NEMETSCHEK Scia

Stade des Alpes - Grenoble
Etudes et Techniques Internationales (ETI)

Tutorial

Shell

Scia Engineer

Table of Contents

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
	ESAS.01	Linear Statics 3D

Manual: Scia Engineer Tutorial Shell

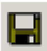
Revision: 06/2009

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).

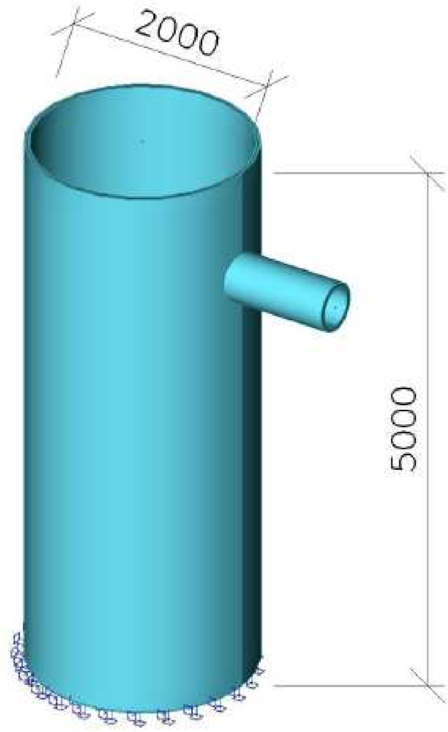
Introduction

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:

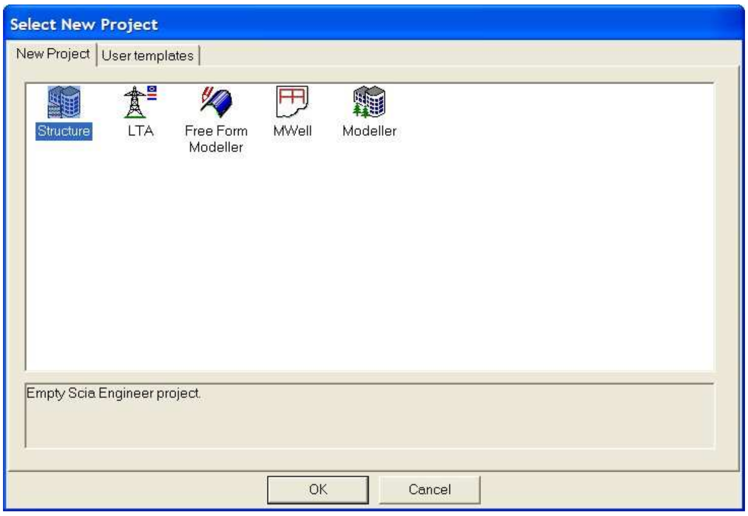
2. If the shortcut is not installed, click [**Start**] and choose **Programs > SCIA ENGINEER 2009.0 > SCIA ENGINEER 2009.0**.

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.

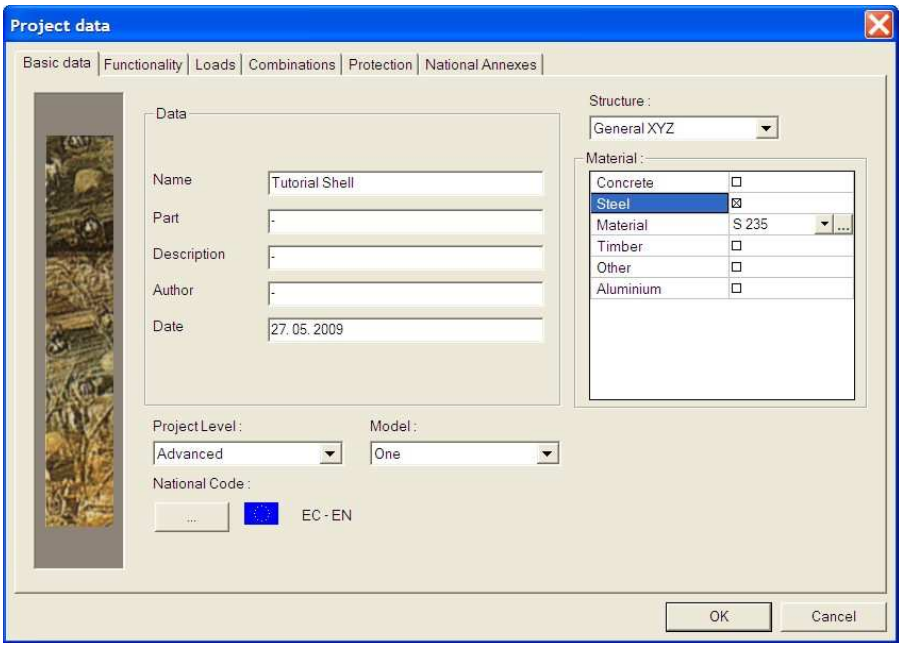
For this Tutorial a new project is created.

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**].



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



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

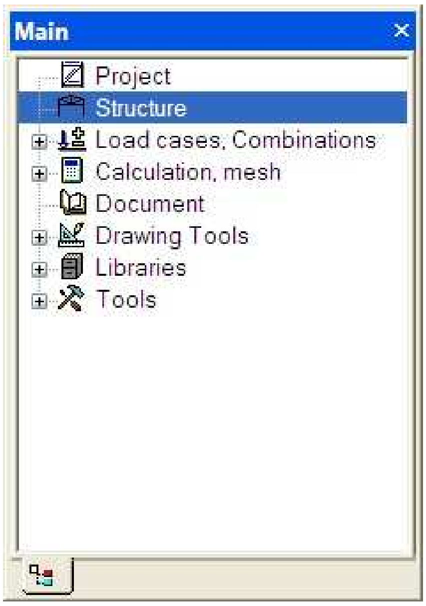
Inputting the geometry

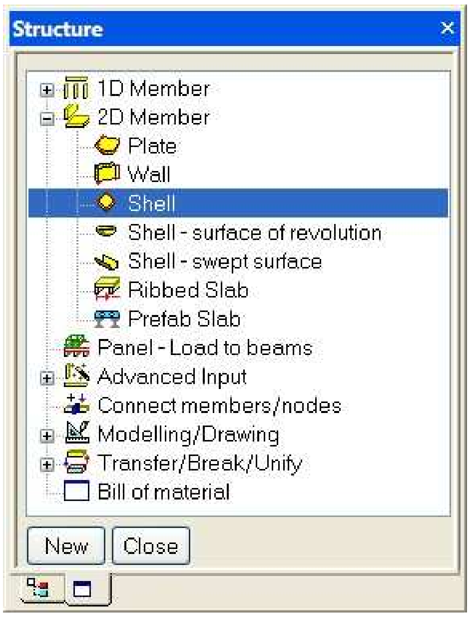
If a new project is started, the geometry of the construction has to be entered. The construction can be imported directly, but also templates with parametric blocks, DXF files and other formats can be used.

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**.

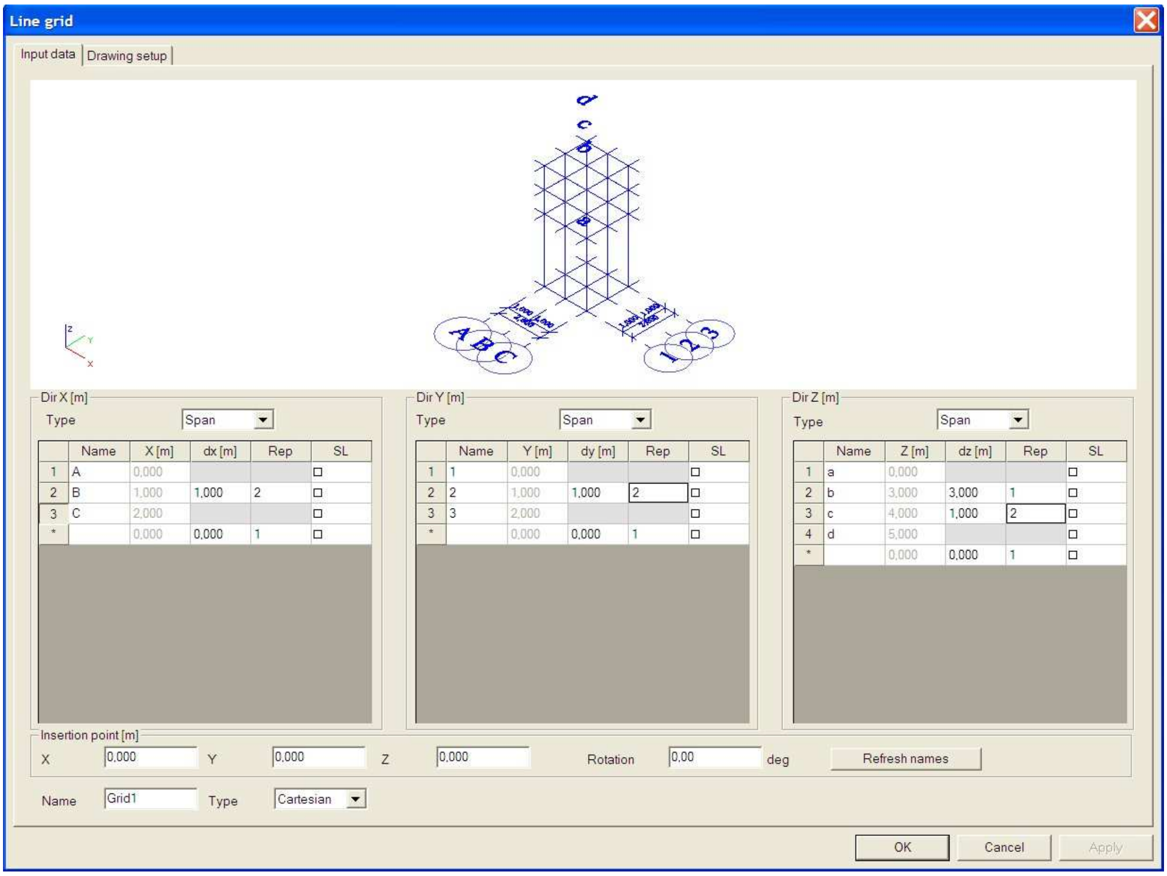




2. In the **Structure menu** you can choose from several structural elements.

Line grid

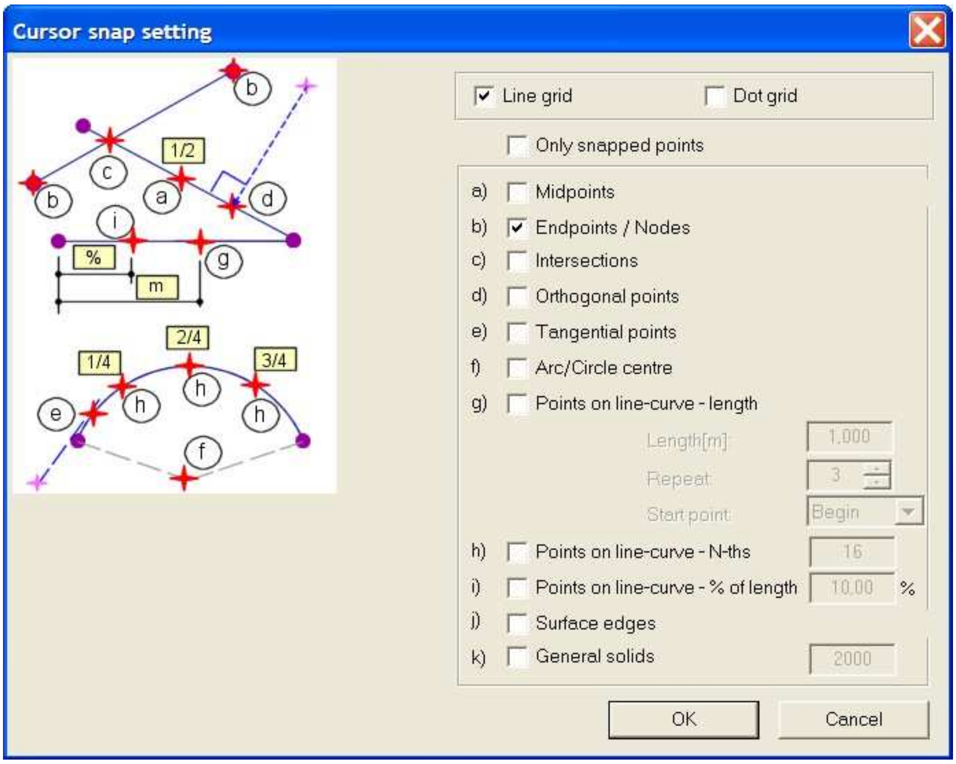
To simplify the input of the geometry, the line grid and snap settings are used. when clicking on the icon  in the toolbar, the following window appears:



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

Cursor snap settings

1. Double-click on the button **Cursor snap settings**  in the command line or press the button [**Snap mode**] bottom right of the screen. The window **Cursor snap setting** appears:



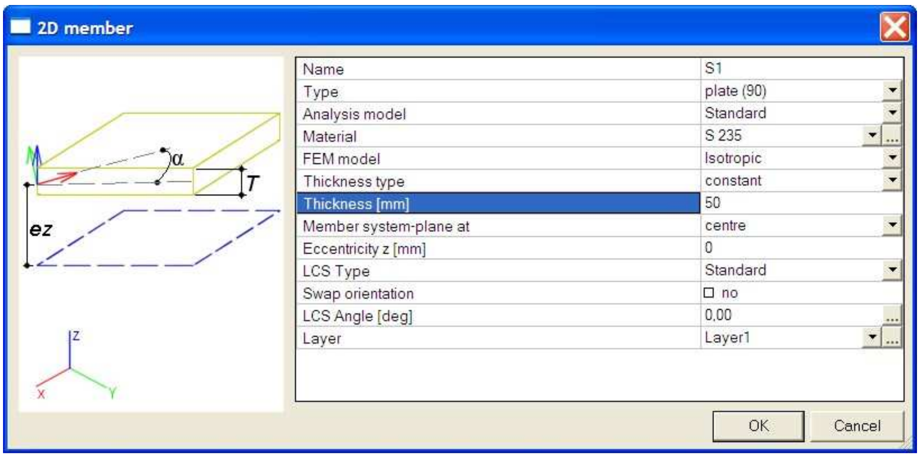
2. Choose the option line grid so the project can be snapped on the grid line you now created.

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**.

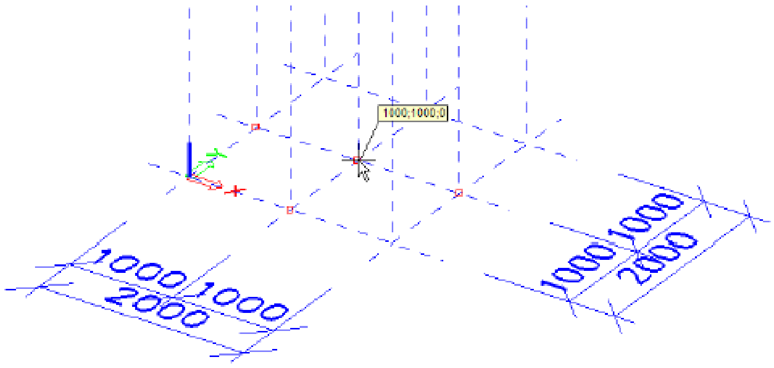


2. In the field thickness, you enter 50 mm.

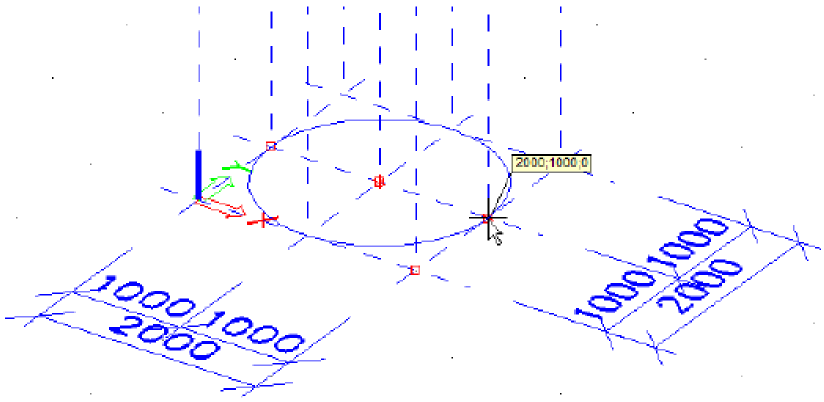
3. Confirm the input with [**OK**].

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):



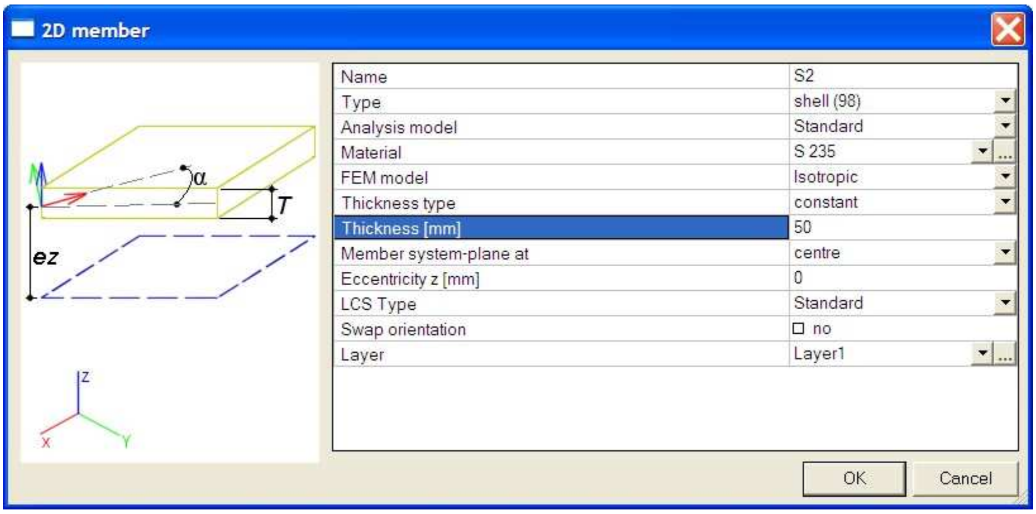
7. Press <**ESC**> to confirm.

After drawing the floor slab, the walls are entered.

Entering a shell element

1. Use the option 

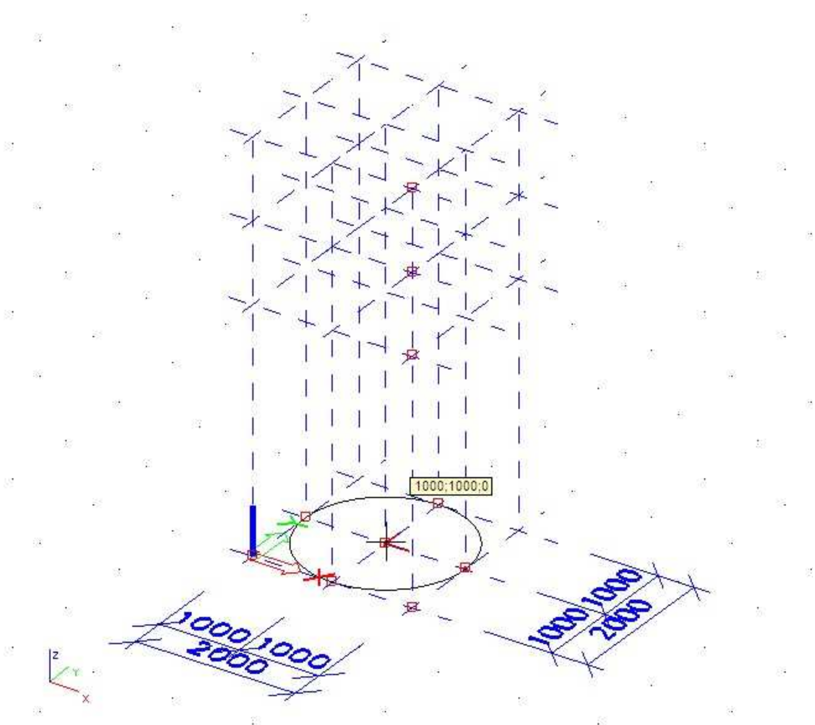
in the **structure menu** to enter a new shell element.



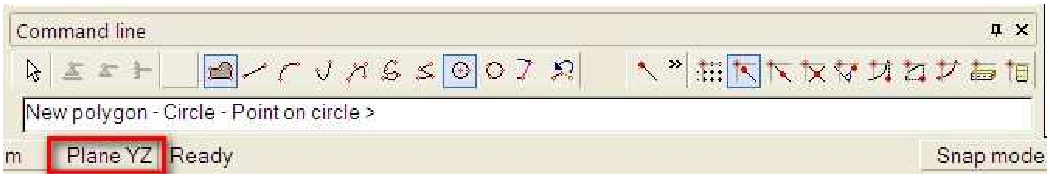
2. Enter 50 mm in the field Thickness.

3. Confirm the input with [**OK**].

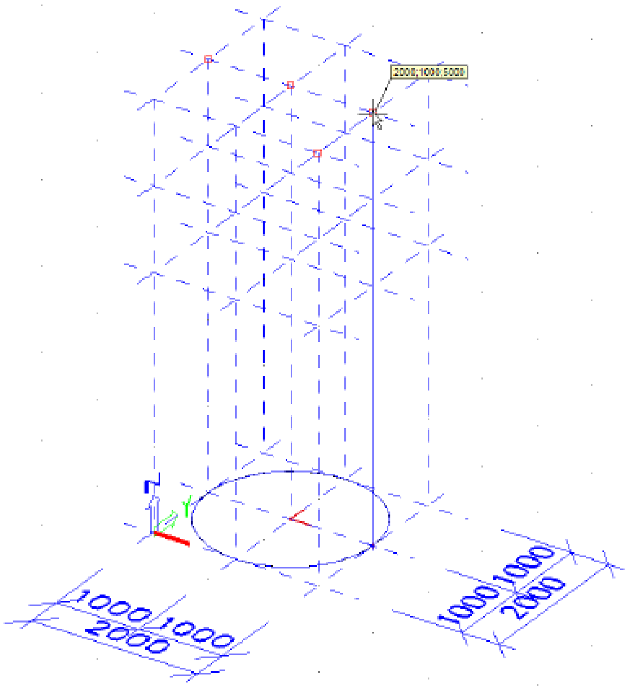
4. Re-enter a circle with the same centre and radius as the floor slab.



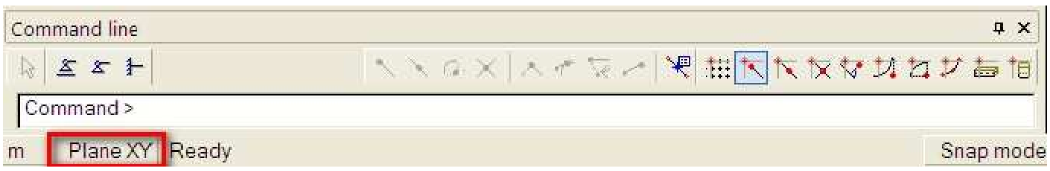
5. Change the active plane in YZ by means of the button bottom left of the command line:



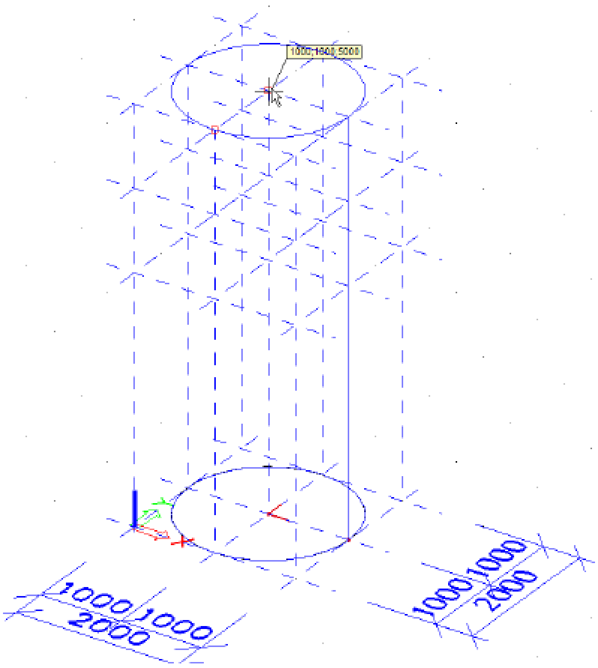
6. Enter the height of the cylinder by clicking on the co-ordinate (2;1;5):



7. Re-activate the XY-plane:



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.*

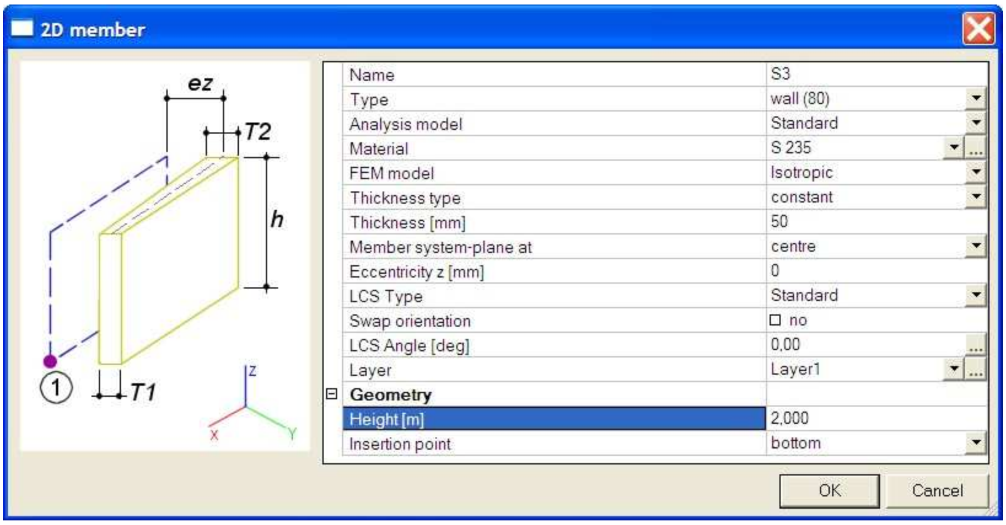
*The button **Zoom All***  *in the toolbar allows you to visualize the entire construction.*

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.

2. To import a new membrane element, use the option  Wall in the **structure menu**.

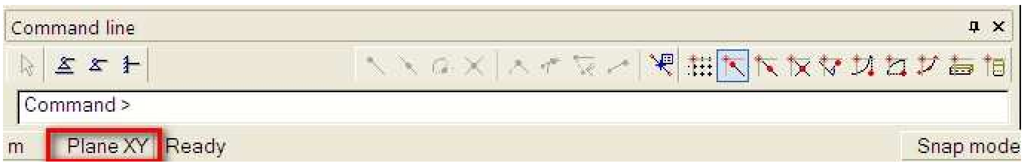


3. **50 mm** is entered in the field Thickness.

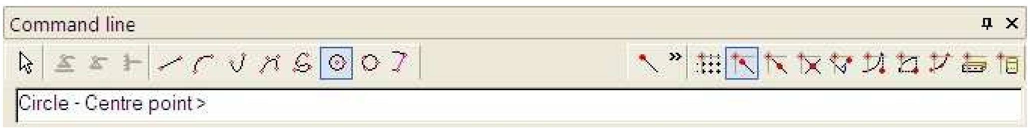
4. The height of the wall is 2 m.

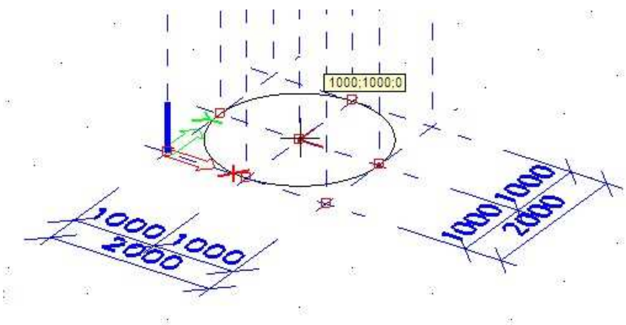
5. Confirm the entry with [**OK**].

6. Change the active plane in XY:



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



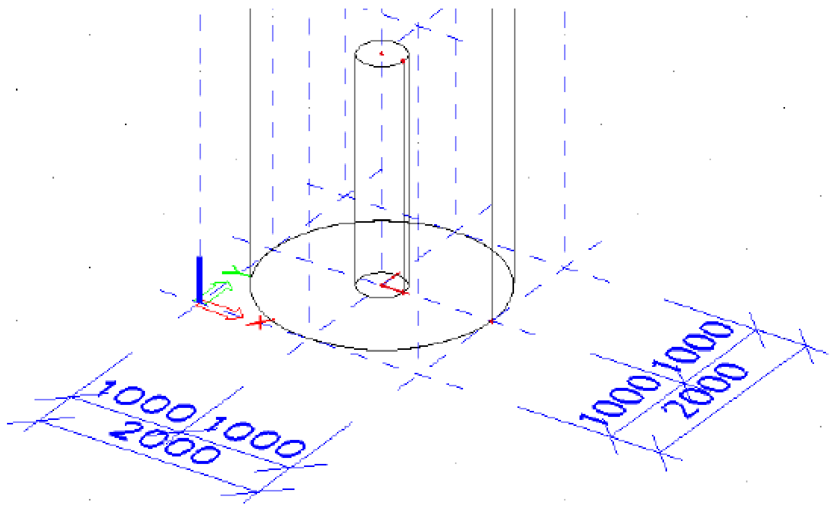


8. Enter the radius of the circle with relative coordinates:

Remark

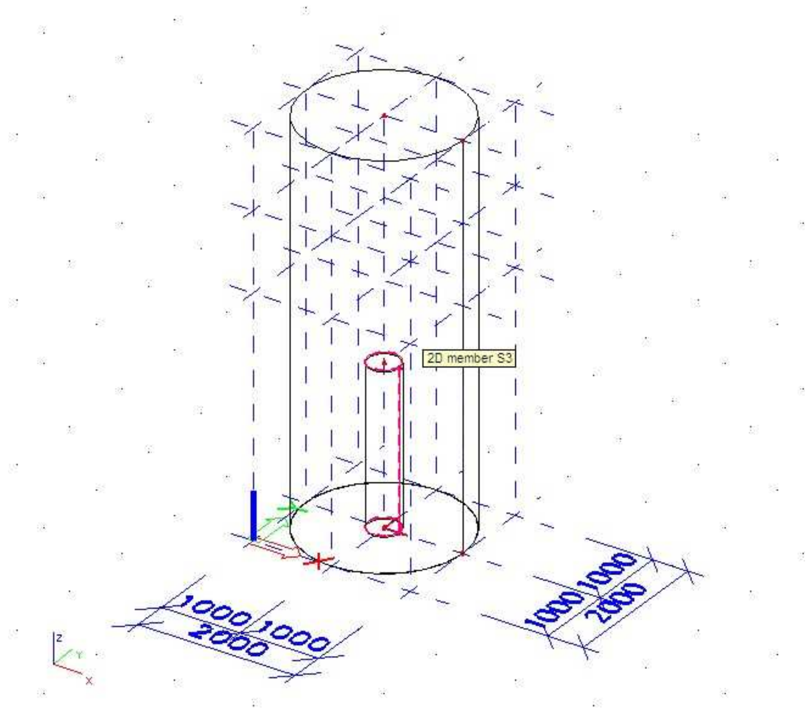
In Scia Engineer you can use absolute or relative coordinates. The difference is the use of the origin and the import of the coordinates. With absolute coordinates, the origin of the UCS is used, these coordinates are entered with the x;y;z coordinates. If the relative coordinates are used, the last 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:

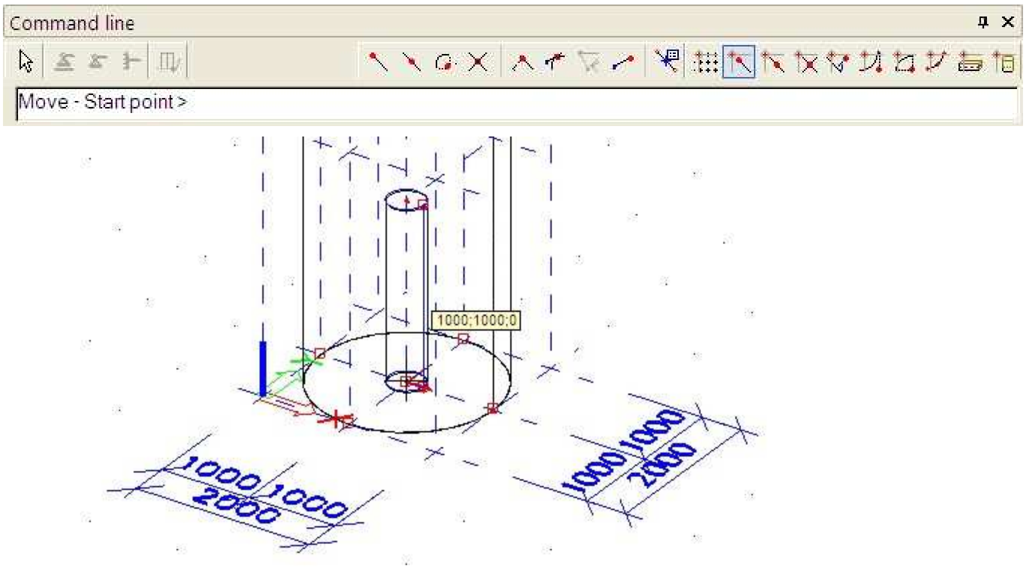


10. The cylinder is being moved with the help of the command move '. This command can be found in the menu bars under **Modify → Move**

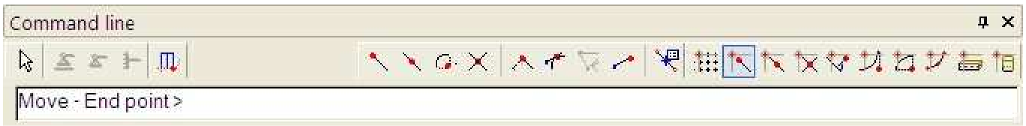
11. First you have to select the element that has to be moved. Confirm afterwards with <**ESC**>:

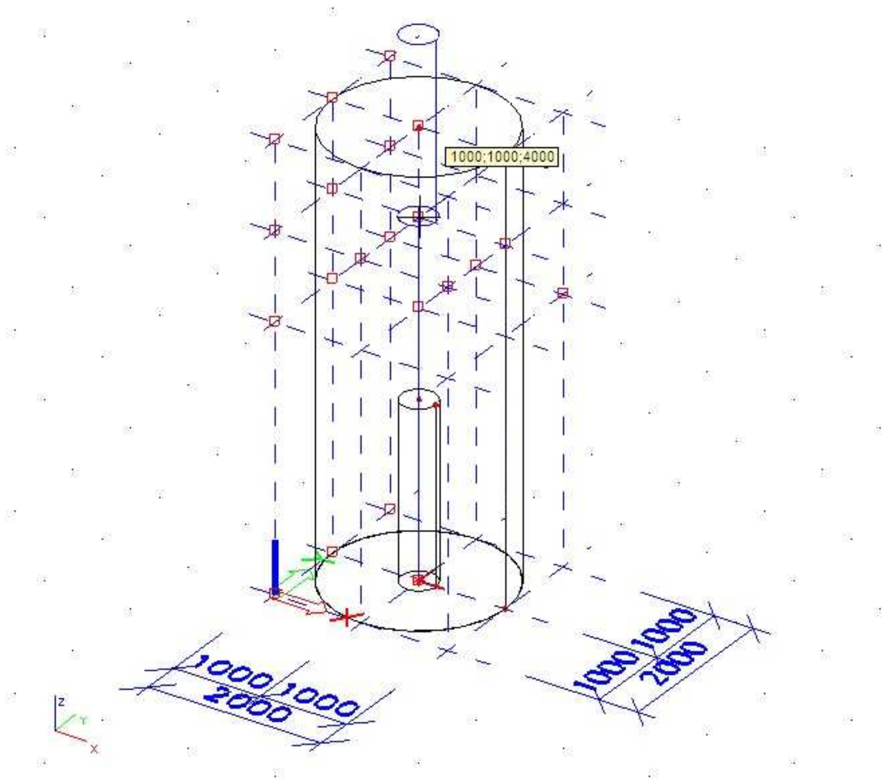


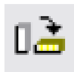
12. Then the start point has to be defined. Point (1;1;0) is used:



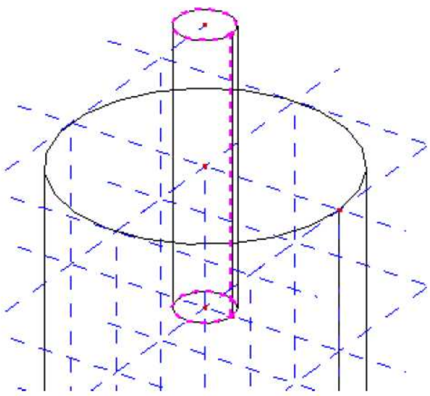
13. Subsequently the end point has to be entered (1;1;4):





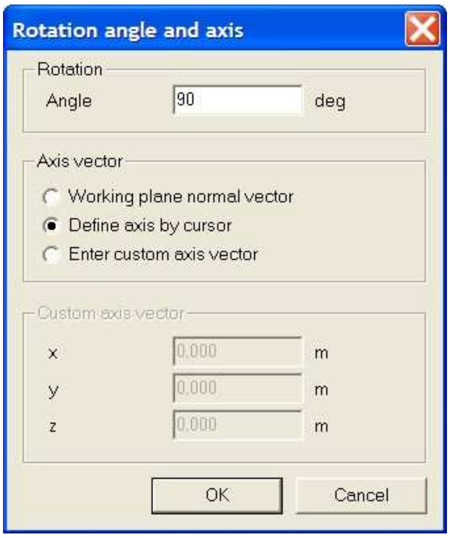
14. Now the cylinder has to be rotated. This is done by using the command rotate  , which you can find in the menu bar **Modify**. This command asks the user to select an element:

15. The cylinder is selected:



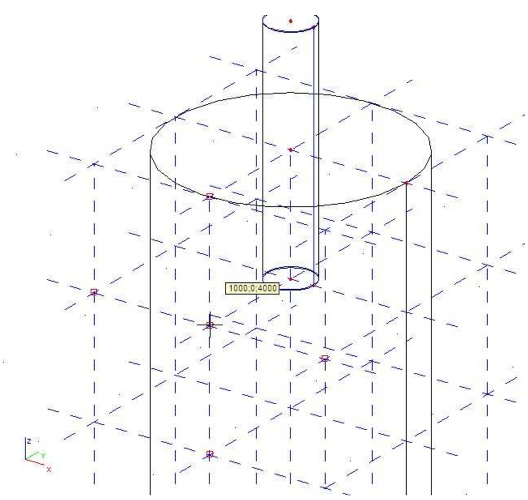
16. Afterwards the icon  is used. This icon can be found at the bottom of the command bar.

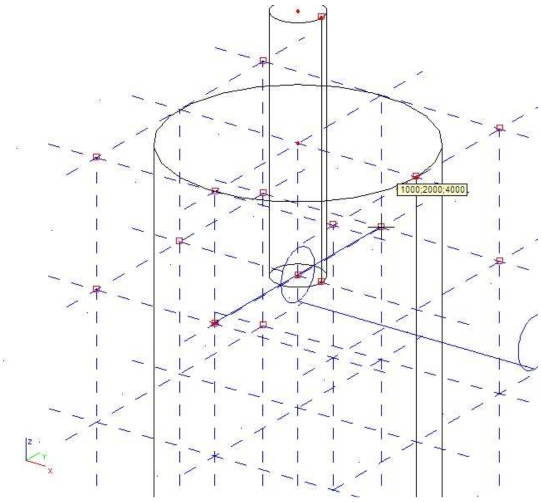
17. The following dialog box is displayed:



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.

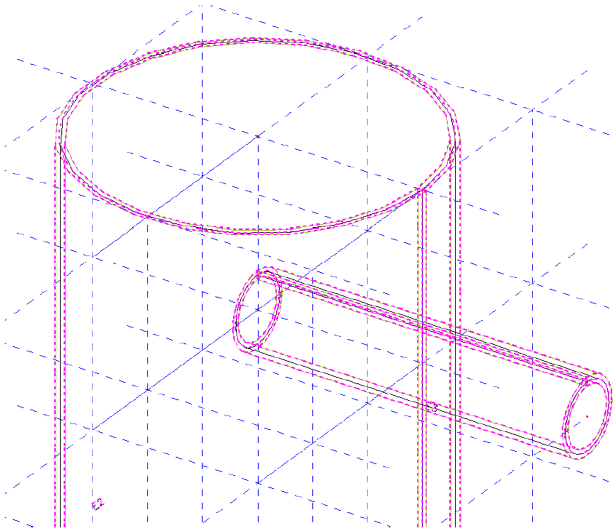
Intersection and cut outs

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

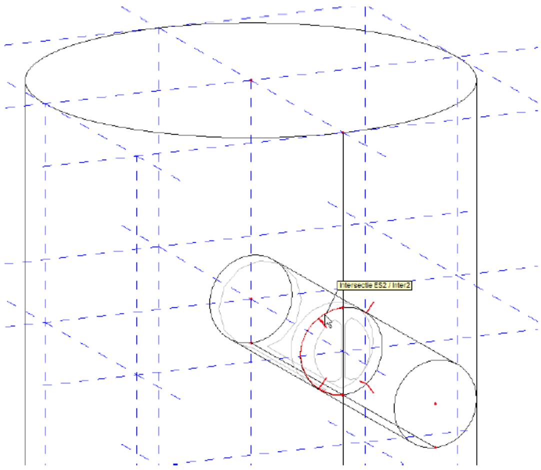
1. To enter a new intersection, use the icon  in the **Structure menu → 2D Member → 2D Member components**.

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.

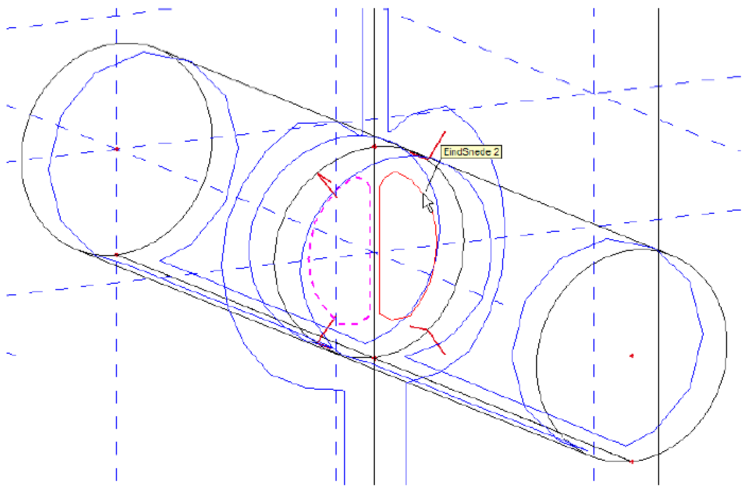
Defining cut-outs

1. Use the icon Cut-out in the **Structure menu** under 2D 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:



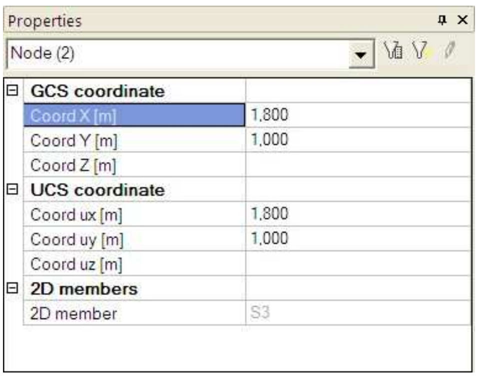
5. Confirm the selection with <**ESC**>.

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

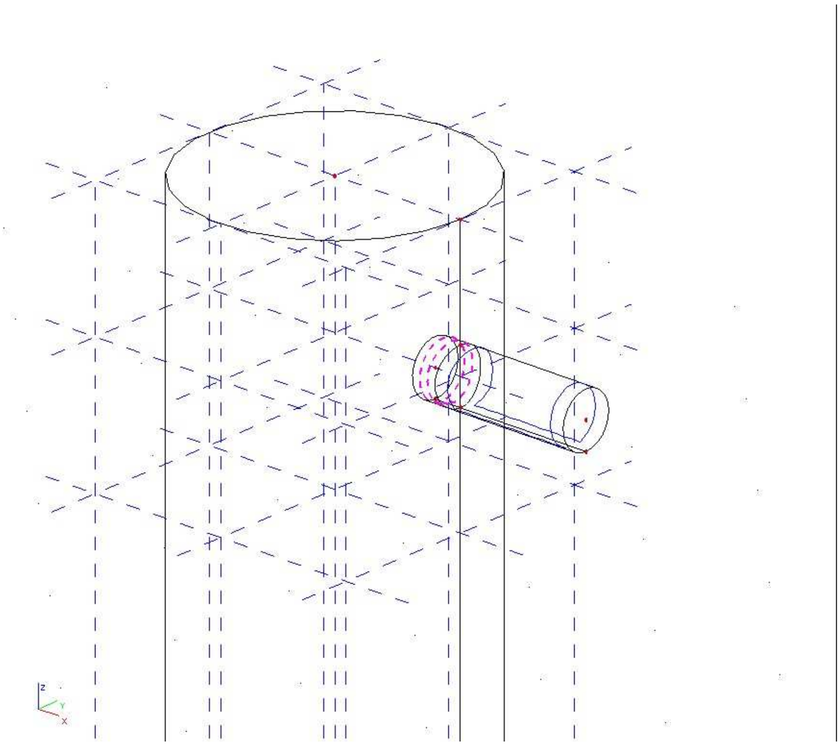


10. Click on the icon Cut-out in the **menu tree** under 2D member components .

11. Select the horizontal cylinder S3.

12. Confirm the entry with <**ESC**>.

13. Select the cut-outs you want to remove:



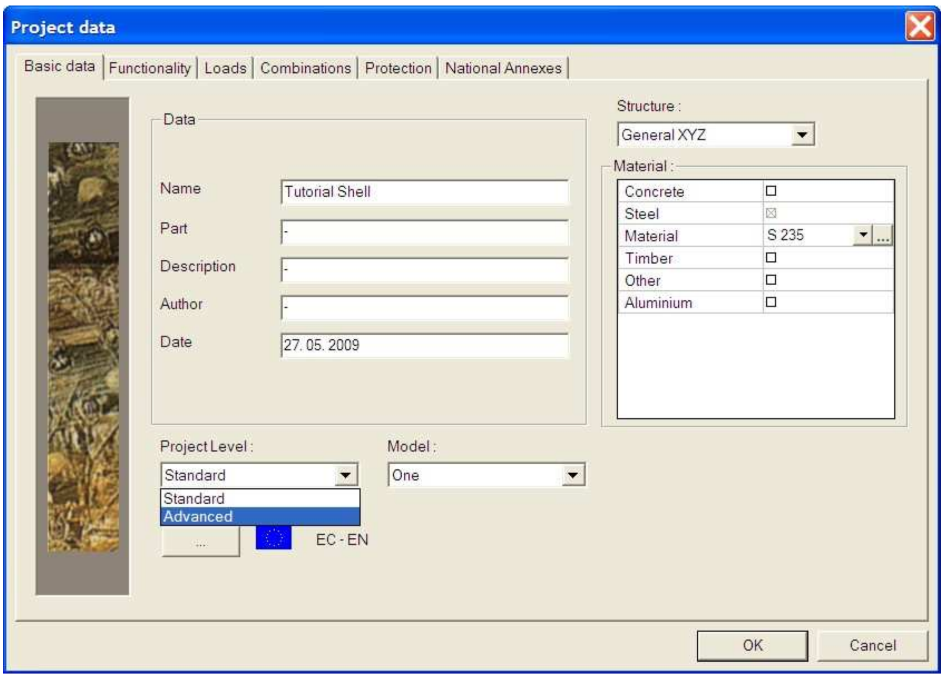
14. Confirm the selection with <**ESC**>.

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

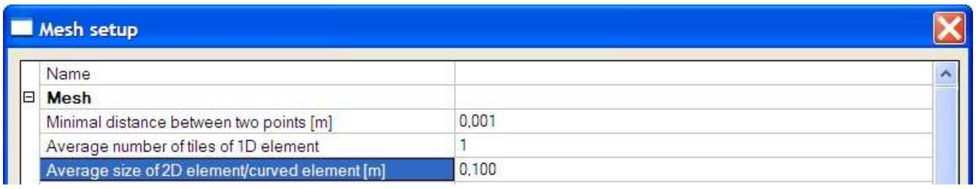
1. To generate the mesh, the advanced mode is first set in the project data. This can be done through the menu bar **Tree, project**:



2. Press the icon  to start the mesh generation.

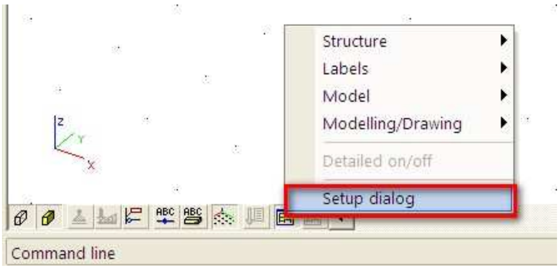
3. To be able to better visualize the result, a mesh refinement is entered. Click on the icon in the menu tree under **Calculation, mesh**.

4. Set the average size of the 2D elements on 0,1:

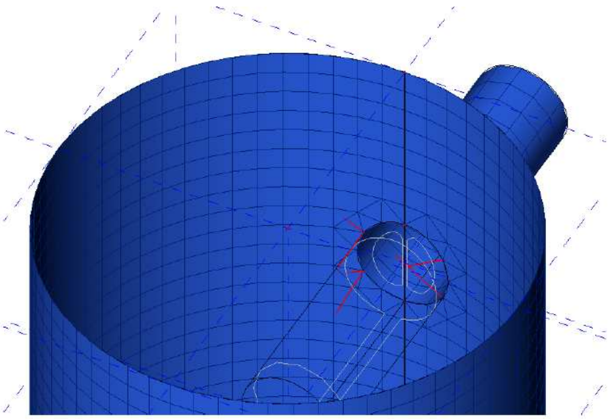


5. Let the mesh regenerate by means of the icon 

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:



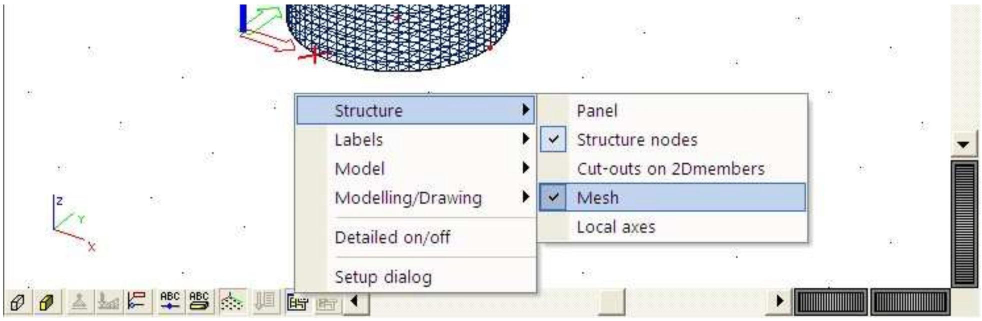
7. The program clearly show the edited geometry:



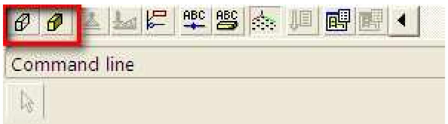
If you want to review the edited structure without generating the mesh, you need to follow the following procedure.

Visualizing the rendered structure

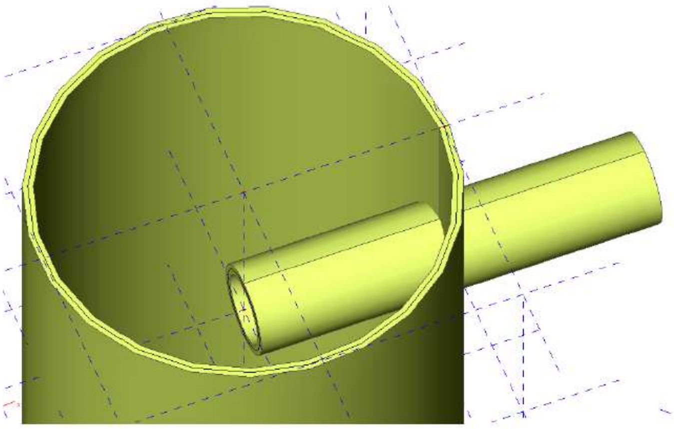
1. First of all the mesh has to be switched off. This can be done through the view parameters above the command line:



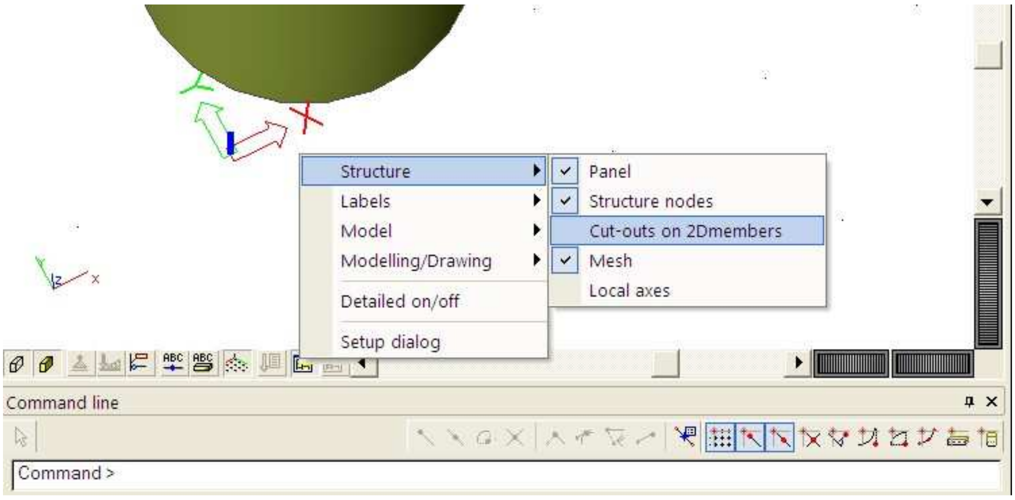
2. Afterwards the structure can be rendered with the render buttons above the command line:



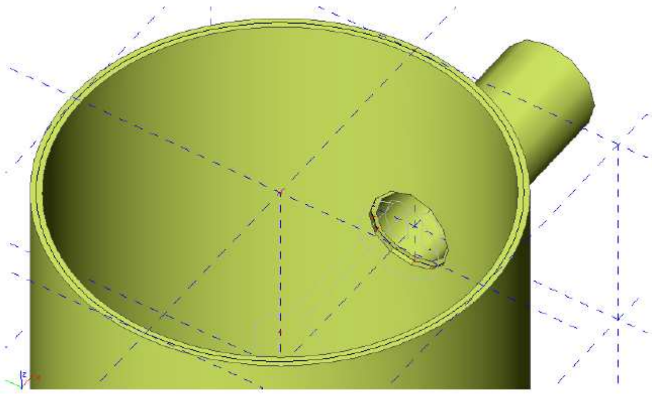
3. At first sight it seems that this structure doesn't match the previous one:



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



5. The structure looks as expected:

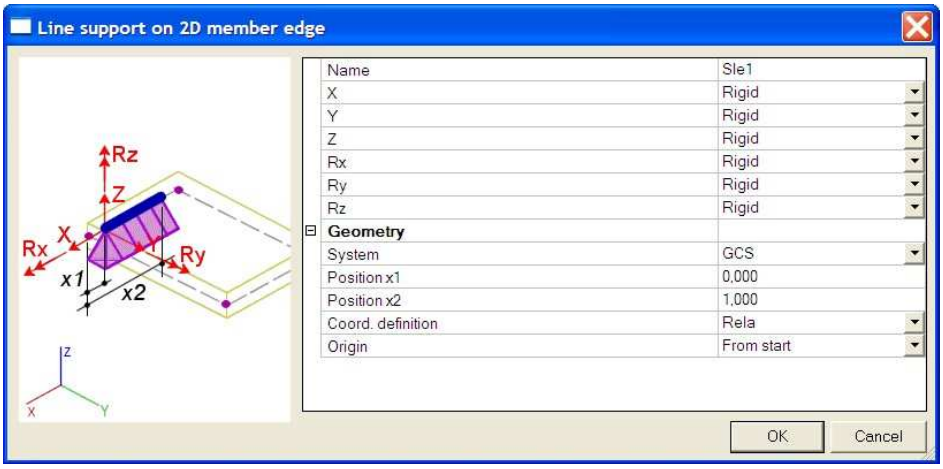


Supports

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**.



2. Since this regards a restraint, both translations and rotations are captured.

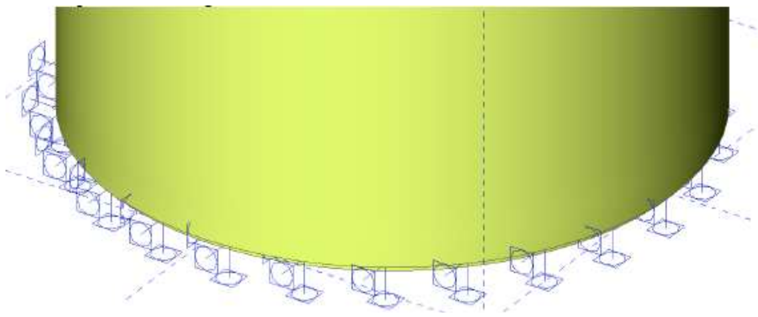
3. Confirm the entry with [**OK**].

4. The floor slab can be selected by using the mouse.

5. Press <**ESC**> to end the entry.

6. Press <**ESC**> once more to end the selection.

7. The supports are visualized:



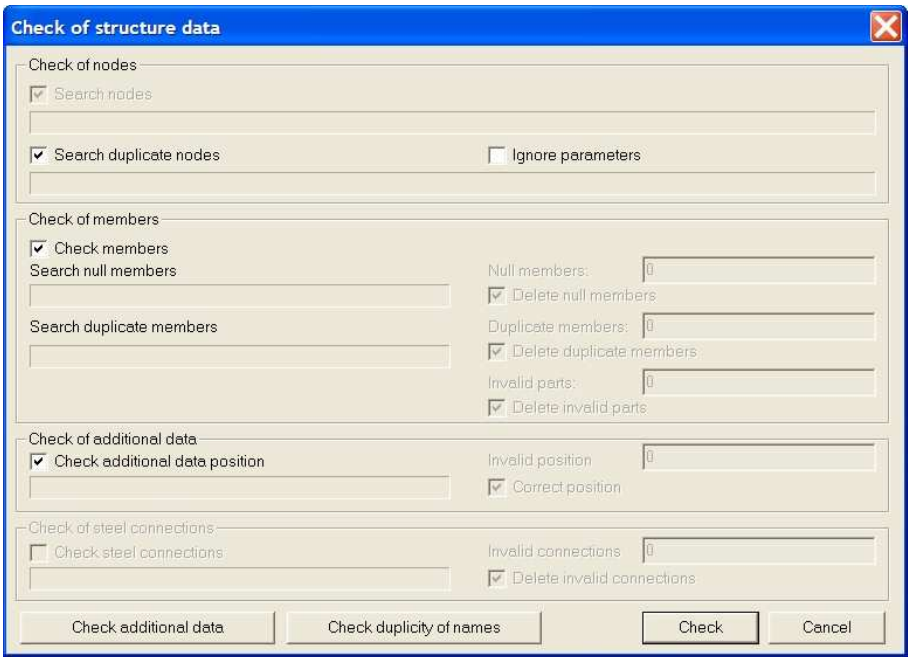
Check structure

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.



3. Click [**Check**] to perform the checks.

4. The window Report data check appears with a notification saying that no problems were found.

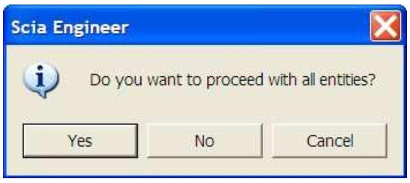
5. Close the check by clicking on [**OK**].

Connecting members/nodes

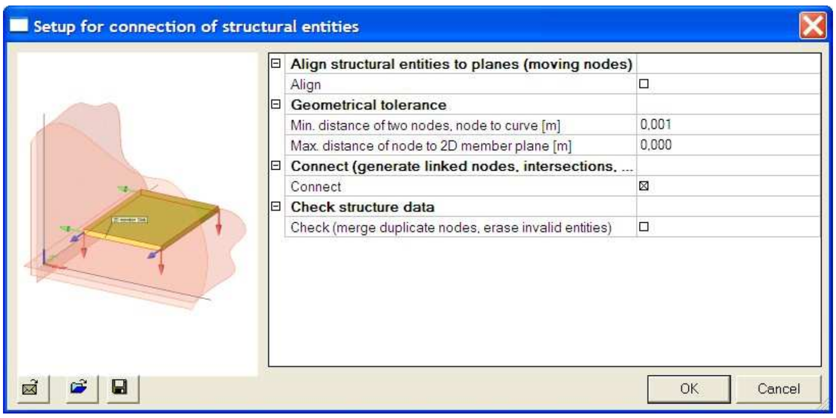
Analogously to the method in a project with 1D elements, the option ‘Connect members/nodes’ (in het construction menu) can be used to connect elements in a project with 2D elements (such as this project).

Connect members/nodes

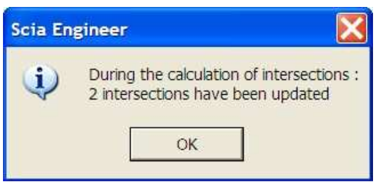
1. If necessary, first deselect the selected elements with the <**ESC**> button.
2. Double-click on the option **Connect members/nodes** in the **Structure menu** or press the button  in the toolbar.
3. A dialog box appears asking if you want to continue with connecting all entities:



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.

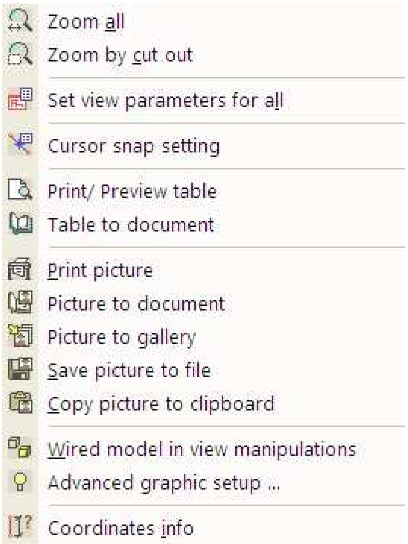
3. Click on the button **View in direction- Z**  for a view in the Z-direction.

The magnifier

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

Editing view parameters through the menu View parameters

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



Remark:

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

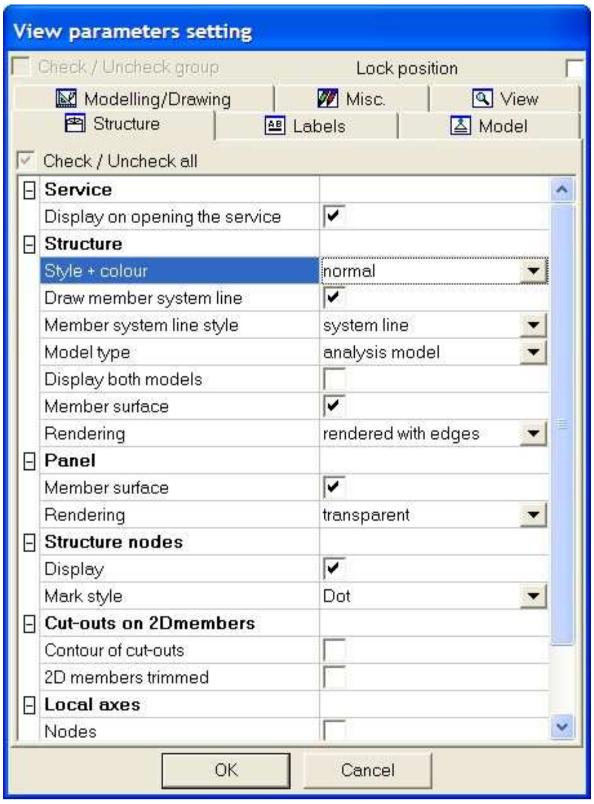
2. Choose the option **Set view parameters for all**. The window **View parameter setting** appears. The menu consists of various tabs. You can set the view parameters for all entities or just for the selected entities.

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:

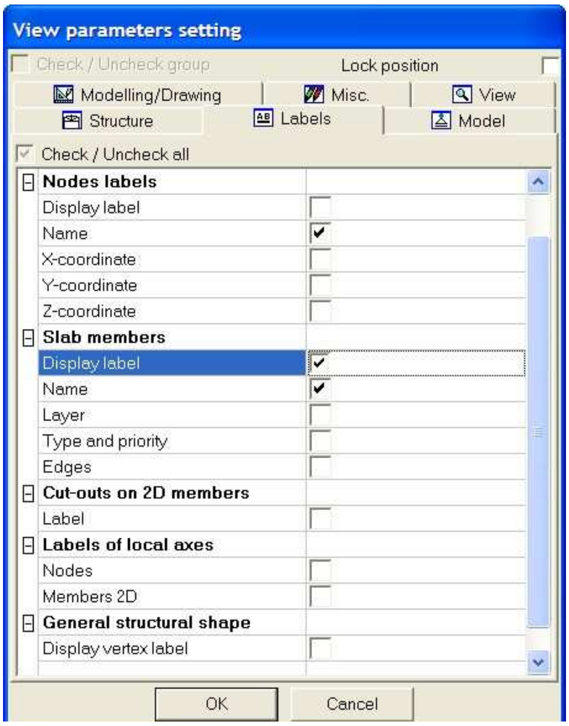
- **Style and colour:** You can display the colours per layer, material, cross-section or structural type.
- **Cross-section:** With this the symbol of the cross-section is displayed on every member.
- **Local axes:** With this the local axes of the elements are activated.



View parameters – Labels and description

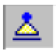
Through the tab **Labels,** the labels of different entities can be displayed. In the group **Members** the following items can be displayed in the label:

- **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.

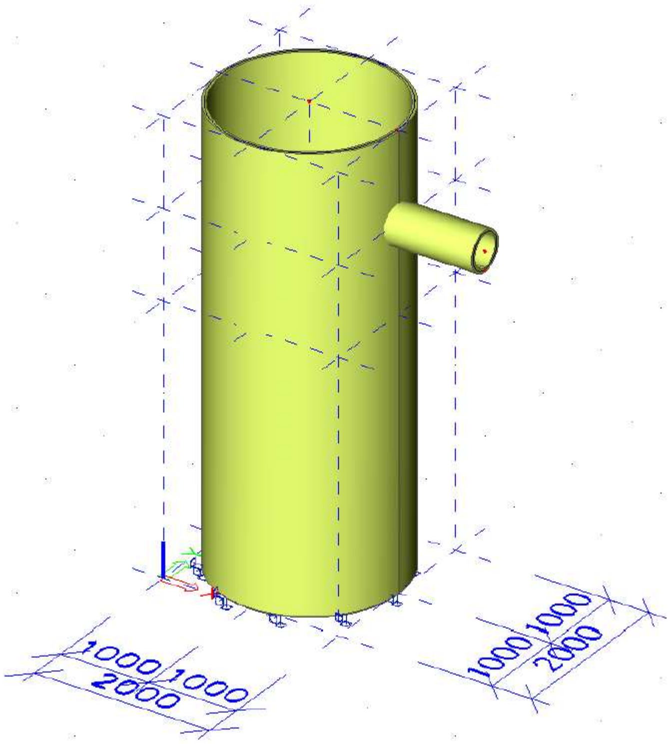


View parameters – shortcuts

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

- **Show/hide surfaces**  to show the surfaces of the cross-sections.
- **Render geometry**  to view the rendered members.
- **Show/hide supports**  to show supports and hinges.
- **Show/hide load**  to show the load case.
- **Show/hide node labels**  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**  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:

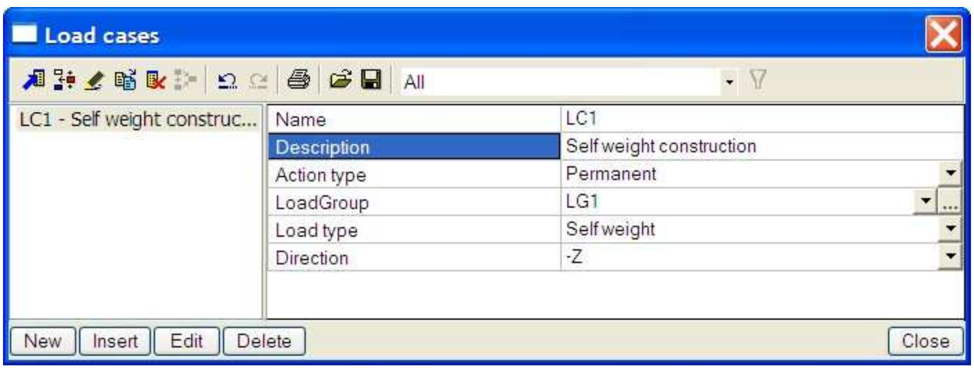
- **LC1**: Permanent load case: self weight of the construction

- **LC2**: Permanent load case: water in vertical cylinder

Defining a permanent load case

1. Double-click on 

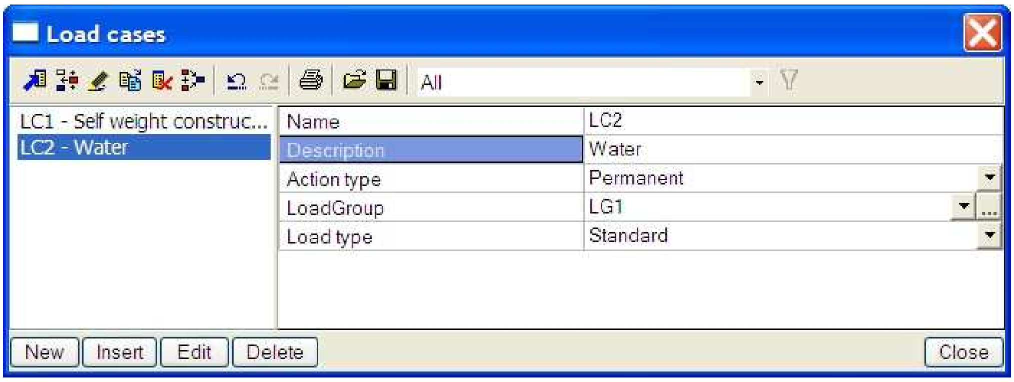
in the **Main window** or click on  in the toolbar.
2. Before loads can be defined, the load cases need to be imported first. Because no load cases were imported in the project, the **Load cases manager** automatically appears.
3. By default, the load type LC1 is made. This load is a permanent load with 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**".



Defining a second load case

1. Click on  or on  to make a second load case.

2. "**Water**" is entered as a description:



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.

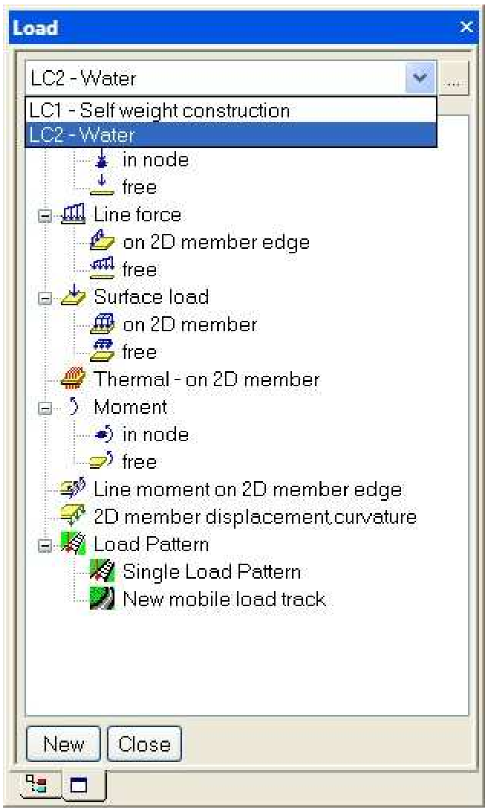
The first load case consists of one load:

- 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**:



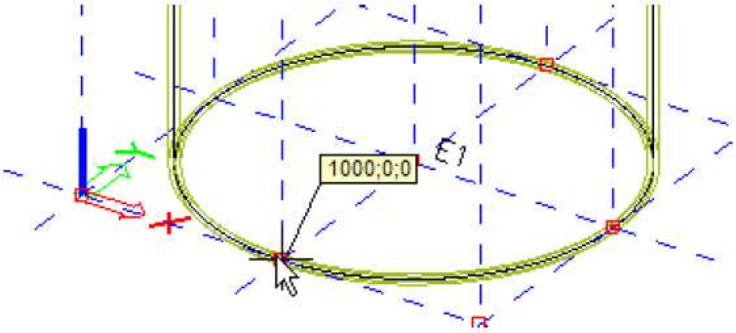
Free load

1. Close the selection by pressing <**ESC**>.

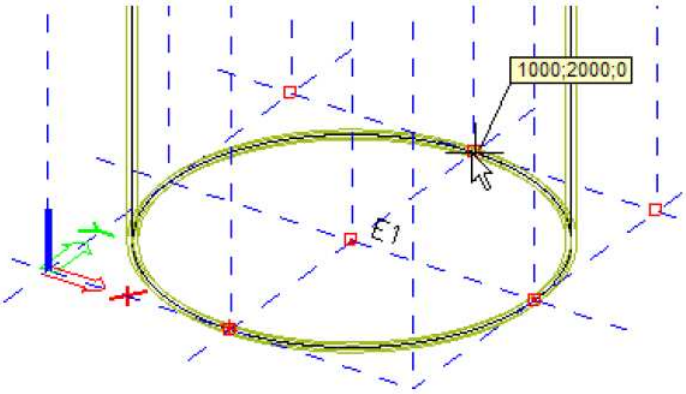
2. Define the new UCS by three points by means of the icon  or through the menu bar **Tools → UC → UCS by three points.**

3. The new UCS is entered as follows:

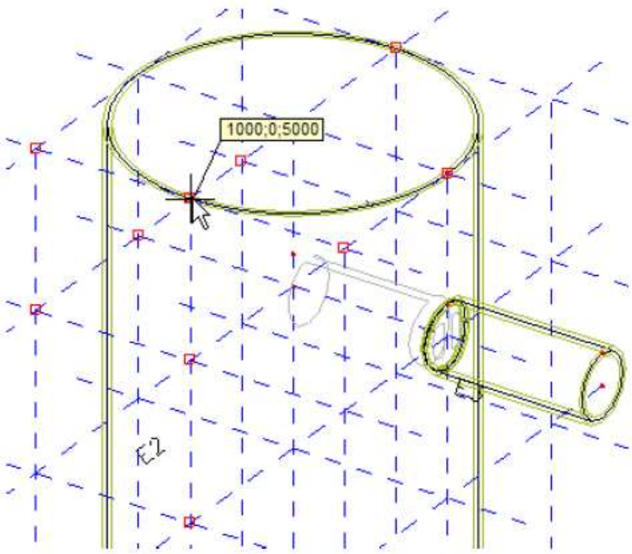
4. Defining the origin:



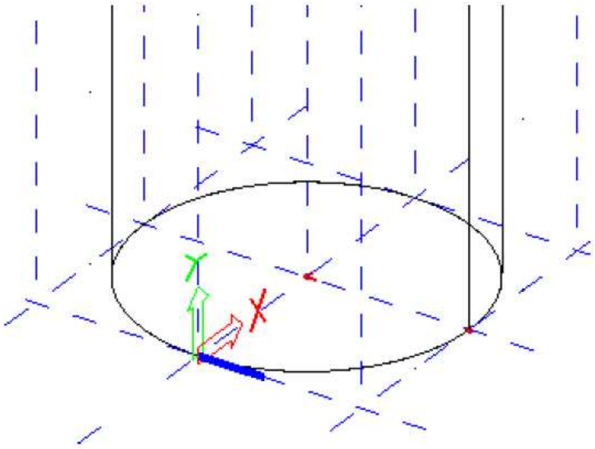
5. Defining the direction of the X-axis:



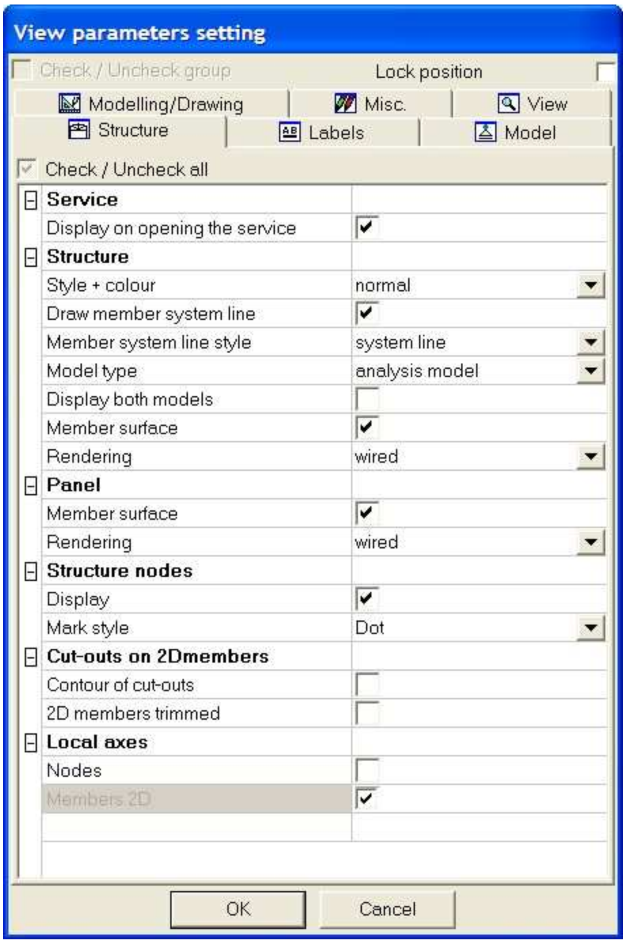
6. Defining the direction of the Y-axis:



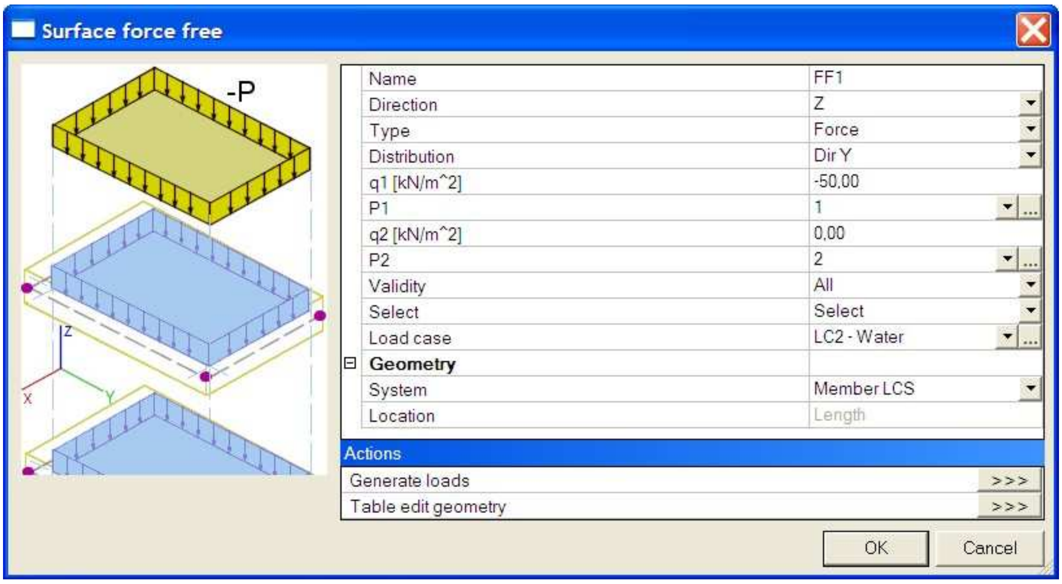
7. The UCS is displayed as follows:



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):



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



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 kN/m² and **q2** = 0 kN/m² 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**].

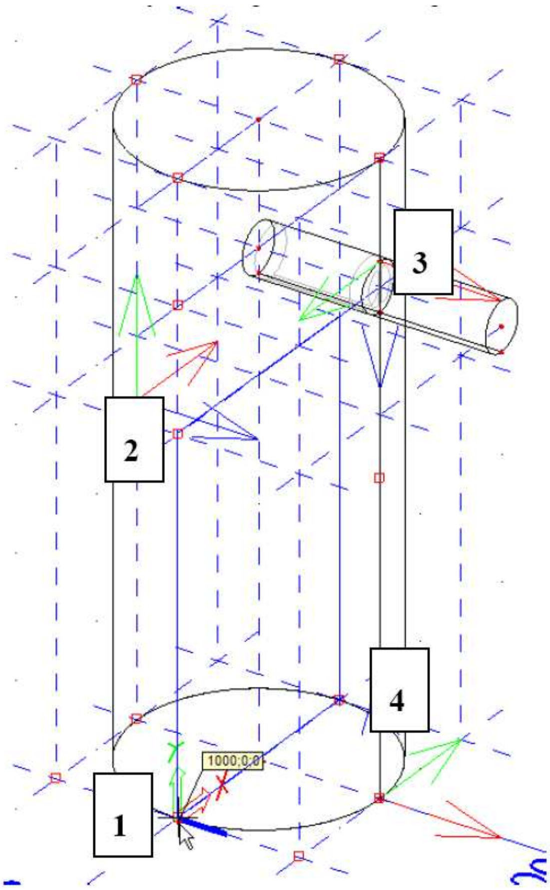
17. Now draw the free load in the geometry in the order as shown on the figure:

punt1 (1;0;0)

punt2 (1;0;4)

punt3 (1;2;4)

punt4 (1;2;4)



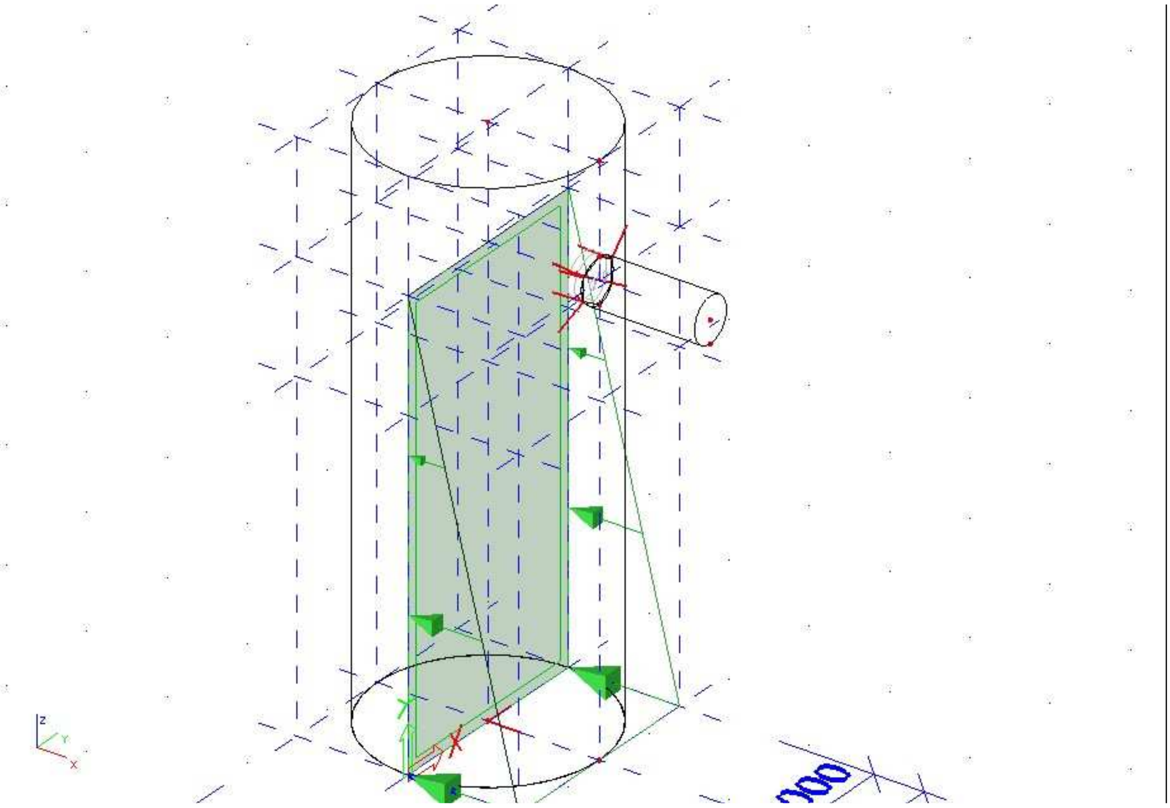
18. The order of drawing the free load is important for the allocation of q1 and q2. Since -50 kN/m² was already attributed to q1, this value needs to be linked to the first insert point of the free load. To the second insert point, 0 kN/m² will be linked.

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

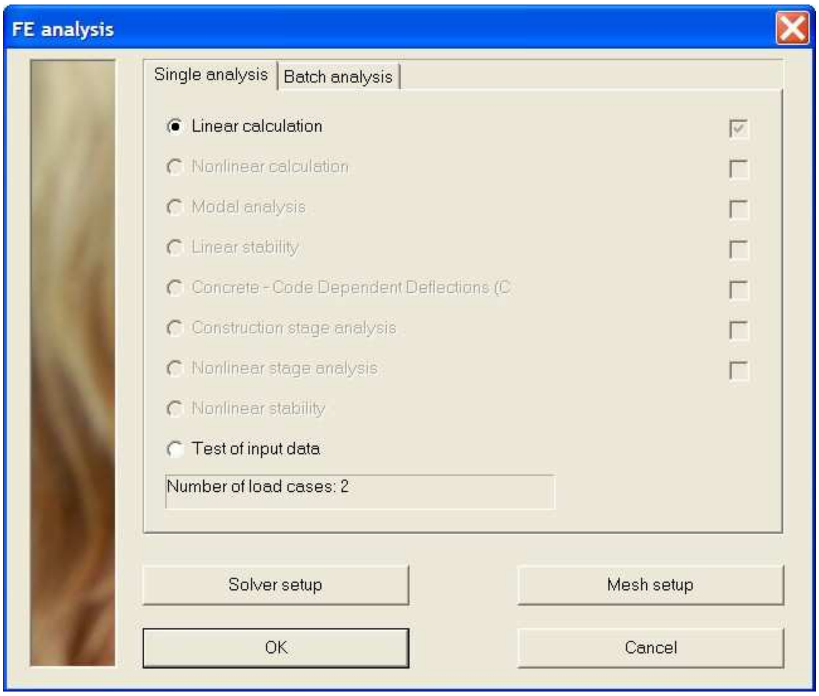
Linear Calculation

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

Performing linear calculation

1. Double-click on Calculation under Calculation, mesh in the **Main window**.

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



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.

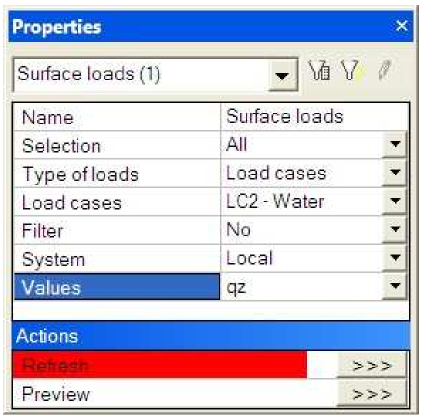
2D data viewer

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**.

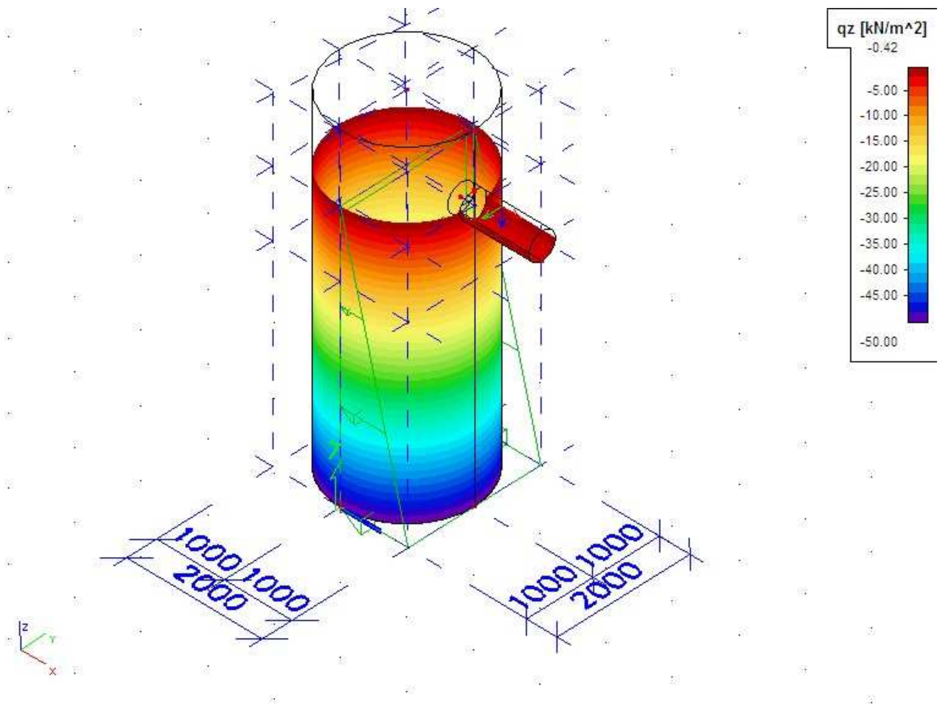
3. The property window is set as follows:

   - 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**



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

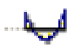
next to **Refresh** to show the results in the graphic window according to the recently set options. The picture below shows the entered loads:

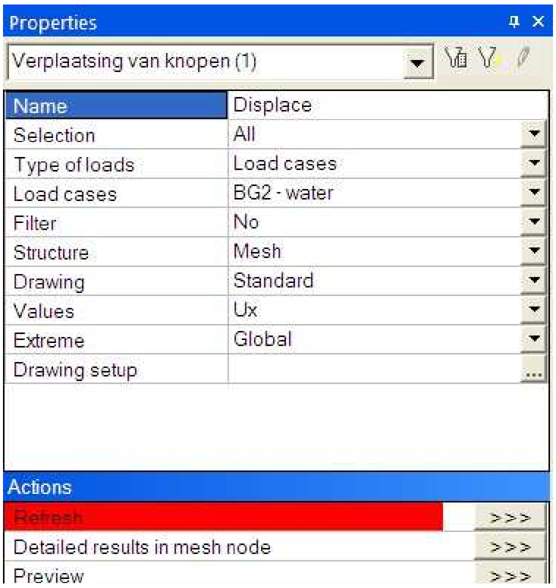


5. On the picture above the user can check how the free load operates on the construction. The figure clearly shows the trapezoidal course of the entered water load in the construction. At the top, the tank is loaded with 0 kN/m², at the bottom this goes to 50 kN/m². The floor slab is also loaded with 50 kN/m². 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.

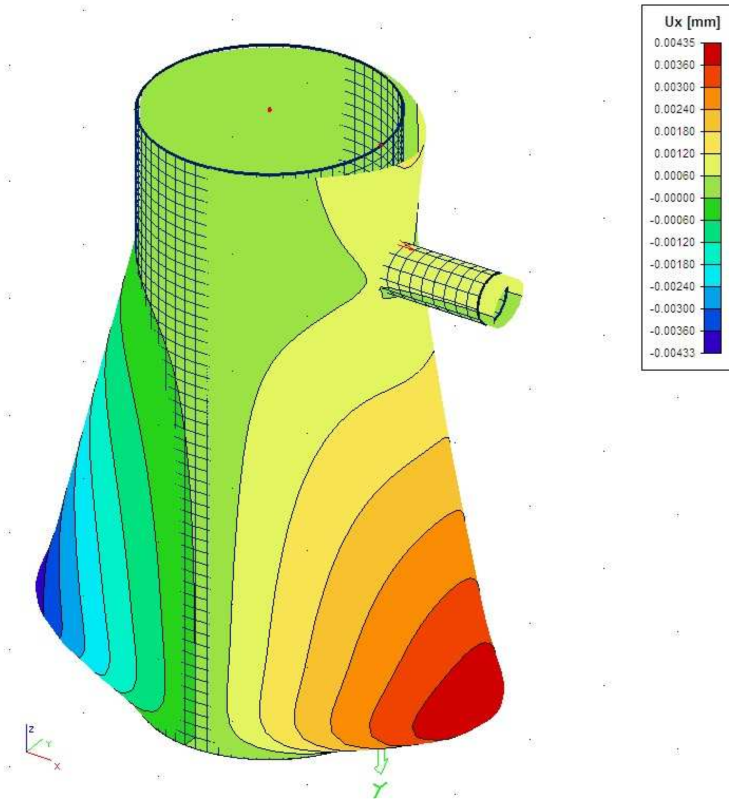
After this check, the other results are reviewed.

View deformations of nodes on 2D element

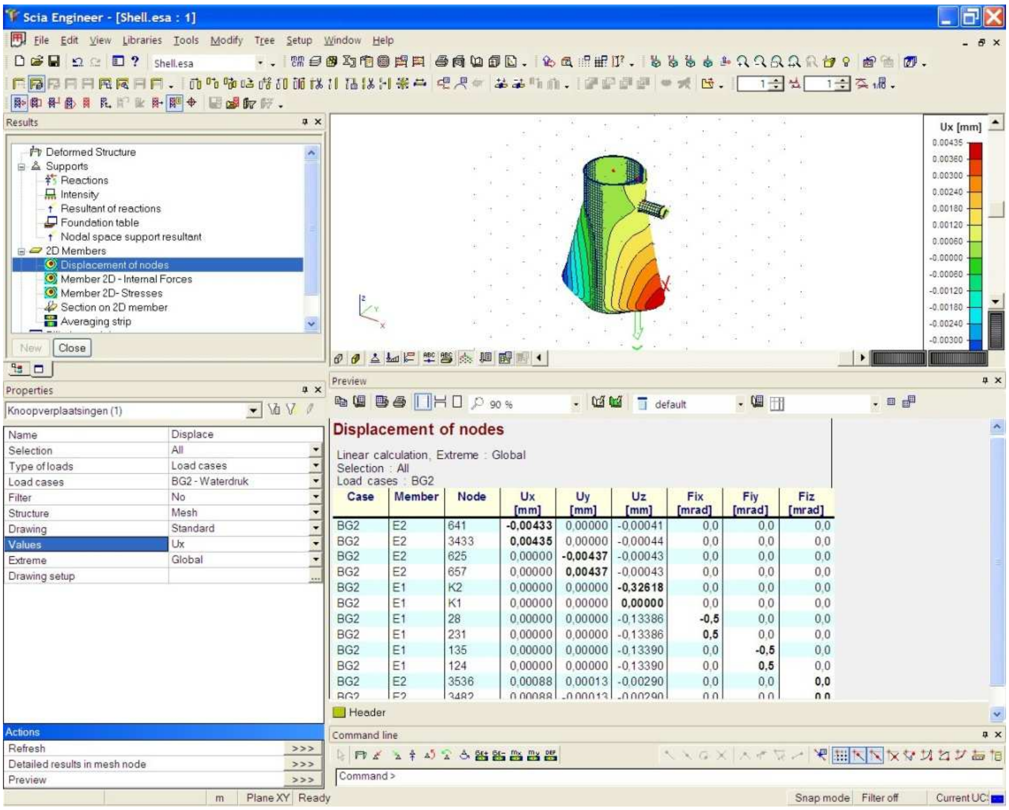
1. Double-click on  in the **Main menu**. The **Result menu** appears.
2. Click under **2D members** on **Displacements of nodes.**
3. The options in the **Property window** are configured as follows:
    - The field **Selection** is put on **All**.
    - The **Load type** is put on **Load case** and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



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:

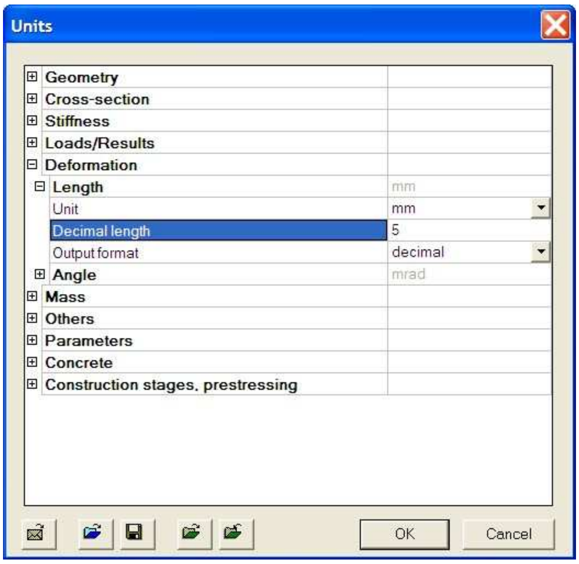


5. To display the results in tabular form, the action **Print Preview** is used. Press the button  next to Print Preview to show the print preview.



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.

7. Click on the icon  and set number of decimals on 5 in **Deformations/length**:



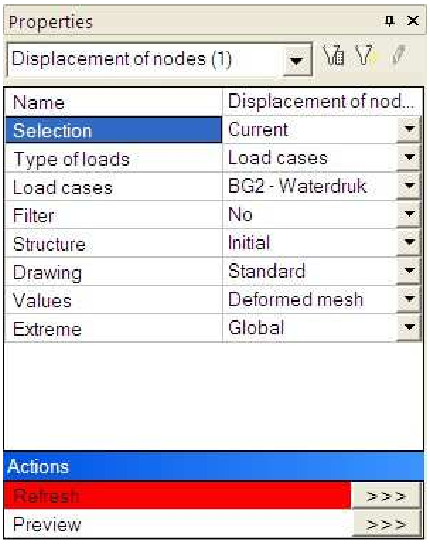
8. Press the button Refresh again to view the modifications on the screen.

Remark:

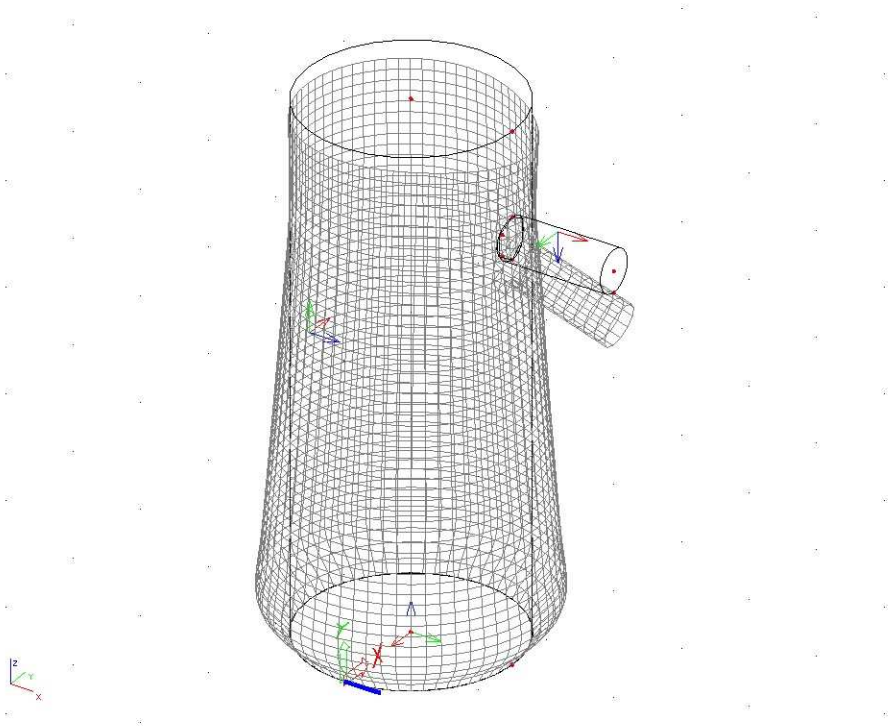
The Print Preview appears between the Graphic Window and the Command line. This screen can be enlarged to display more data at once.

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**



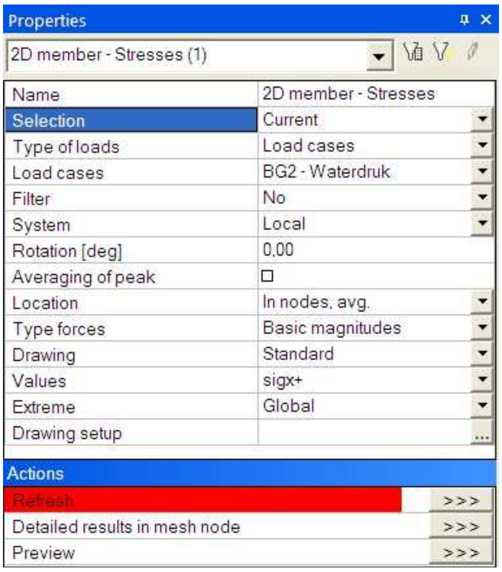
The deformation of the construction can be reviewed in 3D:



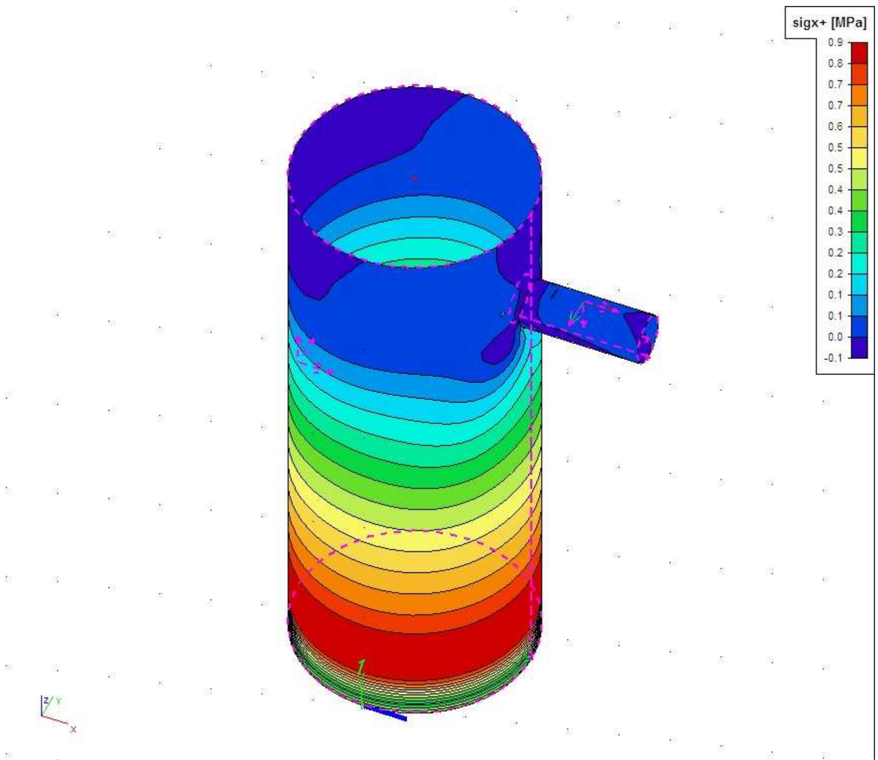
Display stresses

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**+.



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 

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.
   - Open the group **Results** and click on **Member2D - Stresses**. Click [<<< Add**]** to add this item to the document.

5. Click [**Close**] to close the window **New document item** and to return to the document.

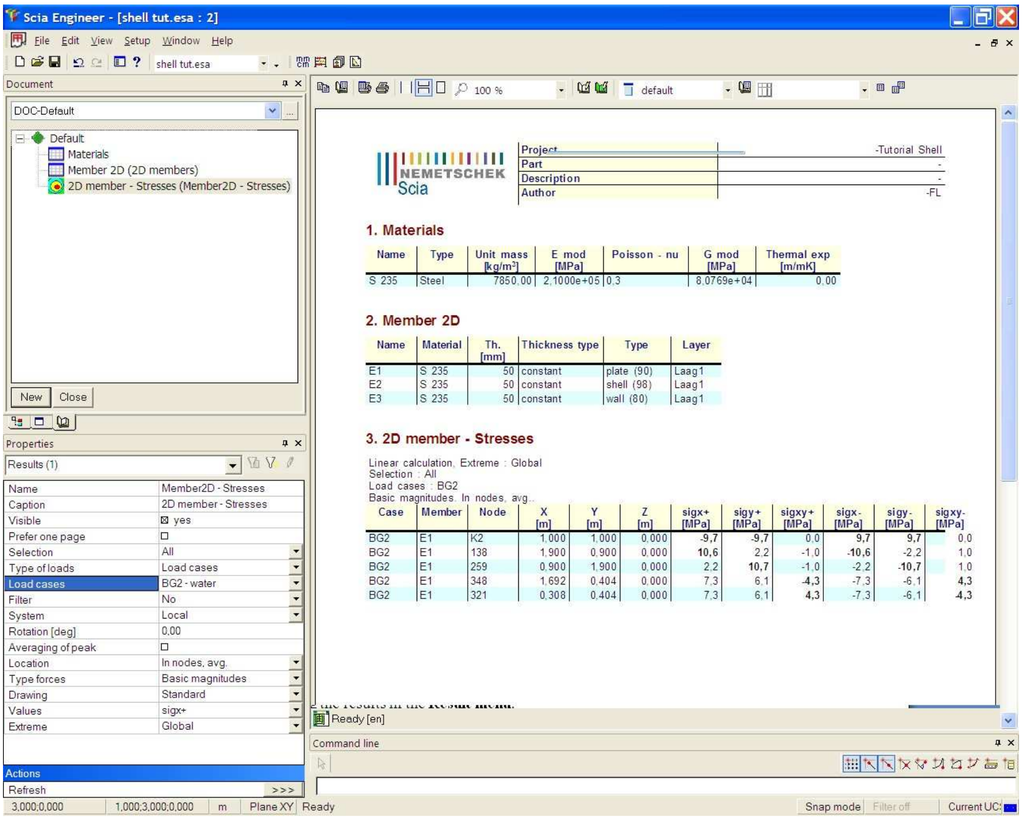
The items that were added to the document, are displayed in the **Document menu**. The order of the items can be edited by dragging the mouse. On the right side of the screen the Print Preview of the document is shown.

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.



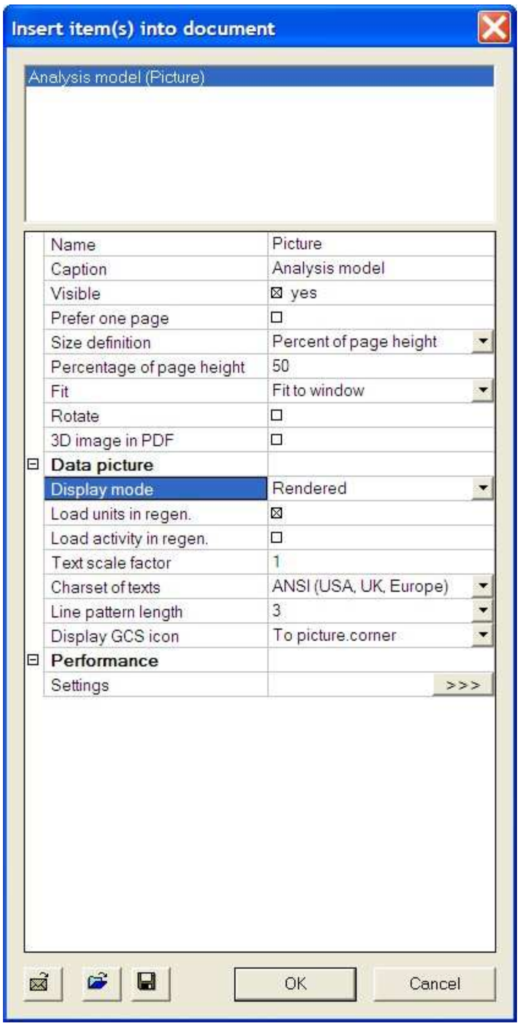
3. Press the button [**Close**] at the bottom of the **Document menu** to close the document and to return to the structure.

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.



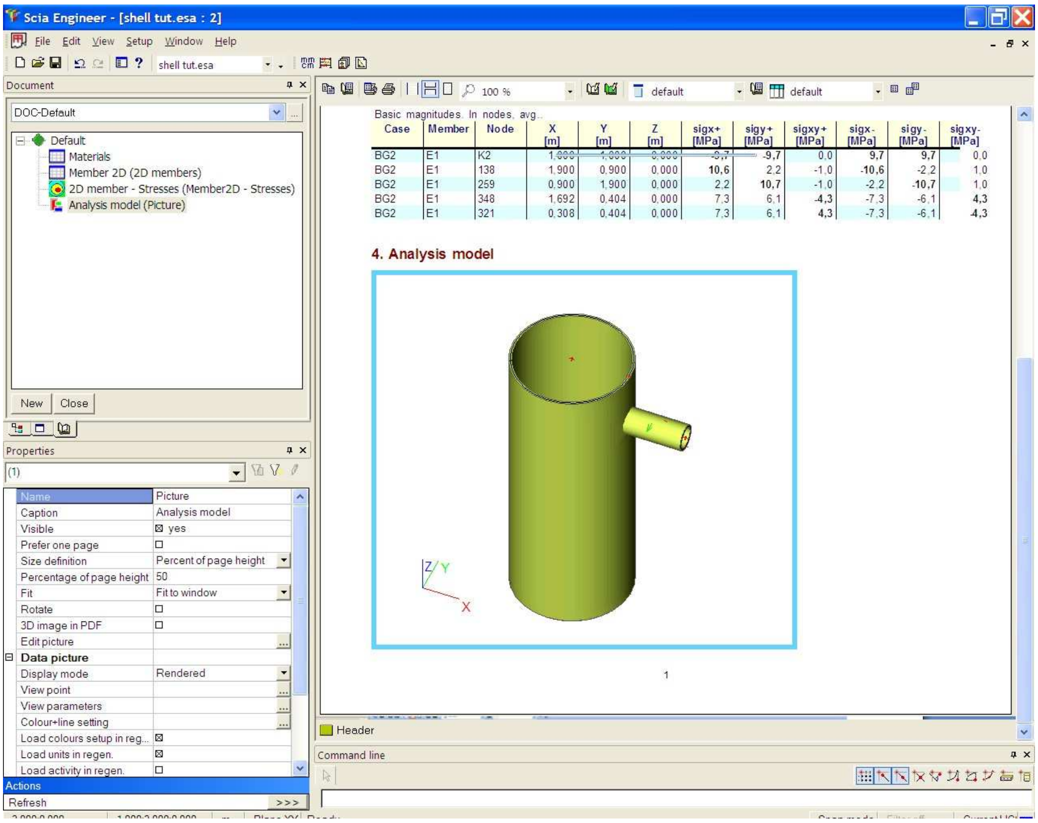
3. The field **Size** is edited to **50** so the figure covers 50% of the page or half a page.

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.