SCIA ENGINEER



TUTORIAL

CONCRETE SLAB (ACI 318-08)

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2018 SCIA nv. All rights reserved.

Table of contents

General Information **1**
Welcome **1**
SCIA Engineer Support **1**
Website **1**
Introduction **3**
Getting started **4**
Starting a project **4**
Project management **6**
Save, Save as, Close and Open **6**
Saving a project 6
Closing a project 6
Opening a project 6
Geometry input **7**
Input of the geometry **7**
Supports 14
Check Structure data **17**
Checking the structure 17
Use of labels 18
Graphic representation of the structure **20**
Input of the Calculation Data **24**
Load Cases and Load Groups **24**
Defining a Permanent Load Case 24
Defining a Variable Load Case 25
Loads **26**
Combinations **30**
Calculation and Mesh generation **33**
Mesh generation **33**
Linear Calculation **34**
Results **36**
Viewing results **36**
Post processing **42**
Automatic mesh refinement **42**
Reinforcement design **46**
Pre-processing **46**
Required areas **48**
Input user reinforcement **50**
Engineering Report **54**
Saving Model Views or Results to Engineering Report **54**
Adding New Items to Engineering Report **55**

General Information

Welcome

Welcome to the SCIA Engineer 15 ACI Plate Tutorial. SCIA Engineer is a design program in Windows with a broad application field: from checking/designing simple frames to the advanced design of complex projects in steel, concrete, cold formed steel and a variety of other materials.
The software allows engineers to model 2D and 3D structures which include flat or curved plates and beam members (straight or curved) as well as other advanced 3D geometry. The complete calculation and design process has been integrated into one program so that the input of geometry, input of calculation information (loads, combinations, and supports), linear and non-linear calculation, output of results, reinforcement design according to various codes and the generation of the calculation documentation are all completed in the same software.
SCIA Engineer is available in three different editions all which require a license to operate:

- Concept
- Professional
- Expert

License version
A licensed version of SCIA Engineer is secured with either a 'dongle', which you apply to the USB port of your computer or a software license on your company's network.
SCIA Engineer is also modular and consists of various modules. The user chooses from the available modules and composes a custom design program, perfectly tuned to the desired needs of the company.
In the general product overview of SCIA Engineer you will find an overview of the different modules that are available.

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

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

SCIA Engineer Support

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

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

By telephone
From USA: 443-542-0637

Via the SCIA Support website
http://www.SCIA-online.com/en/online-support.html

Website

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

Link to eLearning
http://www.SCIA-online.com > Support & Downloads > eLearning

Link to Demo version

http://www.SCIA-online.com > Support & Downloads > Secured Downloads > input username and password > Service Packs > SCIA Engineer > Setup – SCIA Engineer
http://www.SCIA-online.com > Support & Downloads > Free Downloads > input e-mail address > SCIA Engineer > SCIA Engineer Manuals & Tutorials

.

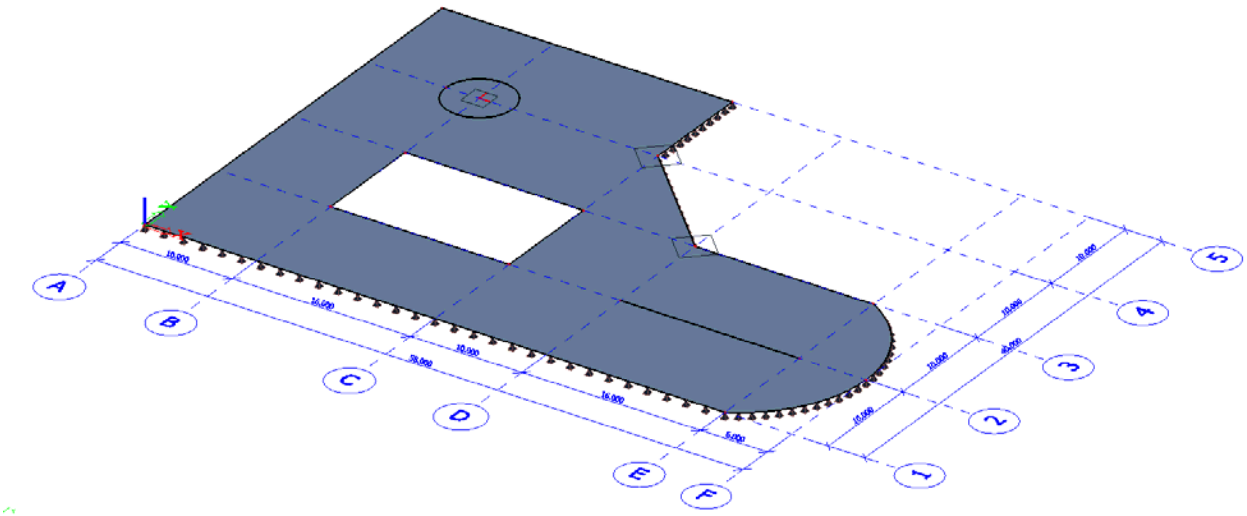
Introduction

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

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

First, we will explain how to create a new project and the set-up of the structure. After the geometry and load input, the structure will be calculated and the results can be viewed.
Next, we will discuss the input of the buckling parameters and we will perform the steel check, the profile optimization and calculate a connection.
The Tutorial ends with a brief introduction to the calculation note.

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



Getting started

Starting a project

Starting the program

Before you can start a project, you need to start the program first. To do this, Double-click on the SCIA Engineer shortcut in the Windows Desktop or if the shortcut is not installed, click **[Start]** and choose **Programs > SCIA Engineer 18.0 > SCIA Engineer 18.0**.

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

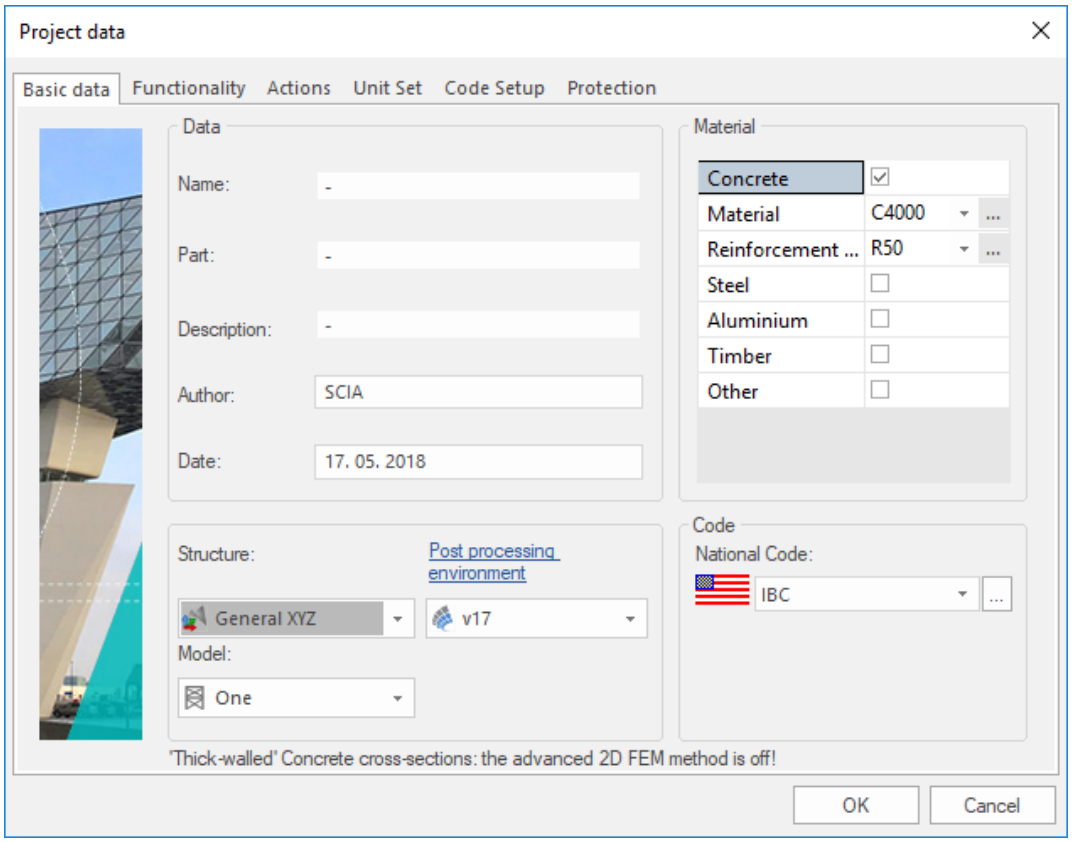
For this Tutorial, you must start a new project.

Starting a new project

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

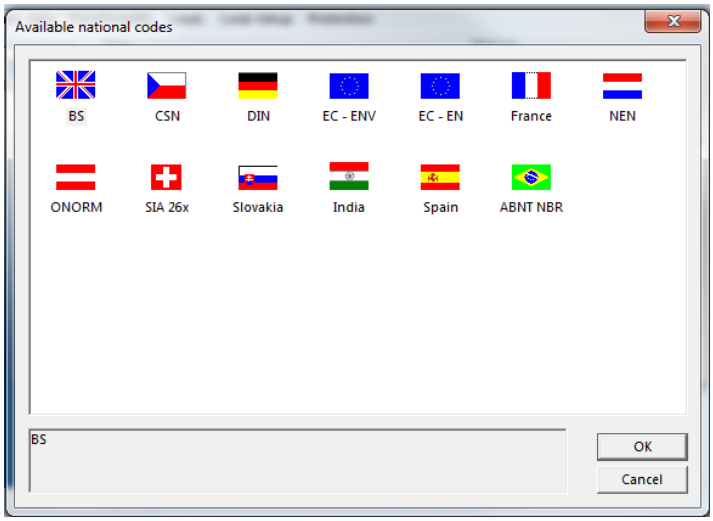
In the **Project dialogue** dialogue, choose for **Blank Project**.

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



3. In the **Data** group, enter your preferred name, part, description, author and data. These data can be mentioned on the output, e.g. in the document and on the drawings.

4. Select **General XYZ** in the **Structure** field. The structure type (Frame XZ, Frame XYZ, Plate XY, General XYZ...) will restrict the input possibilities during the calculation.
5. Choose the **Project level: Advanced** and **Model: One**.
6. In the **Material** group, select **Concrete**.
   Below the item **Concrete**, a new item **Material** will appear.
7. Choose **C4000** from the menu.
8. Click on the rectangular button  below **National Code** to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the project of this tutorial, choose **IBC** which appears as default. The window **Codes in project** shows different available codes. To access a separate code click **[Add]**.
9. The dialogue **Available national codes** is opened and other codes can be chosen and confirmed by clicking **[OK]**.



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

Notes:
*On the **Basic data** tab, you can set a project level. If you choose "standard", the program will only show the most frequently used basic functions. If you choose "advanced", all basic functions will be shown.*
*On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus simplifying the program.*
*On the **Combinations** tab, you will find the values for the partial safety factors. In this Tutorial, we will use the default settings.*

Project management

Save, Save as, Close and Open

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

Saving a project

Click on  in the toolbar.

If a project has not yet been saved, the dialog box **Save as** appears. Click on the arrow in the list **Save** to choose the drive you want to save your project in. Select the file in which you want to put the project and click on **[Open**]. Select the subfolders. Enter the file name in **File name** and click on **[Save]** to save the project.

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

Closing a project

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

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

Opening a project

Click on  to open an existing project.

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

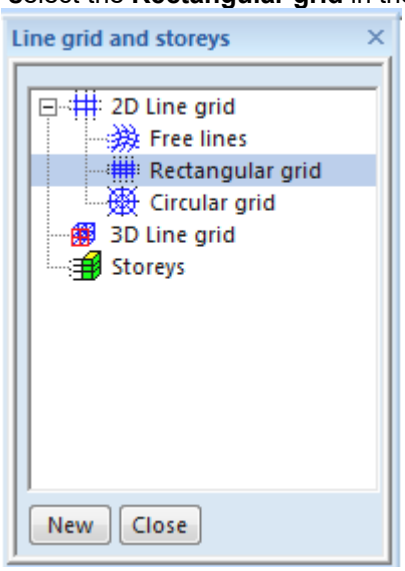
Geometry input

Input of the geometry

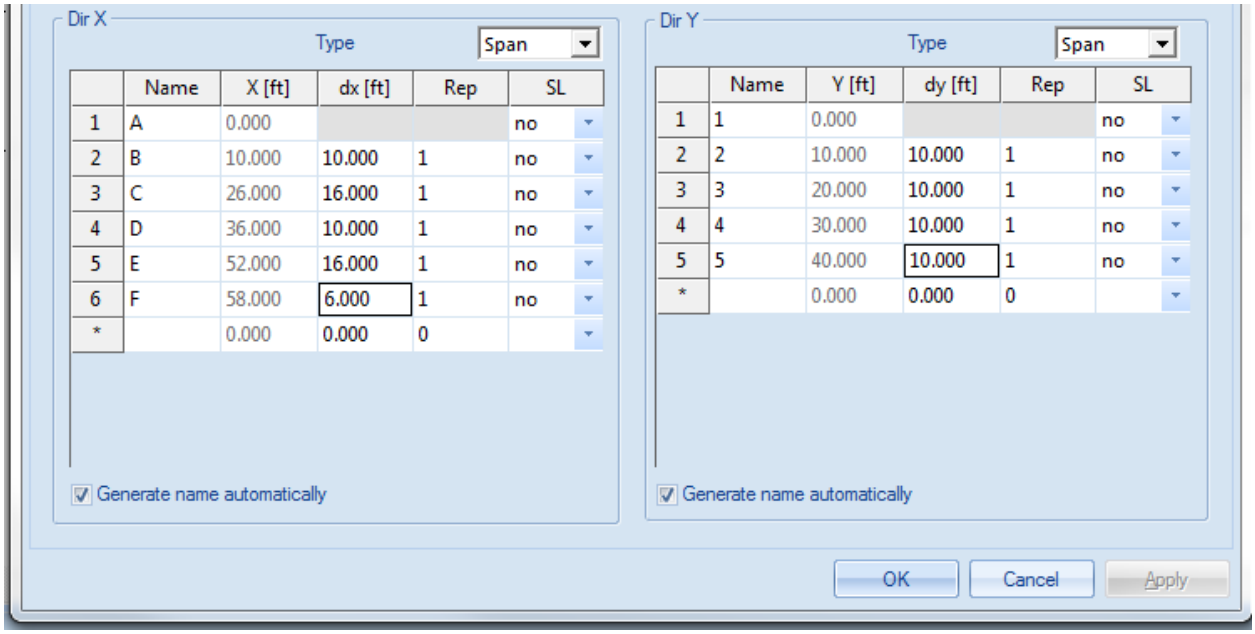
If you start a new project, the geometry of the structure must be entered. The structure can be entered directly, but you can also use for instance templates with parametric blocks, DXF files, DWG files and other formats.

Line grids and storeys

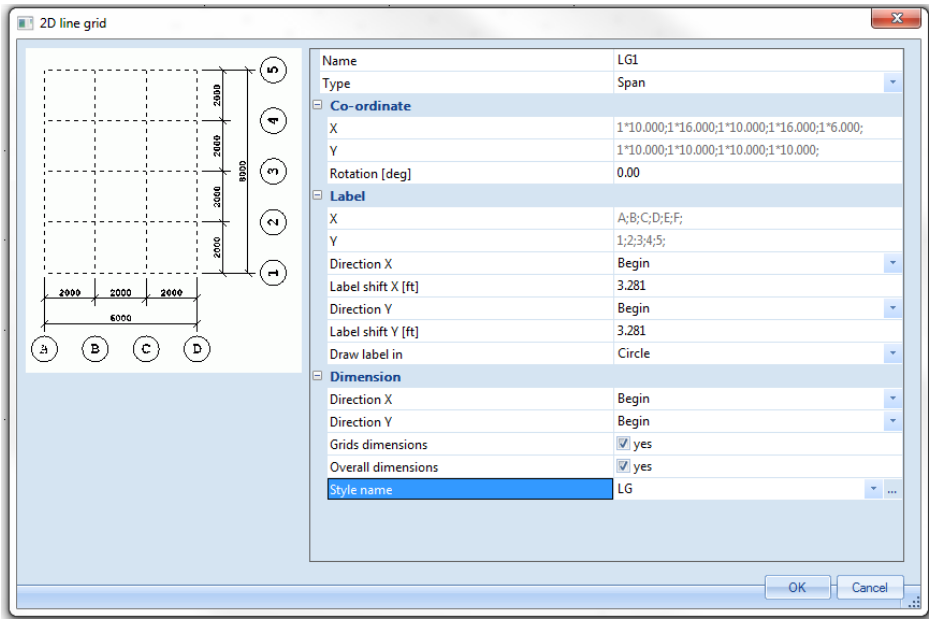
1. The 2D line grids can be inserted through the first option in the Main window**, Line grid and Storeys.** Once in the menu, **s**elect the **Rectangular grid** in the **2D Line grid** menu**.**



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



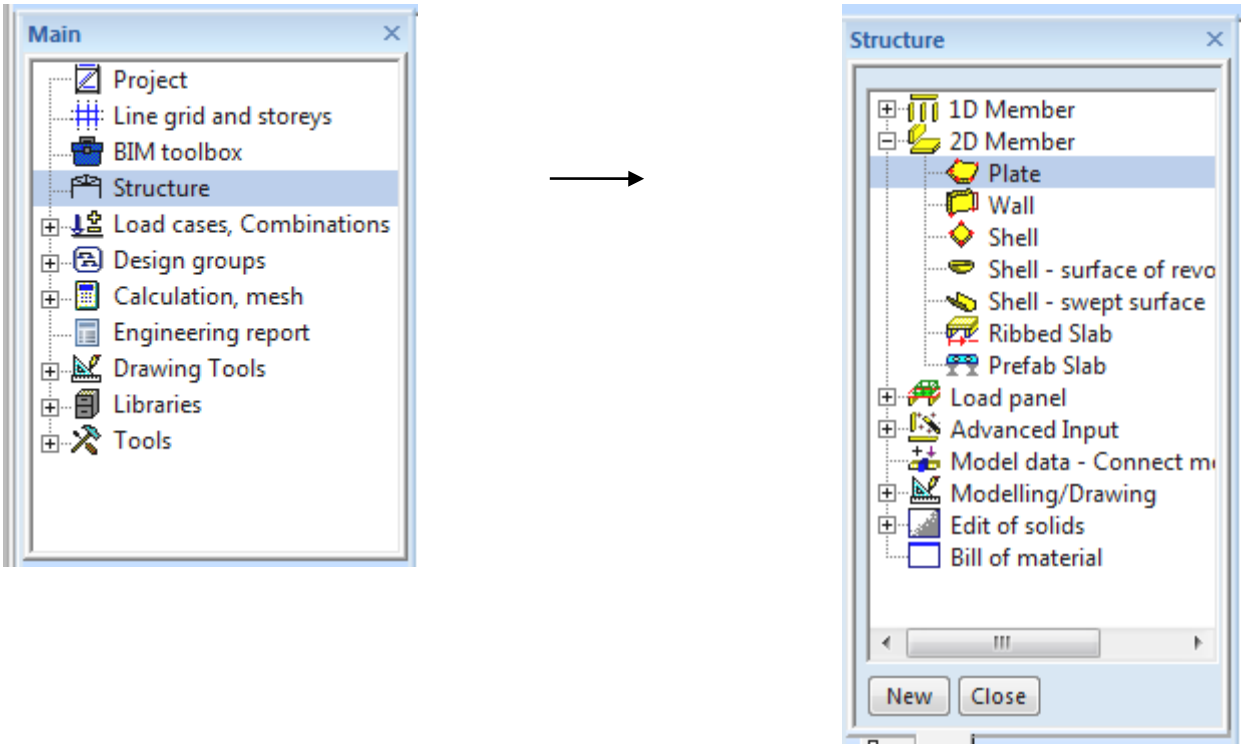
3. Accept the default values in the **2D line grid** window.



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

Structure menu

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



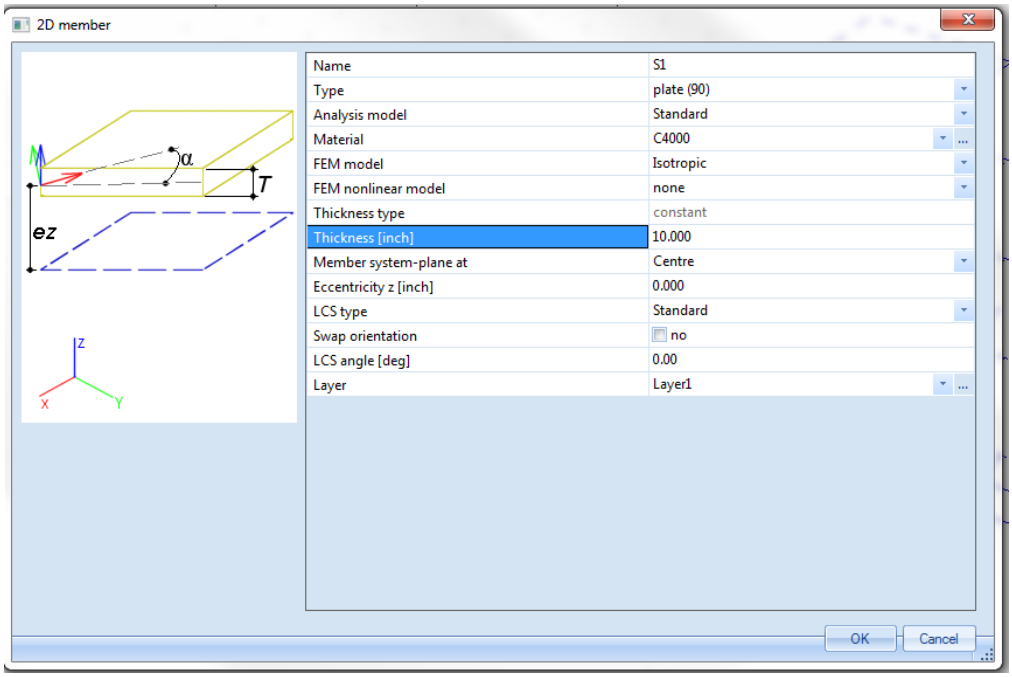
2. In the **Structure menu** different branches will appear, in accordance with the already input items, i.e. support branch will appear if a structure is physically available.

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

Input of a plane 2D member

1. In the structure menu double click on the **Plate** in the **2D member** branch

2. The window **2D member** will be opened.



3. Define the following plate properties: **name=S1**, **type=plate (90)**, **Material=C4000**, **Thickness = 10in.**

4. After accepting with **[OK]** the program asks in the command line for the starting point of the polyline.

5. The buttons in the **command line** allow one to build up the polygonal edges using different line types, or to directly choose a circular or rectangular surface.



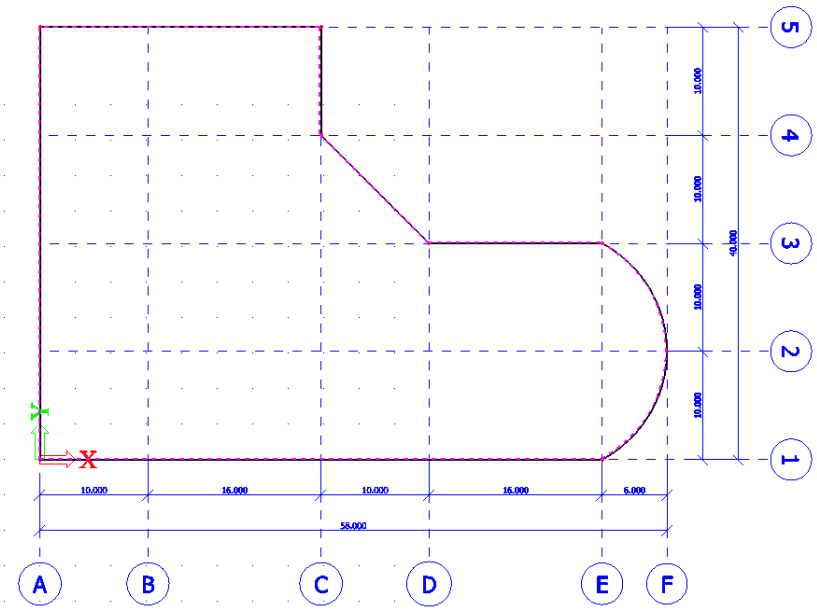
6. The geometry can be input with help of a dot or line grid or with use of the mouse or direct input of coordinates in the command line which the former is used here:

   Starting point: A1 <click>
   E1 <click>
   Choose the New circular Arc symbol on the **Command line**
   New polygon – Circular Arc – Intermediate point:
   F2 <click>

E3 <click>
D3 <click>
C4 <click>
C5 <click>
A5 <click>
A1 <click>

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

   The following picture is now depicted in the screen:



Definition of an opening

1. In the **Structure** menu under **2D member components** create an **Opening**.

•

2. In the command line, quick access buttons are available for quick definition of geometrical outlines like circular or rectangular shapes. By default, definition by drawing of a closed polygon is started, in this example we will define a rectangular opening.

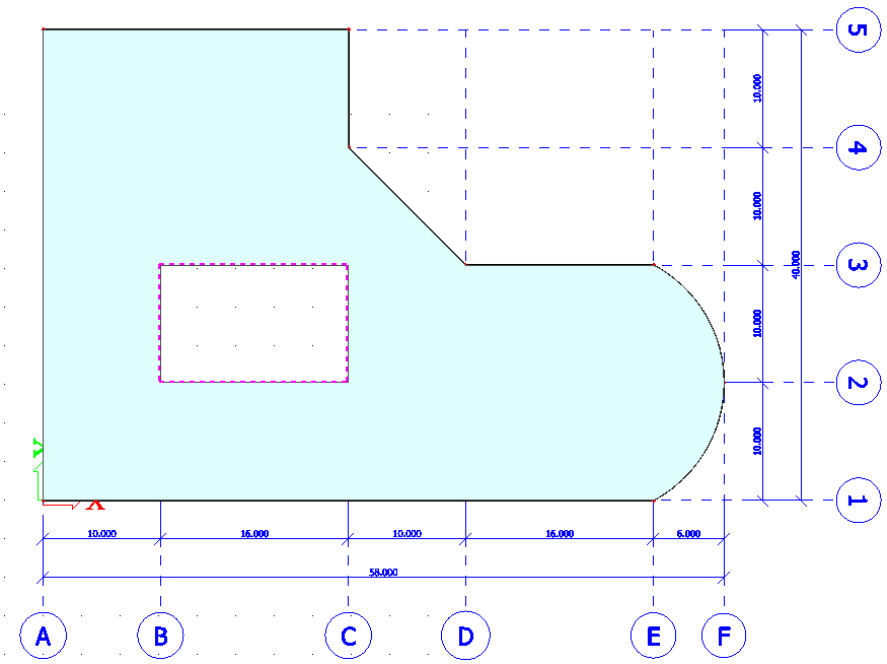
   •

3. The two nodes of its diagonal define a **rectangle**. This is also depicted by the two red dots on the icon.

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

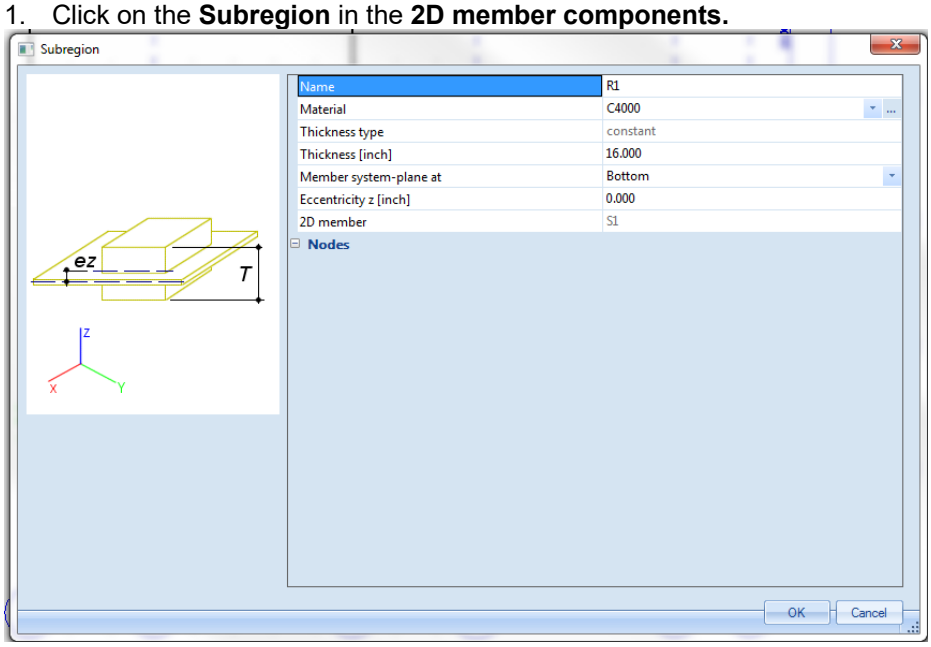
The program displays the proposed rectangle. Accept the **New Rectangle** by clicking the right mouse button.

By setting the rendering  to ON, The result can be visually checked:



Definition of a subregion

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



2. Define the following properties: **name=R1**, **Material=C4000**, **Thickness = 16in, Member system plate at=Bottom.**

3. Define a circular subregion on the slab. First, click on the circle symbol on the command line to define the circle extents.

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

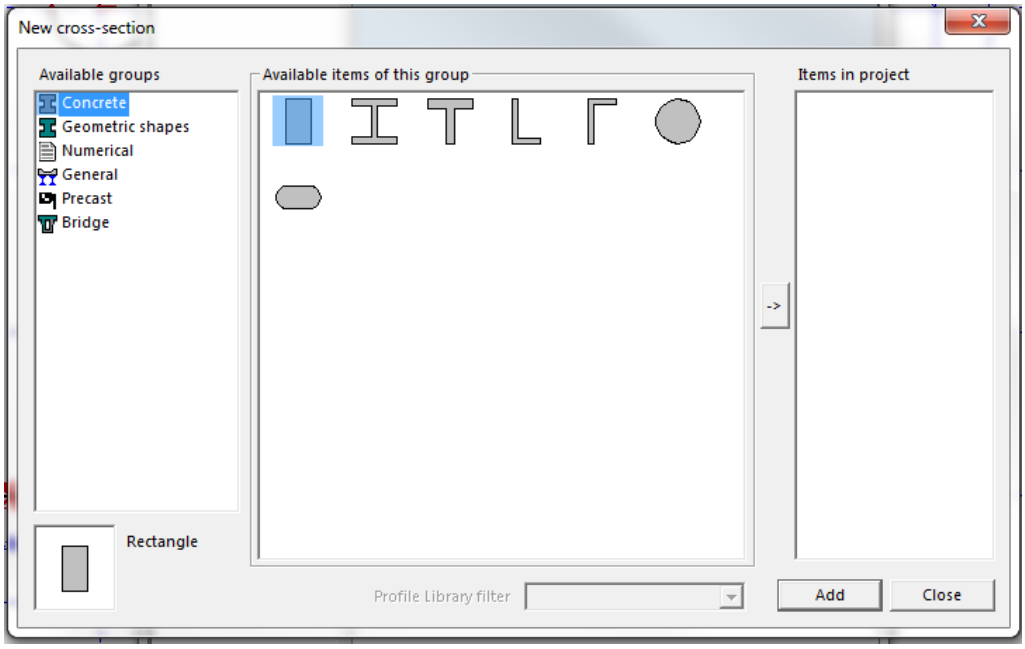
Input of plate rib

1. In the **Structure menu**, under **2D member components** choose **Rib.**

-
2. If no cross-section has been defined in the project, the dialogue **New cross-section** will open.

-

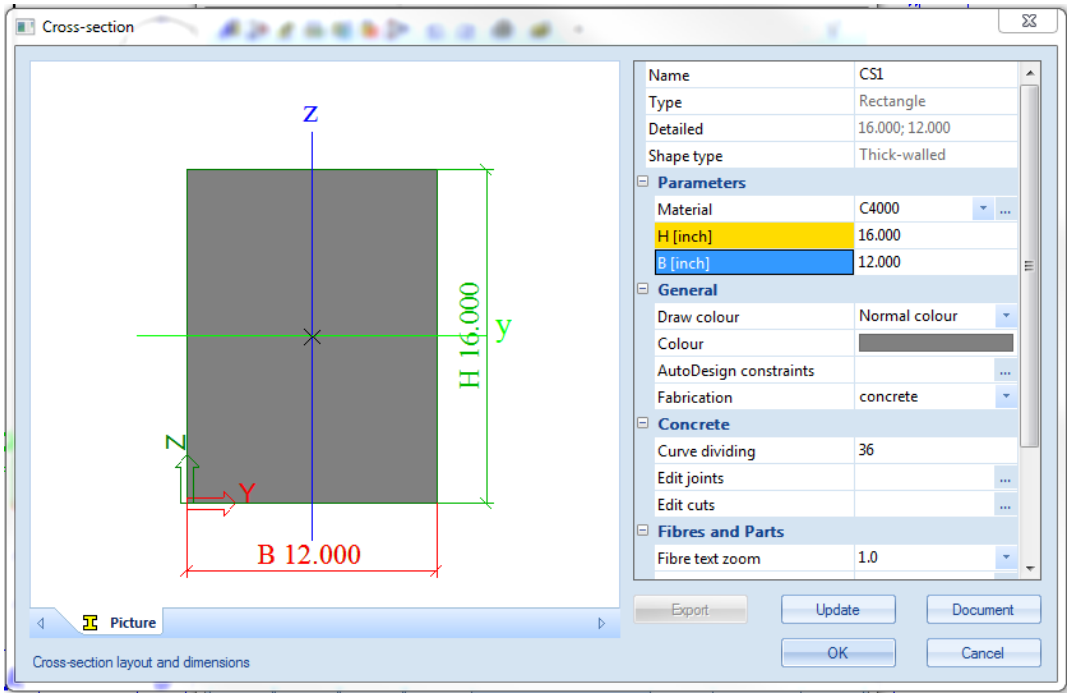
Here we will be able to select and define from the group **Concrete** a **rectangular cross section** as the new rib.



-

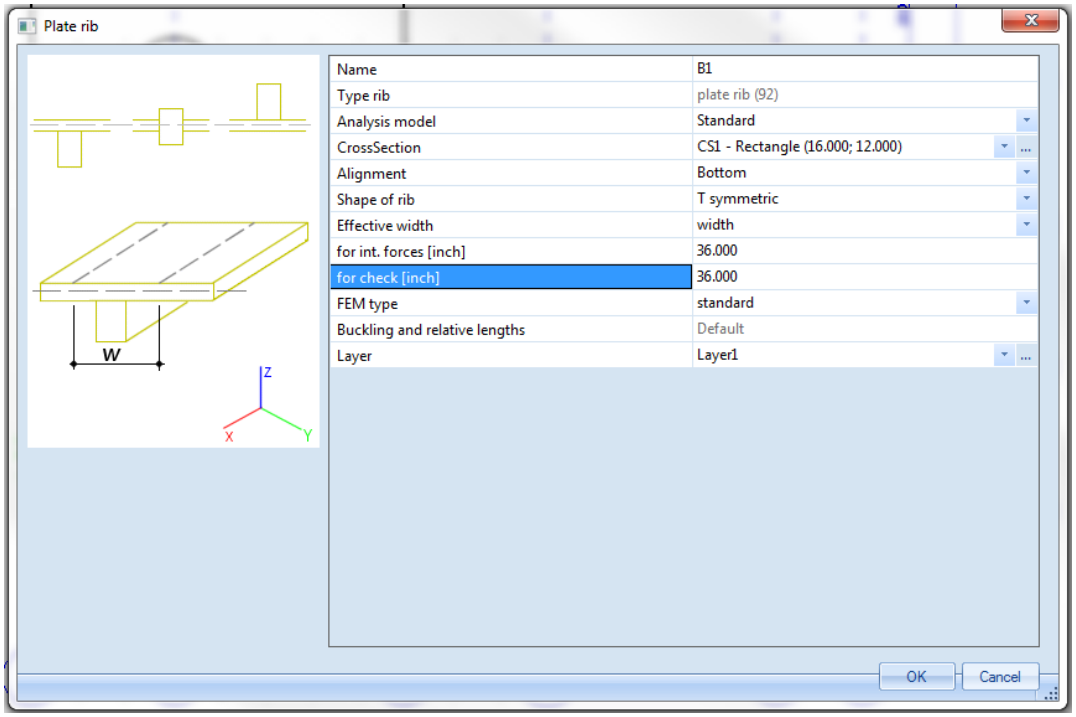
3. Click **[Add]**. This will take us to a new dialogue **Cross-section**. For this Tutorial we will use a rectangular concrete cross-section with **height 16in** and **width 12in** and a concrete grade of C4000.

-



4. Accept the cross-section by pressing the **[OK]** button**.** After this we will **[Close]** both dialogues.

5. In the **Plate rib** window we can define the parameters of the ribs as shown below:

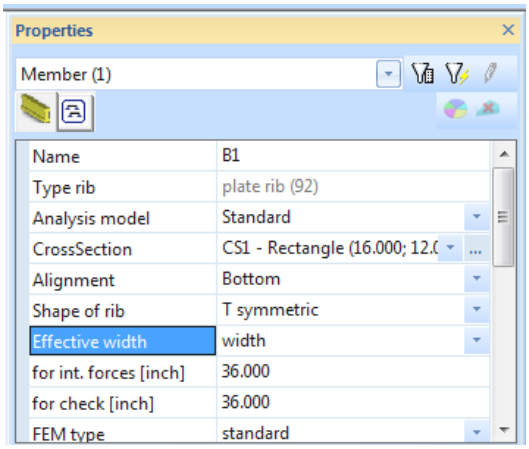


6. After clicking **<OK>** we will have to define the starting and end points of the ribs. After the input in complete, right click or press **<ESC>** to end the command.

Starting point: E2
End point: D2

Notes:

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



Effective Width:

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

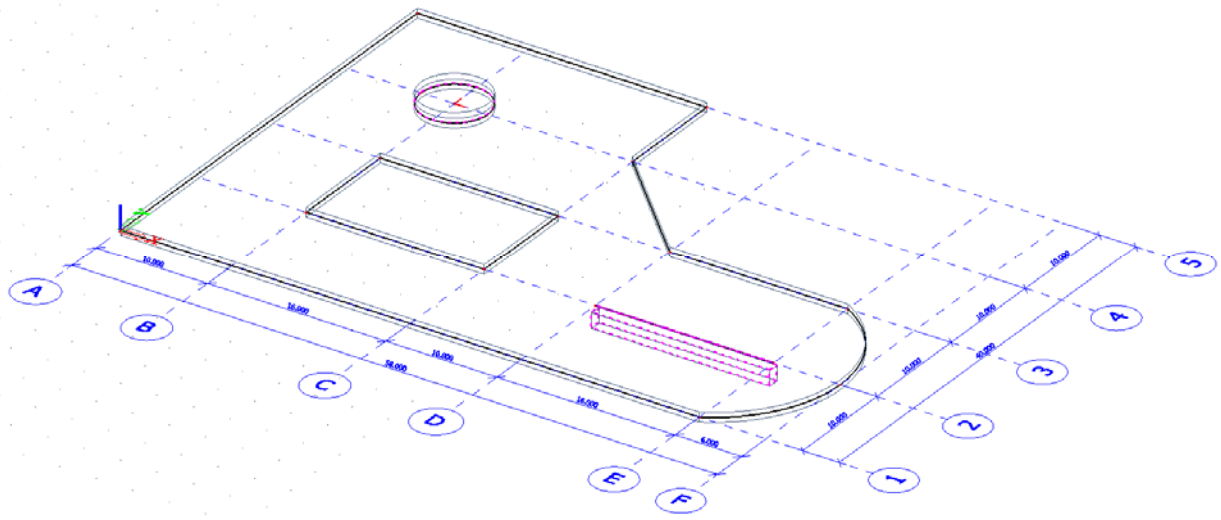
Number of thickness: *The width of the slab for the rib is defined as a factor times the plate thickness. The user enters the factor by hand.*

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

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



By viewing the  surfaces the input values of the geometry can be checked easily:



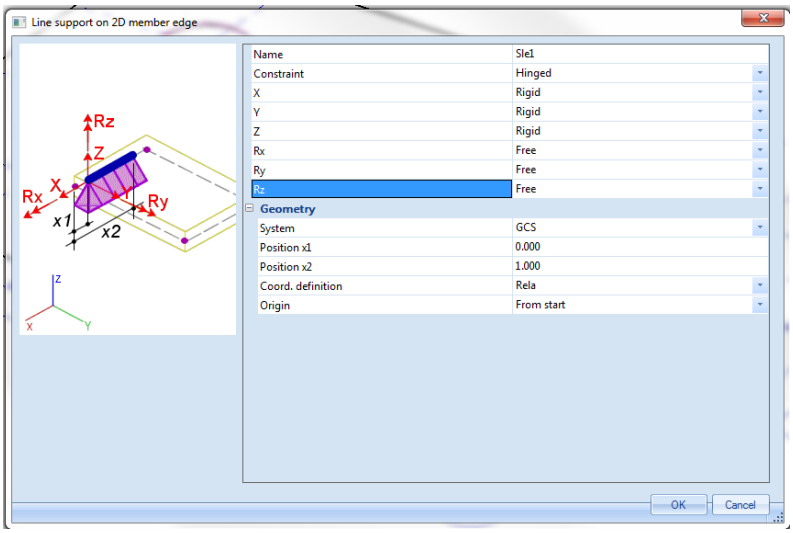
Supports

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

Definition of a support on an edge

1. Select in the **Structure** menu **Model data > Support > line on 2D member edge**

2. The window **Line support on 2D member edge** will pop up.



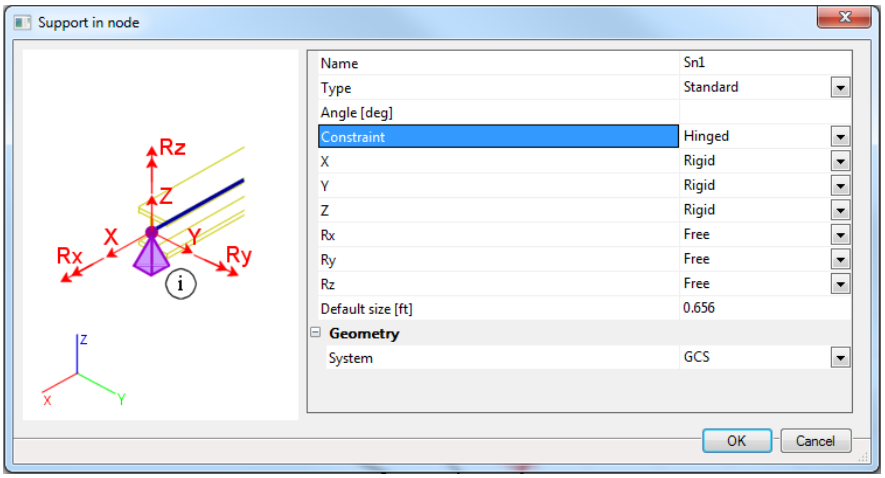
3. Constrain the translational degrees of freedom along the edges of the slab, creating a **Hinged** Support.

4. Select the edges around the slab one by one: edge1, edge2, edge3, edge4, edge5, edge6, edge7.

5. Press **<ESC>** to finish the input command

Input of Nodal supports

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



2. Constrain the translational degrees of freedom of the centroid of the subregion, creating a **hinged** support. This support may for instance account for a column under the slab.

3. Apply the nodal support at B4 point.

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

Check Structure data

After input of the geometry, the input can be checked for errors by means of the option **Check Structure data**. With this tool, the geometry is checked for duplicate nodes, null members, duplicate members and other incorrect data references.

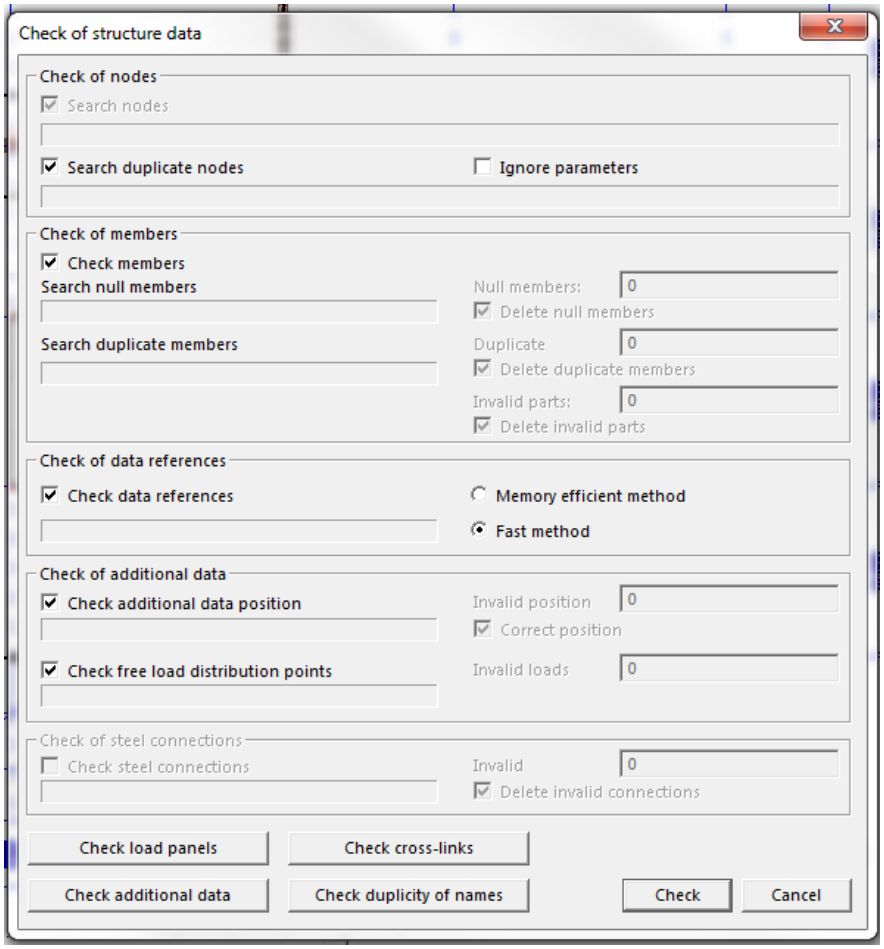
Checking the structure

1. Press on **<Esc>** or click on the button **Cancel Selection** in order to ensure that no entities are selected.
2. Double-click on the **Check Structure data** option in the **Structure Menu** or click on the icon in the toolbar.

•

3. The window **Check of structure data** will pop up for which a different set of checks is depicted.

•

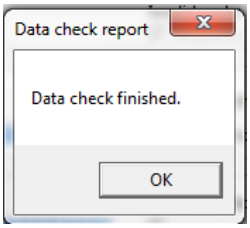


4. In the bottom four checking options for load panels, cross-links, additional data and duplicity of names are available to be selected individually. For this example, click **[Check]**.

•

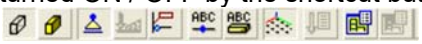
5. The message box **Data Check Report** opens.

•



6. Close the check by clicking **[OK]**.

Use of labels

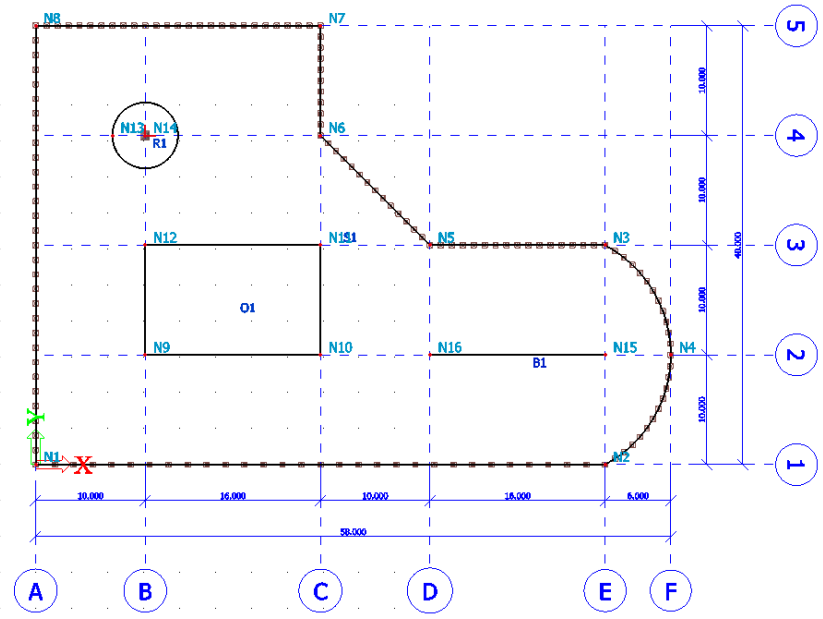
In order to display the names of the members, nodes or support symbols the labels of each item can be turned ON / OFF by the shortcut buttons in the lower left corner of the graphical screen  above the **Command line**.

The 3rd button can visualize supports. 

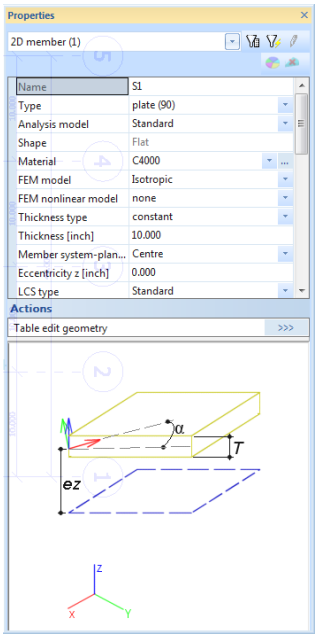
Labels of nodes can be activated by the button  located above the **Command line**

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

A view in direction Z  shows the following:



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



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

Graphic representation of the structure

Edit view

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

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

Editing the view point on the construction

Set view point through the wheels. Bottom right of the graphic window there are three wheels; two are horizontal and one is vertical. With these **wheels** you can **zoom in** on the construction or **turn** it.

1. To be able to zoom in on the construction or to turn the structure, click on the wheel (the cursor will change into a hand), keep the left mouse button pressed and move the wheel
   - OR
   - Set the view point by combining the buttons and mouse.
2. Press CTRL + right mouse button at the same time and move the mouse to **turn** the construction.
3. Press SHIFT + right mouse button at the same time and move the mouse **move** the construction.
4. Press CTRL + SHIFT + right mouse button at the same time and move the mouse to **zoom in or out** on the construction.

Note:

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

Setting a view direction with regard to the global coordinate system

1. Click on the button **View in direction- X** for a view in the X-direction.
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.

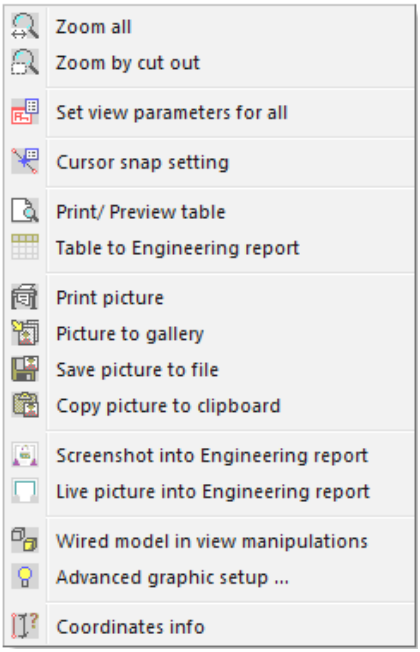
The magnifier

- 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 with the right mouse button. The following shortcut menu appears:



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

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

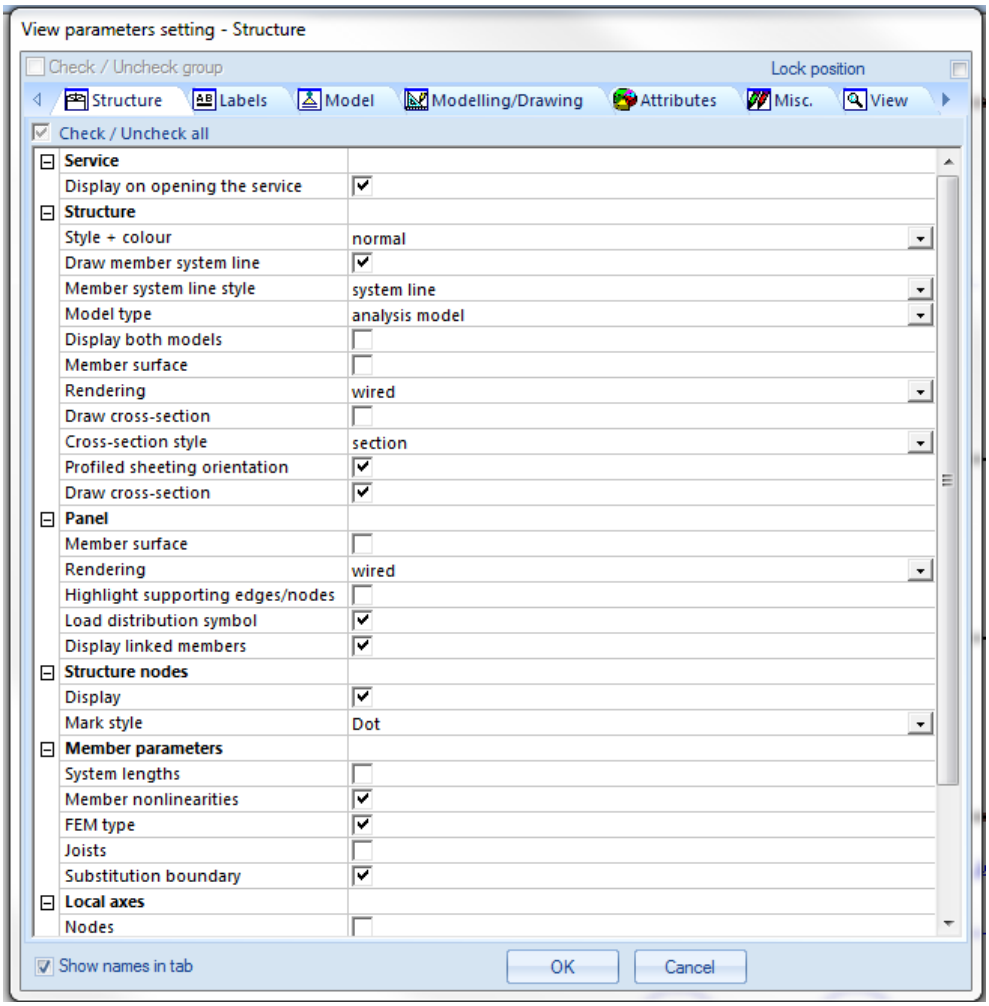
View parameters – Entities

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

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

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

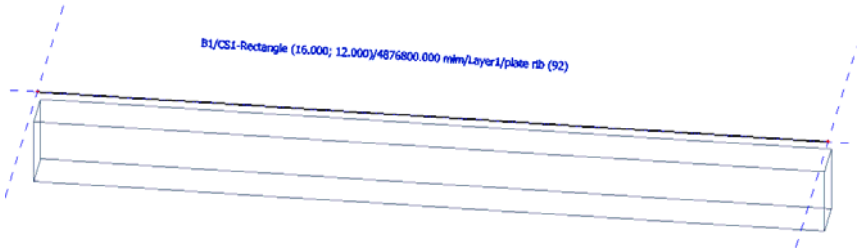
Local Axes: Using this tool the local axes can be shown and modified for nodes, 1D and 2D members.



View parameters – Labels and description

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

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



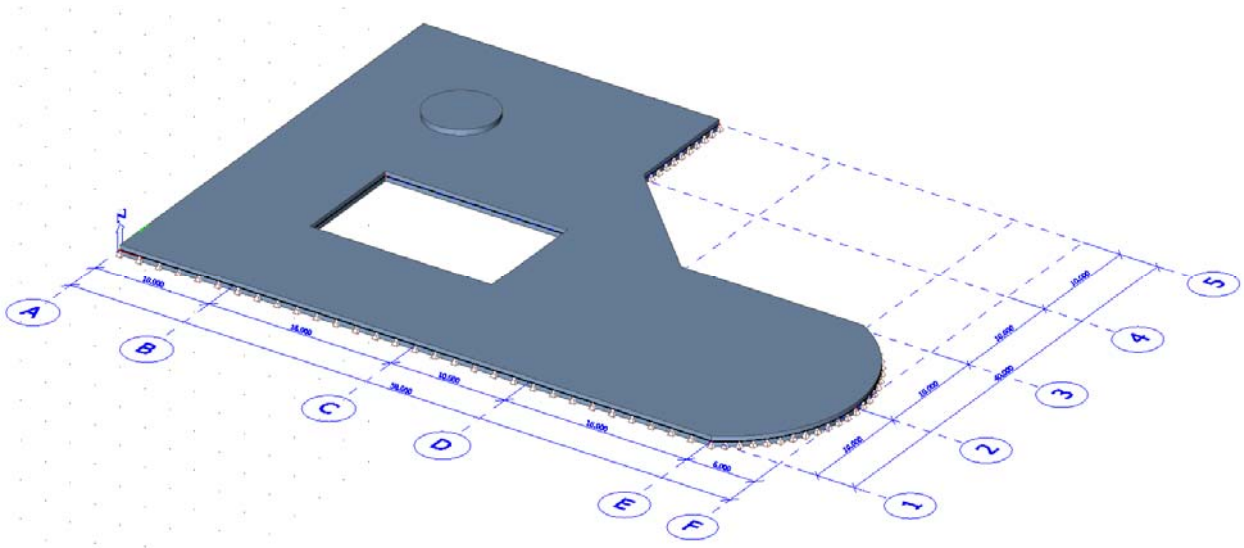
- **Length**: show the length of the beam in the label.
-
- **Display labels**: Only when this tick box is turned ON, will the labels be visible on the graphical screen.

View parameters – Shortcuts

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

- **Show/hide surfaces** to show the surfaces of the elements.
-
- **Render geometry** to view the rendered members.
- **Show/hide supports** to show supports and hinges.
- **Show/hide load** 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.
- **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 of the Calculation Data

Load Cases and Load Groups

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

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

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

In this project, two load cases are entered:

- **LC1**: Dead load
- **LC2**: Live Load Case

Defining a Permanent Load Case

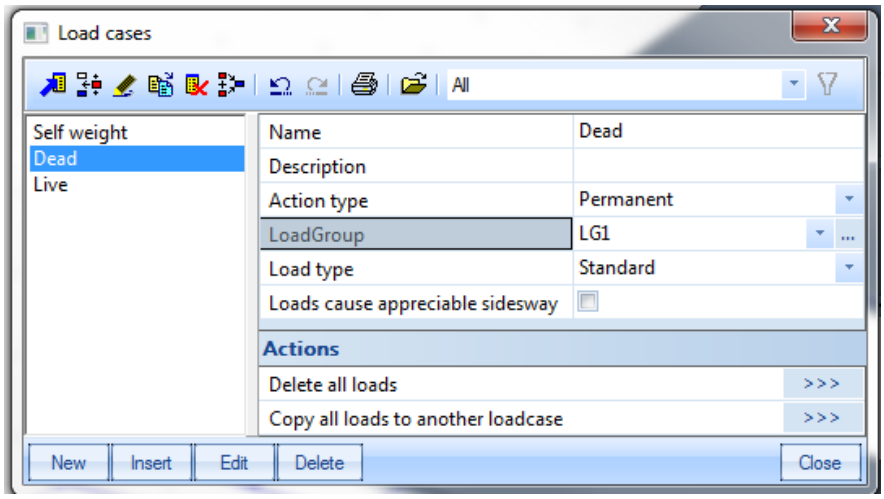
1. Double-click on Load in the **Main window**.

2. Before you can define loads, you first must enter load cases. Since this project does not contain any load cases yet, the **Load Cases Manager** will automatically appear.

3. By default, the load case **LC1** is created. This load is a permanent load of the **Self weight** load type. The self-weight of the structure is automatically calculated by means of this type.

4. In the Description field, you can describe the content of this load case. For this project, enter the description **"Self Weight"**.

5. Click **New** and enter **Dead** load in the Name field. This load has **Permanent** Action type and a **Standard** load type.



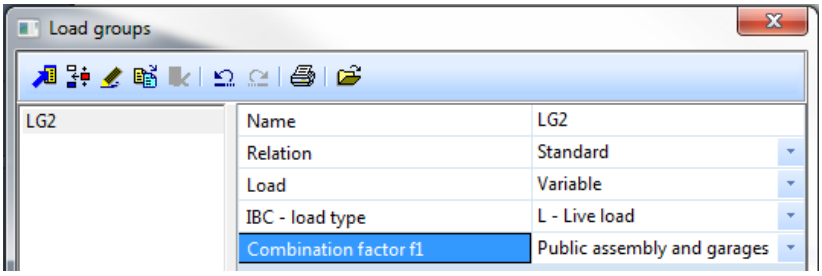
Defining a Variable Load Case

1. Click New or  to create a second load case.

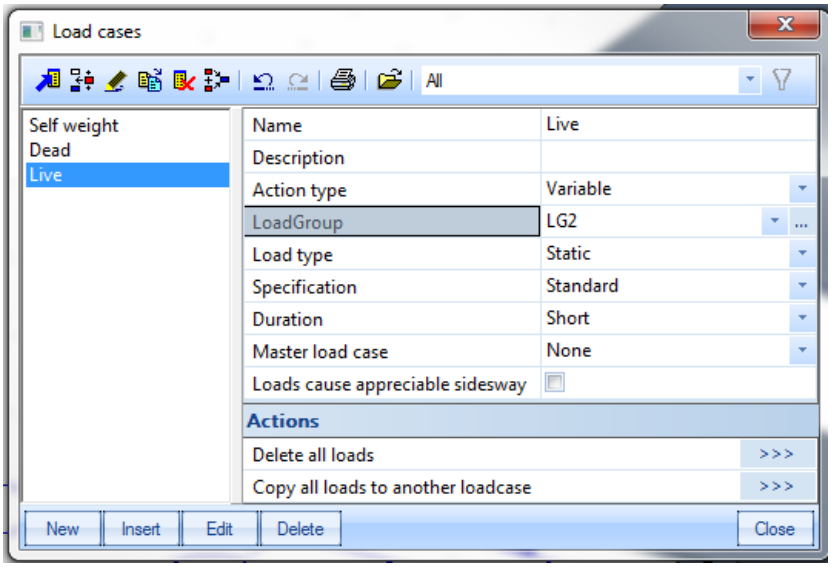
2. Enter the description **"Live"**.

3. As this is a variable load, change the Action type to **Variable**.

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



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



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

Notes:

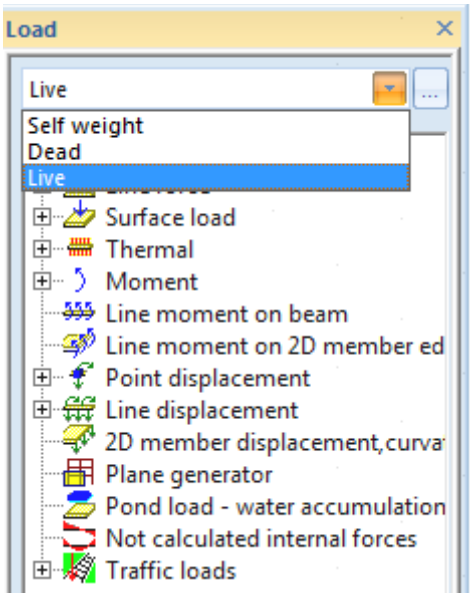
Each load is classified in a group. These groups influence the combinations that are generated as well as the coefficients that are applied. The following logic is adopted.
If the load is divisible, its different components are entered as individual load cases. As long as the load combination does not contain any variable load belonging to another group, no reduction factors may be applied. The different load cases of a divisible load are therefore associated to one variable group.
Load cases of the same type that may not act together, are put into one group, which is made exclusive, e.g. "Wind X" and "Wind -X" are associated to one exclusive group "Wind".

Loads

After input of the Load cases, the **Loads menu** will automatically appear:
- Self weight
- Dead
- Live

Switching between load cases

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

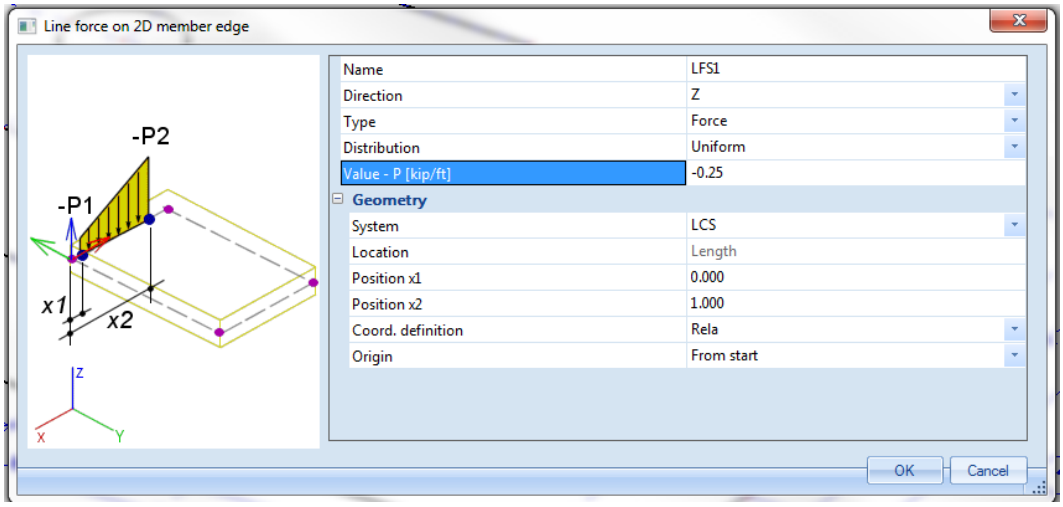


Entering the Self weight

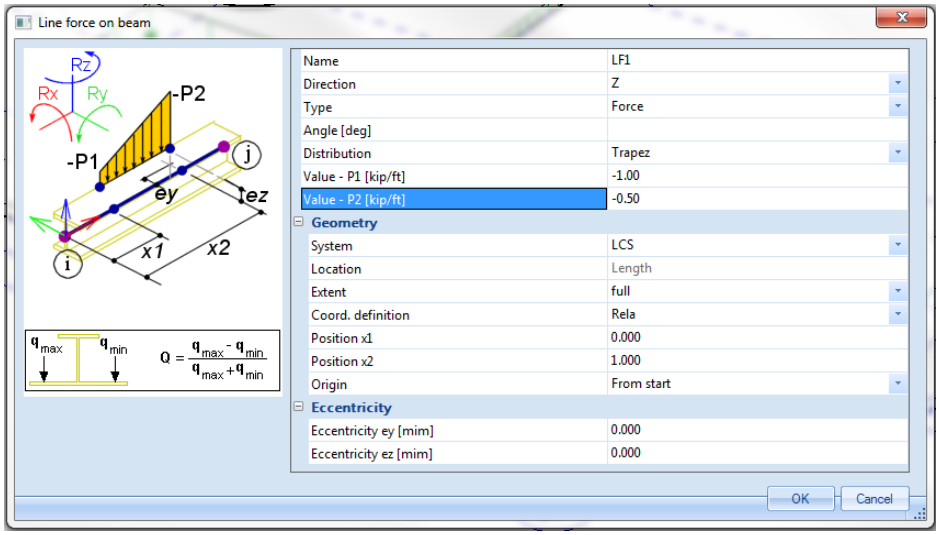
The self-weight loads are automatically generated. The user is not required to calculate these action loads.

Input of dead loads

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



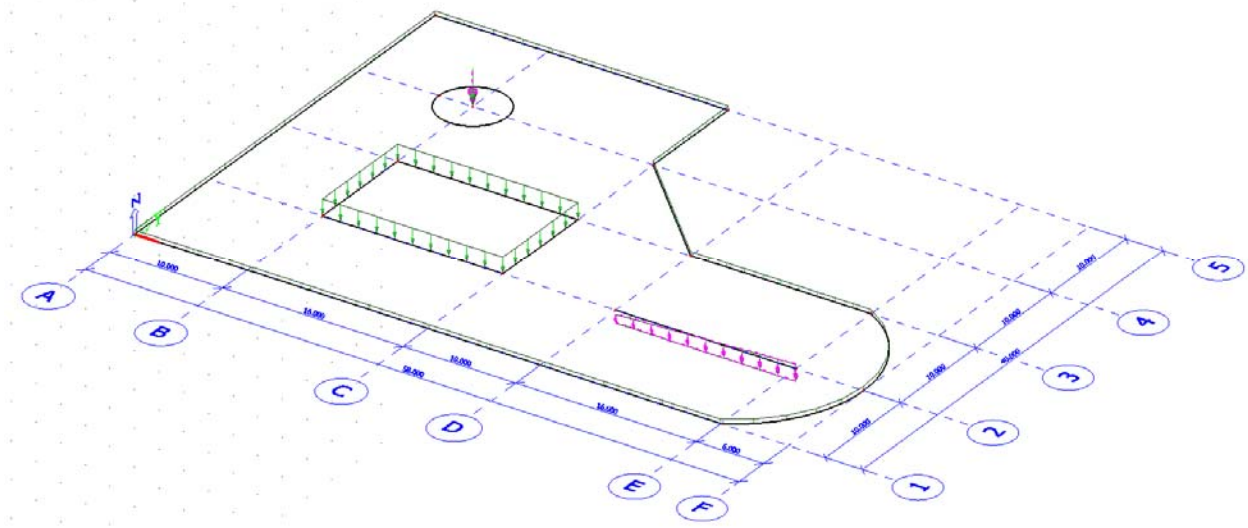
4. Select the outer slab edges of the plate and when finished press **<ESC>.**
5. Similarly insert a **-1.0 kip/ft** line load on the edges of the opening accordingly.
6. Next, apply a trapezoidal load on the beam varying from **-1 kip/ft** to **-0.5 kip/ft**. Select **on beam** in the **Line force**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



7. Finally, input a point load by selecting on **in node** under the **Point Force** heading and apply **-5 kip** load at location B4 on the subregion.

8. Press **[ESC]** to confirm your selection.

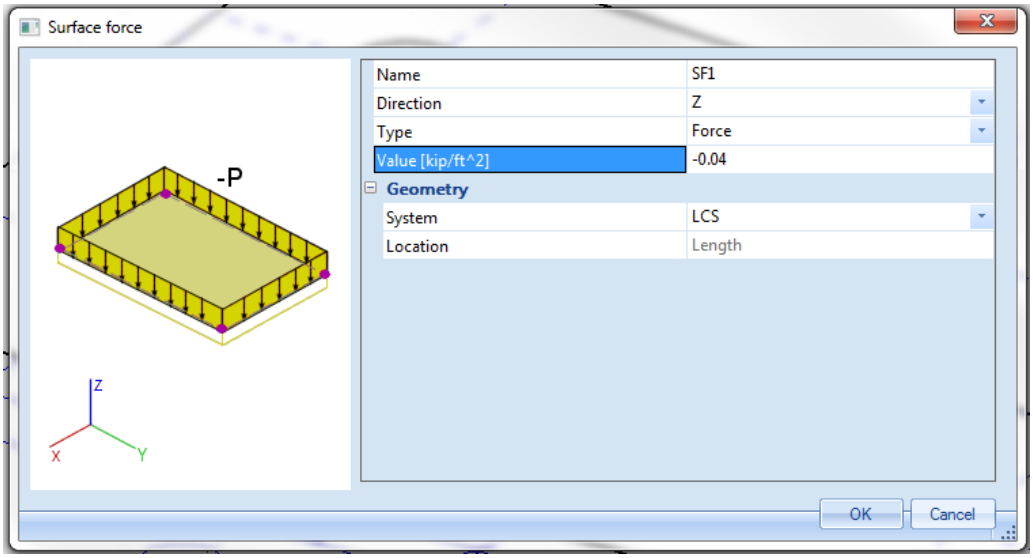
The layouts of the Dead loads are as following



Input of live surface loads

1. Select the **Live** load from the **Load menu**.

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

- 

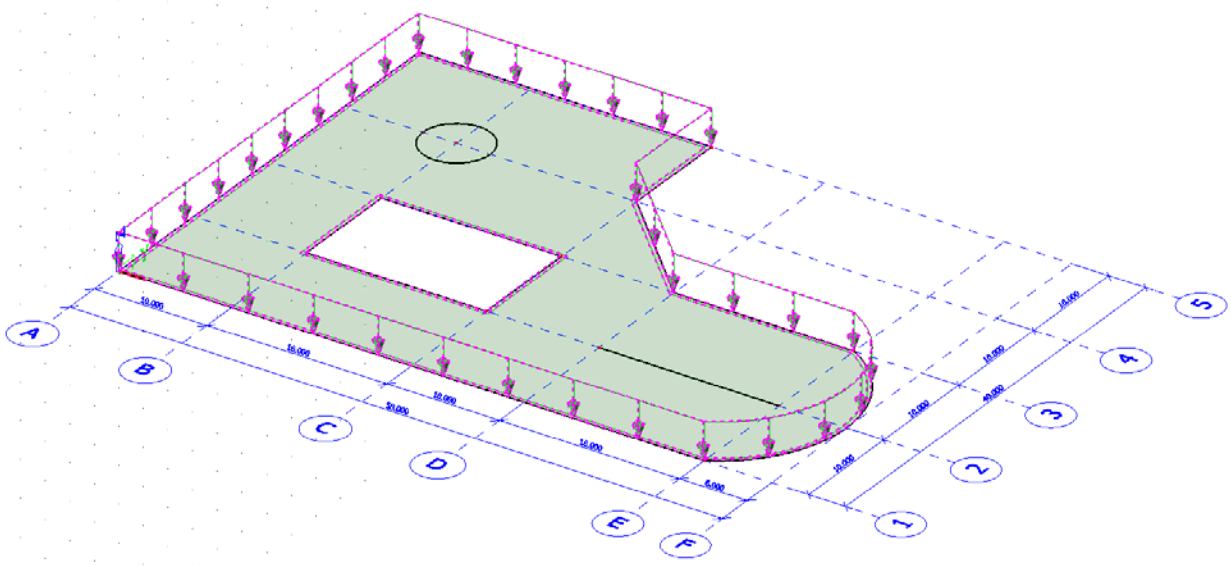
3. Confirm your input with **[OK]**.

-

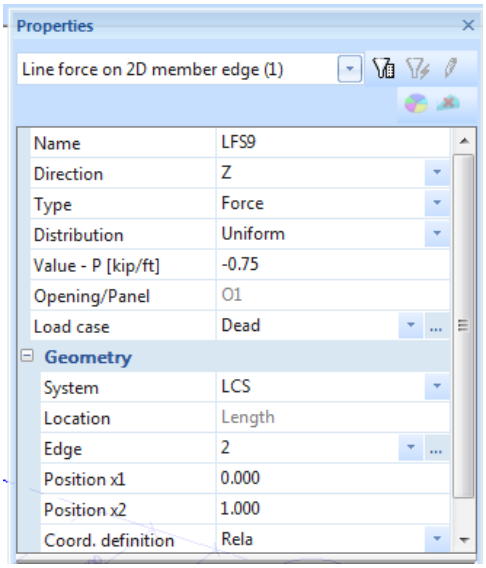
4. The program will then ask the user to define the surface of the free surface load. As the slab has been defined as a closed polygon, selecting any point on the outer edges of the slab would select the surface.

-

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



Adapting a load



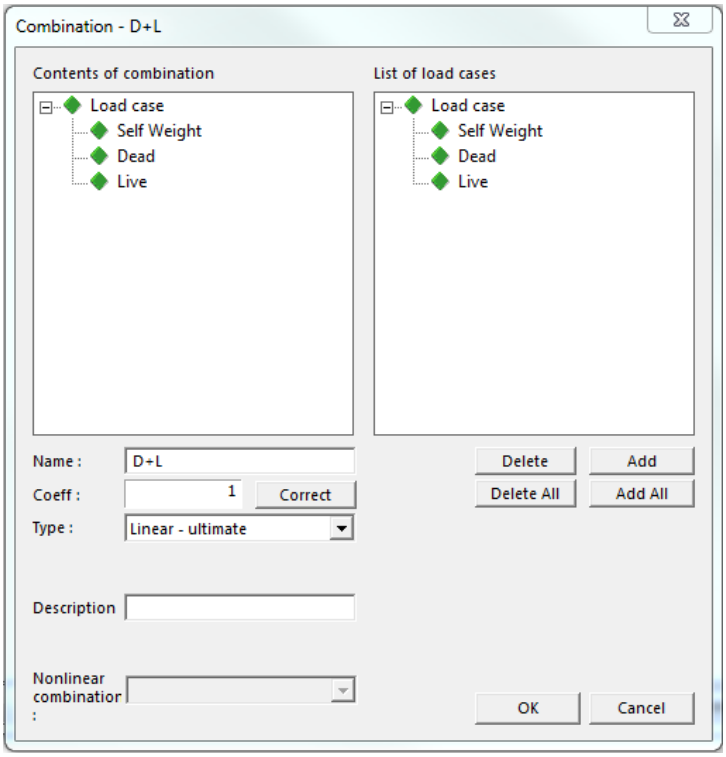
1. Select **Dead** load in the **Load menu**.
2. Select the line loads around the opening by clicking them with the left mouse button.
3. The common properties of the 4 loads are displayed in the **Properties window**.
4. Change the **Value** from **–1.0 kip/ft** to **–0.75 kip/ft** in the Properties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

Combinations

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

Defining Combinations

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



3. By means of the button **[Add all]**, all load cases can be added to the combination.

4. Change the **Type** to **Linear – ultimate.**

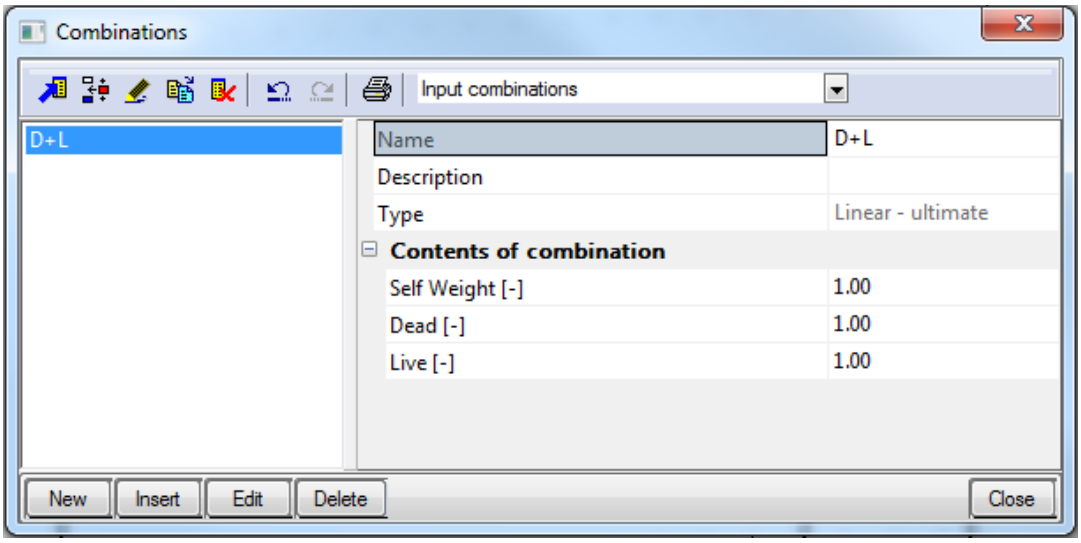
5. Confirm your input with **[OK]**. The **Combination Manager** is opened. Name the combination as **D+L**.

6. Click **[Close]** to close the **Combination manager**.

Notes:

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

Envelope – user defined base for automatic generation of combinations (both ultimate and serviceability)
Linear – user defined combination using user defined combination coefficients
Code Dependant – According to the active code set for the project and the applicable load groups


Combinations
Input combinations
D+L
Name
D+L
Description
Type
Linear - ultimate
Contents of combination
Self Weight [-]
1.00
Dead [-]
1.00
Live [-]
1.00
New
Insert
Edit
Delete
Close

Calculation and Mesh generation

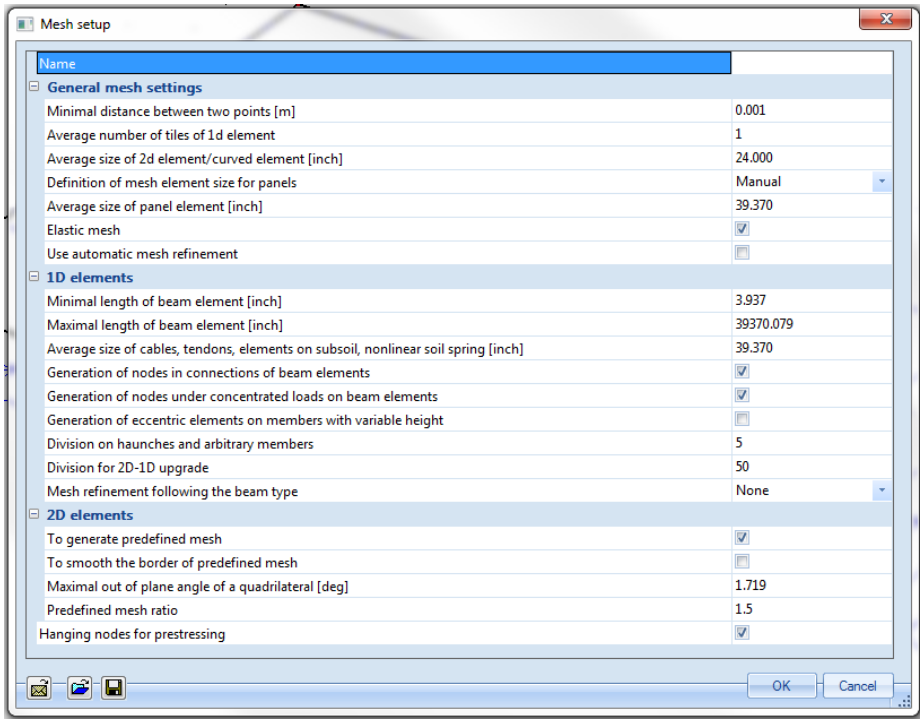
The calculation of the slab will be done according to the finite element method. According to the calculation method a mesh will be generated on the slab and the results will be calculated at each finite element created by the mesh. The result in the middle of a finite element is determined as the average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

Mesh generation

Mesh setup

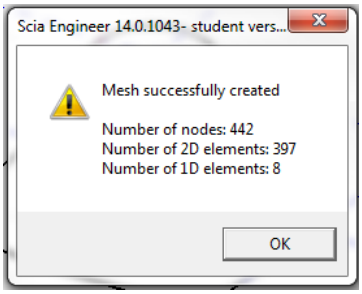
1. In order to generate the mesh, select **Mesh setup** in the **Calculation, mesh** option from the **Main menu**.

-
2. In the **Mesh setup** dialogue box insert **24”** for the **Average size of 2D element/curved element/nonlinear soil spring.**

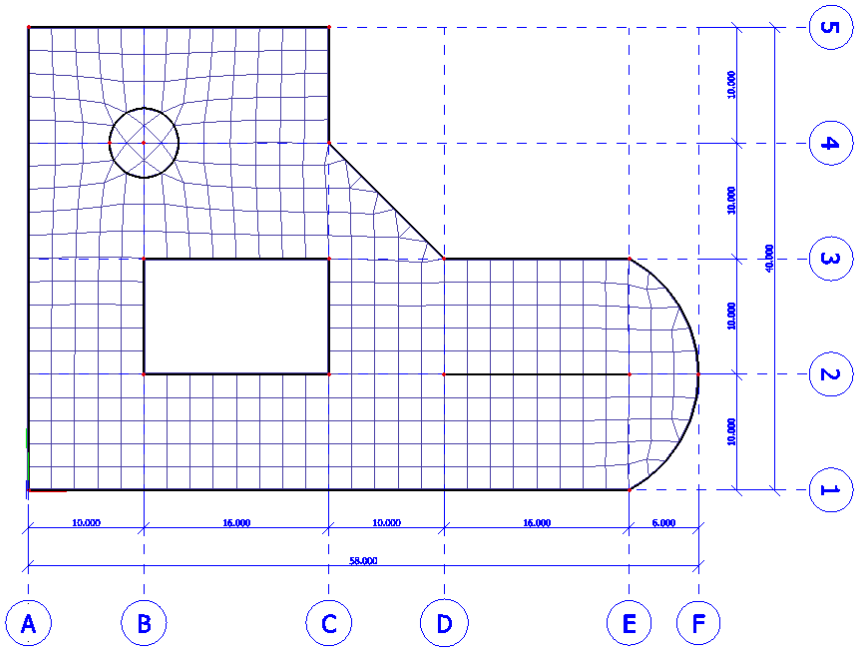


3. Confirm by selecting **[OK]**.

-
4. Choose the Mesh generation option to create the mesh based on the selected finite element size.



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



Linear Calculation

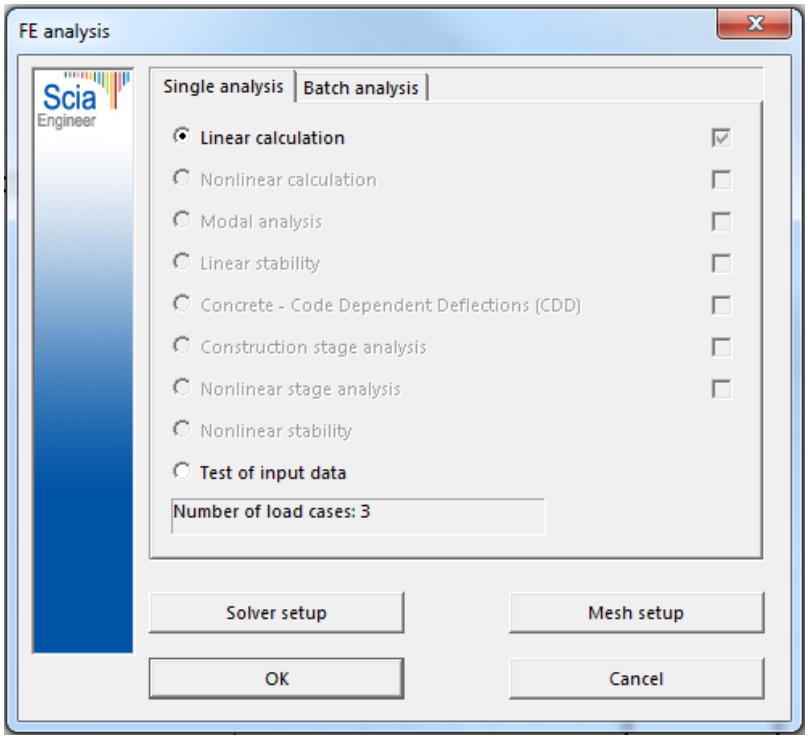
As the calculation model is completely ready, you now can start the calculation.

Solver settings

1. Double-click on below in the **Main window**.

•

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



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

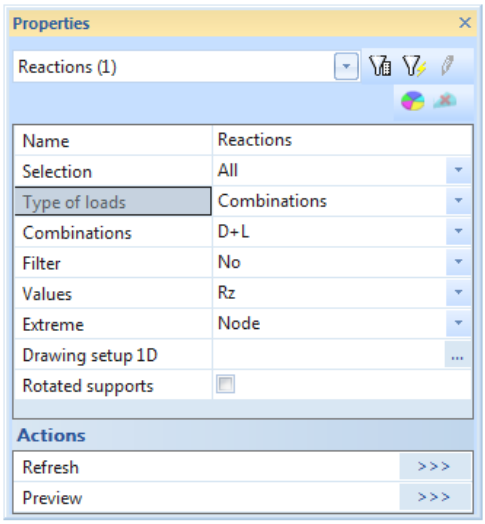
Results

Viewing results

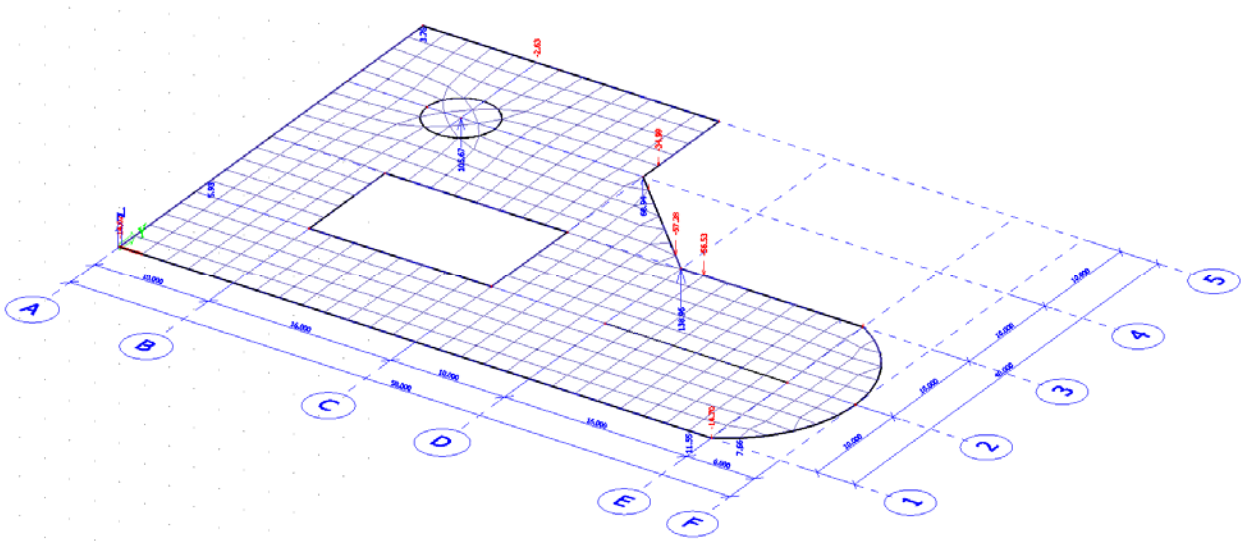
After the calculation is executed, the results can be viewed.

Viewing the Reaction Forces

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

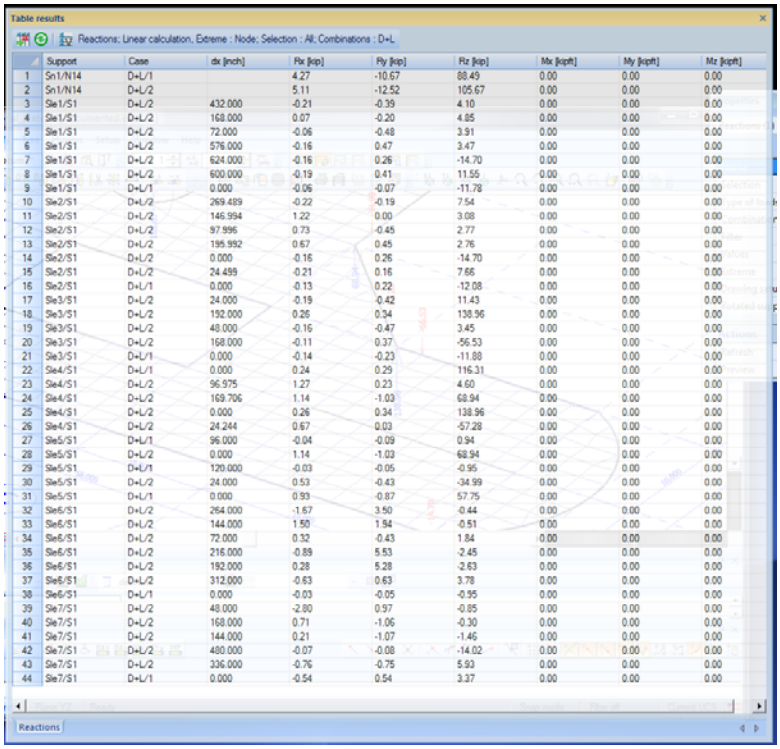


4. The action **Refresh** has a red background, i.e. the graphical screen must be refreshed. Click on the >>> button behind **Refresh** to display the results in the graphical screen in accordance with the set options.



5. To display these results in a table, the **Preview** action is used. Click on the >>> behind **Preview** to open the Preview.

6. Additionally, the results can be accessed in a dynamic table format by using the **Table Results.**



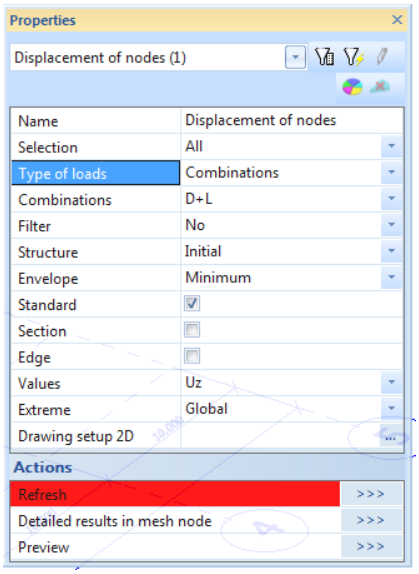
Note:
The Preview appears between the Graphical Screen and the Command line. This screen can be maximized to display more data at one time.

Viewing Results on 2D elements

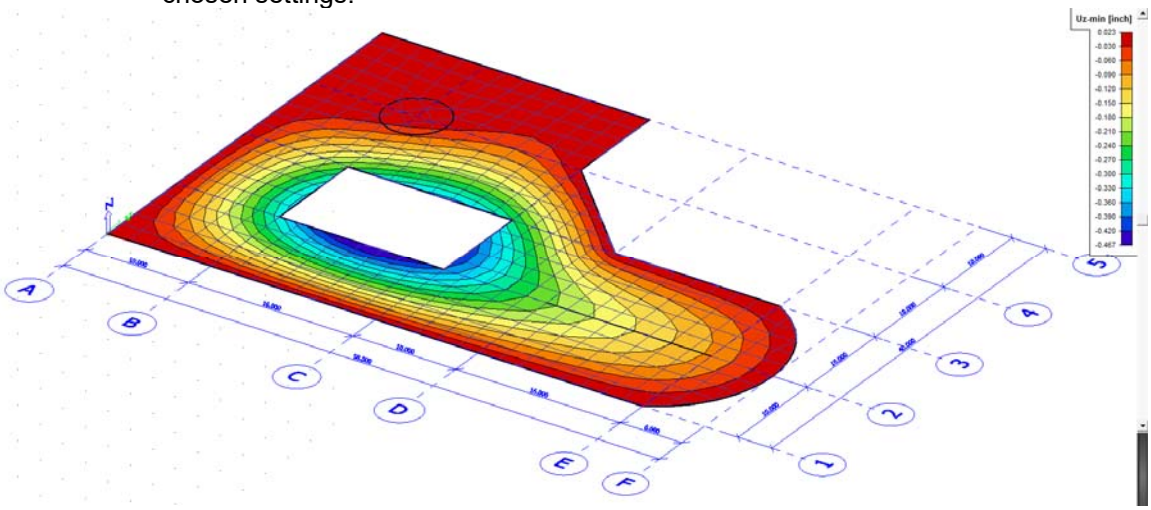
1. Below **2D Members** in the **Results service**, click **Displacement of Nodes**.

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

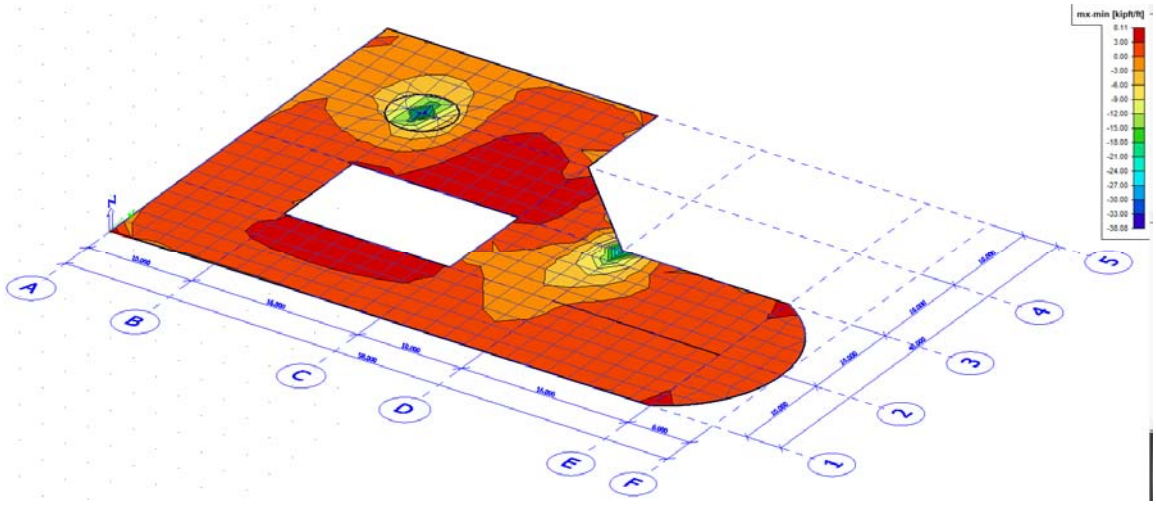
- The **Selection** field is set to **All**
- The **Load type** is set to **Combinations** and Combination to **D+L**
- The **Values** will be reviewed for **Uz**.
- The **Extreme** field is changed to **Global**.



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



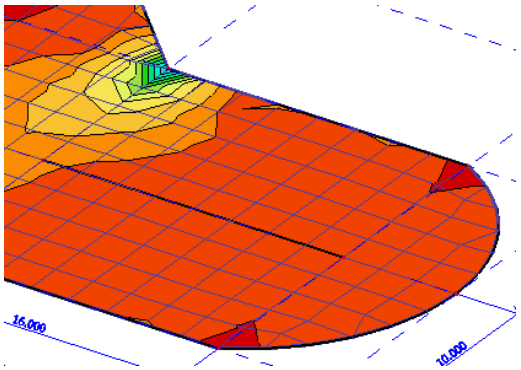
Internal forces can be configured in the **Member 2D – Internal forces** result option. For instance the moment about the x axis **(mx)** has the following graphical contour for the combination D+L.


mx-min [kipft/ft]
8.11
3.00
0.00
-3.00
-6.00
-9.00
-12.00
-15.00
-18.00
-21.00
-24.00
-27.00
-30.00
-33.00
-36.08
A
B
C
D
E
F
1
2
3
4
5

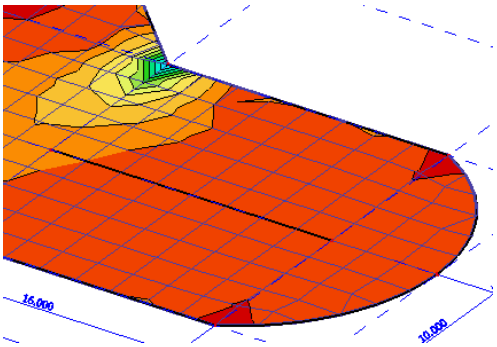
Results for rib

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

- Option Rib not activated:



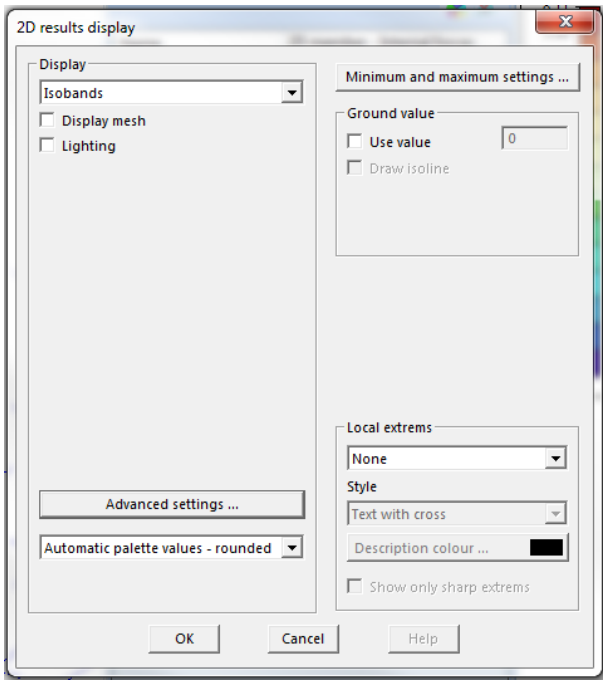
- Option Rib activated:



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

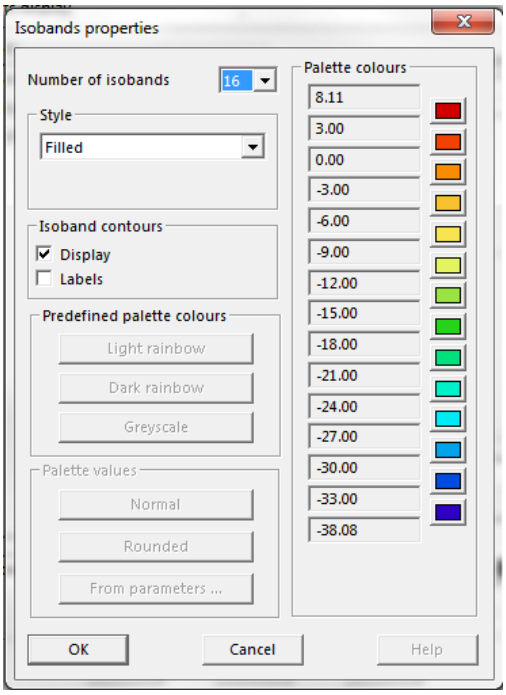
Configuring the Graphical Screen

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



2. For the **Display** the option **Isobands** will be chosen.

3. The button **Advanced settings** allows for the setting of the legend for the graphical screen.



4. Click **[OK]** to accept the settings or **[Cancel]** to ignore the selected settings.

5. Click in the **Property Window**, on the button >>> behind **Refresh** in order to display the results in the graphical screen in accordance with the set options.

6. Click **[Close]** to leave the **Results Menu**.

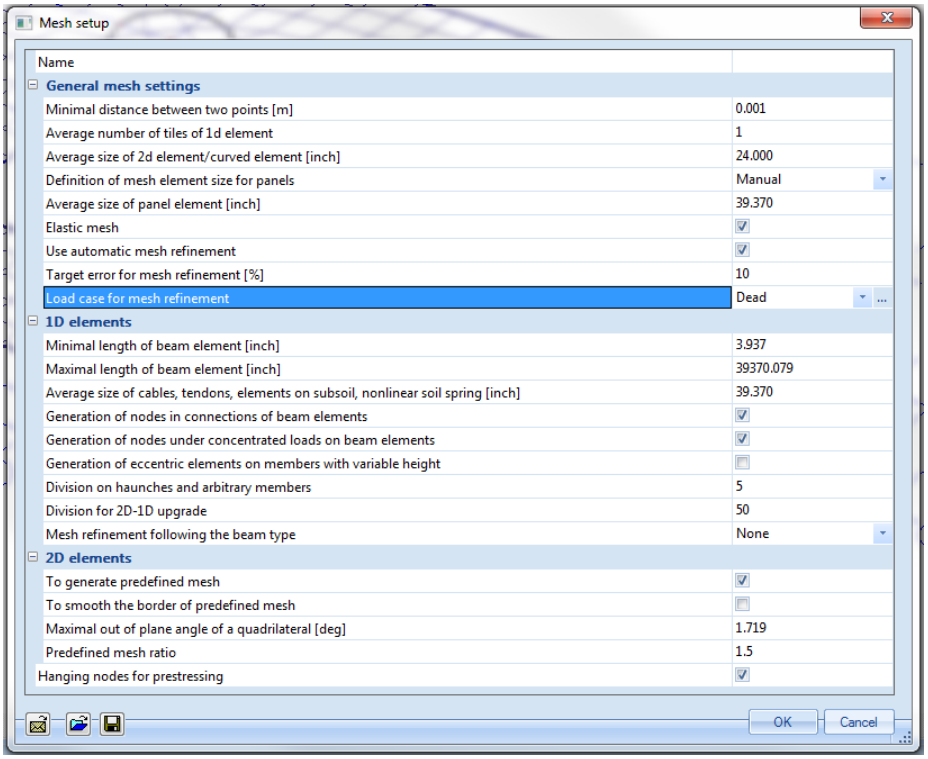
Note:
*To change the font size of the displayed results, you can use the **Setup > Fonts** menu. In this menu, the different sizes of the displayed labels can be changed.*

Post processing

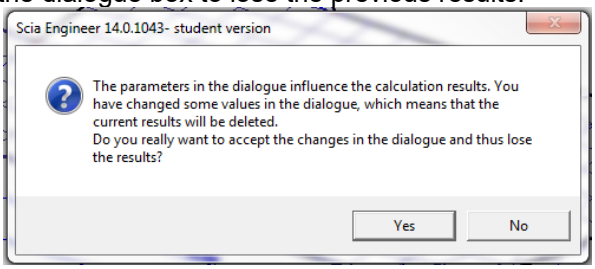
Automatic mesh refinement

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

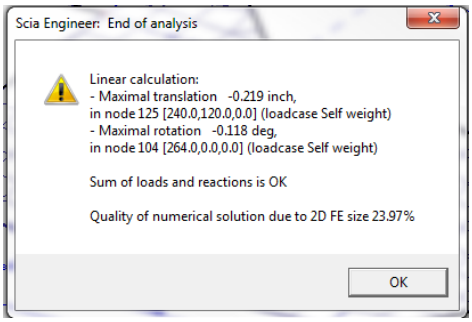
1. Select the **Mesh setup** in the **Calculation, mesh** from the **Main menu**.
2. Choose **10%** for **Target error for mesh refinement** and select **Dead** for **Load case for mesh refinement**.



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



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

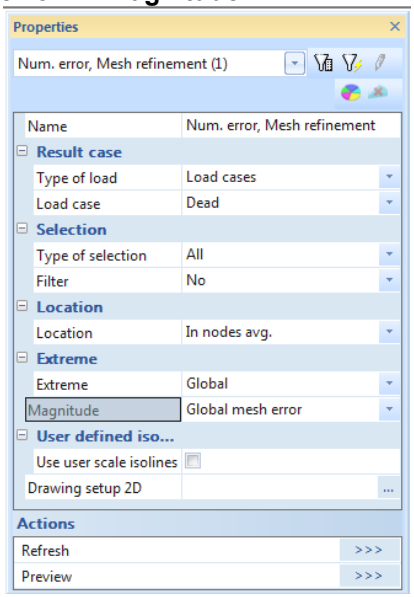


5. The quality of numerical solution is demonstrated in the analysis window. This is the relative % error, as calculated:

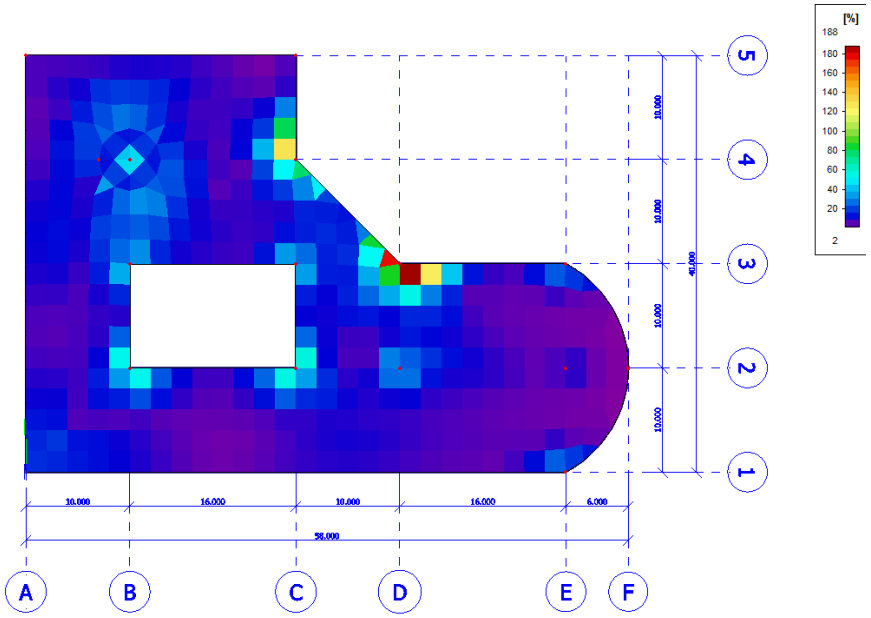
- $\% = \frac{\text{Energetic Norm of Estimated Error}}{\text{Energetic Norm of Displacements}}$ and is numerically acceptable if it is below 30%..

6. The sensitivity of the numerical solution can be visualized through the **Num, error, Mesh refinement** in the **2D members** section of the **Results menu**.

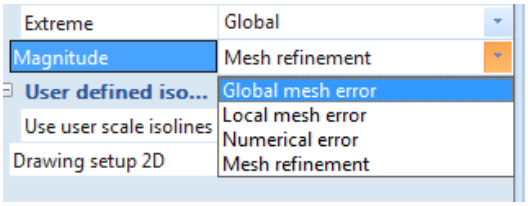
7. In the Properties window select **Load cases** for **Type of loads**, **Dead** for **Load cases**, **Global** for **Extreme** and **Global mesh error** for **Magnitude**.



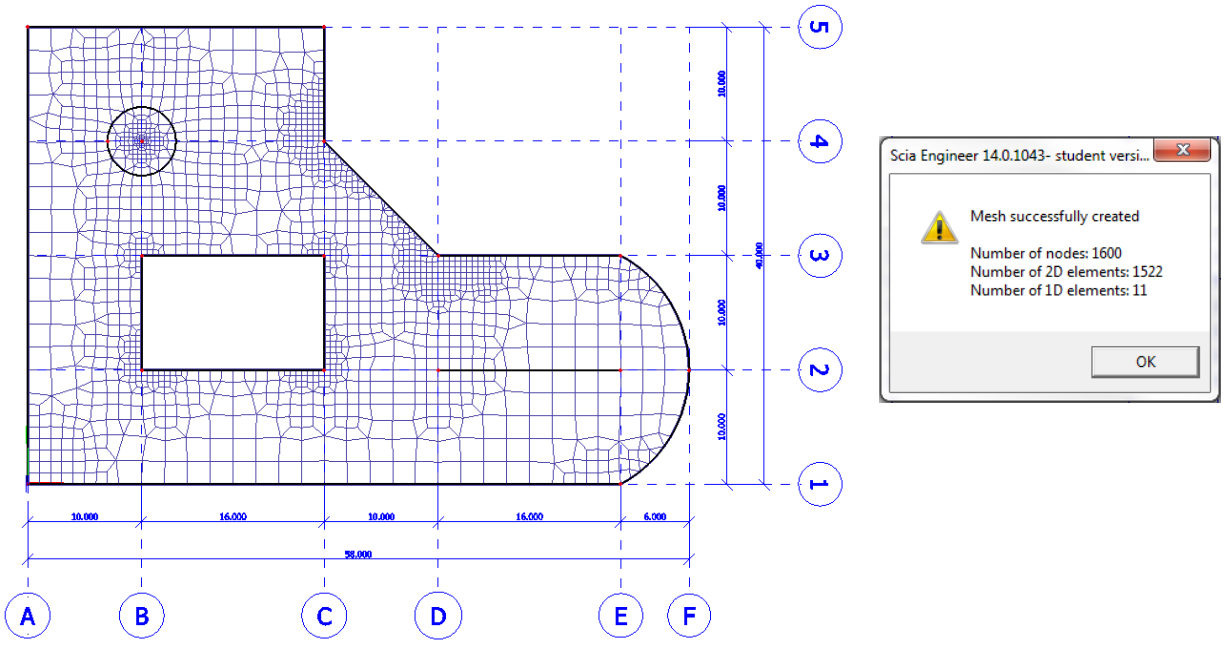
8. The error contour is as follows:



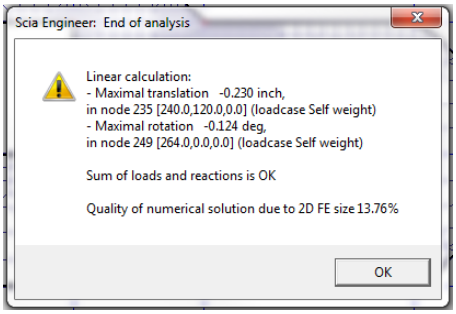
9. Other options for the Magnitude are available as well:



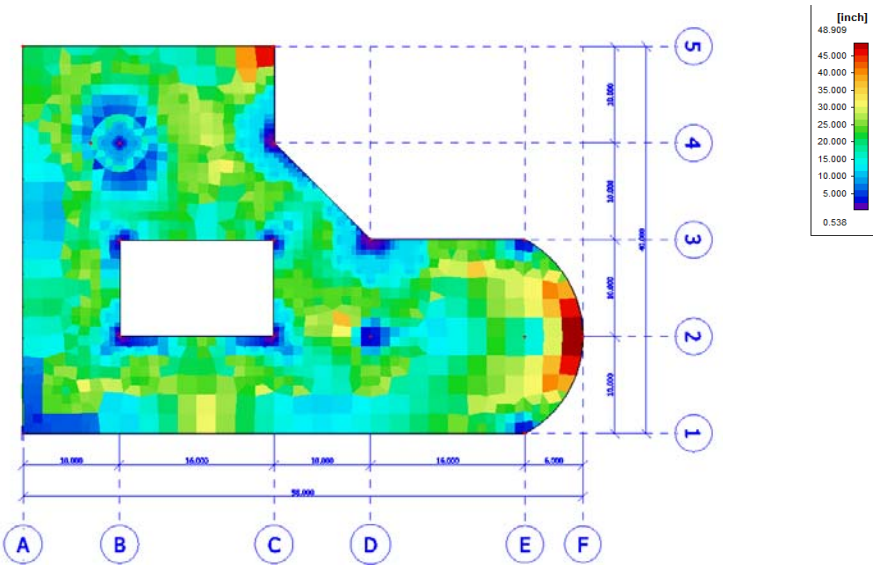
10. The optimal mesh size contours can be viewed through the Mesh refinement option above. In order to apply the error analysis to the previous mesh, simply regenerate the mesh using **Mesh generation** in the **Calculation, mesh** from the **Main menu**. The new mesh will then be created based on the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



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



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

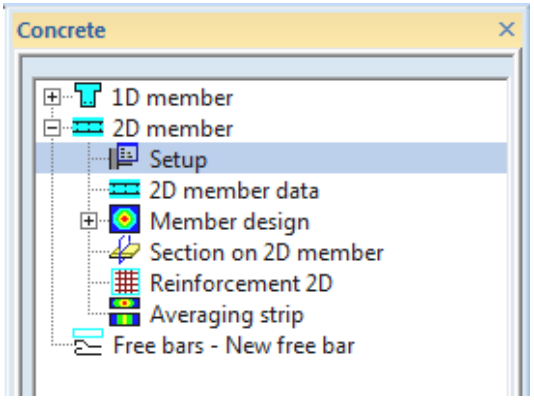


Reinforcement design

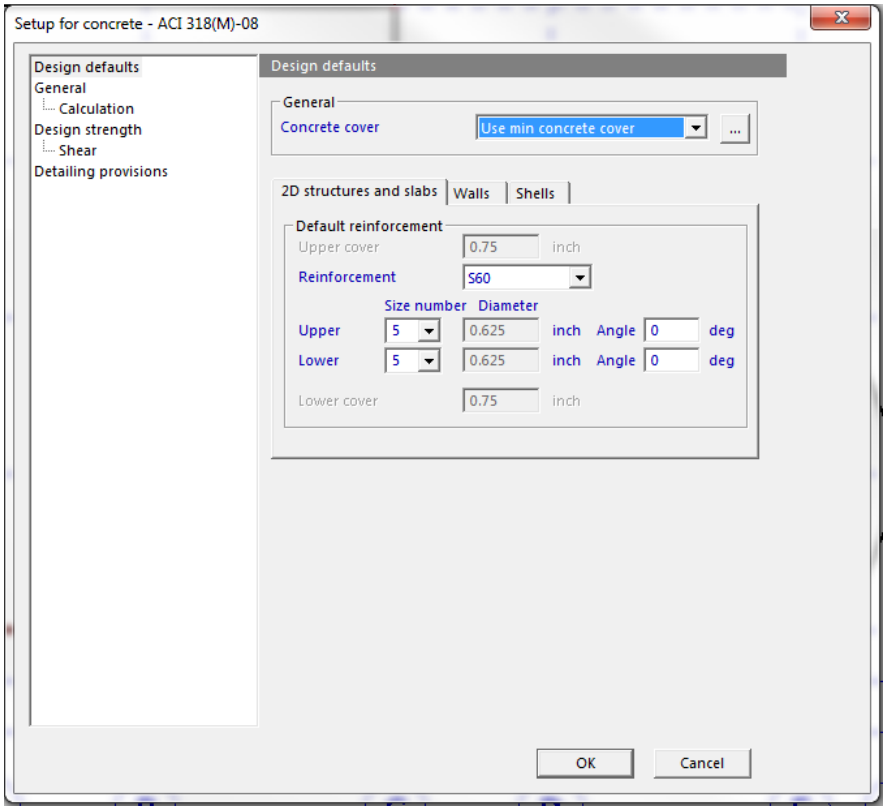
The reinforcement design can be viewed in the concrete menu.

Pre-processing

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

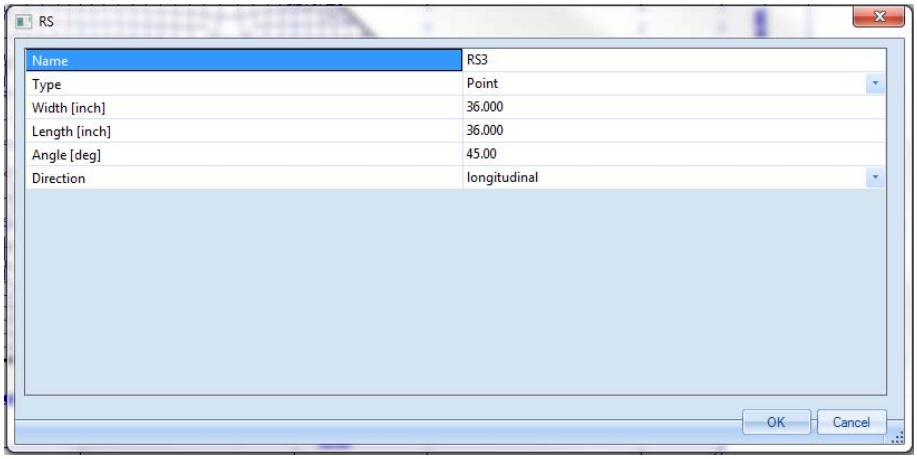


2. Choose **S60** in the **Reinforcement** from the **Design Defaults** tab.



3. Select **[OK]** to confirm.

4. Define **Averaging strips** through the icon in the **Concrete window** at the locations with high potential stress concentrations such as points **D3**, **C4** and **B4**.



5. Choose **36in** for the **Width** and **Length** and **45°** for the **Angle**. Confirm by clicking **[OK]**.

-

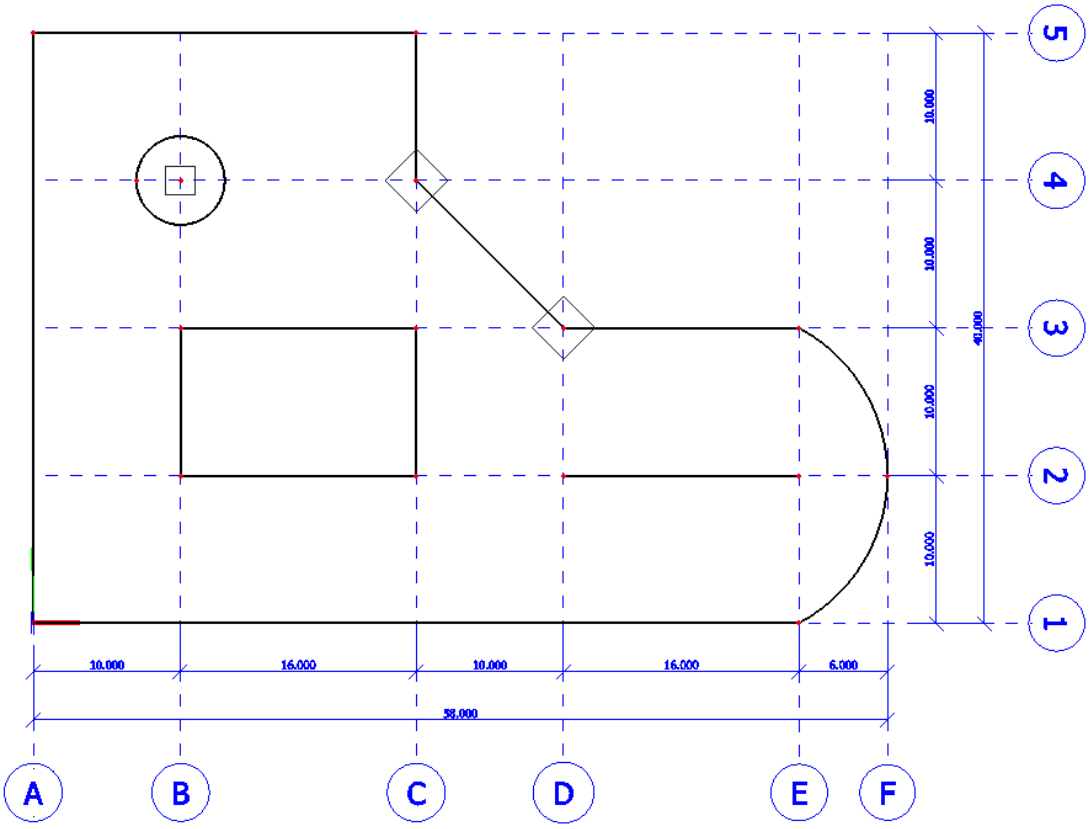
6. Select points **D3** and then **C4** by clicking on these points accordingly. Finalize the selection by **[ESC]**.

Enter point: D3 <click>
C4 <click>

-

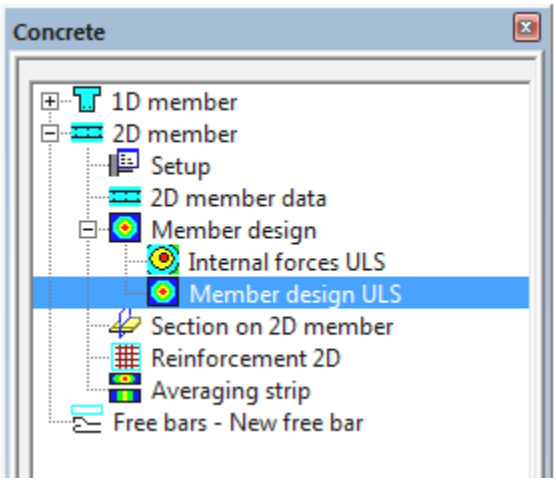
7. Similarly define a **24in x 24in** point at **B4** with a **0°** angle.

8. The layout of the slab will be as follows while showing the averaging strips.

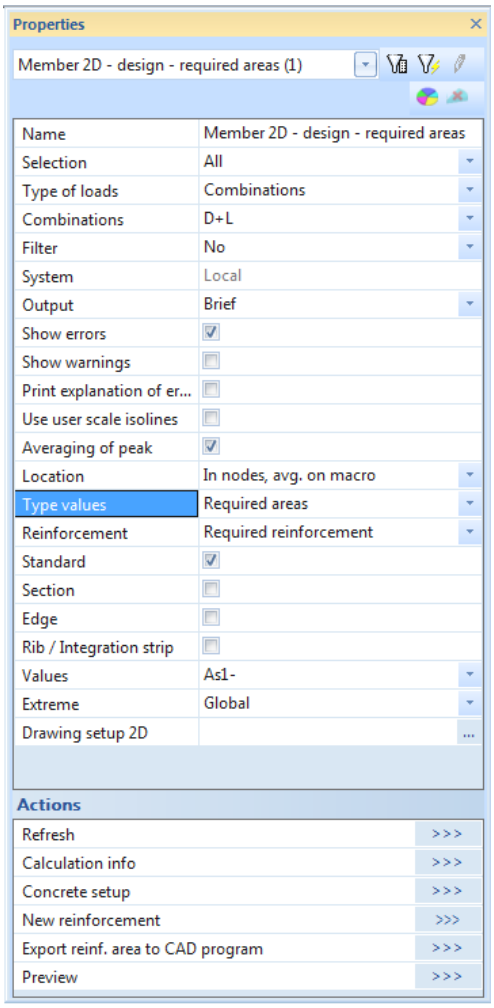


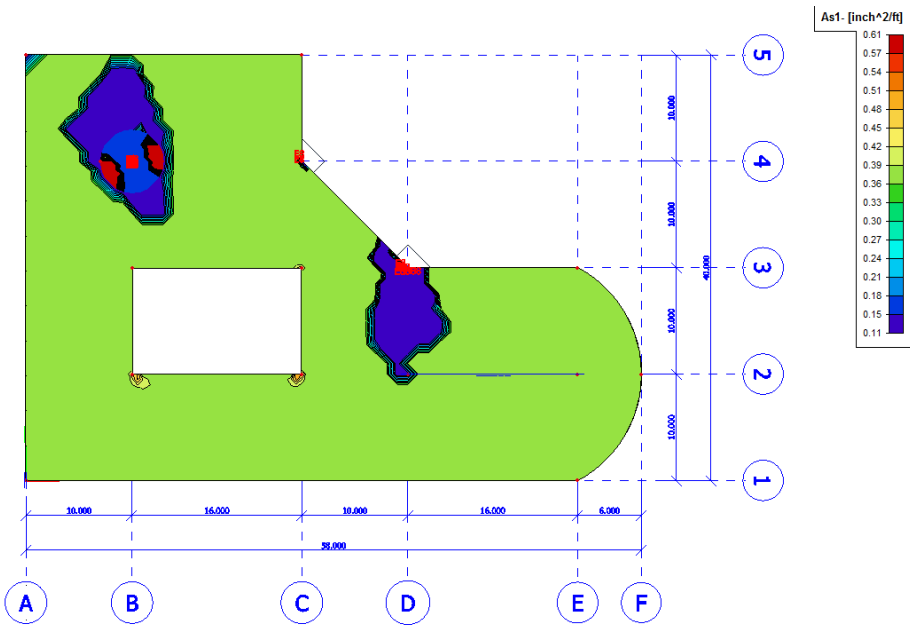
Required areas

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



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



- 

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

As-1 = Lower reinforcement in x-direction

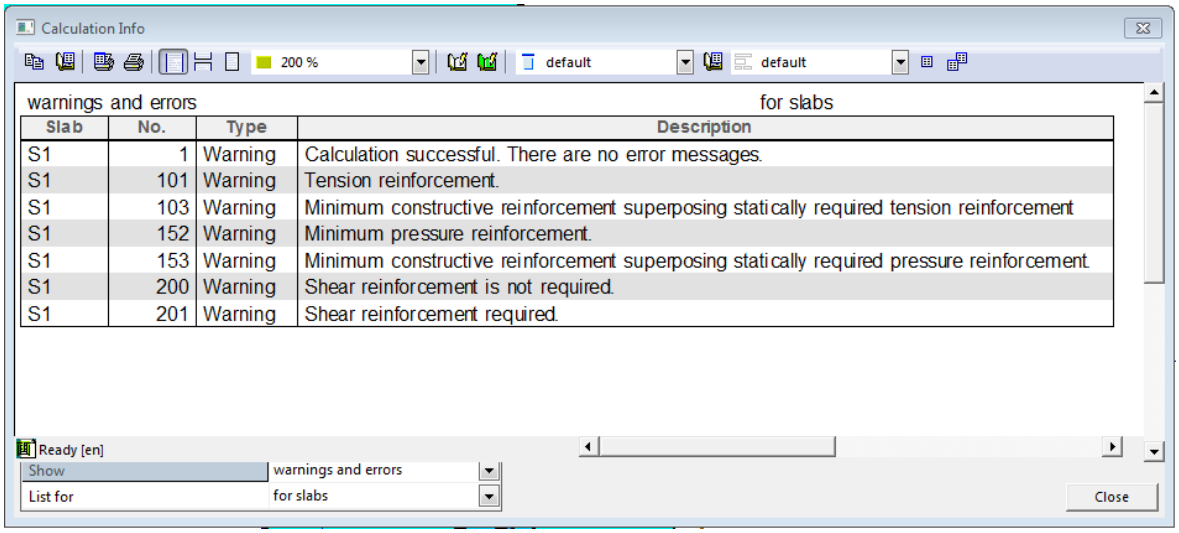
As-2 = Lower reinforcement in y-direction

As+1 = Upper reinforcement in x-direction

As+2 = Upper reinforcement in y-direction

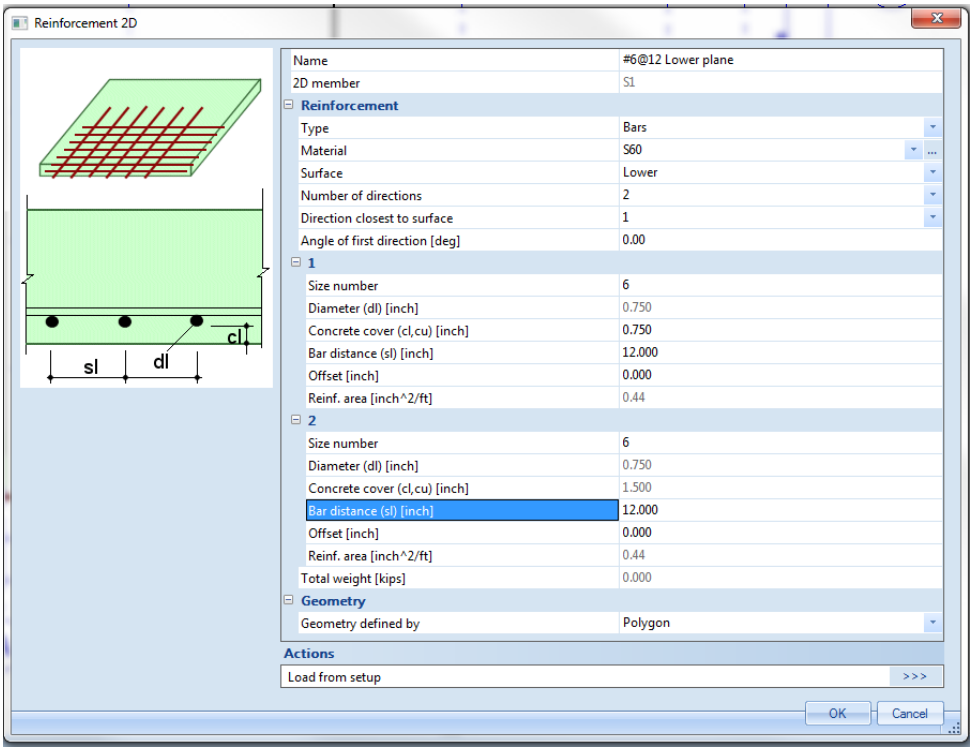
Asw = Shear reinforcement

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



Input user reinforcement

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



2. Choose a name for reinforcement, e.g. **#6@12 Lower** for defining reinforcement with size #6 at a center to center distance of 12” at the lower level of the slab. Choose **Lower** for **Surface**. For **directions 1 & 2** choose **6** for **Size number** and **12** for the **Bar distance** and then click **[OK]**.

- 

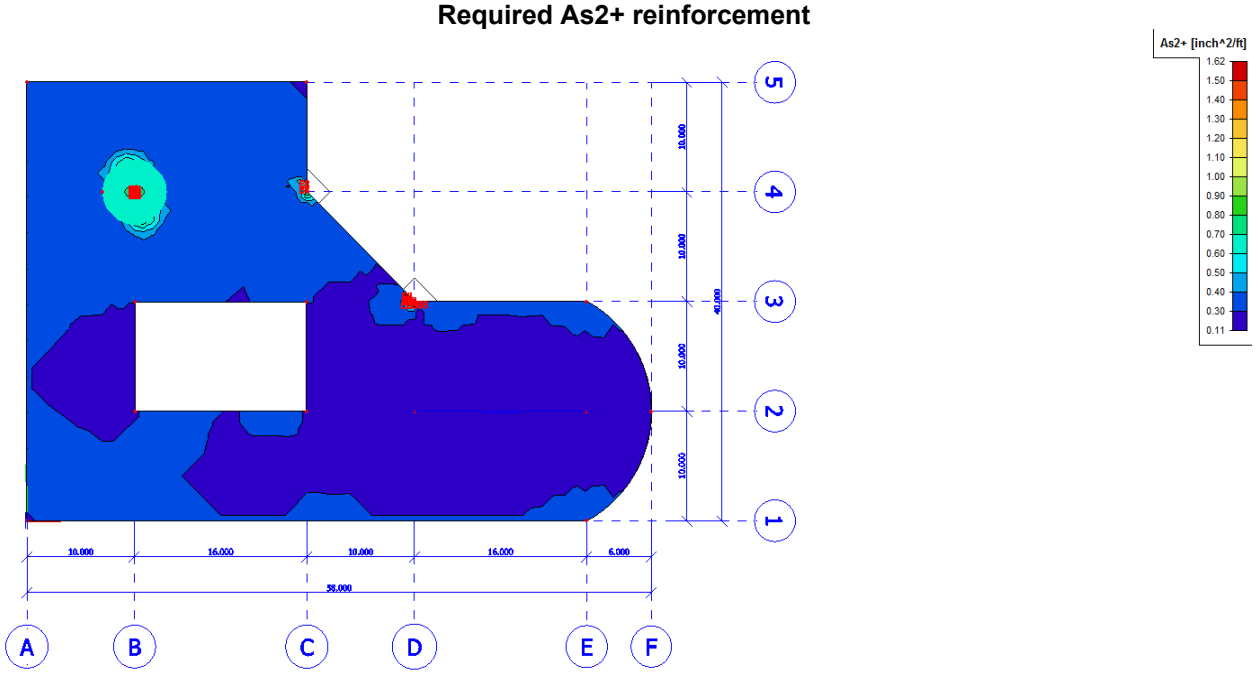
3. Select **Select existing polygon/polyline** above the **Command line** and then click on one edge of the concrete slab

-

4. In a similar fashion, create reinforcement for the upper level. Name this reinforcement e.g. **#6@12 Upper** and choose **Upper** for **Surface** in Reinforcement 2D. Use the **select existing polygon** to apply the reinforcement to the concrete slab.

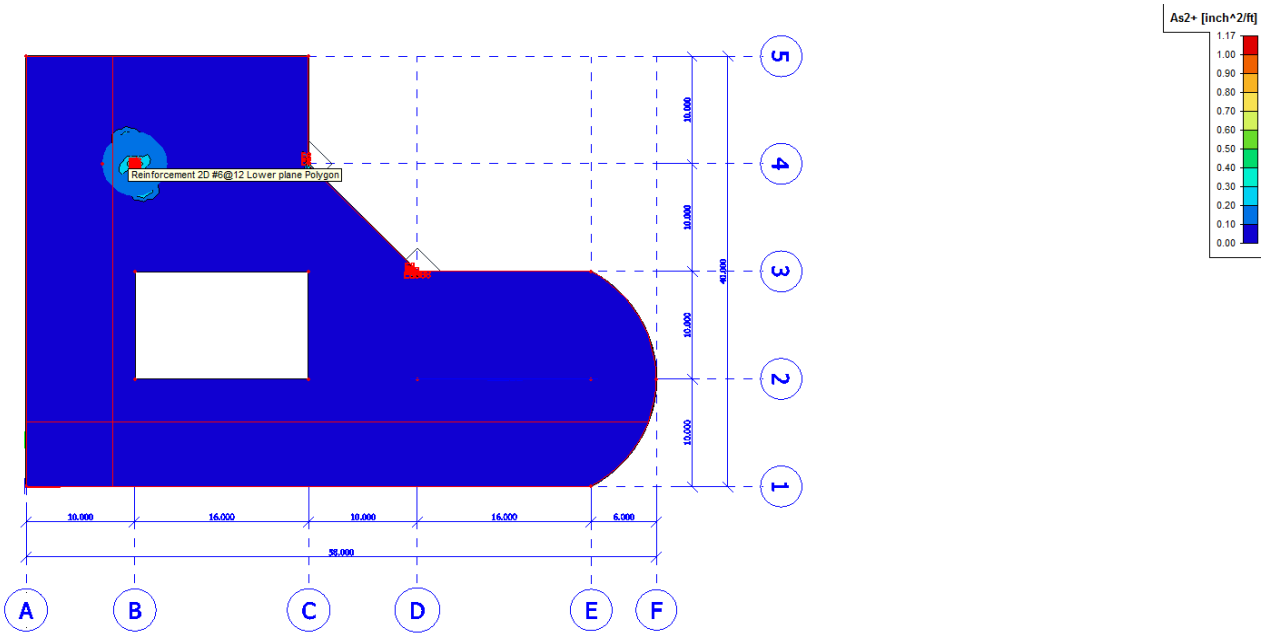
5. Choose **Member design ULS** keep the previously configured input options, except for changing Reinforcement to **Additional reinforcements.** This will allow the user to graphically view the additional required reinforcement at each finite element.

Required As2+ reinforcement



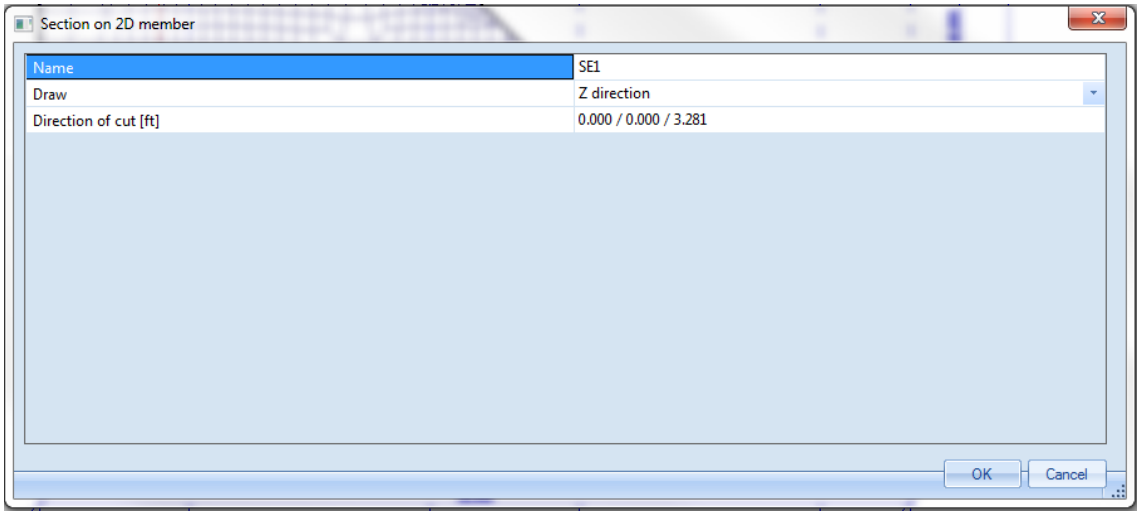
-
-

Additional As2+ reinforcement

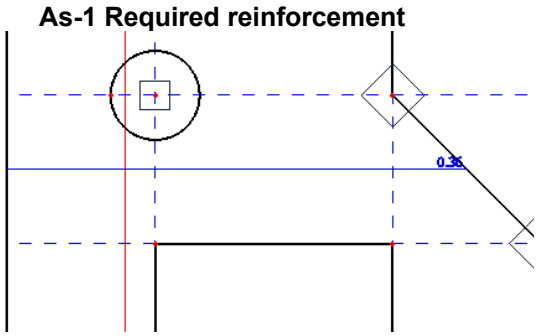


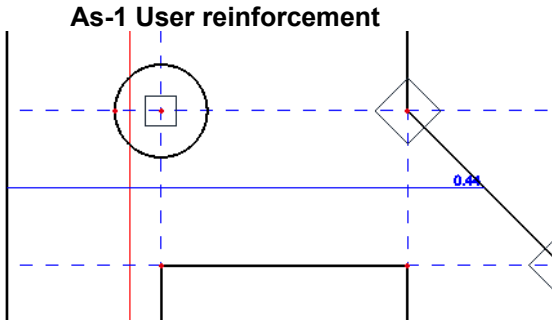
-
-

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

- 

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





-

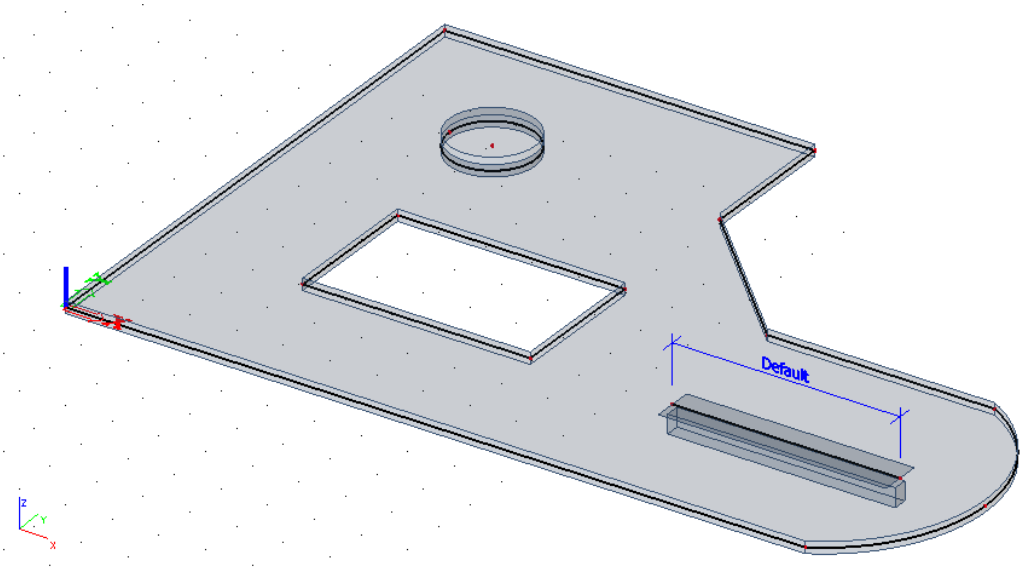
Engineering Report

The final section of the tutorial will focus on how to utilize model information (input, output and graphics) in SCIA Engineer's Engineering Report interface. The Engineering Report is accessed from the Main Service Tree or by the shortcut button  .

Saving Model Views or Results to Engineering Report

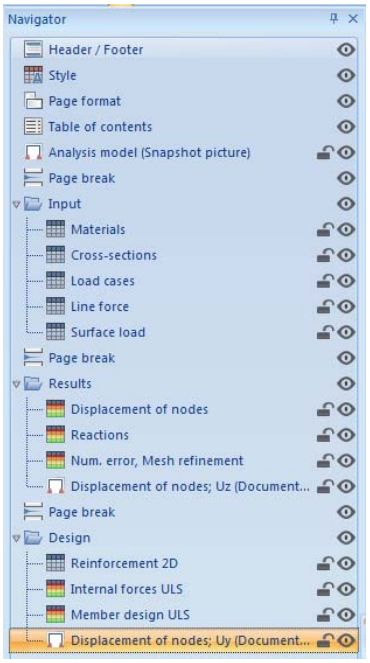
Following actions are accessible via Main menu > Drawing tools, or 'Project' toolbar, or right mouse click in screen

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

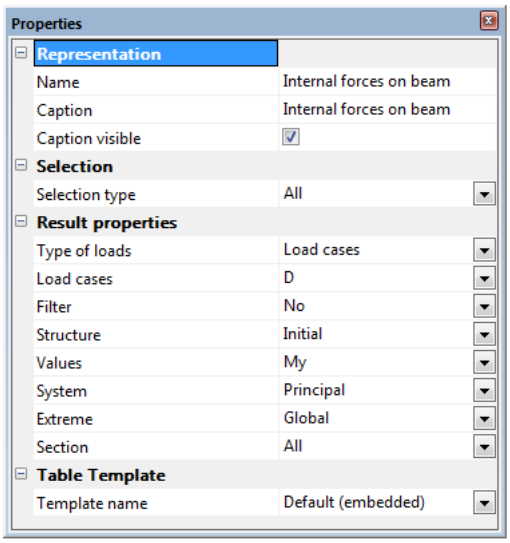


Adding New Items to Engineering Report

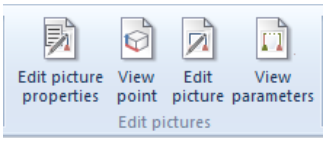
1. Main menu > Engineering Report, or ‘Project’ toolbar 
2. Content of Engineering Report
   - Via “New Items” List and Insert Button 
   - Prepare the Engineering Report as shown in the report navigator



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

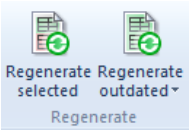


3. Add picture to document
   - Directly via Right Click and Screen Shot into Engineering Report or Live Picture into Engineering Report
   - To make a Live Picture 3D PDF capable, select picture and check “Export to PDF as 3D”
4. Make Changes to Live Pictures
   - Select Live Picture in Navigator and now the Picture Editing buttons become active.
   - Select “Edit Picture Properties” to change size, scale, view point, regeneration settings and result type.



5. Adapt Style of Document
   - Via adding “Style” to Navigator from New Items list
6. Modification of Tables
   - Select specific table in Navigator and click “Edit” button. This enables the user to select and modify specific items in the table, change the table layout and also the size and spacing of the table items.
7. Refresh of Engineering Report

- After adaptations to data or (content of) tables:
  Red Exclamation Point means item regeneration needed.



- Regenerate Selected Items
- Regenerate Outdated Items (Pictures, Tables)

8. The Engineering Report also allows for any information in the “Navigator” to be saved as a report template. These templates are found in the “Report Templates” item in the List of Item.

