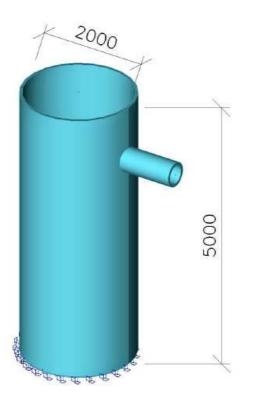
# Introduction

The example of this tutorial can be performed in the three versions of the program. Before you continue, you need to be familiar with the use of your control systems such as working with dialog boxes, menu bars, toolbars, status lines, mouse, etc.

This tutorial describes the most important functions of Scia Engineer for the import of curved shells and free loads.

First of all we'll show how to create a new project and how to build the structure. After the input of geometry and loads, the structure is calculated and the results can be viewed. The text ends with an introduction to draw up a calculation note.

The figure below shows the analysis model of the structure that is being built:



# **Getting Started**

# Starting a project

Before starting a project, the program needs to be run first.

### Starting the program

1. Double-click on the shortcut "Scia Engineer" on the Windows Desktop.

Or:

2. If the shortcut is not installed, click [Start] and choose Programs > SCIA ENGINEER 2009.0 > SCIA ENGINEER 2009.0.

If the program doesn't find the protection, a dialog box will appear showing the reason why the protection was not found. A second dialog box shows the restrictions of the demo version. Click [OK] in both windows.

For this Tutorial a new project is created.

## Creating a new project

- 1. If the dialog box **Open** appears, click [Cancel].
- 2. Click on the icon **New** in the toolbar.
- 3. Click on the icon **Structure** structure, click [**OK**].

Structure	LTA	K Free Form Modeller	Modeller

4. The dialog box Project data appears, in which general data about the project can be inputted.

	Data				Structure :	
CMCT					General XYZ	•
and a					Material :	116
10×-	Name	Tutorial She	ell .		Concrete	
10	Part				Steel	
SOT.	Pan	1-			Material	S 235
and the second	Description	-			Timber	
ant.					Other	
ALC: NO	Author	-			Aluminium	
200	Date	27.05.2009				
		1277.00.2000				
Contraction of the second						
101						
	Project Level :		Model:			
	Advanced	•	One	-		
		- Contraction	Terre			
-1926	National Code					
		EC-EN	V			

- 5. In the group **Data** you can enter the data you want. These data can be mentioned in the output, e.g. in the document and on the drawings.
- 6. Choose Project level: Advanced (since a manual mesh generation will be performed) and Model: One.
- 7. Press the button under **National code** to set a default code for the project. Because of this the available material, combination rules and code checks are determined. For the project of the Tutorial, we choose EC-EN. The window **Codes in project** appears.
  - a. Press the button [Add].
  - b. The dialog box Available national codes appears.
  - c. Select the European flag and click [OK]. You will return to the window **Codes in project** and the **EC-EN** is added.
  - d. Select the flag with the name EC-EN.
  - e. Select the options Active code and click [Close].
  - f. You will return to the window Project data and the EC-EN is the active code.

#### 8. Select General XYZ in the field Structure

The type of structure (Frame XZ, Frame XYZ, Slab XY, General XYZ, ...) restricts the possible input during the calculation. Since a 3D shell element is being designed, the **General XYZ** is chosen.

- 9. Select **Steel** in the group **Material**. Under the item **Steel** a new item **Material** will appear.
- 10. Choose S235 from the menu.
- 11. Confirm the input with [OK].

#### Remarks:

On the tab **Basic data** you can set a project level. If you choose standard, only the frequently used basic functionalities will be displayed. If you choose advanced, all basic functionalities will be displayed.

On the tab **Functionality** you can choose the options you need. The non-selected functionalities, will be filtered out from the menu, which makes the program lighter.

The tab Combinations contains the values of the partial safety factors. For this Tutorial, the default settings are used.

# Input geometry

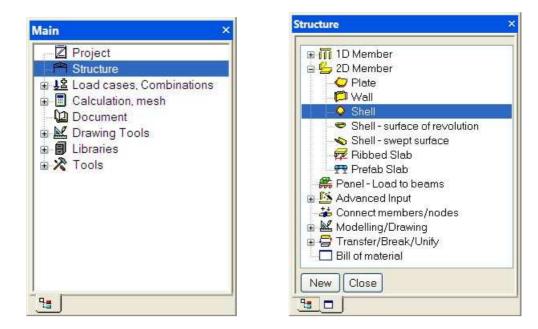
# Inputting the geometry

If a new project is started, the geometry of the construction has to be entered. The construction can be imported directly, but also templates with parametric blocks, DXF files and other formats can be used.

### Geometry

#### Structure menu

1. When starting a new project, the **Structure menu** is automatically opened in the **Main window**. If you want to modify the construction later, you have to double-click on **Structure** in the **Main window**.



2. In the Structure menu you can choose from several structural elements.

# Line grid

To simplify the input of the geometry, the line grid and snap settings are used. when clicking on the icon the following window appears:



Line grid Input data Drawing setup Z 70 Dir X [m] Dir Y [m] Dir Z [m] Туре Span • Туре Span • Туре Span • 
 Name
 X[m]
 dx[m]
 Rep
 SL
 Name Z[m] dz [m] Rep SL Name Y [m] dy [m] Rep SL 1 a 2 b 3 c 4 d 1 1 2 2 3 3 \* 1 A 2 B 1,000 2 1,000 ]0 3,000 4,000 5,000 3 C 1,000 ]0 0,000 0,000 0,000 1 Insertion point [m] X 0,000 0,000 0.000 0,00 Refresh names х Ζ Rotation deg Y Name Grid1 Туре Cartesian 💌 OK Cancel

The values for the line grid are entered as shown on the picture above.

### **Cursor snap settings**

 Double-click on the button Cursor snap settings bottom right of the screen. The window Cursor snap setting appears:

Cursor snap setting	
<b>1 1</b>	✓ Line grid ☐ Dot grid
× 112	Conly snapped points
	a) 🥅 Midpoints
	b) 🔽 Endpoints / Nodes
<u>%</u> 9	c) 🦵 Intersections
	d) 🦵 Orthogonal points
2/4	e) 🦵 Tangential points
1/4 3/4	f) 🔽 Arc/Circle centre
	g) 🦵 Points on line-curve - length
	Length[m] 1.000
*	Repeat 3 🛨
	Start point. Begin 💌
	h) T Points on line-curve - N-ths
	i)
	0 🥅 Surface edges
	k) 🔽 General solids 🛛 2000
	OK Cancel

- 2. Choose the option line grid so the project can be snapped on the grid line you now created.
- 3. Click [OK] to confirm.

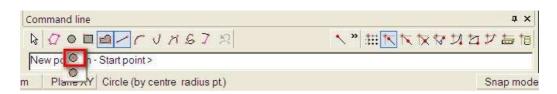
By means of the line grid, the structure is now entered in Scia ENGINEER. First enter the floor slab, then the cylindrical wall.

## Entering a plane 2D element

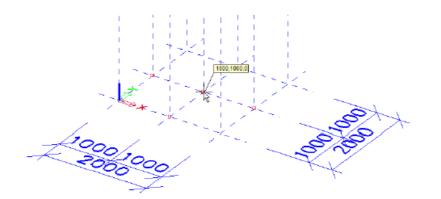
1. To enter a new plane 2D element, you can use the option 2D member -> Plate in the structure menu.

Na	ime	S1
Ту	pe	plate (90)
	, alysis model	Standard
	iterial	S 235 🔹
	M model	Isotropic
	ickness type	constant
	ickness [mm]	50
z / Me	ember system-plane at	centre
	centricity z [mm]	0
	S Type	Standard
Sw	ap orientation	🗆 no
LCS	S Angle [deg]	0,00
Z Lay	yer .	Layer1 👻
		OK Cance

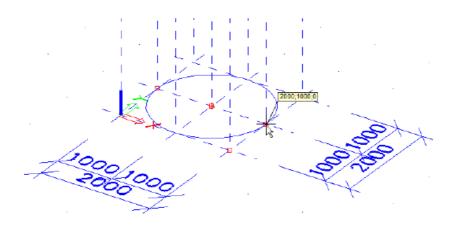
- 2. In the field thickness, you enter 50 mm.
- 3. Confirm the input with [OK].
- 4. The floor slab is now entered by means of the options **New circle** (by centre radius pt) in the command line at the bottom of the screen.



5. Select the centre (1;1;0) of the circle by means of the line grid:



6. Show the radius of the circle by means of the line grid co-ordinate (2;1;0):



7. Press <**ESC**> to confirm.

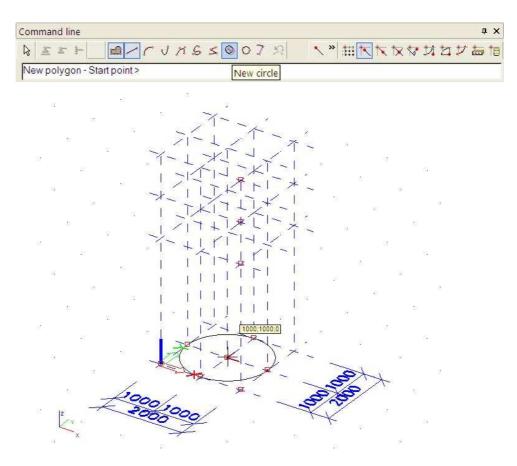
After drawing the floor slab, the walls are entered.

## **Entering a shell element**

1. Use the option Shell in the structure menu to enter a new shell element.

Name	S2
Туре	shell (98)
Analysis model	Standard
Material	S 235
FEM model	Isotropic
Thickness type	constant
Thickness [mm]	50
Member system-plane at	centre
Eccentricity z [mm]	0
LCS Type	Standard
Swap orientation	🗆 no
Layer	Layer1

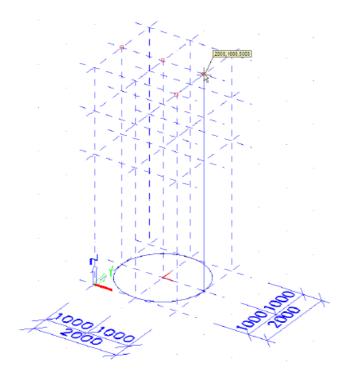
- 2. Enter 50 mm in the field Thickness.
- 3. Confirm the input with [**OK**].
- 4. Re-enter a circle with the same centre and radius as the floor slab.



5. Change the active plane in YZ by means of the button bottom left of the command line:



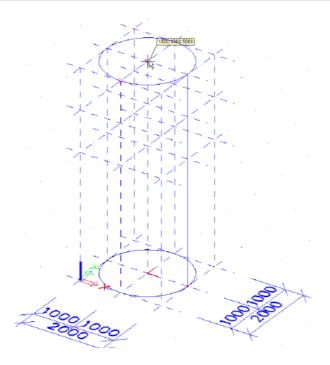
6. Enter the height of the cylinder by clicking on the co-ordinate (2;1;5):



7. Re-activate the XY-plane:

Command line		τ×
≥ = F	3.44.45.41 単業 てきょく スロノノ	なる物
Command >		
m Plane XY Ready		Snap mode

8. Then enter the upper circle by clicking on the centre (1;1;5):



9. The input is ended by pressing <**ESC**>.

After having it entered in Scia Engineer, the entity is always selected. In case of a cylinder, you can tell by the magenta color. To clear the selection, use the **<ESC**> button again.

#### Remarks:

The properties of the selected elements are displayed in the Property window and can be simply edited there.

If there are no cross-sections in the project defined yet, the window **New cross-section** will automatically appear the moment you give a command to input a structural part (column, beam, member).

An input can be ended by pressing <**ESC**> or by clicking the right mouse button.

The button Zoom All



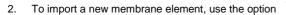
in the toolbar allows you to visualize the entire construction.

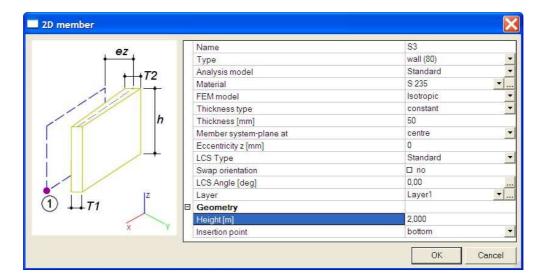
After importing the floor slab and edges, we can import the wiring at the top of the structure. By means of the line grid and snap settings, this wiring can be imported easily:

# Importing the wiring

1. The principle to import this wiring can be analogous to the entry of the first shell element. The command wall is used.

Wall in the structure menu.

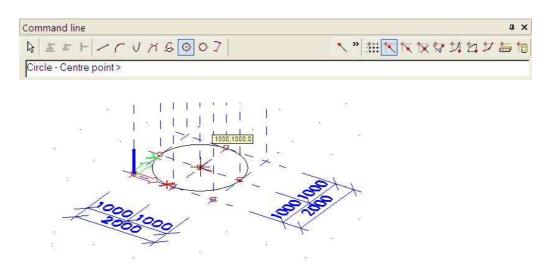




- 3. 50 mm is entered in the field Thickness.
- 4. The height of the wall is 2 m.
- 5. Confirm the entry with [**OK**].
- 6. Change the active plane in XY:

Command line	Ф X
1 2 2 F	
Command >	
m Plane XY Ready	Snap mod

7. Enter a circle with the centre as shown on the picture:



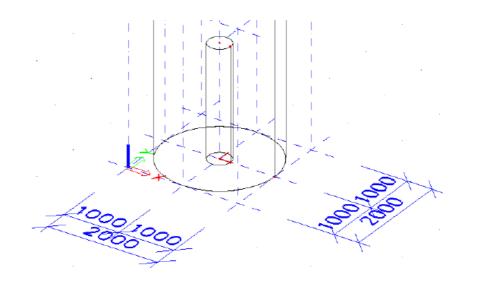
8. Enter the radius of the circle with relative coordinates:

ながあた

#### Remark

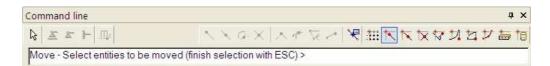
In Scia Engineer you can use absolute or relative coordinates. The difference is the use of the origin and the import of the coordinates. With absolute coordinates, the origin of the UCS is used, these coordinates are entered with the x;y;z coordinates. If the relative coordinates are used, the last entered point will serve as the origin. Relative coordinates are entered with a '@' for the x;y;z coordinates (as shown in the previous example).

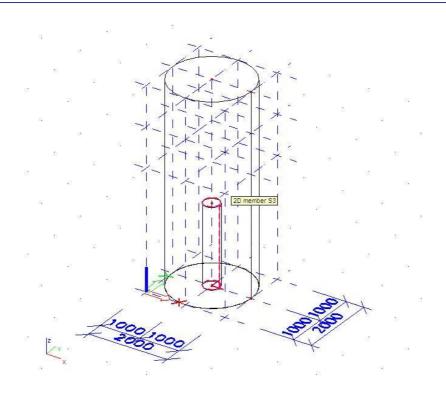
9. The program draws the following cylinder:



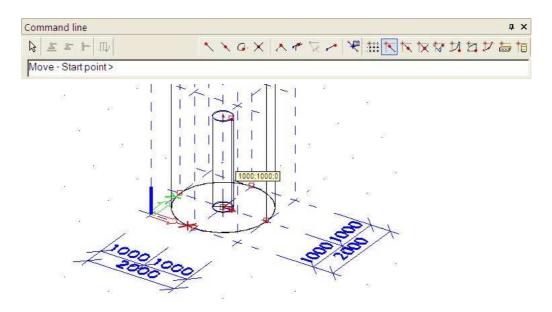
 The cylinder is being moved with the help of the command move bars under Modify → Move This command can be found in the menu

11. First you have to select the element that has to be moved. Confirm afterwards with < ESC>:



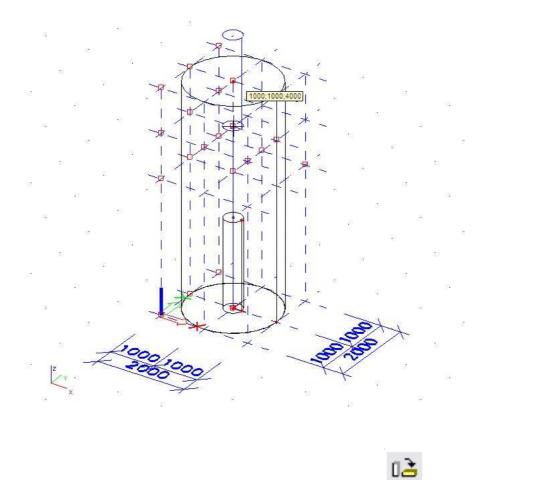


12. Then the start point has to be defined. Point (1;1;0) is used:



13. Subsequently the end point has to be entered (1;1;4):

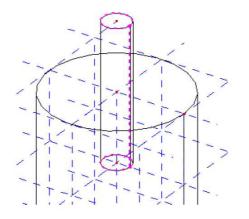
Nerim	
	しているななががが出来 くびやく Xの//



14. Now the cylinder has to be rotated. This is done by using the command rotate , which you can find in the menu bar **Modify**. This command asks the user to select an element:

🖓 🕂 🛣 🗧	
---------	--

15. The cylinder is selected:



\$2 |

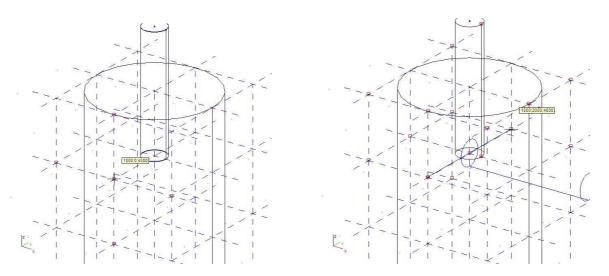
is used. This icon can be found at the bottom of the command bar.

17. The following dialog box is displayed:

16. Afterwards the icon

Rotation — Angle	90	deg
wis vector		
C Workin	ıg plane normal ve	ctor
Define	axis by cursor	
C Enter c	ustom axis vector	
	s vector	m
Custom exi	s vector	m

- 18. Enter an angle of **90**° and select the option **Define axis by cursor.**
- 19. The rotation axis is now determined by the user (1;0;4) and (1;2;4):



20. The cylinder was rotated on the axis with an angle of 90°, defined by the user.

### Intersection and cut outs

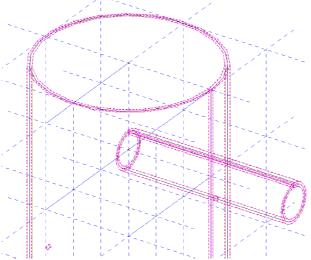
In this paragraph the intersection between the horizontal and vertical cylinder is entered, followed by selecting the present parts of the geometry.

#### **Entering an intersection**

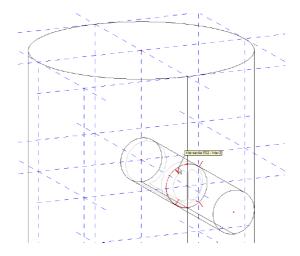
1. To enter a new intersection, use the icon **components**.

in the Structure menu → 2D Member → 2D Member

2. Select the 2D elements (element S2 and S3) between which an intersection has to made:



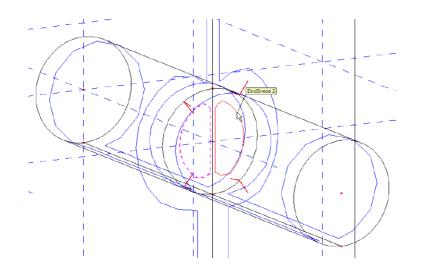
- 3. Confirm the entry with <**ESC**>.
- 4. The made intersection is displayed on the screen:



After entering the intersection, the user can define cut-outs.

### **Defining cut-outs**

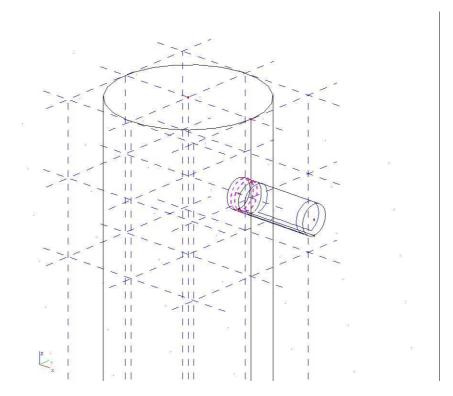
- 1. Use the icon <sup>G</sup> Cut-out in the Structure menu under <sup>G</sup> <sup>G</sup> <sup>2D</sup> member components to enter a cut-out.
- 2. Select the vertical cylinder S2.
- 3. Confirm the entry with **<ESC**>.
- 4. Scia Engineer shows which parts (cut-outs) of the structure can be removed for the calculation. These cut-outs are now being selected:



- 5. Confirm the selection with < ESC>.
- 6. The selected areas are removed from the geometry.
- 7. The definition of the cut-out for the horizontal cylinder happens analogously.
- 8. Select node N5 and N8 manually.
- 9. In the property-window, input Coord X [m] = 1,8

N	ode (2)	▼ ¼ V Ø
	GCS coordinate	
	Coord X [m]	1,800
	Coord Y [m]	1,000
	Coord Z [m]	
Ξ	UCS coordinate	
	Coord ux [m]	1,800
	Coord uy [m]	1,000
	Coord uz [m]	
Ξ	2D members	
	2D member	S3

- 10. Click on the icon Cut-out in the menu tree under a CD member components .
- 11. Select the horizontal cylinder S3.
- 12. Confirm the entry with <**ESC**>.
- 13. Select the cut-outs you want to remove:



- 14. Confirm the selection with **<ESC**>.
- 15. The selected areas are removed from the geometry.

There are various possibilities to visualize the cut-outs. In this example the mesh is generated.

# Visualizing the mesh

1. To generate the mesh, the advanced mode is first set in the project data. This can be done through the menu bar **Tree, project**:

	Data			Structure :		
	Data			General XYZ	•	
Contraction of the				Material:		
	Name	Tutorial She	11	 Concrete		
ALC: NO		1		Steel		
Contractory of	Part	-		Material	S 235	·
100	Description	-		 Timber		
1000	Description	-		Other		
100	Author	-		 Aluminium		
1 Case				-		
位单人	Project Level :		Model :			
	Standard	-	One			
	Standard Advanced	EC-EN				

- 2. Press the icon to start the mesh generation.
- 3. To be able to better visualize the result, a mesh refinement is entered. Click on the icon in the menu tree under **Calculation, mesh**.
- 4. Set the average size of the 2D elements on 0,1:

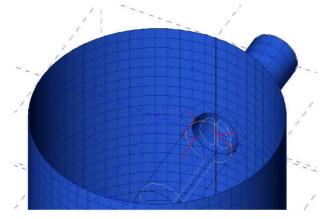
	Mesh setup		E
	Name		1
Ξ	Mesh		
	Minimal distance between two points [m]	0.001	
1	Average number of tiles of 1D element	1	
	Average size of 2D element/curved element [m]	0,100	

- 5. Let the mesh regenerate by means of the icon
- 6. Set the following options for the mesh (on the tab structure) in "view parameters for all", which you can find at the top of the command line:

0 0 4 14 14 14 15 15 1	
	Setup dialog
×	Detailed on/off
Z T	Modelling/Drawing
38 	Model
	Labels 🕨
15	Structure 🕨

Mesh		
Draw mesh	~	
Free edges		
Display mode	wired	*

7. The program clearly show the edited geometry:



If you want to review the edited structure without generating the mesh, you need to follow the following procedure.

# Visualizing the rendered structure

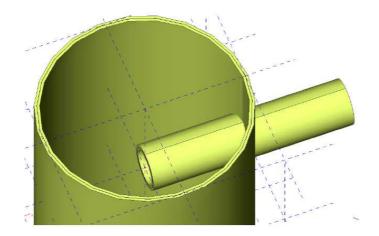
1. First of all the mesh has to be switched off. This can be done through the view parameters above the command line:

			5 2	8
*	Structure	•	Panel	
10	Labels	• •	Structure nodes	*
10	Model	•	Cut-outs on 2Dmembers	· · · ·
z	Modelling/Drawing	• 🗸	Mesh	
×	Detailed on/off		Local axes	
	Setup dialog			
🕑 🕖 🛓 🕍 🎏 👭 🚳 💭	le e 1	_		

2. Afterwards the structure can be rendered with the render buttons above the command line:

00	
Comma	nd line
13	

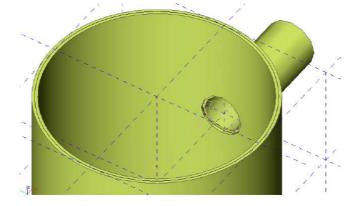
3. At first sight it seems that this structure doesn't match the previous one:



4. However, when the cut-outs are activated on 2D elements at the view parameters:

	Structure Labels Model Modelling/Drawing Detailed on/off Setup dialog	<ul> <li>Panel</li> <li>Structure nodes</li> <li>Cut-outs on 2Dmembers</li> <li>Mesh Local axes</li> </ul>	
Command line	1100		またななななな

5. The structure looks as expected:



## **Supports**

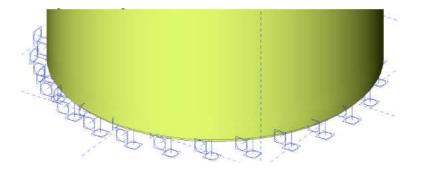
The input of the geometry can be completed by entering supports.

### **Entering supports**

1. To be able to enter supports, you can use the option Model data → Support → Line on 2D element in the Structure menu.

	Name	Sle1
	X	Rigid
	Y	Rigid
	Z	Rigid
<b>Rz</b>	Rx	Rigid
7	Ry	Rigid
	Rz	Rigid
X	Geometry	
Ry	System	GCS
x1	Position x1	0,000
x2	Position x2	1,000
	Coord. definition	Rela
17	Origin	From start
		A

- 2. Since this regards a restraint, both translations and rotations are captured.
- 3. Confirm the entry with [OK].
- 4. The floor slab can be selected by using the mouse.
- 5. Press  $\langle ESC \rangle$  to end the entry.
- 6. Press <**ESC**> once more to end the selection.
- 7. The supports are visualized:



# **Check structure**

After importing the geometry, the entry can be checked on errors by means of the option **Check structure data**. With this tool, the geometry is checked on duplicate nodes, zero-members, duplicate members, ...

## **Checking the structure**

- 1. Double-click on the option **Check structure data** in the **Structure menu** or press the button in the menu bar.
- 2. The window Check of structure data appears, on which the various checks are shown.

Check of nodes	
🔽 Search nodes	
	<b>—</b>
Search duplicate nodes	Ignore parameters
Check of members	
🔽 Check members	
Search null members	Null members: 0
	Delete null members
Search duplicate members	Duplicate members: 0
	Delete duplicate members
	Invalid parts: 0
	Delete invalid parts
Check of additional data	Invalid position
Check additional data position	Invalia position
	Correct position
Check of steel connections	
Check steel connections	Invalid connections 0
	Delete invalid connections

- 3. Click [Check] to perform the checks.
- 4. The window Report data check appears with a notification saying that no problems were found.

Data check	report	×
Data check	finished. No proble	ems found.
	ОК	

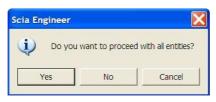
5. Close the check by clicking on [OK].

# **Connecting members/nodes**

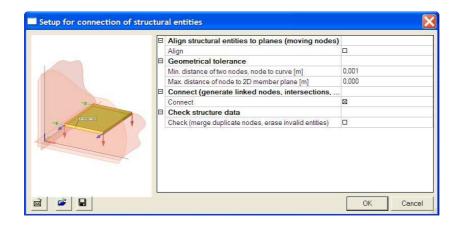
Analogously to the method in a project with 1D elements, the option 'Connect members/nodes' (in het construction menu) can be used to connect elements in a project with 2D elements (such as this project).

### **Connect members/nodes**

- 1. If necessary, first deselect the selected elements with the <**ESC**> button.
- 2. Double-click on the option **Connect members/nodes** in the **Structure menu** or press the button in the toolbar.
- 3. A dialog box appears asking if you want to continue with connecting all entities:



4. Answer 'Yes'. Subsequently the dialog box to set the connection of the construction entities appears. Close with OK.



5. Confirm the previous command. Since intersections were previously made, Scia Engineer shows that they were refreshed:



# **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.

#### Remark:

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.
- 2. Click on the button View in direction- Y

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

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

#### Editing view parameters through the menu View parameters

- 1. Click in the graphic window on the right mouse button. The following shortcut menu appears:
  - Coom all
     Zoom by cut out
     Set view parameters for all
     Cursor snap setting
     Print/ Preview table
     Table to document
     Print picture
     Picture to document
     Picture to gallery
     Save picture to file
     Copy picture to clipboard
     Wired model in view manipulations
     Advanced graphic setup ...
     Coordinates info

#### Remark:

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. The menu consists of various tabs. You can set the view parameters for all entities or just for the selected entities.

#### **View parameters – Entities**

Through the tab entities 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 colours per layer, material, cross-section or structural type.
- Cross-section: With this the symbol of the cross-section is displayed on every member.
- Local axes: With this the local axes of the elements are activated.

Check / Uncheck group	Lock position	1	
Modelling/Drawing	Misc. 🛛 🔍 View		
🕾 Structure	🕮 Labels 🔋 👗 Model		
Check / Uncheck all			
Service		^	
Display on opening the servi	ce 🔽		
Structure			
Style + colour	normal 💌		
Draw member system line	<b>v</b>		
Member system line style	system line 📃 💌		
Model type	analysis model 📃 💌	1	
Display both models			
Member surface			
Rendering	rendered with edges 📃 💌		
- Panel			
Member surface	<b>v</b>		
Rendering	transparent 💌		
Structure nodes			
Display			
Mark style	Dot 💌		
Cut-outs on 2Dmembers			
Contour of cut-outs		-	
2D members trimmed			
- Local axes			
Nodes		Y	

### View parameters – Labels and description

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

- Name: Show the name of the cross-sections in the label.
- Cross-section type: Show the cross-section type in the label.
- Length: show the length of the member in the label.

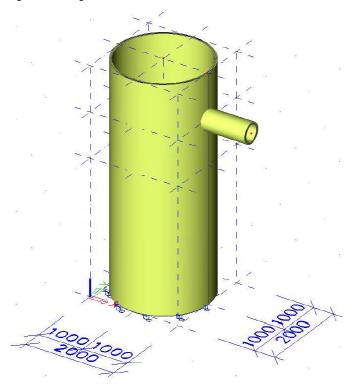
1)	Check / Uncheck group	Lock position	ſ
	Modelling/Drawing	Misc.   🔍 Vie Labels   👗 Model	W
7	Check / Uncheck all		
8	Nodes labels		^
	Display label		1
	Name		
	X-coordinate		
	Y-coordinate		
	Z-coordinate		
Ξ	Slab members		
	Display label	<ul> <li>Image: A set of the set of the</li></ul>	
	Name		
	Layer		
	Type and priority		
	Edges		
Ξ	Cut-outs on 2D members		
	Label		
Ξ	Labels of local axes		
	Nodes		
	Members 2D		
Ξ	General structural shape		5
	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 <sup>build</sup> to show the load case.
- Show/hide node labels
   ABC
   to view the label of the nodes.
- Show/hide member labels
   to view the label of members.
- Show/hide load labels to show the value of the load.
- Set load case for view
   to edit the active load case.
- Fast adjustment of view parameters on the whole construction III to quickly access to the options from the menu View parameters.

After rendering the following structure is obtained:



# Input calculation data

# Load cases and load groups

Every load is attributed to a load case. A load case can contain several load types.

Properties, that are decisive when generating combinations, are attributed to every load case. The action type of a load case can be permanent or variable.

Every load case is connected to a load group. The group contains information about the category of the load (service load, wind, snow, ...) and about the appearance (standard, together, exclusive). With an exclusive group, the different loads, attributed to the group, cannot occur together in a norm combination. With standard combinations, the combination generator allows the simultaneous occurrence of loads of the same group.

The way in which the load cases are defined, is decisive for the load combinations that are produced by the generator. It is recommended to thoroughly study the chapter about the loads and combinations from the reference manual.

In this project two load cases are imported:

- LC1: Permanent load case: self weight of the construction
- LC2: Permanent load case: water in vertical cylinder

### Defining a permanent load case

- 1. Double-click on Load in the Main window or click on in the toolbar.
- 2. Before loads can be defined, the load cases need to be imported first. Because no load cases were imported in the project, the **Load cases manager** automatically appears.
- 3. By default, the load type LC1 is made. This load is a permanent load with load type **Self weight.** By means of this type, the self weight of the construction is automatically calculated.
- 4. Since only self weight is attributed to the first load case in this project, the load type is set to Self weight.
- 5. In the field Description you can describe what this load case contains. For this project the description will be "Self weight Construction".

🎜 🤮 🗶 🛍 👠 🐎 🛛 🕰	🖆 😂 🖬 🛛 Ali	• 17	
LC1 - Self weight construc	Name	LC1	
240	Description	Self weight construction	
	Action type	Permanent	
	LoadGroup	LG1	-
	Load type	Selfweight	
	Direction	-Z	
	Load type	Selfweight	

### Defining a second load case

1. Click on New or on to make a seco

to make a second load case.

2. "Water" is entered as a description:

/1 }} <u>∕</u> № № № № № №	🗠 🚭 😂 🖬 🛛 All	- Y	
.C1 - Self weight construc.	Name	LC2	
C2 - Water	Description	Water	
	Action type	Permanent	
	LoadGroup	LG1	•
	Load type	Standard	

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

#### Remark: load groups

Every load is divided into a group. These groups influence the combinations that are generated as well as code dependent factors that are applied. The following logic is pursued.

Variable load cases, that have nothing to do with each other, can be linked to the various variable groups. Per group you set the category of the load (see LC1). The combination factors from the Eurocode are generated, based on the present load group. As soon as two load cases, belonging to different groups in a generated combination, are present, the decrease factors are applied for the transient loads.

If the load can be divided, you enter the various parts as separate load cases. As long as no variable load, belonging to another group in the load combination is present, no decrease factors can be applied. The different load cases of a divisible load can be linked to one single variable group.

The load cases of the same type, but that cannot occur together, can be put in one single group, which you make exclusive. For example "Wind X" and "Wind –X" are linked to one singe exclusive group "Wind".

# Loads

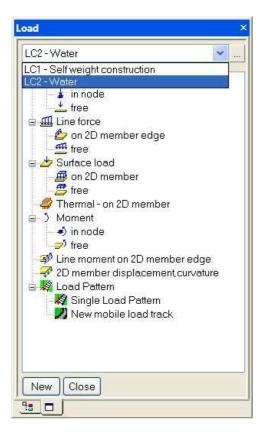
After the input of Load cases the Load menu automatically appears.

- The first load case consists of one load:
- Self weight of the construction.

The program automatically attributes the self weight to the construction.

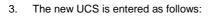
### **Changing between load cases**

Activate LC2 by indicating this load case in the Load menu:

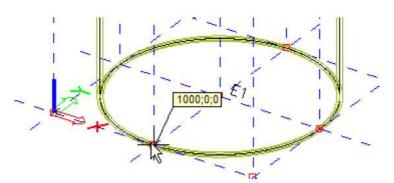


## **Free load**

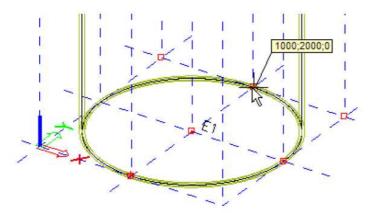
- 1. Close the selection by pressing <**ESC**>.
- 2. Define the new UCS by three points by means of the icon three points.



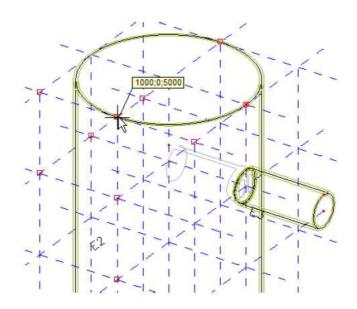
4. Defining the origin:



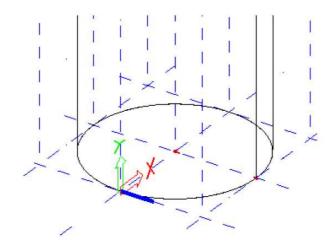
5. Defining the direction of the X-axis:



6. Defining the direction of the Y-axis:



7. The UCS is displayed as follows:



8. Set the local axes of the 2D elements visible with **Set view parameters for all → Structure → Local** axes (activation by right mouse button in the active window):

View parameters setting	
Check / Uncheck group	Lock position
Modelling/Drawing	Misc. 🕴 🔍 View
🕾 Structure	🗉 Labels 🔰 👗 Model 📗
Check / Uncheck all	
Service	
Display on opening the servic	e 🔽
Structure	
Style + colour	normal 💌
Draw member system line	~
Member system line style	system line 📃
Model type	analysis model 📃 🚽
Display both models	
Member surface	
Rendering	wired 🗾
🛛 Panel	
Member surface	
Rendering	wired 💽
Structure nodes	
Display	V
Mark style	Dot 🗾
Cut-outs on 2Dmembers	
Contour of cut-outs	
2D members trimmed	
🛛 Local axes	
Nodes	
Members 2D	
OK	Cancel

9. Click on Surface load - free in the Load menu. The dialog box Surface force free appears.

Surface force free		
	Name	FF1
-P	Direction	Z
Ar when	Туре	Force
	Distribution	DirY
	q1 [kN/m^2]	-50,00
ALL LE	P1	1
	q2 [kN/m^2]	0,00
ALL VIL	P2	2 🗾 🗾
	Validity	All
	Select	Select
2	Load case	LC2 - Water
	Geometry	
X	System	Member LCS 🔹
	Location	Length
ACCESSION AC	tions	
G	ienerate loads	>>>
Т	able edit geometry	>>>
		OK Cancel

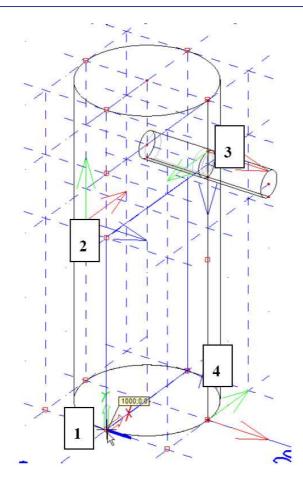
- 10. Set **Element LCS** in the geometry, because we want to load the loads on the structure according to the local axes of the 2D elements.
- 11. Set Direction Z, this is the direction in which the free load is going to work with regard to the LCS of the 2D element.
- 12. Set **Division Direction Y**, this is the direction in which the power will vary with regard to the previous defined UCS.
- 13. Enter the values for  $q1 = -50 \text{ kN/m}^2$  and  $q2 = 0 \text{ kN/m}^2$  as shown on the picture.
- 14. Validity All shows that the load operates on the structure in both the +z as well as the -z direction of the UCS. If the option would be put to +z, only the front part of the structure would be loaded. The origin of the UCS is in the middle of the structure and the +z axes refers to the front part.
- 15. Select "Select", by this the user can select the 2D elements that need to be loaded by the water load.
- 16. Confirm the entry with [OK].
- 17. Now draw the free load in the geometry in the order as shown on the figure:

punt1 (1;0;0) punt2 (1;0;4)

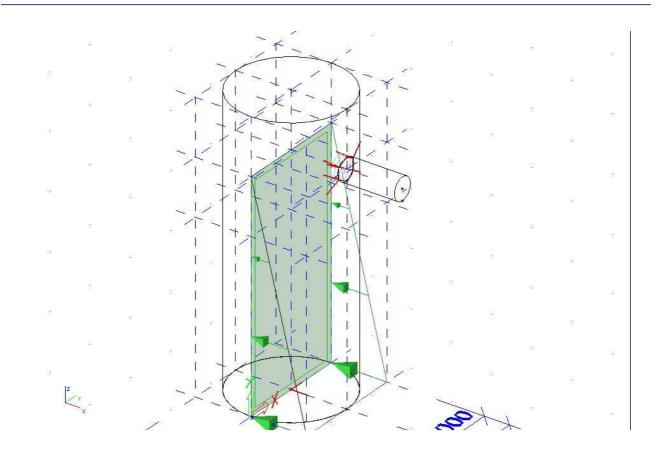
punz (1,0,4

punt3 (1;2;4)

punt4 (1;2;4)



- 18. The order of drawing the free load is important for the allocation of q1 and q2. Since -50 kN/m<sup>2</sup> was already attributed to q1, this value needs to be linked to the first insert point of the free load. To the second insert point, 0 kN/m<sup>2</sup> will be linked.
- 19. Press <**ESC**> to end the entry.
- 20. Click on Update 2D member selection in the action menu.
- 21. Select all elements.
- 22. Press <**ESC**> to end the entry.



# Calculation

# **Linear Calculation**

The analysis model is now completely finished, the calculation can be started.

### **Performing linear calculation**

- 1. Double-click on Recalculation under Calculation, mesh in the Main window.
- 2. The window FE analysis appears. Click [OK] to start the calculation.

	Ð
Single analysis Batch analysis	
Linear calculation	<u>v</u>
C Nonlineer calculation	
C Modal analysis	
C Linear stability	
Concrete - Code Dependent Deflections (C	
C Construction stage analysis	
C Nonlineer stage analysis	
C Nonlinear stability	
C Test of input data	
Number of load cases: 2	
Solver setup	Mesh setup
ОК	Cancel
	<ul> <li>Linear calculation</li> <li>Nonlinear calculation</li> <li>Modal analysis</li> <li>Linear stability</li> <li>Concrete - Code Dependent Deflections (C.</li> <li>Construction stage analysis</li> <li>Nonlinear stability</li> <li>Test of input data</li> <li>Number of load cases: 2</li> </ul>

3. After the calculation a window appears telling that the calculation has finished. Click [OK] to close this window.

# Results

#### **View results**

After performing the calculation, the results can be viewed. First of all it is visually checked if the free load has been entered correctly.

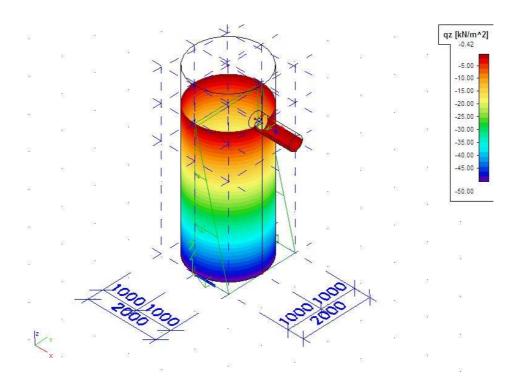
### 2D data viewer

- 1. Double-click on 2D data viewer in the menu tree under Calculation, mesh. The menu 2D data viewer opens.
- 2. Click on Surface loads.
- 3. The property window is set as follows:
  - The field Selection becomes All
  - The field Load type becomes Load cases
  - The field Load cases becomes LC2 Water
  - The field System becomes Local
  - The field Values becomes qz

		7 Ø
Name	Surface loads	
Selection	All	
Type of loads	Load cases	-
Load cases	LC2 - Water	
Filter	No	-
System	Local	
Values	qz	-

4. The action **Refresh** has a red background, which means that the graphic window needs to be refreshed. Press the

button next to **Refresh** to show the results in the graphic window according to the recently set options. The picture below shows the entered loads:



5. On the picture above the user can check how the free load operates on the construction. The figure clearly shows the trapezoidal course of the entered water load in the construction. At the top, the tank is loaded with 0 kN/m<sup>2</sup>, at the bottom this goes to 50 kN/m<sup>2</sup>. The floor slab is also loaded with 50 kN/m<sup>2</sup>. Here it is again clear how powerful the free load is in Scia Engineer. In addition, checking the 2D data is very convenient when using the free load since the user has 100% control on the input of the free load in this way.

After this check, the other results are reviewed.

## View deformations of nodes on 2D element

- 1. Double-click on in the Main menu. The Result menu appears.
- 2. Click under 2D members on Displacements of nodes.
- 3. The options in the **Property window** are configured as follows:
  - The field Selection is put on All.
  - The Load type is put on Load case and load case on LC2.
  - The Values are asked for Ux.
  - The Field **Construction** is edited into **Mesh**.

Verplaatsing van kr	- Va V 🦉	
Name	Displace	
Selection	All	
Type of loads	Load cases	
Load cases	BG2 - water	
Filter	No	
Structure	Mesh	
Drawing	Standard	
Values	Ux	
Extreme	Global	
Drawing setup		
Actions		
Retrest		>>>
Detailed results in n	nesh node	>>>

4. The action **Refresh** has a red background, which means that the graphic window needs to be refreshed. Press the button next to **Refresh** to show the results in the graphic window according to the recently set

options: Ux [mm] 0.00360 0.00300 0.00240 0.00180 0.00120 0.00060 -0.00000 -0.00060 -0.00120 -0.00180 -0.00240 -0.00300 -0.00360 --0.00433 -Z Y Jy Y

5. To display the results in tabular form, the action **Print Preview** is used. Press the button Preview to show the print preview.

Pie get view Loranies Tools Modely Tore Setup Window Help         Pie get view Loranies Tools Modely Tore Setup Window Help         Pie get view Loranies Tools Modely District Tools (1) and (1) a													a : 1]	🌾 Scia Engineer - [Shell.e:	
Image: Status       Image: Status       Image: Status         Image: Status       Image: Status       Image: Status       Image: Status         Image: Status       Image: Status       Image: Status	- 7									eln	Window H	s Setup			
Image: Standard designed and a constrained and constrained and a constrained and constrai				000											
Image: Structure       Image: Structure         Image: Structure	•														
Subt       a         A Supports       Supports         A Support Subtement of reactions       Foundation table         Comments       Foundation table         Statistic Comments       Foundation table		≤ 1.8 .	1 1 🕂	1÷	1		<ul> <li>1節節</li> </ul>	<b>古 福</b> 小 前	四月前一	日米草。	11 11 18				
Produced Structure         P													🕅 🕈 🔛 🎜 🕅 🕅	» \$P	
Image: Structure       Image: Structure         Image: Structure	Ux [mm]				, e - e		- W - +0	14				a x		sults	
a Supports       # Peackors         interesty       Peached or feactions         Poundation table       Poundation table         Notes space support resultant.       Peached on table         Poundation table       Peached on table         Member 2D - Internal Forces       Peached on table         Peached on aD member       Peached on table         Peached on aD table       Peac	0.00435														
Image: Section of 2D remained Forces         Image: Section of 2D remain	0.00360 -					700	-					^			
Imposity       Provide or reactions         Poundation table       Not space support resultant.         2D Members       Section 2D member forces         Member 2D Stresses       Not on 2D member.         Provide on table       Not space support resultant.         Impose 2D members       Impose 2D members         Provide on 2D members       Impose 2D members         Section All       Impose 2D	0.00300														
Possubar of reactions         © Displacements of notables         © Displacements of notables         © Member 2D-Stresses         © Stresses         BG2         Erectom All         E	0.00240				8	1									
Worksbors       Use and uses         20 Mombers       Use and uses	0.00180														
and cases       Bi2       Value       <	0.00120					1									
Owner Displace         Owner Displace           Image State         Image Displace           Image Displace         Im	0.00060				6 10 10 10	1							esultant		
Womber 2D - Internal Forces       Image: Construction of 2D member         Womber 2D - Stepsess       Image: Construction of 2D member         Womber 2D - Stepsess       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member         Image: Construction of 2D member       Image: Construction of 2D member	-0.00000			a 10 a 10		110		1							
Womber 2D-Stesses	-0.00060 -			20 <sup>(A)</sup>		11/2									
Section on 2D member         Image: Close         Image	-0.00120					IIIX									
Normalize         a _ i = i = i = i = i = i = i = i = i = i	-0.00180					110					2				
New         Cose         Image: C	-0.00240			2 8		- C			36	¥	-	~		Averaging strip	
	-0.00300		×.		8 - E	13									
Image: Constraint of the second sec						· · ·			8 🔨 II	Lat 12 ABC	001			New Close	
Operation         Operation           oppowerplaastingen (1)         Image: Comparing and the parameter of th										1	Littlends Trend street				
Oopvarpassinger (1)         Image of the formed of the	4									and a second second		a ×		operties	
Displacement of nodes           Displacement of nodes           Linear calculation, Extreme : Global           Colspan="2">Displacement of nodes           Colspan="2">Displacement of nodes           Displacement of nodes           Linear calculation, Extreme : Global           Colspan="2">Displacement of nodes           Displacement of nodes <th co<="" td=""><td></td><td>- = =</td><td></td><td>- 🕼 🖽</td><td>ault</td><td>🖬 🧻 def</td><td>- 🖾 (</td><td>96</td><td>10,00</td><td>5<i>6</i>      -</td><td></td><td>h 17 0</td><td>• 5</td><td>noonverplaatsingen (1)</td></th>	<td></td> <td>- = =</td> <td></td> <td>- 🕼 🖽</td> <td>ault</td> <td>🖬 🧻 def</td> <td>- 🖾 (</td> <td>96</td> <td>10,00</td> <td>5<i>6</i>      -</td> <td></td> <td>h 17 0</td> <td>• 5</td> <td>noonverplaatsingen (1)</td>		- = =		- 🕼 🖽	ault	🖬 🧻 def	- 🖾 (	96	10,00	5 <i>6</i>      -		h 17 0	• 5	noonverplaatsingen (1)
silection       Al       Inear calculation, Extreme : Global         perfloads       Load cases       BG2       Vatarduk       Selection : All         Selection : All       Selection : All       Selection : All       Selection : All         Selection : All       Selection : All       Selection : All       Selection : All         Selection : All       Selection : All       Selection : All       Selection : All         wing ad cases       BG2       Vatarduk       Ima       Imradl       Imradl       Imradl         awing Selup       Standard       V       BG2       E2       841       -0.00435       0.0000 - 0.00041       0.0       0.0         drame       Global       V       BG2       E2       843       0.00001 - 0.00437       0.00043       0.00       0.0       0.0         awing setup       Ima       Ima       K1       V       V       V       V       V       0.0 </td <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>5</td> <td>of node</td> <td>cement</td> <td>Displa</td> <td></td> <td></td> <td></td>								5	of node	cement	Displa				
thear calculation, Extreme : Global         ad cases       BG2 -Wateduk         ad cases       BG2 -Wateduk         ber       No         kade       Member       Node         teir       No         sandard       Gase       Member       Node       Imm       Imm       Imm       Immade         admas       Ur       Case       Member       Node       Imm       Imm       Immade       Immade         admas       Ur       Case       622       E2       641       0.00001       0.0001       0.0       0.0         admas       Ur       Case       622       E2       6413       0.00001       0.0001       0.0       0.0       0.0         admas       Global       Global       Gase       622       E2       657       0.00000       0.00001       0.00       0.0											14	-			
Stand classes     BG2-Waterdruk     V       Meric     No     V       Meric     Mesh     Vode     Imm]     Imm] <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>lobal</td> <td>xtreme : C</td> <td></td> <td></td> <td>-</td> <td></td> <td></td>								lobal	xtreme : C			-			
Mesh         Mesh         Mesh         mml         mml<												-			
Bit         Bit<         Bit         Bit         Bit <td></td> <td></td> <td>Fiz</td> <td>Fiv</td> <td>Fix</td> <td>Uz</td> <td>Uy</td> <td>Ux</td> <td>Node</td> <td>Member</td> <td>Case</td> <td>+</td> <td>No</td> <td>Iter</td>			Fiz	Fiv	Fix	Uz	Uy	Ux	Node	Member	Case	+	No	Iter	
among         Junication         BG2         E2         433         0.00435         0.00000         -0.00044         0.0         0.0         0.0           drame         Global         Image         Junication         BG2         E2         825         0.000000         -0.00043         0.0         0.0         0.0           drame         Global         Image         Junication         BG2         E2         825         0.00000         -0.00043         0.0         0.0         0.0           awing setup         Image         Junication         BG2         E1         K2         0.00000         0.00043         0.0         0.0         0.0           BG2         E1         K2         0.00000         0.00000         -0.00043         0.0         0.0         0.0           BG2         E1         K2         0.00000         0.00000         -0.00000         0.00000         0.00000         0.00000         0.00000         0.00000         0.00         0.0         0.0           BG2         E1         K1         0.00000         0.00000         -0.00000         0.00000         -0.00000         0.00000         -0.00000         -0.00000         -0.00000         -0.00000         -0.00000 <td></td> <td></td> <td>[mrad]</td> <td>[mrad]</td> <td>[mrad]</td> <td></td> <td>[mm]</td> <td>[mm]</td> <td></td> <td></td> <td></td> <td>•</td> <td>Mesh</td> <td>ructure</td>			[mrad]	[mrad]	[mrad]		[mm]	[mm]				•	Mesh	ructure	
Biology         Description         Description <thdescription< th=""> <thdescription< th=""> <th< td=""><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>•</td><td>Standard</td><td>awing</td></th<></thdescription<></thdescription<>												•	Standard	awing	
Bit Mark         Bit Car         E2         667         0,00000         0,00027         0,000437         0,00047         0,00047         0,00047         0,00010         0,000200												-	Ux	lues	
BG2         E1         K2         0,00000         0,00000         -0,032618         0,0         0,0         0,0           BG2         E1         K1         0,00000         0,00000         0,00000         0,00         0,0         0,0           BG2         E1         K1         0,00000         0,00000         0,00000         0,00         0,0         0,0           BG2         E1         Z8         0,00000         0,00000         -0,13366         0,5         0,0         0,0           BG2         E1         231         0,00000         0,00000         -0,13366         0,5         0,0         0,0           BG2         E1         135         0,00000         -0,13366         0,5         0,0         0,0           BG2         E1         124         0,00000         -0,13390         0,0         0,5         0,0           BG2         E2         3536         0,00088         0,0013         -0,00290         0,0         0,0           BG2         E2         3536         0,00088         -0,0013         -0,00290         0,0         0,0           BG2         E2         3482         0,00013         -0,00290         0,0         0,												-	Global		
BG2         E1         K1         0,00000         0,00000         0,00000         0,0         0.0           BG2         E1         28         0,00000         0,00000         -0,1336         -0,5         0.0         0.0           BG2         E1         28         0,00000         -0,0336         -0,5         0.0         0.0           BG2         E1         231         0,00000         -0,1336         0.5         0.0         0.0           BG2         E1         135         0,00000         -0,1336         0.5         0.0         0.0           BG2         E1         124         0,00000         -0,03390         0.0         -0.5         0.0           BG2         E2         3536         0,00001         -0,03390         0.0         0.0         0.0           BG2         E2         3536         0,00081         -0,00290         0.0         0.0         0.0           BG2         E2         3482         0.00084         -0,00131         -0,00290         0.0         0.0         0.0           BG2         E2         3482         0.00084         -0,00131         -0,00290         0.0         0.0         0.0														awing setup	
BG2         E1         28         0,00000         0,013386         -0.5         0.0         0.0           BG2         E1         231         0,00000         0,00000         -0,13386         0.5         0.0         0.0           BG2         E1         135         0,00000         0,00000         -0,13386         0.5         0.0         0.0           BG2         E1         135         0,00000         -0,13380         0.0         -0.5         0.0           BG2         E1         124         0,00000         -0,13390         0.0         0.5         0.0           BG2         E1         124         0,00000         -0,00290         0.0         0.0         0.0           BG2         E2         3482         0,00088         0,0013         -0,0290         0.0         0.0         0.0           BG2         E2         3482         0,00088         -0,0013         -0,0290         0.0         0.0         0.0           BG2         E2         3482         0,00088         -0,0013         -0,01290         0.0         0.0         0.0															
BG2         E1         231         0,00000         0,00000         -0,13386         0,5         0,0         0,0           BG2         E1         135         0,00000         0,00000         -0,13386         0,6         0,0         -0,6         0,0           BG2         E1         135         0,00000         0,00000         -0,13390         0,0         -0,6         0,0           BG2         E1         124         0,00000         0,00000         -0,01390         0,0         0,0         0,0           BG2         E2         3536         0,00008         0,0013         -0,00290         0,0         0,0         0,0           BG2         E2         3548         0,00013         -0,00290         0,0         0,0         0,0           BG2         E2         3548         0,00013         -0,00290         0,0         0,0         0,0           BG2         E2         3482         0,00013         -0,00290         0,0         0,0         0,0           BG2         E2         3482         0,00013         -0,00290         0,0         0,0         0,0           BG2         E2         3482         0,00013         -0,0013         -0,															
BG2         E1         135         0,00000         0,013300         0.0         -0.5         0.0           BG2         E1         124         0,00000         0,00000         -0.13390         0.0         0.0           BG2         E1         124         0,00000         0,00000         -0.05         0.0           BG2         E2         124         0,00001         0,00001         0.00         0.0         0.0           BG2         E2         2         3538         0,00013         -0.00290         0.0         0.0         0.0           BG2         E3         1482         0,00013         -0.00290         0.0         0.0         0.0           BG2         E3         1482         0,00013         -0.00290         0.0         0.0         0.0           BG2         E3         1482         0,00013         -0.00290         0.0         0.0         0.0															
BG2 E1 124 0,00000 0,00000 -0,13390 0,0 0,5 0,0 BG2 E2 3556 0,00088 0,00013 -0.00290 0,0 0,0 0,0 BG2 E2 3482 0,00088 0,00013 -0.00290 0,0 0,0 0,0 Header toms Command line															
BG2         E2         3536         0,00088         0,00013         -0,00290         0,0         0,0         0,0           BG2         E2         12482         0,00088         0,00013         -0,00290         0,0         0,0         0,0           Header         Image: Command line         Im															
IBG2 IE2 2492 INDONERLINDONSLINDOPEN NI NI NI NI Header Command line															
sons Command line															
ions Command line							Concerne Vision			r	E Heade				
											_	_		ions	
			. Sm [mu]					- In. In. 141	0 A 844 8			>>>		fresh	
stalled results in mesh node >>> PFI 1 + A) T & B B B B B B B A A A A A A A A A A A A	NUNE	NXX X		VAL B	C X G X			5666		-	Sec.				
eview >>> Command >										d>	Comman	>>>			

- 6. Since the deformations Ux and Uy are small, it would be better to increase the number of decimals of the deformations in the settings.
- Click on the icon
   and set number of decimals on 5 in Deformations/length:

÷	Geometry	
ŧ	Cross-section	
Ð	Stiffness	
Ŧ	Loads/Results	
Ξ	Deformation	
Ξ	Length	mm
	Unit	mm
	Decimal length	5
	Output format	decimal
ŧ	Angle	mrad
Đ	Mass	
ŧ	Others	
ŧ	Parameters	
ŧ	Concrete	
Đ	Construction stages, prestressing	

8. Press the button Refresh again to view the modifications on the screen.

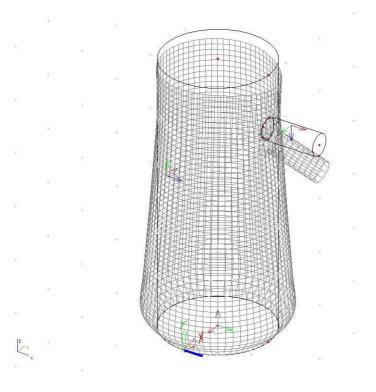
#### Remark:

The Print Preview appears between the Graphic Window and the Command line. This screen can be enlarged to display more data at once.

- 9. If the following adjustments are performed in the property window:
  - Change the field Structure into Initial
  - The field Values becomes Deformed mesh
  - Change the field Selection into Current and select manually element S2 and S3

Disalaraaraatataa	des (1) 🗸 🗸	12
Displacement of no		
Name	Displacement of n	od
Selection	Current	-
Type of loads	Load cases	
Load cases	BG2 - Waterdruk	
Filter	No	
Structure	Initial	
Drawing	Standard	
Values	Deformed mesh	
Extreme	Global	
Actions		>>
	2	12

The deformation of the construction can be reviewed in 3D:



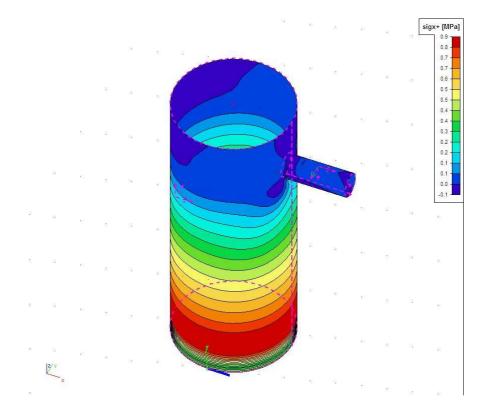
# **Display stresses**

Analogously to displaying the deformations, the stresses can also be reviewed in the menu of the results:

- 1. Click under 2D elements on 2D element stresses.
- 2. The options in the **Property Window** are configured as follows:
  - The field Selection becomes Current. Manuall select S2 and S3.
  - The Load Type is put on Load case and load case on LC2.
  - The Values are asked for sigx+.

Properties	12.17	a.
2D member - Stresses (1)	<ul> <li>Va V</li> </ul>	8
Name	2D member - Stresses	
Selection	Current	
Type of loads	Load cases	•
Load cases	BG2 - Waterdruk	
Filter	No	
System	Local	
Rotation [deg]	0,00	
Averaging of peak		
Location	In nodes, avg.	•
Type forces	Basic magnitudes	
Drawing	Standard	•
Values	sigx+	
Extreme	Global	•
Drawing setup		
Actions		
	>>	>
Detailed results in mesh n	ode _>>	>
Preview	>>	>

3. On the graphic window, the course of the stresses is shown under influence of the free load:



# Document

This last part of the tutorial shows how to draw up a calculation note.

## **Drawing up document**

- 1. Double-click on Document in the Main window or click on in the toolbar. The Document appears.
- 2. The project data are automatically displayed in the header of the document.
- 3. Click on the button [New] at the bottom of the Document menu. The window New document item appears.



- 4. By means of this window, various data can be inserted in the document.
  - Open the group Libraries en click on Materials. Click [<<< Add] to add this item to the document.
  - Open the group Structure and click on 2D members. Click [<<< Add] to add this item to the document.</li>
  - Open the group Results and click on Member2D Stresses. Click [<<< Add] to add this item to the document.</li>
- 5. Click [Close] to close the window New document item and to return to the document.

The items that were added to the document, are displayed in the **Document menu**. The order of the items can be edited by dragging the mouse. On the right side of the screen the Print Preview of the document is shown.

## Showing results in the document

- 1. Click in the **Document menu** on **2D element Stresses**. The properties of this table are displayed in the **Property window**. The configuration of the parameters for the display of the results in the **Document** happens completely analogously to viewing the results in the **Result menu**.
- The selection field is put on All.
- The Load type is put on Load cases and load case is put on LC2.
- The Values are asked for sigx+.
- The field Extreme is edited into **Global**.
- 2. Press the button \_\_\_\_\_ next to **Refresh** to show the table according to the recently set options.

068 20		1.000												-
	Shell tut.esa													
ocument		Φ×	nu 55	80 %	100 %	-	63 64	📕 default		- 🖼 🏢		-	•	
DOC-Default	~		ľ											
		_												
Default Materials						Project			Ĩ	-			-Tutorial S	hell
	0 (2D members)					Part								-
	er - Stresses (Member2D - Stress	(295		EMETS	GHEK	Descriptio	n							
Co memor	a de cases (i lenidenzo de cas		00	Ja		Author			1					-FL
			1. Mate	erials										
			Name	Туре	Unit ma [kg/m <sup>3</sup>			isson - n	u G m	od Th	hermal ex [m/mK]	p		
			S 235	Steel		00 2,1000			8,0769			00		
				and the second	1. A.S. 194		10 10 1 10 10 10 10 10 10 10 10 10 10 10			1997 B				
			2 Man	ber 2D										
						less s		_						
			Name	Material	Th. [mm]	Thickness	type	Туре	Layer					
			E1	S 235	50	constant		e (90)	Laag1					
			E2	S 235		constant		ll (98)	Laag1					
New Close			E3	S 235	50	constant	wall	(80)	Laag1					
			in the second second											
operties		ф×	3. 2D r	nember	- Stress	es								
			2018	1 632 1	2 83 10 10									
	• % V	1	Linear ca	liculation, t	Extreme : C	Global								
esults (1)			Selection Load cas	All BG2										
esults (1) ame	Member2D - Stresses 2D member - Stresses	9	Selection Load cas Basic ma	: All ses : BG2 sgnitudes. I	n nodes, a	ivg					ing second 1			
asults (1) ame aption	Member2D - Stresses	9	Selection Load cas	All BG2	n nodes, a	ivg.	Y	Z	sigx+	sigy+	sigxy+	sigx-	sigy-	sig xy-
asults (1) ame aption sible	Member2D - Stresses 2D member - Stresses Ø yes	0	Selection Load cas Basic ma	: All ses : BG2 sgnitudes. I	n nodes, a	ivg	Y [m] 1,000	Z [m] 0.000	sigx+ [MPa] -9,7	sigy+ [MPa] -9,7	sigxy+ [MPa] 0.0	sigx - [MPa] 9,7	sigy- [MPa] 9,7	sigxy- [MPa] 0.0
asults (1) ame aption sible efer one page	Member2D - Stresses 2D member - Stresses Ø yes All		Selection Load cas Basic ma Case BG2 BG2 BG2	: All ses : BG2 gnitudes. Member E1 E1	n nodes, a Node K2 138	xyg X [m] 1,000 1,900	[m] 1,000 0,900	[m] 0,000 0,000	[MPa] -9,7 10,6	[MPa] -9,7 2,2	[MPa] 0,0 -1,0	[MPa] 9,7 -10,6	[MPa] 9,7 -2,2	[MPa] 0,0 1,0
esults (1) ame aption isible refer one page election ype of loads	Member2D - Stresses 2D member - Stresses 23 yes 24 All Load cases		Selection Load cas Basic ma Case BG2 BG2 BG2 BG2	: All ess : BG2 gnitudes. I Member E1 E1 E1	n nodes, a Node K2 138 259	X [m] 1,000 1,900 0,900	[m] 1,000 0,900 1,900	[m] 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2	[MPa] -9,7 2,2 10,7	[MPa] 0,0 -1,0 -1,0	[MPa] 9,7 -10,6 -2,2	[MPa] 9,7 -2.2 -10,7	[MPa] 0.0 1.0 1.0
asults (1) ame aption sible efer one page alection /pe of loads oad cases	Member2D - Stresses 2D member - Stresses 20 yes 21 All Load cases 22 - water	• • •	Selection Load cas Basic ma BG2 BG2 BG2 BG2 BG2	: All des : BG2 denitudes. I Member E1 E1 E1 E1 E1 E1	n nodes, a Node K2 138 259 348	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
esults (1) ame aption isible refer one page election ype of loads opad cases liter	Member20 - Stresses 2D member - Stresses 3 yes All Load cases BG2 - water No	4 4 4 4	Selection Load cas Basic ma Case BG2 BG2 BG2 BG2	: All ess : BG2 gnitudes. I Member E1 E1 E1	n nodes, a Node K2 138 259	X [m] 1,000 1,900 0,900	[m] 1,000 0,900 1,900	[m] 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2	[MPa] -9,7 2,2 10,7	[MPa] 0,0 -1,0 -1,0	[MPa] 9,7 -10,6 -2,2	[MPa] 9,7 -2.2 -10,7	[MPa] 0.0 1.0 1.0
esults (1) ame aption refer one page election ype of loads oad cases liter ystem	Member2D - Stresses 2D member - Stresses 2 January	• • •	Selection Load cas Basic ma BG2 BG2 BG2 BG2 BG2	: All des : BG2 denitudes. I Member E1 E1 E1 E1 E1 E1	n nodes, a Node K2 138 259 348	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
esults (1) ame aption sible lection pad cases lter ystem otation [deg]	Member2D - Stresses 2D member - Stresses 3 yes 4 All Load cases BG2- water No Local 0.00	4 4 4 4	Selection Load cas Basic ma BG2 BG2 BG2 BG2 BG2	: All des : BG2 denitudes. I Member E1 E1 E1 E1 E1 E1	n nodes, a Node K2 138 259 348	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
sults (1) ame aption sible efer one page election alection ype of loads optionals ter ystem optionals retaging of peak	Member/20 - Stresses           20 member - Stresses           aves           All           Load cases           BG2- water           No           Local           0,00	* * *	Selection Load cas Basic ma BG2 BG2 BG2 BG2 BG2	: All des : BG2 denitudes. I Member E1 E1 E1 E1 E1 E1	n nodes, a Node K2 138 259 348	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
sults (1) ame aption sible efer one page efer one page per of loads tad cases ter vata cases ter vatem ter vatem tation [deg] reraging of peak cation	Member2D - Stresses 2D member - Stresses 2 ves All Load cases BG2 - water No Local 0,00 In nodes, avg.	* * *	Selection Load cas Basic ma BG2 BG2 BG2 BG2 BG2	: All des : BG2 denitudes. I Member E1 E1 E1 E1 E1 E1	n nodes, a Node K2 138 259 348	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
suits (1) ame pption sible eferone page eferone page eferone page of loads and cases ter rester rester rester restation (deg) creation cation cation pe forces	Member2D - Stresses 2D member - Stresses 3 yes 4I Load cases BG2-water No Local 0,00 1 In nodes, avg. Basic magnitudes	· · · ·	Selection Load cas Basic ma BG2 BG2 BG2 BG2 BG2	: All des : BG2 denitudes. I Member E1 E1 E1 E1 E1 E1	n nodes, a Node K2 138 259 348	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
suits (1) ame aption refer one page refer one page election page of loads bad cases liter suits of the set bad cases liter verging of peak coation uppe forces rawing	Member2D - Stresses 2D member - Stresses 2D member - Stresses 2D - Stres	• • •	Selection Load car Basic mu BG2 BG2 BG2 BG2 BG2 BG2 BG2	: All press: BG2 press: BG2 press: BG2 member E1 E1 E1 E1 E1 E1 E1 E1 E1 E1	Nodes, a Node K2 138 259 348 321	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
suits (1) ame offer one page election election of dessos tetre reter ret	Member2D - Stresses 2D member - Stresses 0 All Load cases BG2- water No Local 0.00 0 in nodes, avg. Basic magnitudes Standard sigx+		Selection Load car Basic my BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2	: All press: BG2 press: BG2 press: BG2 member E1 E1 E1 E1 E1 E1 E1 E1 E1 E1	Nodes, a Node K2 138 259 348 321	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
suits (1) ame aption sible efer one page election ype of loads pade cases pade cases pad	Member2D - Stresses 2D member - Stresses 2D member - Stresses 2D - Stres		Selection Load car Basic my BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2	: All press: BG2 press: BG2 press: BG2 member E1 E1 E1 E1 E1 E1 E1 E1 E1 E1	Nodes, a Node K2 138 259 348 321	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3	[MPa] 9,7 -2.2 -10,7 -6.1	[MPa] 0.0 1.0 1.0 4,3
suits (1) ame offer one page election election of dessos tetre reter ret	Member2D - Stresses 2D member - Stresses 0 All Load cases BG2- water No Local 0.00 0 in nodes, avg. Basic magnitudes Standard sigx+		Selection Load car Basic my BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2	: All press: BG2 press: BG2 press: BG2 member E1 E1 E1 E1 E1 E1 E1 E1 E1 E1	Nodes, a Node K2 138 259 348 321	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3 -7,3	[MPa] 9,7 -2,2 -10,7 -6,1 -6,1	[MPa] 0.0 1.0 4,3 4,3
suits (1) ime ption ption ptiole efer one page lection per of loads ad coses ter ter ter ter ter cabon pe forces awing tutes	Member2D - Stresses 2D member - Stresses 0 All Load cases BG2- water No Local 0.00 0 in nodes, avg. Basic magnitudes Standard sigx+		Selection Load car Basic my BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2 BG2	: All es : BG2 genitudes I Member E1 E1 E1 E1 E1 E1 E1 E1 E1 E1	Nodes, a Node K2 138 259 348 321	X [m] 1,000 1,900 0,900 1,692	[m] 1,000 0,900 1,900 0,404	[m] 0,000 0,000 0,000 0,000	[MPa] -9,7 10,6 2,2 7,3	[MPa] -9,7 2,2 10,7 6,1	[MPa] 0.0 -1.0 -1.0 -4,3	[MPa] 9,7 -10,6 -2,2 -7,3 -7,3	[MPa] 9,7 -2,2 -10,7 -6,1 -6,1	[MPa] 0.0 1.0 1.0 4,3

3. Press the button [Close] at the bottom of the Document menu to close the document and to return to the structure.

## Adding a picture to the document

- 1. Press the button **Print picture** in the toolbar.
- 2. Choose the option **Picture to document** from the menu to send the current picture on the graphic window to the document.

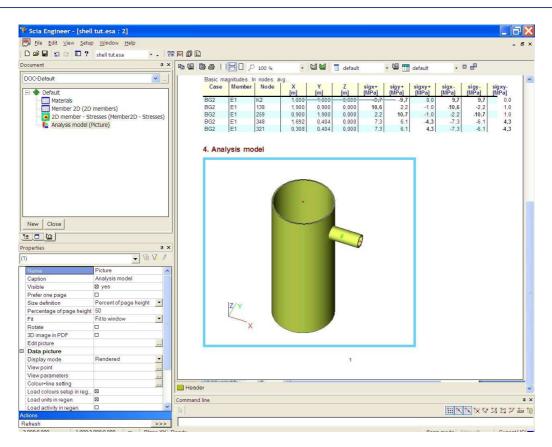
The window Add picture in document appears.

-	Name	Picture	
	Caption	Analysis model	
	Visible	⊠ yes	
	Prefer one page		
	Size definition	Percent of page height	•
	Percentage of page height	50	
	Fit	Fit to window	•
	Rotate		
	3D image in PDF		
Ξ	Data picture		
	Display mode	Rendered	•
	Load units in regen.		
	Load activity in regen.	1	
	Text scale factor Charset of texts	ANSI (USA, UK, Europe)	
		3	_
	Line pattern length Display GCS icon	To picture.comer	
Ξ		To picture.comer	_
	Settings	>>	~

3. The field **Size** is edited to **50** so the figure covers 50% of the page or half a page.

4. The field **Display mode** is edited to **Rendered** so the picture is also displayed rendered in the document.

- 5. Confirm the entry with [OK] so the picture is sent to the document.
- 6. Press in the toolbar to open the **Document**.
- 7. Click in the Document on Picture. The picture is displayed in the Print Preview of the Document.



8. Press the button [Close] at the bottom of the Document menu to close the document and to return to the structure.

# Epilogue

In this syllabus the basic functionalities were introduced regarding the input of:

- Curved shells
- Intersection between 2D elements
- Cut-outs
- Free load

by means of an example.

After studying the text and entering an example, the user should be able to model and calculate simple shell structures.