



TUTORIAL MODELLING

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 2024 SCIA nv. All rights reserved.

Table of Contents

Table of Contents3	
Input of structural entities – 1D elements4	
Example 1: Frame	4
Example 2: Frame	11
Example 3a: Hall	16
Example 3b: Hall	24
Example 4: Purlins	30
Example 5: Bridge	37
Example 6: Carrousel	44
Example 7: Bearing frame	48
Input of structural entities – 2D elements	
Example 8: Rectangular plate	53
Example 9: Slab on an elastic foundation (subsoil)	58
Example 10: Slab with ribs	66
Example 11: Precast wall	69
Example 12: Balcony	72
Example 13: Tank	75
Example 14: Swimming pool	79
Example 15: Cooling Tower	85
Example 16: Steel hall with concrete plate	90
Example 18: Detailed study of a column base	94

Input of structural entities – 1D elements

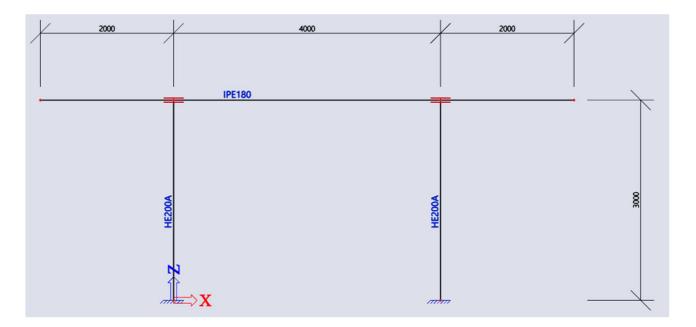
Example 1: Frame

1. Input of geometry

• Project data: Frame XZ – Steel S 235

	DATA		MATERIAL	
-	PAIA			
	Name:	-	Concrete	
	an and		Steel	A second s
	Part:	·	Material	\$ 235 ¥
_			Masonry	
- 1	Description:		- Aluminium	
	Author:		Timber	
	, idd for t		 Steel fibre concrete 	
	Date:	29/01/2024	Other	
E	Structure:	🜓 Frame XZ 🔹 👻	CODE	
			National Code:	
			EC - EN	×
			National annex:	
			Standard EN	<u>ب ب</u>

• Geometry:

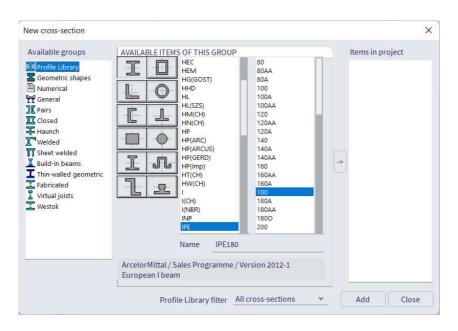


Adding cross-sections:

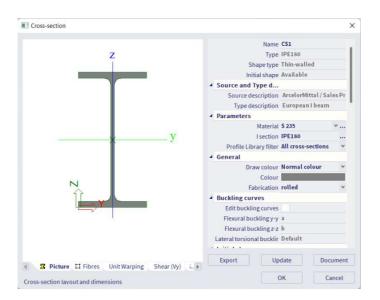
To add cross-sections click on "Library" > "Cross-sections"

	R Pa	
	E Layers	
	🗾 Materials	Ctrl+M
ſ	I Cross sections	Ctrl+J
	🚺 Image gallery	
	Paperspace gallery	
	Load cases, combinations	•
	Loads	•
Î	Structure and analysis	•
Ĩ	Tools	•
	Steel	•
	Subsoil and foundation	•
	Drawing tools	

Add a new cross-section from the library on the left, choose an I-shaped cross-section on the right for the beams:



When you click on 'Add' a new window opens where you can change some parameters of the cross-section:



Use the same approach to add a cross-section for the columns, HE200A for this example.

• Inputting elements:

You can narrow down the content of the Input Panel by setting the category filter to '1D Members'. There are 3 options to define a 1D element:

💼 All workstations 🗸
🥔 All tags 🗸
Ctrl+B
Ctrl+Shift+B

Choose 'Column' from the input panel and change the properties in the pop-up window like shown below:

III Column		
Nar	ne B1	
ez Ty	pe column (100)	¥
Analysis mor	lel Standard	~
Cross-section	on CS2 - HE200A	۷
Alp	na O	Y
Member system-line	at Centre	Y
(J) ez [m/	n] 0,00	
L L	CS standard	*
FEM ty	pe standard	*
Lay	er Layer1	۰
✓ Buckling		
System lengths and buckling settin	gs Default	
Secondary memb	er	
(i) Geometry		
Length [r	n] 3,000	
Insertion po	nt Bottom	*
Structural model		
		OK Cancel

Click 'OK' and draw the left column by entering '0 0' and clicking ENTER to end the action. You can now click ESC to close the action or enter another coordinate: enter '3 0' for the right column, click enter and close the action with ESC. Note that coordinates are separated with a SPACE in between them. Because we are in a 2D environment, we only need to fill in 2 coordinates.

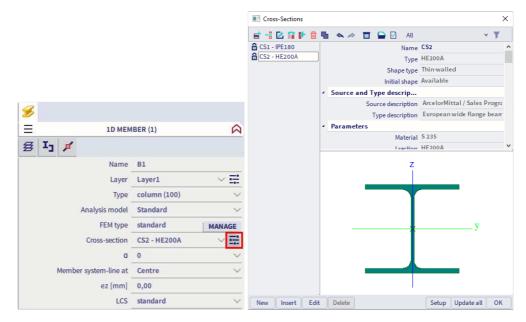
For the beams, choose 'Beam' from the Input Panel and change the properties as shown below:

III Horizontal beam			×
	Name	B3	
~	Туре	beam (80)	*
α	Analysis model	Standard	*
1/1	Cross-section	CS1 - IPE180	×
A z'	Alpha	0	¥
	Member system-line at	Centre	*
	ez [mm]	0	
ez	LCS	standard	*
(i)	FEM type	standard	*
~	Layer	Layer1	×
1	▲ Buckling		
	System lengths and buckling settings	Default	
	Secondary member		
	Geometry		
	Length [m]	8,000	
	Insertion point	and the second se	v
	Structural model		
			OK Cancel

Click 'OK' to start modelling. The beam is entered from the beginning (left side) so again enter coordinates first '-2,5 3' and ESC to close the action.

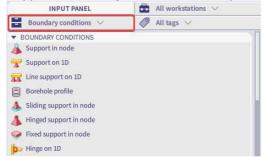
• Modifying geometry:

After you model a certain element, the properties can always be modified from the property panel. For example if you want to change the cross-section of an element: select the element and click on the 'Manage' icon next to the cross-section property. From there you can access the cross-section editor gain by clicking on 'Edit.



• Add supports:

Supports can easily be found in the Input Panel by filtering the category to 'Boundary conditions':



Enter a fixed support on the bottom of the columns by choosing 'Support in node' from the Input Panel and changing the properties as shown below:

Support in node			>
	Name	Sn1	
	Туре	Standard	~
	Angle [deg]		
/	Constraint	Fixed	*
7 /	x	Rigid	~
	Z	Rigid	¥
X Ry	Ry	Rigid	*
(i)	Default size [m]	0,200	
O	▲ Geometry		
俞	System	GCS	*
			OK Cancel

Simply select the nodes where you want these supports to be applied to and end action with ESC.

Note: whenever you choose a certain action, the SCIA Spotlight will tell you what the next step should be:

New support - Select node >

1. Changing the view

Navigate:

The are two ways to navigate around the model in SCIA:

- By using the mouse and keyboard

CRTL + right mouse button = rotate. SHIFT + right mouse button = pan. Click mouse wheel = pan. Roll mouse wheel = zoom.

Note: if you want to rotate around a certain element you first have to select it.

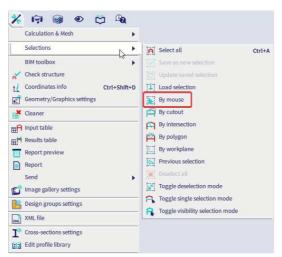
- By using the Navicube



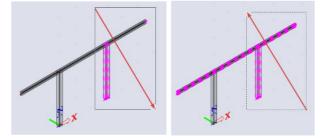
You can click on all sides of the Navicube to show an orthogonal view of your model.

Selecting objects:

There are different ways to select an object, by default it is set to 'By mouse' if you go to 'Tools' > 'Selections':



If you click and drag the mouse depending on the direction you will select elements that fall inside of the box partly or completely:



Left to right: elements must fall inside the box completely to be selected. Right to left: elements that fall inside the box partly will be selected. You can also search for any object if you know the name of it. Type 'sel' in the SCIA Spotlight followed by a SPACE and the name of the object you wish to search:



The element will then be selected. If the element has a SPACE in the name, you'll have to add brackets ("") around the name when you use the 'sel' command.

2. Actions after the input of the geometry.

We recommend to execute two actions before you start to run the calculation:

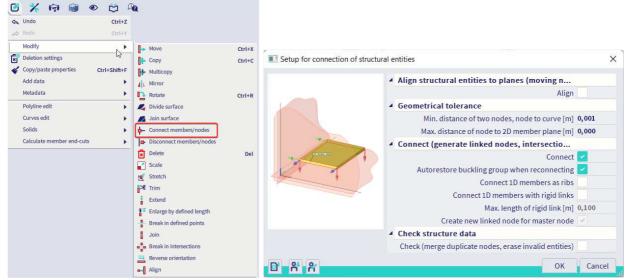
• Tools > Check structure

This checks your structure for double nodes, double elements and deletes them. It also checks if there are any incorrect entities defined both in elements and additional data:

00LS 25.54	Check of structure data				>
💥 🗑 🞯 👁 🖻 🖓	CHECK OF NODES				
Ealculation & Mesh	Search nodes				0%
Selections	Search duplicate nodes		Ignore paran	neters	0%
BIM toolbox	CHECK OF MEMBERS				
Check structure	Check members Search null members	0%			
Geometry/Graphics settings	Search duplicate members	0%			
Cleaner				0 f parts	
A Input table	CHECK OF DATA REFERENCES				
E Results table	Check data references	0%	 Memory efficities Fast method 	ent method	
Report preview	CHECK OF ADDITIONAL DATA				
Report	Check additional data posi	tion 0%			
Send Image gallery settings	Check free load distributio			0	
Design groups settings	CHECK OF STEEL CONNECTION	15			
XML file	Check steel connections	0%		0 d connections	
Cross-sections settings	Check load panels	Check cross-l	inks		
Edit profile library	Check additional data	Check duplicity o	fnames	Check	Cancel

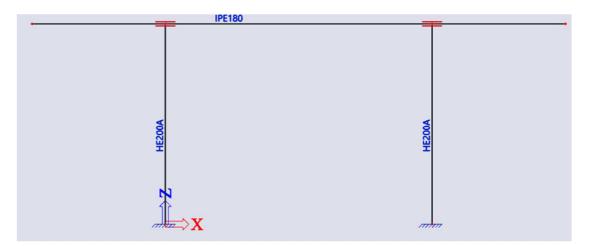
• Edit > Modify > **Connect members/nodes**

This allows Scia to search for contacts between two intersecting elements and adds a rigid connection in these locations:



Note : you can either do this for a number of selected nodes or all nodes in the model when nothing was selected before choosing this action.

In this specific example there were two connections made between the columns and the beam in the upper nodes of the column. These are marked with two parallel red lines so that you know there is a connection:



Elements which share a node are automatically connected.

Note: by default an option is activated in the solver settings to always connect nodes before the calculation. So there's no need to manually connect nodes, but you can to check if everything was modeled properly before calculating:

E analysis		×
Calculations	Mesh setup	
Linear analysis Load cases: 1 Other processes	Average number of 1D mesh element 1 Average size of 1D mesh element on < 0,200 Average size of 2D mesh element [m] 0,500 Connect members/nodes	
Test input of data	Setup for connection of structural en Advanced mesh settings	••
Save project after analysis	✓ Solver setup	
	Specify load cases for linear calculati	
	Advanced solver settings	

Example 2: Frame

- 1. Input of geometry
- Project data: Frame XZ Steel S 235:

	Project data			×
	Basic data Fi	unctionality Actions	Unit Set Protection	
		DATA		MATERIAL
	de la	Name:	·	Concrete Steel
		Part:	·	Material \$ 235 v
		Description:	-	Masonry
	E A	Author:	<u>.</u>	Timber
		Date:	29/01/2024	Other
		Structure:	🖞 Frame XZ 🔹 👻	CODE National Code: EC - EN ¥ National annex: Standard EN ¥
Geom	etry:		×	OK Cancel
		IPE160		<u>IPE160</u>
				VOO
	HE200A			Ē
	X C			E

• Adding cross-sections:

To add cross-sections click on "Library" > "Cross-sections". For this example we are using I-shaped cross-sections, you can find them in the library under this icon:

E

We are going to need two type of cross-sections: IPE160 for the beams, HE200A for the columns.

• Inputting elements:

First approach:

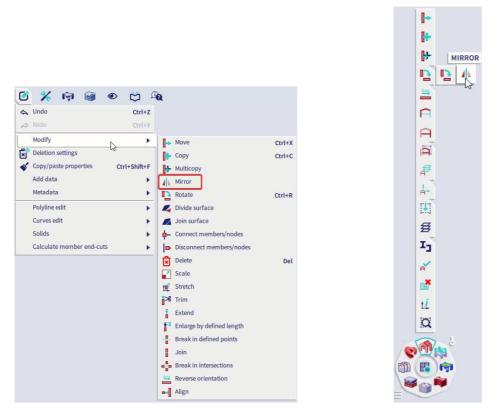
-Left side of the frame: in the Input Panel choose for 1D member under '1D members'.

INPUT PANEL All workstations ∨ All categories ∨ All tags ∨ Collos & STOREYS 3 Do line grid D MEMBERS Column Ctri+B Column Ctri+Shift+B Haurch on 1D Arbitary profile Copening on 1D

Choose the HE200A cross-section for the columns, click 'OK', enter the coordinates '0 0' followed by ENTER, enter the coordinates '0 5' followed by ESC to end the action.

Repeat the above steps for the beams with cross-section IPE 160 and coordinates '0 5' and '5 6,5'.

-Next use the mirror option under Edit > Modify or search it under the structure branch of the Process Toolbar:



After activating the mirror function follow the next steps, which are also shown in the SCIA Spotlight, and mirror the left side of the frame to the right:

- 1- Select the elements you want to mirror, end with ESC.
- 2- Choose the first node of the mirror plane.
- 3- Choose the second node of the mirror plane (tip: enable tracking guides under the snap settings).
- 4- Choose 'No' in the next pop-up window to keep the left side of the frame as well.

Second approach:

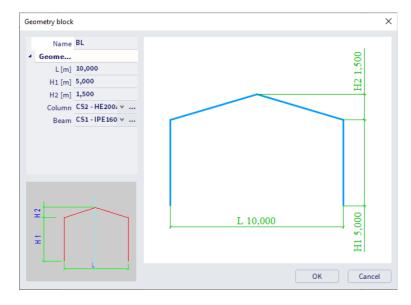
There are also some predefined blocks available in Scia which you can use to quickly model a structure. In the Input Panel set the category filter to 'Import & Blocks' and choose for 'Catalog blocks':

INPUT PANEL	\blacksquare All workstations \lor
📑 Import & Blocks 🖂	🥔 All tags 🗸
IMPORT & BLOCKS	
Catalog blocks	
User blocks	
Predefined shapes	
Import project from esa file	
Import DWG, DXF, VRML97	

Under 'Frame 2D' choose the upper left option which allows you to design a symmetrical frame based on the dimensions:

r						
Block selection manager						×
Available groups	AVAILABLE IT	EMS OF THIS (GROUP		Items in pr	oject
Curve Frame 2D Beam Tower2D 22 Truss 2D 22 Truss 2D Arc			\bigcap			
Frame						
			ilter	v	ОК	Close

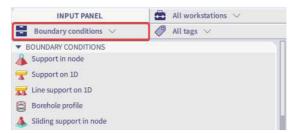
In the next window you can fill in the dimensions we showed you in the beginning of this exercise and choose a cross-section for both columns and beams:



After you click 'OK' and 'OK' once more the only thing left to do is define an insertion point so enter the coordinates '0 0'.

Add supports:

Filter the category of the Input Panel to 'Boundary conditions' and choose for 'Support in node':



Change the properties in the next window as shown below and select the nodes where you want to apply the support to, bottom two nodes for this example.

	Name	Sn1		
		Standard		
	Angle [deg]			
	Constraint	Fixed		
7/	x	Rigid		
	Z	Rigid		
X Ry	Ry	Rigid		
(i)	Default size [m]	0,200		
\sim	▲ Geometry			
Î	System	GCS		
			ОК	Cancel

2. Manipulations

• Moving nodes

You have a number of option to move a node, first select one and after:

- Drag the node with the left mouse button.
- Change the coordinates in the property panel:

	NOD	E (1)	2
6			
	Name	N5	
 GCS COORDINATE 	_		
	X [m]	10,000	
	Z [m]	5,000	
 UCS COORDINATE 			
	ux [m]	10,000	
	uz [m]	5,000	
 MEMBERS 			
	Member	B3	
	Member	84	

Note that if you select multiple elements there is an icon for every element in the selection on top of the property panel. In the current selection there are both members and nodes.

- Move it with the below option from the Process Toolbar:



- Use a hotkey (CTRL+X by default for moving). Hotkeys can be assigned to any command in the SCIA Spotlight:

Move		
l → Move	Ctrl+X	

3. Actions after the input of the geometry

We recommend to execute two actions before you start to run the calculation:

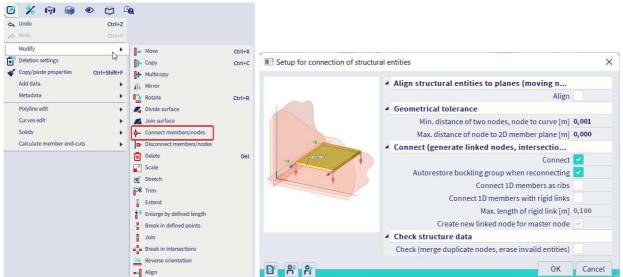
• Tools > Check structure

This checks your structure for double nodes, double elements and deletes them. It also checks if there are any incorrect entities defined both in elements and additional data:

100LS 26.64	Check of structure data				
🗡 河 🎯 👁 🗂 🕰	CHECK OF NODES				
Ealculation & Mesh	Search nodes				0
Selections	Search duplicate nodes	Ignore parameters			
BIM toolbox	CHECK OF MEMBERS				
Check structure	Check members Search null members	0%		0	
Coordinates info Ctrl+Shift+D Geometry/Graphics settings	Search duplicate members	0%		0	
Cleaner				0 d parts	
A Input table	CHECK OF DATA REFERENCES				
There are an a second sec	Check data references		Memory effic	ient method	
		0%	Fast method		
Report preview	CHECK OF ADDITIONAL DATA				
Report	Check additional data posit				
Send		0%			
🚰 Image gallery settings	Check free load distribution	n points 0%		0	
Contraction and the settings	CHECK OF STEEL CONNECTION	S .		_	
XML file	Check steel connections	0%		0 d connections	
▲ Cross-sections settings	Check load panels	Check cross-li	inks		
Edit profile library	Check additional data	Check duplicity of	names	Check	Cancel

• Edit > Modify > Connect members/nodes

This allows Scia to search for contacts between two intersecting elements and adds a rigid connection in these locations:



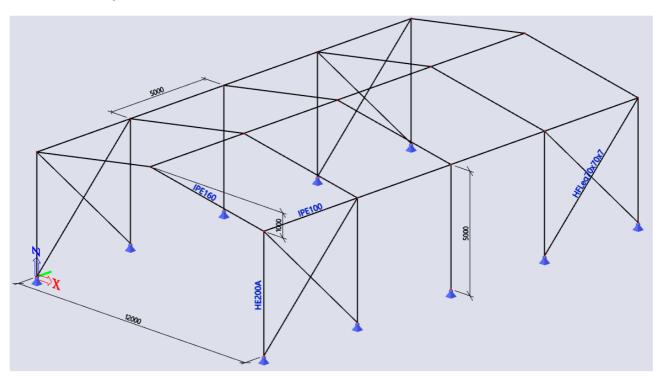
Note: you can either do this for a number of selected nodes or all nodes in the model when nothing was selected before choosing this action. For this specific example no additional connections were made because all 1D elements share a node with the adjacent member.

Example 3a: Hall

• Project data: Frame XYZ – Steel S 235

		Unit Set Protection	
	DATA		MATERIAL
1	Name: Part:		Concrete Steel Material \$235
1	Description:	ā	Aluminium
	Author: Date:	- 29/01/2024	Timber Steel fibre concrete Other
	Structure:	🦨 Frame XYZ 🔹 👻	CODE National Code:
			National annex:

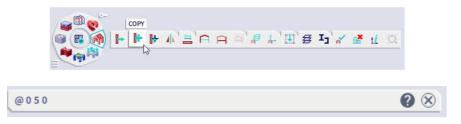
• Geometry:



1. Input of the geometry

- For the first frame you can use the steps as described in **example 2.**
- **Copy** the first frame using one of the below approaches:

-Copy: choose the below option from the Process Toolbar, select the first frame, click on a start point and type @ followed by the coordinates of location where you want to copy the frame to:



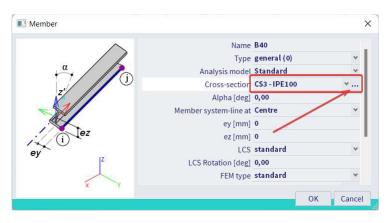
Repeat the above to make 5 copies of the first frame in total.

You can also use the default hotkey CTRL+C to copy. If you want to copy or move in a certain direction you can also use the tracking guides in the snap settings. Aim in a certain direction and simply add a value followed by ENTER.

-Multicopy: choose the below option from the Process Toolbar, select the first frame and fill in the below properties (make sure you don't add the lower nodes in the selection):

		'A A	¤' ₽ ↓'⊞	₿ Ij	' 🖌 📑	tĹ	α
Multicopy					×		
Number of	copies 5	+	Connect selected with new beams				
Insert th	e very last co	ру	Copy additional	data			
DISTANCEV	ECTOR	_	HOW TO DEFINE	THE DISTA	NCE ?		
Define dista	ance by curso	or 🗌	• between two	o copies			
x	0,000	m	from origina	l to the las	st copy		
v	5	m	HOW TO DEFINE	THE ROTAT	FION ?		
	0,000		 between two 	o copies			
Z	0,000	m	from origina		st copy		
ROTATION			ROTATION AROU	ND			
rx	0,00	deg	current UCS				
ry	0,00	deg	distance vec	tor			
rz	0,00	deg	ОК	Cance	el		

Because you enabled the upper right option right after you click 'OK' a new window appears to choose a cross-section which is going to be applied to the additional members. Click on the manage option next to the cross-section property and add another IPE 100 for the purlins.



• Add supports:

We are going to select all the nodes with Z=0 to add supports on them. There are a few ways to do this quickly:

-Click on one side of the Navicube or type 'view -y' in the SCIA Spotlight

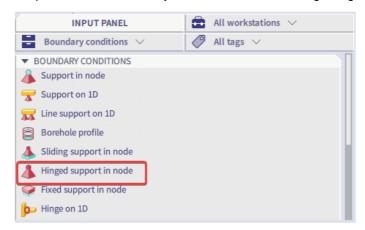
😋 View -Y								63	0		0						
View -Y																0	\otimes
													/				
									1		3	8		1	b .		
													6				
	ų.	=>	Х								4			12			
														₽	۲.		
														Ŧ	-		
	-	_	-	_					-	-	_				2		
				10	_	-	_							F			
														F			

You now have a side view of the structure which makes it easy to drag from left to right to only select the nodes on the bottom.

-Select just one node with a z-coordinate equal to '0,00'. In the property panel you can then right click on the coordinate and expand selection to all nodes with the same z-coordinate:

	#			
	Ξ	NOD	E (1)	
	3			
		Name	N4	
	▼ GCS COORDINATE			
		X [m]	10,000	
		Y [m]	0,000	
		Z [m]	0,000	
Isola	te selection		ß	
+ Expa	nd selection		10,000	
Subt	ract from selection		0,000	
		uz [m]	0,000	
	▼ MEMBERS			
		Member	B3	

When you used one of the above methods to select the correct node you can add a hinged support by filtering the category of the Input Panel to 'Boundary conditions' and choosing 'Hinged support in node':



2. Actions after the input of the geometry

• Tools > Check structure

• Edit > Modify > Connect members/nodes Or you can simply type "Connect members/nodes" in the SCIA Spotlight. Note that you can do this action both for a selection or the whole structure (select nothing to do the latter).

Note: connecting the elements is done by default when you perform a calculation:

FE analysis		×
Calculations	Mesh setup	
Linear analysis Load cases: 1	Average number of 1D mesh elements (1 Average size of 1D mesh element on cu 0,200 Average size of 2D mesh element [m] 0,500	
Other processes	Connect members/nodes	
Engineering report regeneration Engineering reports: 1	Setup for connection of structural entit	
Save project after analysis	Advanced mesh settings	
ouve project arter analysis	▲ Solver setup	
	Specify load cases for linear calculation	
	Specify combinations for linear stability	
	Specify combinations for nonlinear sta	
	Advanced solver settings	
	 Engineering report 	
	Specify reports for regeneration	
Calculate		

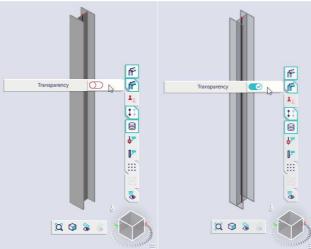
3. Structural model

The structural model is used to make a better representation of the analytical model. It is also necessary when you input steel connections, reinforcement anchoring etc.

• You can visualize this representation by going to view > visualization > structural model. You also need a specific module in your license for this and it should be enabled in the project data:

Project data				×
Basic data Functionality Actions	Unit Set Protection			
GENERAL		D	ETAILED	
and the second sec	Property modifiers	4	Nonlinearity	
115	Model modifiers		Beam local nonlinearity 🔽	
	Parametric input		Support nonlinearity/basic soil spr 🔽	
	Climatic loads 📝		Initial imperfections	
	Mobile loads		Geometrical nonlinearity	
	Dynamics		General plasticity	
	Stability 🔽		Cables	
	Nonlinearity 🗾		Friction support/Soil spring	
	Structural model 💟		Subsoil	
	IFC properties		Pad foundation check	
	Prestressing		Steel	
	Bridge design		Plastic hinge analysis	
Constant of	Construction stages		Fire resistance checks	

• We provided shortcuts in the view toolbar to show both volumes and rendering (with possible transparency):



• You can switch back to the representation of the analytical model by going to: View > visualization > analysis model.

💌 🛱 🚇	
lundo view change	
💰 Redo view change	
Visibility	•
Zoom	•
Views	•
Clippingbox	•
Visualization	Analysis model - volumes
🖺 Global UI settings	Analysis model - axes
User configuration	Structural model
👠 Colours & lines	Generate structural model
A [®] Fonts settings	
Structural beam types settings	
Dimension lines settings	
June grid manager Ctrl+S	ihift+G
Dot grid settings	
G Wired model in view manipulations	

Attention: everything that is set in the structural model is not taken into account in the calculation. Therefor we provided a separated part in the property panel for these kind of settings:

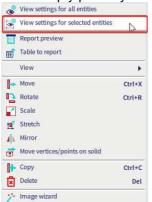
	Mode	Automatic \checkmark
	Priority definition	according to member \sim
	Priority value	100
114	Perp. alignment	default 🗸
	Eccentricity def.	whole member \vee
1014	BEGIN POINT	
	Eccentricity y [mm]	0
	Eccentricity z [mm]	0
	END-CUTS	
A.	x-gap begin [mm]	0
	x-gap end [mm]	0

4. Display of screen

• View settings can be changed for all elements in a project by right clicking on an empty part of your screen and choosing:



• View settings can be changed for a selection of elements by selecting those elements, right clicking on an empty part of your screen and choosing:



The above settings allow you to change the color by cross-section to give you a nice overview of which elements have a different cross-section:

Check / Uncheck gro	- M 🔗 🛛	Lock position
Check / Uncheck all		
- Service		
Display on opening the service		
- Structure		
Style + colour	colour by c	ross-section 🔹
Draw member system line	 	
Member system line style	system line	•
Model type	analysis mo	odel 🗾 👻
Display both models		
Member surface		
Rendering	transparent	t 🗸
Draw cross-section		

5. Activity and visibility

• Activity by layers

Layers can be added from the layer manager if you go to libraries > layers:

🖻 📲 🗹 🕩 🛢 ۹	🖬 🔟 All	• Y
Columns	yes	Name Beams
Beams	yes	Comment
Bracings	yes	Colour
Panels	no	Structural model only no
		Current used activity 🔽 yes

For each layer there are two options available:

- « Structural model only » : if this option is enabled for a certain layer, none of the elements in this layer will be taken into account in the calculation.
- « Current used activity » : if this option is disabled, the elements in this layer are hidden from the display but still present in the calculation.

For the last option to have effect, the visibility by layer should be active from view > visibility > by layers.

• Activity by selection

Apart from using the layers to show certain elements in your model you can also switch to activity by selection. Simply select the elements you want to show/hide and choose for one of the below options in the process toolbar:

ŀ +
8+
₽
<u>P</u>
Ē
HIDE UNSELECTED
AAA
A-
H
番
I]
A
1 ¹
a
🞯 🖪 🛤 🖽 🖽 🚛 🚛 🎼

The toggle switch and the draw invisible members icon can be used to show all elements again or to draw elements in a light color when they're hidden:

• Colors, lines and fonts

Besides what you see you can also change about every color and line type of what you see in the graphical displace by going to view > colors & lines/Font settings:

VIEW		
•	🛱 🕰	
S	Undo view change	
8	Redo view change	
	Visibility	•
	Zoom	•
	Views	•
	Clippingbox	
	Visualization	•
	Global UI settings	
	User configuration	•
٠.	Colours & lines	
A	Fonts settings	
-	Structural beam types settings	
<mark>بھر</mark>	Dimension lines settings	
÷	Line grid manager Ct	rl+Shift+G
0	Dot grid settings	
6	Wired model in view manipulatio	ns

6. Saving a file

You can save a file by going to file > save/save as.

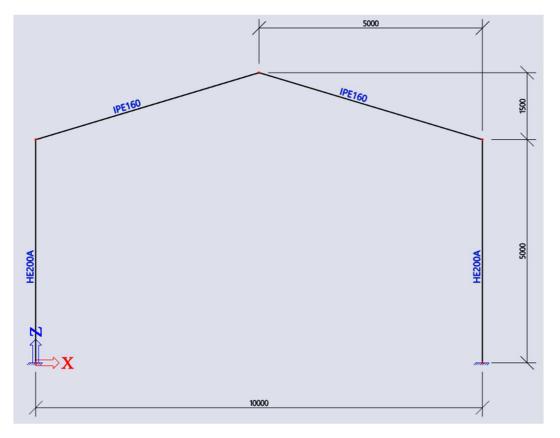
If you close the file a pop-up window will always be shown to ask if you want to save before closing. In this window you have the option to delete mesh and results of both calculation and cross-section properties:

Ask to save			×
Clean mesh, results of e	calculation and	2D FEM cro	oss-section
Save changes to C:/Users/martijn/Downlo	ads/ehethethe	th.esa?	
	Yes	No	Cancel

This will significantly reduce the file size, but the project needs to be recalculated after it was saved with this option enabled.

Example 3b: Hall

- 1. Input of the geometry
- Project data : Frame XYZ Steel S 235



• Insert the first frame of the hall as shown in example 2.

2. Input table

The table input allows you to numerically introduce or edit project data. Numerical data can also be handled simply by a Copy/Paste from SCIA Engineer into Excel and vice versa. You can show the table input by going to Tools > Input Table

%	i 🗑 🕘 💿		₽ <mark>@</mark>
	Calculation & Mesh		•
	Selections		•
粹	Explode line grid		
	BIM toolbox		•
ep.	Clash check		
A	Check structure		
11	Coordinates info	Ctr	l+Shift+D
	Geometry/Graphics settings		
*	Cleaner		
⊞A	Input table		
EM	Results table	3	
	Report preview		
	Report		
	Send		
Ľ	Image gallery settings		

4	Name	Туре	Beg. n	End node	Cross-section		Lengt	Layer			LCS Ro	Memi
L	S1	column (100) 🗸	KI	К2	CS1 - HE200A 🗸 🖓	H	5,000	Laag1	V	ΞĦ	0,00	Centr
2	S2	beam (80) 🗸	К2	КЗ	CS2 - IPE160	H	5,220	Laag1	V	퍮	0,00	Centr
3	S3	beam (80) 🗸	К4	КЗ	CS2 - IPE160 × 2	#	5,220	Laag1	\sim	Ξ÷	0,00	Centr
1	54	column (100) 🗸	К5	К4	CS1 - HE200A 🗸 🗸	H	5,000	Laag1	\sim	Ħ	0,00	Centr

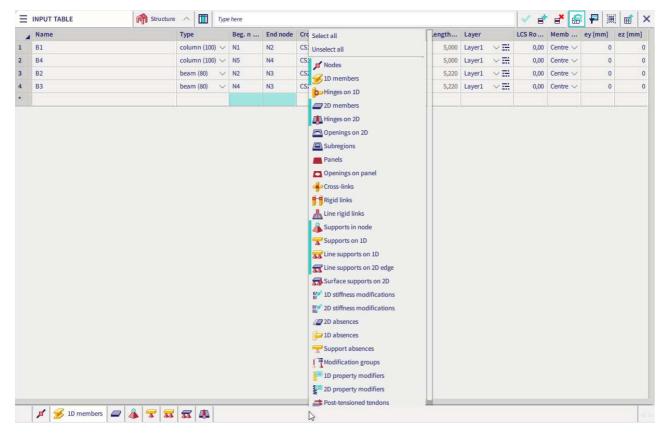
This table contains all different elements in the project with their properties in the columns. You can edit the properties from the input panel by simply typing a new value or string in the cell of the property you want to change. You can also create new elements from the table input by copying the row of an existing element.

Note:

At the top of the table input you can change between a matrix of the structural elements, the loads and the library items:

41	Name	Туре	Beg	Structure	oss-section		Lengt	Layer	LCS Ro	Mem
	S1	column (100) 🗸	К1	Construction sta Loads	51 - HE200A	\vee Ξ	5,000	Laag1 🗸 🎞	0,00	Centr
2	S2	beam (80) 🗸	К2	Libraries	52 - IPE160	~ ==	5,220	Laag1 🗸 🚟	0,00	Centr
3	53	beam (80) 🗸 🗸	K4	КЗ	CS2 - IPE160	~ =	5,220	Laagl 🖂 🖽	0,00	Cent
	S 4	column (100) 🗸	K5	K4	CS1 - HE200A	~ =	5,000	Laag1 🖂 🖽	0,00	Centr

On the bottom of the table input you can find icons for the different type of elements in SCIA. If you miss a certain element type or want to remove one from the table input simply right click on the bottom line and select which items you want to view:



3. Copy with the input table

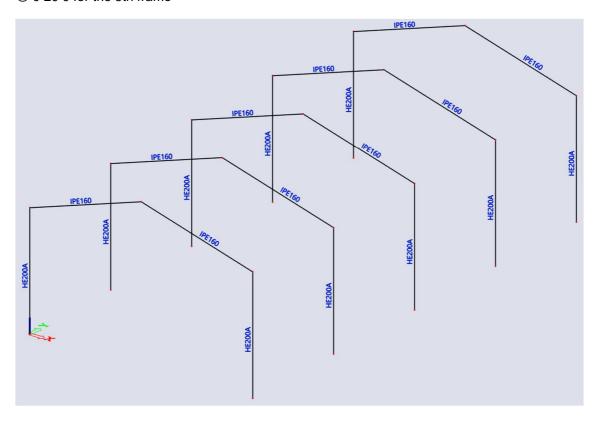
- Select the 4 elements which take part of the frame by clicking on the row number and holding SHIFT
- In the command line enter the vector along which you want to copy the frames: @050
- Click on the copy icon in the table input

With these steps we have created a second frame.

-	Name	Туре	Beg. n	End †	Cross-section		Lengt	Layer	LCS Ro	Mem
1	S1	column (100) 🗸	KI	К2	CS1 - HE200A	\sim Ei	5,000	Laag1 🗸 🚟	0,00	Centr
2	S2	beam (80) 🗸 🗸	К2	КЗ	CS2 - IPE160	\sim II	5,220	Laagl 🗸 🚟	0,00	Centr
3	53	beam (80) 🗸 🗸	K4	КЗ	CS2 - IPE160	~ #	5,220	Laag1 🖂 🖽	0,00	Centr
4	S 4	column (100) 🗸	К5	K4	CS1 - HE200A	\sim IH	5,000	Laag1 🖂 🖽	0,00	Centr
•										

Repeat the above but with steps of 5m for the vector:

- @ 0 10 0 for the 3rd frame @ 0 15 0 for the 4th frame
- @ 0 20 0 for the 5th frame



4. Add model data to the project with the input table

We are now going to add supports on our frame with a little help of the input table

- We are going to sort the table on Z-coordinate by clicking on the column name for this coordinate Note: if you don't see the nodes in the input table you can add it by right clicking on the bottom line

- Select all the cells with the name of the nodes with a Z-coordinate equal to zero.
- Copy the above selected cells
- Paste these cells in a new Excel sheet.

4	Name	X [m]	Y [m]	Z [m] †	Member	2D member	Memb	Memb	Interse	Linked	Coord.
	NI	0,000	0,000	0,000	B1						
	N5	10,000	0,000	0,000	B4						
	NS	10,000	5,000	0,000	B6						
	N10	Solart mint	od momber		_						
	N13	Сору									
	N15	Dente	ß								
	N18	Copy value	to editbox								
	N20		10.000								
	N23	Search									
1000	N25	Copy value	to filter								
r.	N2	0,000	0,000	5,000	B1; B2						

- There should already be 2 supports in your model from example 2. Copy all these cells and again past them in an excel sheet.

	INPUT TA	BLE		1 miles	Structu	re 🔿		Type h	ere			4	=	đ	6	₽	1	Ē	>
	Name	Туре	Con	straint		X		Y	Z	Rx	Ry		Rz	- 0	Node				
(Sn1	Standard 🗸	Fixe	ed	\sim	Rigid	\sim	Rigid 💛	\sim Rigid \sim	Rigid \lor	Rigid	\sim	Rigid	~	N1		Ľ		
ř.	Sn2	Standard 🗸	Fixe	ed	V	Rigid	~	Rigid ~	Rigid \sim	Rigid \lor	Rigid	~	Rigid	~	N5				
Ĭ																			
	11												1			_			
	E		_		1-	1.5.0	122	T T										_	
_				n node	T	35	-												

- Your excel sheet could look something like this:

	Α	В	С	D	E	F	G	Н	1	J	K	L
1	Name		Name	Туре	Constrain	t X	Y	Z	Rx	Ry	Rz	Node
2	N1		Sn1	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N1
3	N5		Sn2	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N5
4	N8											
5	N10											
6	N13											
7	N15											
8	N18											
9	N20											
0	N23											
1	N25											

- Now copy the rows of the cells describing the supports until you have 10 rows in total. Replace the Node names with the node names you also pasted in the sheet:

Name	Name	Туре	Constraint	х	Y	Z	RA.	Ry	Rz	Node
N1	Sn1	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N1
N5	Sn2	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N5
N8	Sn3	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N8
N10	Sn4	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N10
N13	Sn5	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N13
N15	Sn6	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N15
N18	Sn7	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N18
N20	Sn8	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N20
N23	Sn9	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N23
N25	Sn10	Standard	Fixed	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid	N25

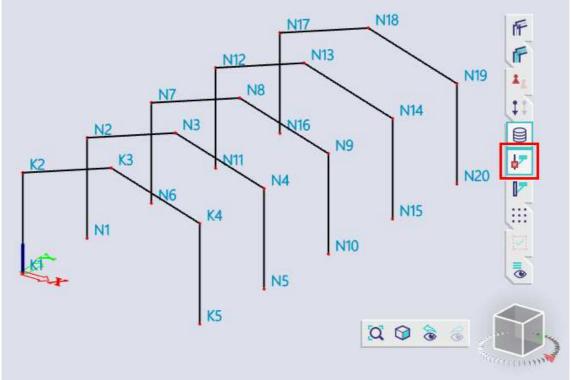
- Copy all the cells with support properties from excel and paste them in the support tab of the input table in Scia:

-	Name	Туре	Constraint	х	Y	Z	Rx	Ry	Rz	Node	
1	Sn1	Standard 🗸 🧹	Fixed 🗸 🗸	Rigid 💛	Rigid 💛	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid \lor	N1	
2	Sn2	Standard 🗸	Fixed 🗸	Rigid 🗸	Rigid \sim	Rigid \lor	Rigid 🗸	Rigid 💛	Rigid 🗸	N5	
3	Sn3	Standard 🗸	Fixed 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	N8	
4	Sn4	Standard 🗸	Fixed \sim	Rigid \sim	Rigid \sim	Rigid 🗸	Rigid 🗸	Rigid 🗸 🧹	Rigid 🗸	N10	
5	Sn5	Standard 🗸	Fixed \sim	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	N13	
5	Sn6	Standard 🗸	Fixed \lor	Rigid 🗸	Rigid 💛	Rigid 🗸	Rigid 🗸	Rigid 💛	Rigid 🗸	N15	
7	Sn7	Standard 🗸	Fixed \sim	Rigid \sim	Rigid $\!$	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	N18	
3	Sn8	Standard 🗸	Fixed 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	N20	
9	Sn9	Standard 🗸	Fixed \sim	Rigid \sim	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	Rigid 🗸	N23	
10	Sn10	Standard 🗸	Fixed 💛	Rigid 🗸	Rigid 🗸	Rigid \lor	Rigid 🗸	Rigid 💛	Rigid 💛	N25	
•											

- Still in the table input, try to create a new element in the 1D elements tab. Fill in a begin node and an end node. Scia will then automatically create a new element with a random cross-section in the library. You can change it from the table input by clicking on the drop down menu. Assign a IPE100 cross-section:

-	Name	Туре		Beg. n	End node	Cross-section	t	Length
16	B12	beam	(80) 💊	N14	N12	CS2 - IPE160 ~	2 #	5,22
17	B13	beam	(80) 🕓	N16	N17	CS2 - IPE160 🗸	ΞĦ	5,22
18	B16	beam	(80) 🕓	N19	N17	CS2 - IPE160 🗸	Ŧ	5,22
19	B17	beam	(80) ~	N21	N22	CS2 - IPE160 V	1 #	5,22
20	B20	beam	(80) 🔨	N24	N22	CS2 - IPE160 ~	Ξ÷	5,22
21	B21	beam	(80) ~	N2	N6	CS3 - IPE100 🗸		5,00
*					1			

- Insert the remaining edge beams by using the same approach. To know what node names you should enter you can show them easily with this option near the Navicube:



5. Actions after the input of the geometry.

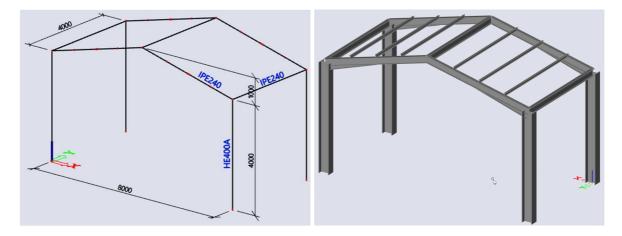
- Tools > Check structure
- Edit > Modify > Connect members/nodes
 Or simply write "Connect members/nodes" in the SCIA Spotlight

Note: members are automatically connected when the below option is enabled in the mesh settings:



Example 4: Purlins

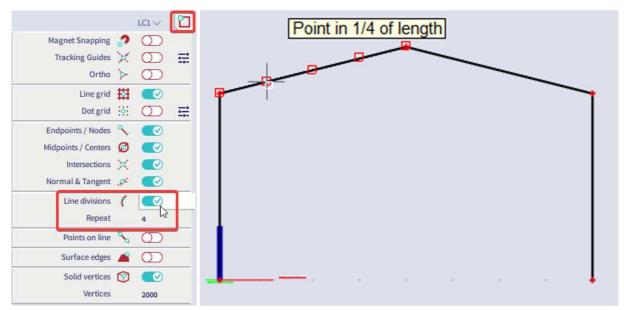
- 1. Input of the geometry
- Project data: Frame XYZ Steel S 235 Purlins IPE100



വ

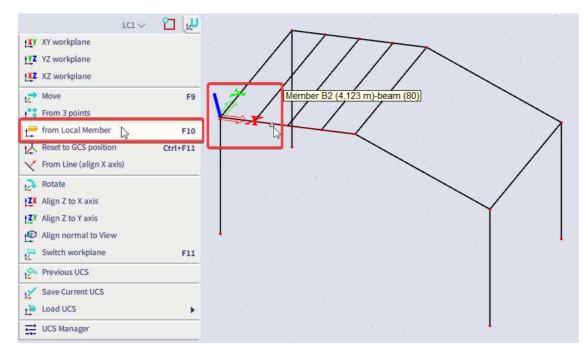
- Input of purlins:
 - Enter the purins on the left part of the roof

At the top right of your screen click on "Snapping" and make sure the option "Line divisions" is enabled. Fill in a value of "4" to divide every line element into 4 equal parts:

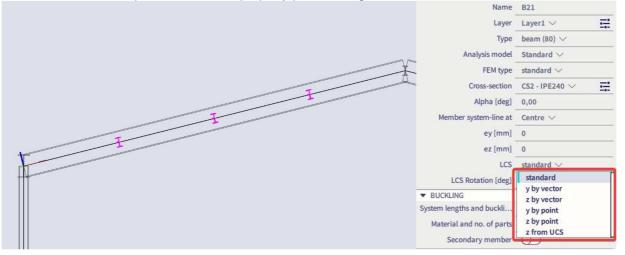


- We are going to rotate the purlins so that their LCS Z-directions points in the same direction as the LCS of the frame beams.

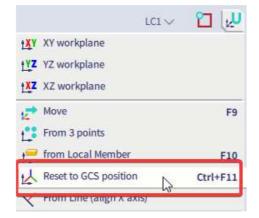
First, change the orientation of the UCS by choosing "From local member" in the Coordinate system settings and select a beam of the main frame:



- Select the purlins and in the property panel change the LCS definition to "z from UCS":

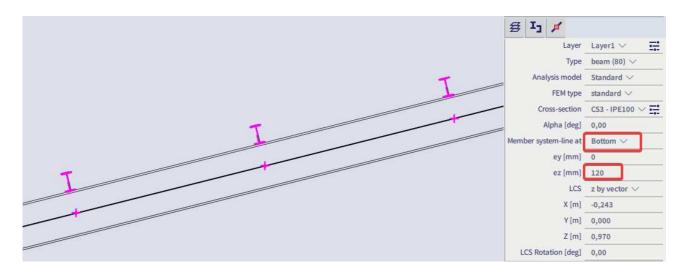


 Reset the UCS back to its original position by choosing "Reset to GCS position" from the coordinate system settings:



• Input of eccentricity:

- Select the purlins.
- Change member system line to "bottom"
- Change z eccentricity to 120mm (half of the main beams).



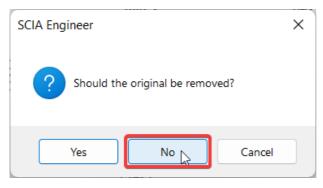
• Copy the purlins to the other side of the roof

Select the 3 purlins on the left side of the roof and choose the mirror option in the process toolbar



Or go to Edit > Modify > Mirror

Draw a mirror plane by clicking on the two top nodes of the roof and click on "No" when the window shown below appears:

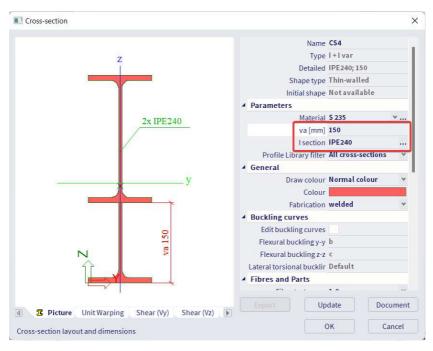


• Input of haunch

First approach

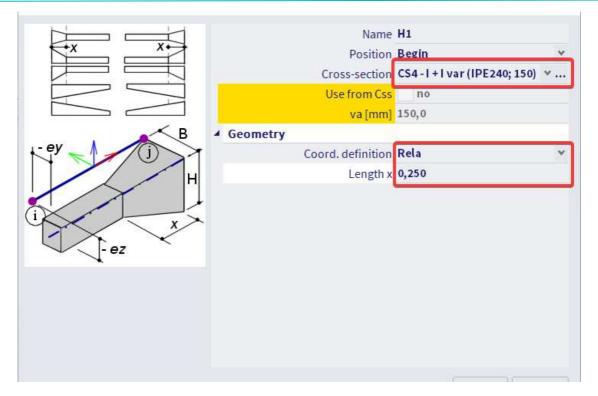
We need a new cross-section which defines the haunch first. Go to the cross-section library and under "Haunch" choose for the first type:

New cross-section									×
Available groups	AVAILABLE I	TEMS OF	THIS GROU	JP				Items in proje	ct
Profile Library Geometric shapes Numerical	Ŧ	Ŧ	Ŧ	Ŧ	Ŧ	I		CS1 - HE400A CS2 - IPE240 CS3 - IPE100	
ff General] [] Pairs [] Closed - Haunch	Ŧ	Ţ	I	I	I				
Welded Sheet welded Build-in beams Thin-walled geometric Fabricated							->		
【 Virtual joists 】 Westok									
I+Ivar		Profile	.ibrary filte	r All cr	oss-sectior	ns ¥	. C	Add	Close

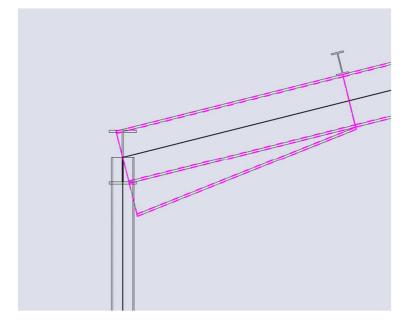


Now go to the input panel, filter category on 1D members and choose for "Haunch on 1D":

INPUT PANEL	ullet All workstations $$
1D Members 🗸	🥔 All tags $$
▼ 1D MEMBERS	
援 1D member	- Beam Ctrl+B
Column Ctrl+Shift+B	루 Rib
Haunch on 1D	🤫 Arbitrary profile
Opening on 1D	🔲 Internal node on 1D



And click on the element where you want to add the haunch on, the beams of the frame in our case:



Note that a haunch is some additional data attached to a certain 1D element. If you want to select the haunch, always show the volumes and click on the haunch itself. If you select the system line the 1D member itself will be selected. Haunch data can also be copied to other elements: select the haunch, right click, choose "copy add data" and select the items where you want to add the same haunch on.

2nd approach

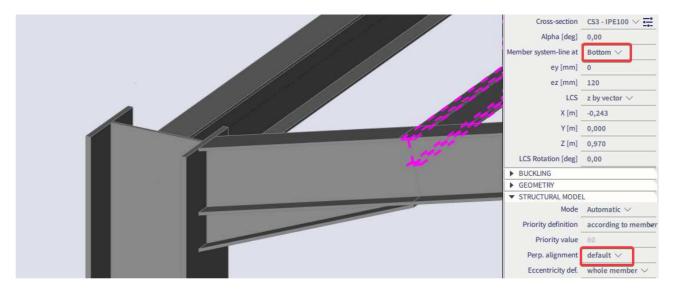
Go to the input panel, filter category to "1D member" and choose for "arbitrary profile":

INPUT PANEL		All workstatio	ons 🗸			
1D Members 🗸		🥔 All tags 🗸				
1D MEMBERS		1911				
📕 1D member		😑 Beam		Ctrl+		
Column	Ctrl+Shift+B	루 Rib				
Haunch on 1D		🔫 Arbitrary profile 💦				
J Opening on 1D		Internal node on 1D				
Arbitrary profile				×		
		Name	AP			
	Geometry					
		Coord. definition	Rela	*		
j (j		Span 1	Edit Span			
		length 1	1,000			
	Span prop(1)					
		Type of Css(1)	param. haunch	*		
a <u>11 i</u> 12 12		Cross-section1(1)	CS4 - I + I var (IPE240; 150)	×		
		Css 1 param(1)	from DB	۲		
		Css 2 param(1)	variable	۷		
		Use from Css	no			
		va [mm]	10,0			
		Alignment(1)	default	٣		
			ОК	Cancel		
			UK	cancel		

2. Structural model

Go to the main menu and choose for View > Visualization > Generate structural model

Note that the eccentricities for the analysis model and the structural model are separate settings.



3. Actions after the input of the geometry.

- Tools > Check structure
- Edit > Modify > Connect members/nodes
 Or simply write "Connect members/nodes" in the SCIA Spotlight

Note: members are automatically connected when the below option is enabled in the mesh settings:

4	Mesh setup	
	Average number of 1D mesh element	1
	Average size of 1D mesh element on (0,200
	Average size of 2D mesh element [m]	0,500
	Connect members/nodes	
	Setup for connection of structural en	

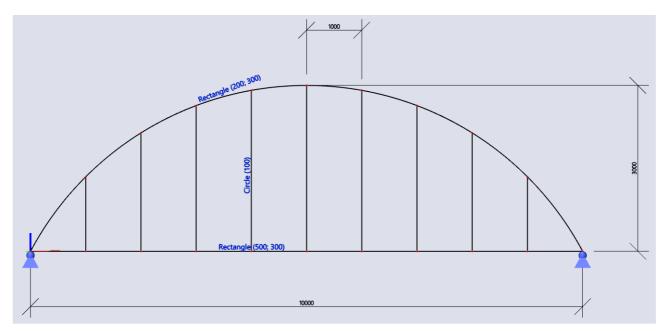
Example 5: Bridge

1. Input of geometry

• **Project data:** Frame XZ – Steel S 235 – Concrete C25/30

	DATA		MATERIAL	
	Name:	-	Concrete 🔽	-
20	Marrie.		Material C25/30 v	
	Part:		Reinforcement m B 500B ¥	
_			Steel 🔽	
-1	Description:			
	Author:	2	Masonry	
	rudior.		Aluminium	
	Date:	29/01/2024	Timber	
			Steel fibre concre	
			Other	1
	Structure:	🗘 Frame XZ 🛛 👻	CODE	
			National Code:	
			EC-EN 🛩	
NO TREAS SAME AN			National annex:	
			Standard EN 👻	J.

• Geometry:



• Input a straight beam

Filter the Input Panel workstation to "Structure" and under 1D members choose for "Beam":

Horizontal beam			2
	Name	B12	
\land	Туре	beam (80)	*
α	Analysis model	Standard	*
	Cross-section	R500x300 - Rectangle (500; 300)	
z'	Alpha	0	*
	Member system-line at	Centre	۷
	ez [mm]	0	
ez	LCS	standard	~
(i) \	FEM type	standard	*
	Layer	Layer1 👻	
	▶ Buckling		
	▲ Geometry		
	Length [m]	10,000	
	Insertion point	begin	۷
		OK Car	ncel

In this window you can also directly define a new rectangular cross-section by clicking on the three dots next to the cross-section dropdown menu.

Close the window by clicking on "OK" and input the beam in the origin by entering "0 0" in the SCIA Spotlight. End the action with ESC.

• Input a curved beam

Filter the Input Panel workstation to "Structure" and under 1D members choose for "1D member":

II Member			×
	Name	B12	
\sim	Туре	beam (80)	~
α	Analysis model	Standard	×
	Cross-section	R200x300 - Rectangle (200; 300)	×
z'	Alpha	0	*
J (J)	Member system-line at	Centre	~
	ez [mm]	0	
ez	LCS	standard	v
i	FEM type	standard	~
~	Layer	Layer1	۰
1	A Buckling		
	System lengths and buckling settings	Default	
	Secondary member		
	Structural model		
		OK Car	icel

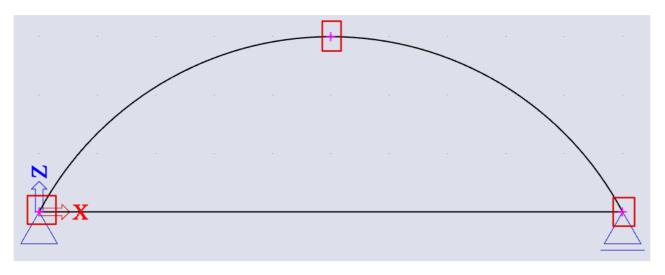
In this window you can also directly define a new rectangular cross-section by clicking on the three dots next to the cross-section dropdown menu.

After closing the above window by clicking on "OK" below the SCIA Spotlight some additional option should appear which let you decide the shape of the element that you are going to input. In this case choose for a circular arc defined by three points:



Afterwards you can insert the coordinates of this arc in the SCIA: 1^{st} point: 0 0 (or click on the left node of the rectangle (500;300)) 2^{nd} point: 5 3

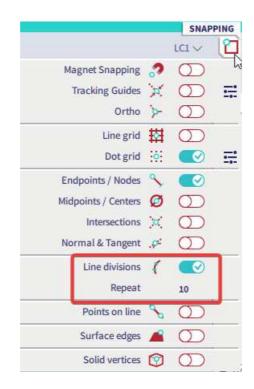
3rd point: 10 0 (or click in the right node of the rectangle (500;300))



• Input the vertical elements:

In the status bar you can go to "Snapping" and enable "line divisions".

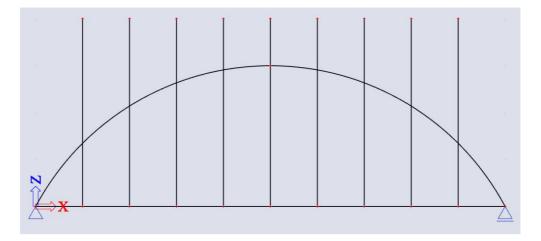
This allows you to have a snap point after each 10th of any element with the below settings:



Filter the Input Panel workstation to "Structure" and under 1D members choose for "Column":

	Name	B12	
ez	Type	column (100)	*
	Analysis model	Standard	Y
	Cross-section	C100 - Circle (100)	۷
	Alpha	0	Y
(j)	Member system-line at	Centre	4
	ez [mm]	0	
	LCS	standard	v
	FEM type	standard	Y
	Layer	Layer1	۲
	Buckling		
(i) 🖌 🗠 🗠	▲ Geometry	_	
	Length [m]	4,000	
	Insertion point	Bottom	*

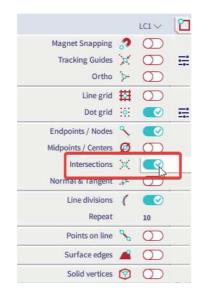
Define a circular cross-section from this window, define a length of 4m and close the window by clicking on "OK". Now you can use the previous enabled snap settings to add 9 columns because the straight beam was now divided in 10 equal pieces to snap to.



Now we are going to change the height of these vertical elements so that they match with the curved beam.

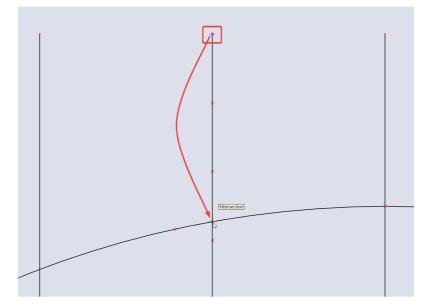
1st method

- Enable the intersections snap in the snap settings:



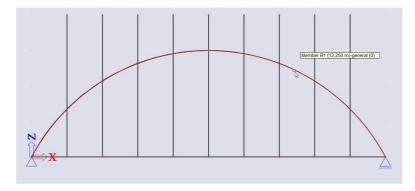
- Select the top node of a column.
- Drag the node with the left mouse button to the intersection with the curved beam.

This has to be done for each individual node because the lengths of the columns differ:

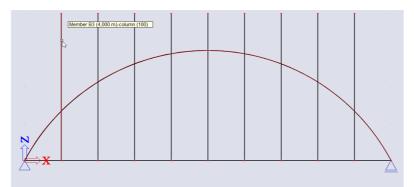


2nd method

- In the main menu go to "Edit" > "Modify" > Trim
- Select the curved beam and end with ESC:



- Select each column individually above the curved beam, this is the part that is going to be subtracted:

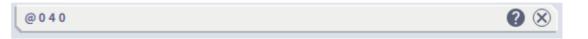


2. From 2D to 3D

Go to "File" > "Project Settings" and change "Structure" from "Frame XZ" to "Frame XYZ"

• Copy 2D bridge

Select the whole (CTRL+A) structure and press CTRL+C to copy. Select some node of the bridge and enter the below:



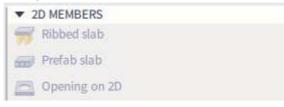
Note that by entering "@" before the coordinates you actually define a vector so it doesn't matter from which point you start the copy.

Next a message will appear with the question whether you want to copy additional data as well. In this case this is about the supports so you can click "Yes":

SCIA Engineer	×
	selected entities contain additional data. d-data as well?
	Yes No

General XYZ

If we would want to add a 2D bridge deck now you will see this is not possible as there is no such option in the input table:



The reason is because in the project data we choose for "Frame XYZ" instead of "General XYZ". Go to "File" > "Project Settings" and change "Structure" from "Frame XYZ" to "General XYZ". Now you will have options in the input panel to also define 2D elements:

	2D MEMBERS	
	Plate	Ctrl+T
W	Composite deck	
12	Metal deck	
7	Ribbed slab	
	Prefab slab	
	Wall	Ctrl+Shift+H
P	Shell	
	Shell - surface of revolution	
-	Shell - swept surface	
	Subregion	
	Opening on 2D	
(E)	Internal node on 2D	
Z	Internal edge	
4	Intersection	
8	Cutout	

3. Actions after the input of the geometry

- Tools > Check structure
- Edit > Modify > Connect members/nodes
 Or simply write "Connect members/nodes" in the SCIA Spotlight

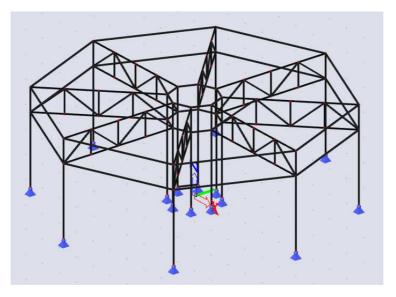
Note: members are automatically connected when the below option is enabled in the mesh settings:

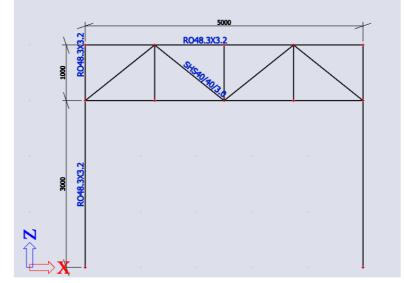
4	Mesh setup	
	Average number of 1D mesh element	1
	Average size of 1D mesh element on (0,200
	Average size of 2D mesh element [m]	0,500
	Connect members/nodes	
	Setup for connection of structural en	

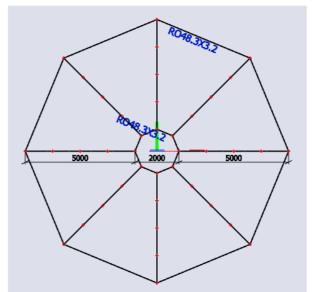
Example 6: Carrousel

1. Input of geometry

• Project data: Frame XYZ – Steel S 235







• Input of a single frame

Filter the Input Panel workstation to "Structure" and under 1D members choose for "Column". Enter a column of height 4m with cross-section RO48.3x3.2 on starting points "1 0 0" and "6 0 0".

Filter the Input Panel workstation to "Structure" and under import & blocks choose for "Catalog blocks". Choose the below template and change the values as shown below:

Block selection manager						×
Available groups	AVAILABLE IT	EMS OF THIS	GROUP		Items in pr	oject
冊 Frame 3D 舟 Tower ↑ Frame 2D ○ Curve ○ Ream						
Truss 2D Truss 3D Truss 3D Truss 3D Arc						
	- V					
Truss.nN						
			-ilter	¥	ОК	Close
Geometry block						×
n 2 m 2 l[m] 5,000 h[m] 1,000 hu[m] 0,000 hd[m] 0,000 Type V Bb CS1 - RO48					*	
Bu CS1 - RO48 D CS2 - SHS4 V CS2 - SHS4	· •					
	m=1					

We are going to use the same cross-section for the whole cross-section. Close the windows with "OK" and enter the truss on coordinates 1 0 5.

2. Manipulations

• Multicopy

Select all the elements (CTRL+A) and in the process toolbar in the structure branch choose for "Multicopy":

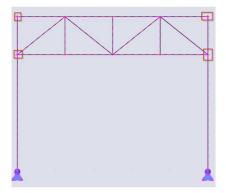


In the window that pops up fill in the below values:

Multicopy			×
	of copies	8 + - copy	Connect selected nodes with new beams Copy additional data
	E VECTOR istance by cu	rsor	HOW TO DEFINE THE DISTANCE ? • between two copies
x y z	0,000	m m m	from original to the last copy HOW TO DEFINE THE ROTATION ? between two copies
ROTATIO	N		• from original to the last copy ROTATION AROUND
rx ry	0,00	deg deg	• current UCS distance vector
rz	360,00	deg	OK Cancel

In case you enabled "Connect selected nodes with new beams" after clicking "OK" a new window will open which allows you to define a cross-section for the automatically generated connection beams.

<u>Note</u>: in the first step of the multicopy we selected all the elements in the model, so now all nodes between the copies will be connected with a beam. If we only wanted to connect the nodes from the truss we had to select all the 1D elements, but only the nodes that are part of the frame:



2. Actions after the input of the geometry

- Tools > Check structure
- Edit > Modify > Connect members/nodes
 Or simply write "Connect members/nodes" in the SCIA Spotlight

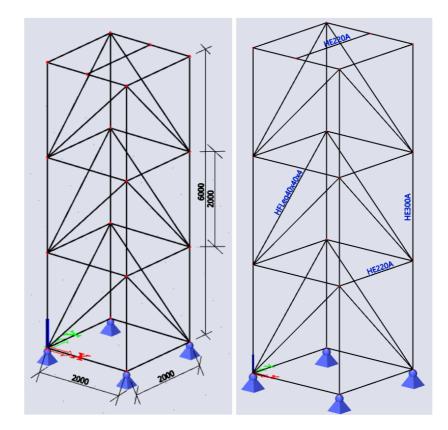
Note: members are automatically connected when the below option is enabled in the mesh settings:

4	Mesh setup	
	Average number of 1D mesh element	1
	Average size of 1D mesh element on (0,200
	Average size of 2D mesh element [m]	0,500
	Connect members/nodes	
	Setup for connection of structural en	

Example 7: Bearing frame

1. Input of geometry

• Project data: General XYZ – Steel S 235



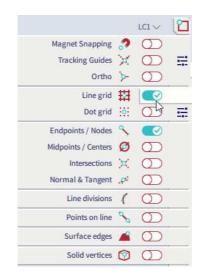
• Use of the line grid

Go to the input panel and under Grids & Stories choose for "3D line grid", fill in the values shown below:

ut data Drawing setup						
₽ [×]	•				a a a a a a a a a a a a a a a a a a a	A TANALANA C
IR X [M]	DIR Y [M]			DIR Z [M]		
rpe Span ~	Туре	Span	~	Туре	Span	~
Name X[m] dx[m] Rep SL	Name Y[m]	dy[m] Rep	SL	Name		ep SL
L A 0,000 no	1 1 0,000		no Y		0,000	no 🗸
2 B 2,000 2,000 1 no	2 2 2,000 * 0,000	2,000 1	no v		2,000 2,000 3 4,000	no v
0,000 0,000 0	0,000	0,000 0	v		4,000 6,000	no v
					0,000 0,000 0	no *
Generate name automatically	Generate name au	tomatically		Generate na	ame automatically	
Generate name automatically	Generate name at	tomatically		Generate na		
		Rotation	0,00	deg	Refresh names	

Click on "OK" and enter the grid in the origin so that the grid dimensions fall in line with the coordinates in Scia. Enter "0 0 0" in the SCIA spotlight. End with ESC.

In the snap settings make sure the line grid is enabled:



Use the grid to draw the structure as shown on the first page.

2. Loads

Enter the below load cases by going to the loads dropdown menu in the status bar and clicking on "Manage load cases".

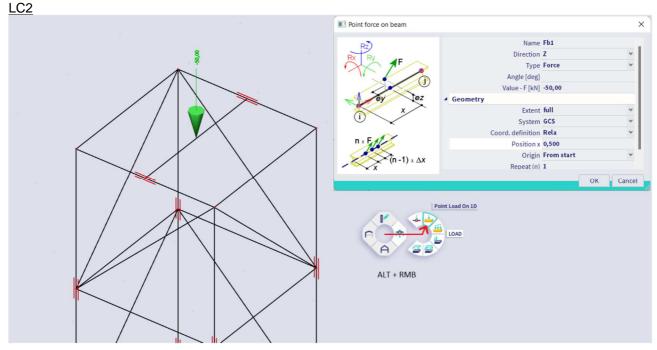
LC3 V
Ctrl+I

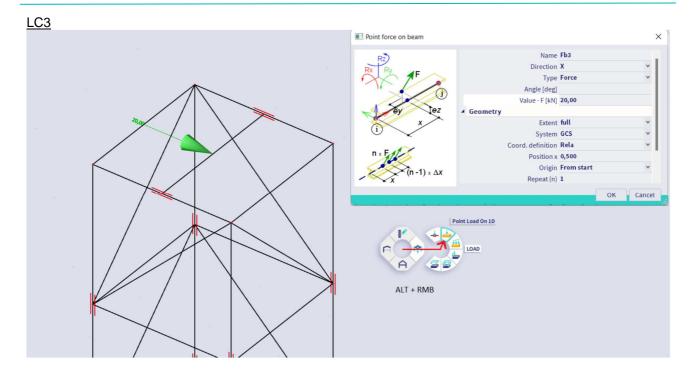
LC1: Self weight (is automatically created)

LC2: Vertical loads (Variable)

LC3: Horizontal loads (Variable)

You can now switch between the load cases in the dropdown menu and enter te below loads on the upper transfer beam:





3. 3D displacements and 3D stress

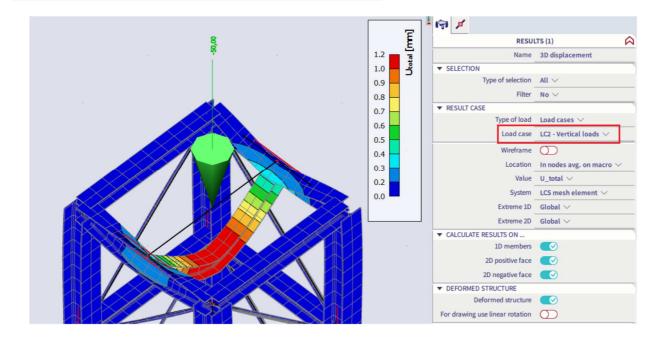
Now we can perform a calculation and have a look at the results with the options in the process toolbar.

Calculation:



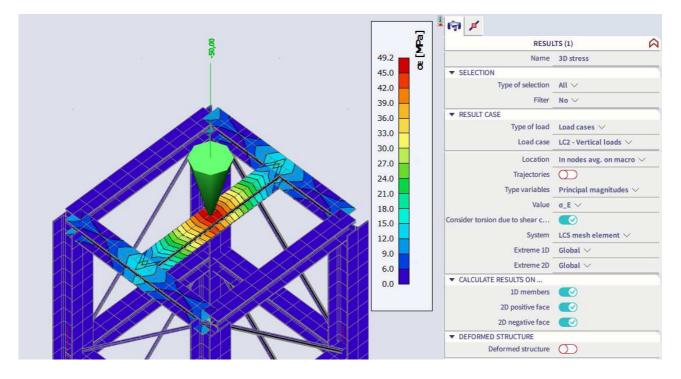
3D deformations:



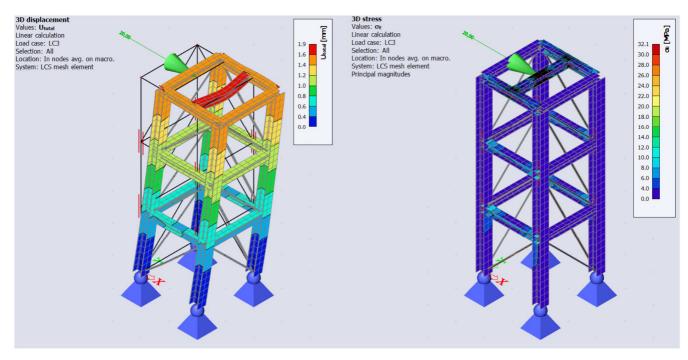


3D stress:



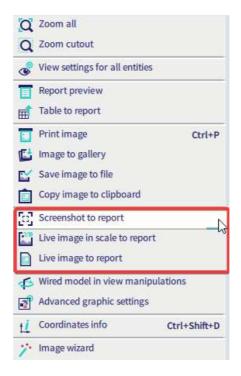


Same for LC3:



4. Images

You can add different kinds of images to the engineering report by right clicking on your mouse on an empty part of your screen (and without any active selection):



If you choose for a live image, the image will stay connected with the model. Any changes you make to the model will also be visible on the image and view settings can still be edited later on. A screenshot will be a rasterized image of the current state of your model and is not editable.

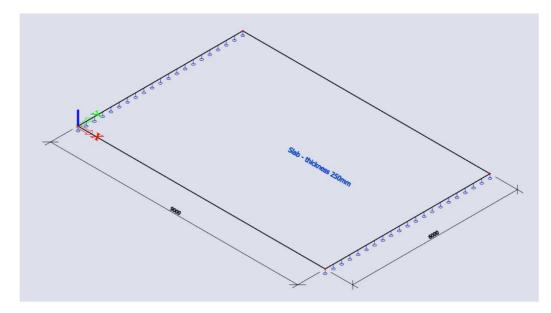
Send the image to a report and have a look at it via the report option in the process toolbar:

						0 5.0	· + ×	🐌 + =		R	eport_1 [Esa1	Lesa] - Enginee	ring report			-		×
						Home	View											
	+ +					Paste & Cut	← Undo • → Redo •	Report	Insert	Regenera	te Regenerate outdated V	Edit picture properties	View DWG	G colour	ø	Edit		
	₩					Clipboard	Undo	properties	Document item		outdated	Edit pictures			Tables / Pictures	external file		
	ŋ					Cipbourd			ement; U_total	neg	enerate	curt pictures	currexternar	pictures	indica / i licturea	External mes		_
	-						Values:	Jtotal							-			
	A						Linear ca Load cas Selection	e: LC3							1.9 E	0		
	P						Location		/g. on macro. ement						1.9 1.6 1.4 1.2			
	(T and									19.00	*				1.0			
	4										1 Nor				0.6			
	Ē														0.0			
	¥									Ale a	1							
	I]																	
	Ă										1							
	di se																	
	t i																	
	Q										3							
16		REPORT									-							
-) 🛋	E 🗹	İ	www.scia.net Pag	je 1 💽	5						# E	91%			+
1		13				1		5 C	10 - E		2						<i></i>	

Input of structural entities – 2D elements

Example 8: Rectangular plate

- 1. Input of geometry
 - Project data: Plate XY Concrete C30/37



• Input of the slab:

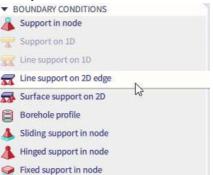
In the input panel filter the workstation to "structure" and under 2D members choose for "Plate". In the next window fill in a thickness of 250mm and leave the other values as default. Below the SCIA spotlight there should appear some additional options to change the shape of what you are going to input. Choose a rectangle so that we only have to define the 2 opposite corners of the slab:



Afterwards enter the coordinates "0 0 0" and "9 6 0".

• Input of the supports

Filter the input panel category to "Boundary conditions" and choose for "Line support on 2D edge":



Set the restriction as sliding and click on the left and right 6m edge of the slab we just created to add the supports.

2. Loads

Enter the below load cases by going to the loads dropdown menu in the status bar and clicking on "Manage load cases".

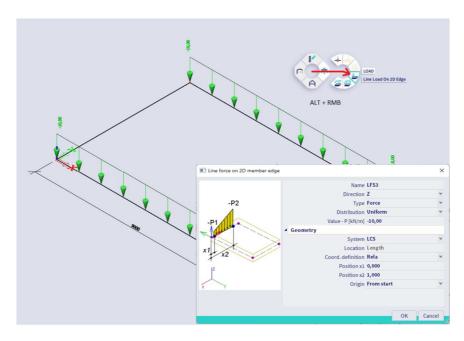


LC1: Self weight (is automatically created)

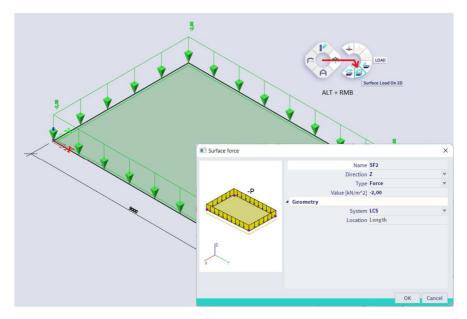
LC2: Walls (Permanent)

LC3: Service loads (Variable)

You can now switch between the load cases in the dropdown menu and enter te below loads on the slab: <u>LC2</u>



<u>LC3</u>:



3. Finite element mesh

• Generate the mesh

In the main menu go to "Tools" > "Calculation & mesh" > "Generate mesh"

	Calculation & Mesh	•	2	Calculate	Ctrl+Shift+F5
	Selections	•	~	Hidden calculation	
撑	Explode line grid		A	Autodesign	
	BIM toolbox		0	Solver settings	
A	Check structure			Generate mesh	
11	Coordinates info	Ctrl+Shift+D	B	Mesh settings	3
1	Geometry/Graphics settings		-		

Note: the generation of the mesh is something that is done automatically whenever you run a calculation.

• Graphical display of the mesh elements

In the view settings there is an option to visualize the mesh that will be used for the finite element calculation. Right click on an empty part of your screen, choose for "view settings for all entities" and enable the mesh option in the structure tab:

			🖸 Zoom all	
View parameters setting			Q Zoom cutout	
Check / Uncheck gro	Lock position		View settings for all entities	
Check / Uncheck all			Report preview	
	1		Table to report	
+ Service		- 11	Print image Ctu	rl+P
+ Structure		- 11	Image to gallery	
+ Panel		_ [] []		
+ Structure nodes				
Mesh			Copy image to clipboard	
Draw mesh		- 11	Screenshot to report	
			Live image in scale to report	
riee edges	L		Live image to report	
Display mode	wired	<u> </u>		
+ Local axes			Direct model in view manipulations	
Show names in tab	OK Apply Cand		Advanced graphic settings	
	Apply Can	lei	Coordinates info Ctrl+Shi	ft+D

• Mesh refinements

To have a finer mesh you can either apply local mesh refinements from the option you gave in the input table:



Or you can change the default mesh size in the calculation setup:

FE analysis		×
Calculations	Mesh setup	
Line and line	Average number of 1D mesh element 1	
Linear analysis Load cases: 3	Average size of 1D mesh element on c 0.200	
	Average size of 2D mesh element [m] 0,250	
ther processes	Connect members/nodes 🗹	
Test input of data	Setup for connection of structural en	
	Advanced mesh settings	
Save project after analysis	Solver setup	
	Advanced solver settings	
	Deve setup	
Calculate		

This will change the mesh size for all the elements in the model when there are no refinements defined on them.

4. Results

• Results on the plate

In the Process Toolbar go to the "Results" branch and click on 2D internal forces:

ma	2D INTERNAL FORCES	
	📲 🖷 ଜ ନ 🗟 🌭 🌺 📲 ୍ୟା ଜି ଲା 🖓 📾 🐂	

In the Process Toolbar go to the "Results" branch and click on 2D stress/strains:

ma		2D STRESSES/STRAINS	
	ት 🗑 🕅 🖓 📾 🐯	And a second s	

Specify which part of the results have to be shown and how they are shown in the result properties. Settings of which:

-System: choose between axis of the elements or axis of the mesh elements.

-Location: Scia provides 4 different locations and types of averaging the results calculated by the finite element calculation. Have a look at our FAQ to read about the difference: https://www.scia.net/en/support/faq/results/results-2d-members-what-influence-option-location

-Type of values: choose between basic, principal, design or resultant magnitudes. Have a look at our FAQ to read about the difference:

https://www.scia.net/en/support/faq/scia-engineer/results/calculation-1d-and-2d-results

-Drawing setup 2D: make changes to the colors, legends, min./max. settings etc. to change the visualization of the results.

• Accuracy of the results

An infinitely small 2D mesh will give you the most correct results from the finite element calculation. But because you typically don't have so much time to wait for that kind of calculation we have to decide a proper mesh size. If you change between the 4 location types in the result properties, they should not be much different from each other. If so, you will have to choose for a finer mesh setting.

General rule of thumb is to choose a mesh size of about the thickness of the scoped 2D element.

• Line reaction forces

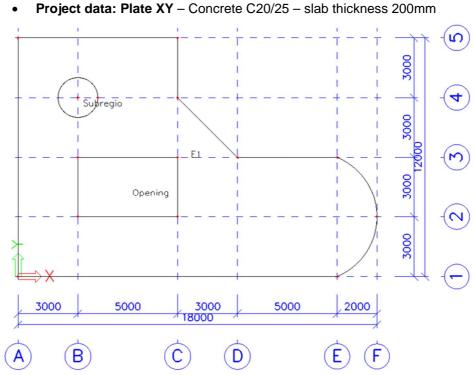
In the Process Toolbar go to the "Results" branch and click on "reaction":



With the "course" setting set to "precise" Scia will show the exact results from the calculation. You can choose from "Trapezoidal" or "Average" for a an averaged out result of the line reaction forces.

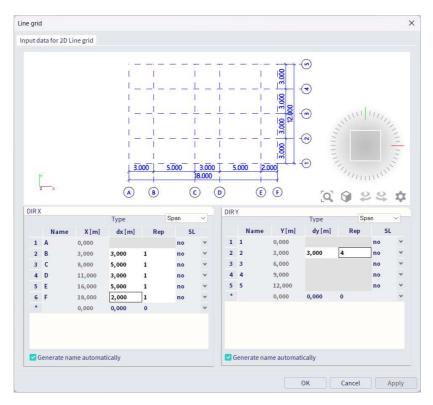
Example 9: Slab on an elastic foundation (subsoil)

1. Input of the geometry



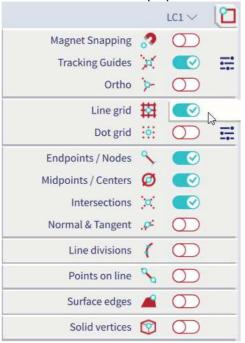
• Input of a grid:

In the Input Panel choose for "Rectangular Grid" under "GRIDS & STOREYS". Fill in the below values, click "OK", "OK" again and afterwards enter the coordinates of the origin: 0 0 0, all separated with a SPACE. Press "ESC" to end the action.



• Input of a slab:

With the help of the grid we are now going to create a 2D element. Therefor make sure the line grid snap is enabled under the snap options:



Then in the input panel under category "2D members" choose for "plate" and fill in the properties of the slab. Click "OK" and start drawing the slab. Notice that right underneath the SCIA Spotlight you can change the shape of the next segment you will be drawing. For our plate element we need straight lines and circular arcs:



Finish the plate by selecting the first node again, clicking "ESC" or right clicking and choosing for "Confirm action".

• Input of additional data:

Now we are going to create a circular subregion and a rectangular opening. In the input panel choose for "Opening on 2D" under 2D members and "Subregion" after, we are going to use the below shapes to create the openings and subregions:

	NEW	RECTANGLE							
0 4	00	₫ #	1	5	\cap	Λ	S	S	4
NEW CIRC	LE (BY CENTI	RE & RADIUS	PT.)						

If you want to enter a value for the radius make sure the tracking guides are enabled in the snap options. Point in any orthogonal direction and fill in the value for the radius.

To simulate the ground underneath the slab we are going to apply a surface support. Look for "Surface support on 2D" under "Boundary conditions" in the input panel. Copy a predefined subsoil to the project library and click "OK" to assign it to the slab.

2. Loads

Insert the following load cases: LC 1: Self weight LC 2: Walls on the outer edges (Perm.) > Line force 10 kN/m LC 3: Freestanding walls (Perm.) > Line force 6,5 kN/m LC 4: Service load (Var.) > Surface load 2 kN/m² LC 5: Service load on subregion (Var.) > Surface load 1,5 kN/m²

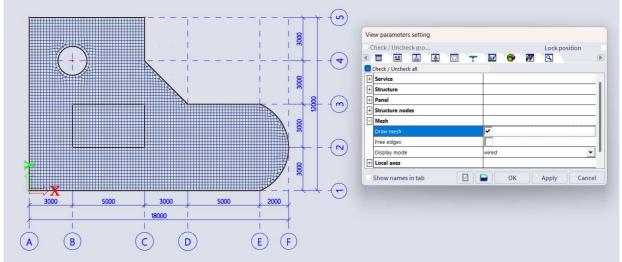
3. Finite elements mesh

• Mesh settings:

"Tools" > "Calculation & mesh" > "Mesh settings" We recommend a 2D mesh setting of 1 to 2 times the thickness of the 2D element.

• Generate the mesh:

"Tools" > "Calculation & mesh" > "Generate Mesh" You can now also visualize the mesh in the view settings:



4. Results

Results on the slab

Go to the "Results" branch of the Process Toolbar and choose for 2D internal forces:



Set the result properties and click on "Refresh" at the bottom.

• Results on the subregion

Repeat the above but before refreshing select the subregion and change the "Type of selection" in the result properties to "Current:



• Results on the foundation support

Go to the "Results" branch of the Process Toolbar and choose for 2D contact stresses:



Note: Convention for soil stresses: positive value = compressive stress, negative value = tensile stress.

5. Eliminate tension in subsoil

Go to File > Project settings and under the "Functionalities" tab enable support nonlinearity:

GENERAL	DETAILED
Model modifiers	A Nonlinearity
Parametric input	Beam local nonlinearity
Climatic loads	Support nonlinearity/basic soil spring
Mobile loads	Initial imperfections
Dynamics	Geometrical nonlinearity
Stability	General plasticity
Nonlinearity 🔽	Compression-only 2D members

Go to the "Loads" branch of the process toolbar and choose for "Nonlinear combination". Then create some nonlinear combinations and perform the nonlinear analysis by enabling it in the calculation window:



Note: if you do a nonlinear analysis with the above functionality enabled by default Scia ignores the possible tensional capacity of any surface support. You could also define a nonlinear function for a standard support to achieve the same.

6. Concrete settings

Overall settings

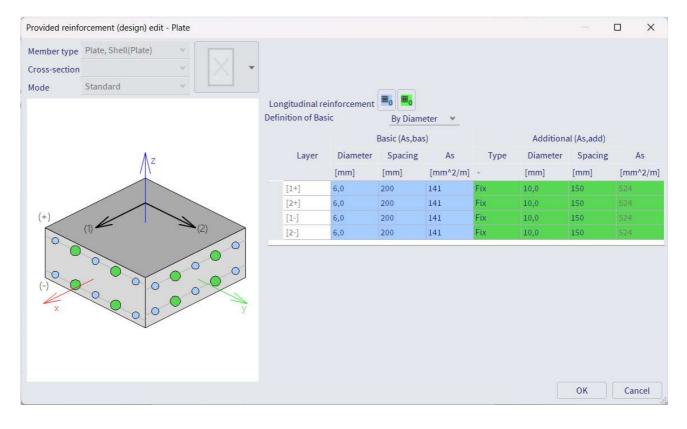
In the process toolbar go to the "Concrete" branch and choose for "Concrete settings":



These settings will affect all the elements in the project which were not overwritten yet with member data (see further).

For each type of element you can define a template for the reinforcement design. Do so with the below settings for type "Plate":

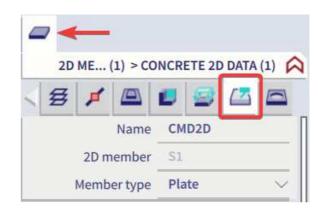
	te setup 👻 View sett 👻 Loa	derault	Find	National annex:
Descriptio	n	Symbol	Value	Default Unit Chapter Code Struc Chec
<all></all>	Q	<all> ρ</all>	<all> ρ</all>	Provided reinforcement (design)
Design def	aults			≓ -: □ ▶ = < <i>∧ ∧</i> ? -
A Reinfor	rcement			Plate
⊳ Bea	am / Rib			
⊳ Bea	am slab			
Col	lumn			
🔺 Pla	ate			
4	Longitudinal			
	Design of provided reinforcement			Name Plate
	Design template of provided reinfo		Plate	Description Basic and
	✓ Upper(z+)		G	Member tyr. Plate, Sh
	Type of cover	Type _{c+}	Auto	Cross-sectic Rectangle
	Diameter of first layer	d _{s1+}	10,0	Mode Standard
	Angle of first layer direction	α ₁₊	0,00	
	Diameter of second layer	d _{s2+}	10,0	
	Angle of second layer direction	α ₂₊	90,00	
	Lower (z-)			
	Type of cover	Type _{c-}	Auto	
	Diameter of first layer	d _{s1-}	10,0	
	Angle of first layer direction	α ₁₋	0,00	
	Diameter of second laver	d-o	10.0	



• Member data

Every concrete element in Scia has member data attached to it which follow the general settings. Whenever you change a certain setting in here, the general settings will be overwritten.

Member data are accessible via the selection of and element:



Or by showing the concrete member data labels in the view settings:

	Check / Uncheck gro Check / Uncheck all Check / Uncheck all	Lock position	
	Display on opening the service	•	
th	Concrete + reinforcement Display		
	Concrete member data SaT detail data		
	Drawing directions for design		
	Show names in tab	Cancel	
		I I M	
	\	<u> </u>	$-\frac{1}{2}\frac{1}{2}$

7. Reinforcement design of plate

• Theoretical reinforcement

In the Process Toolbar go to the "Concrete" branch and choose for "Concrete 2D reinforcement design":



-As + = Amount of reinforcement needed for upper surface (positive LCS Z-axis) of the 2D element -As - = Amount of reinforcement needed for lower surface (negative LCS Z-axis) of the 2D element -Direction 1 is by default the LCS X-direction of the element

-Direction 2 is by default the LCS Y-direction of the element

Directions can be changed in the concrete settings or concrete member data.

• Provided reinforcement

Repeat the command you did for the theoretical reinforcement but in the result properties change the "Type of values" to "Provided":

▼ EXTREME 2D			
Averaging of peak	\bigcirc		
Location	In nodes avg. o \lor		
System	LCS mesh elem \vee		
Extreme	Global 🗸		
Type of values	Required 🗸 🗸 🗸		
Values	Required		
	Required - Static		
 LIMIT STATE CONDI 	Required - Not c		
Design ULS	Provided		
Design SLS (crack w	Provided - Uniza		
Design SLS (reinf. st			

In the template we set in the concrete settings Scia is now going to compare with the theoretically needed amount of reinforcement to calculate the provided amount of reinforcement.

• User reinforcement

In the Process Toolbar go to the "Concrete" branch and choose for "2D reinforcement":



You can add reinforcement of any kind onto the slab. To take into account this information in the calculation of provided reinforcement, enable the below option in the result properties:

▼ EXTREME 2D	
Averaging of peak	\bigcirc
Location	In nodes avg. o \vee
System	LCS mesh elem \vee
Extreme	Global 🗸
Type of values	Provided \checkmark
Values	Nø,prov,1+ 🗸 🗸
Consider user reinf	

• 2D reinforcement checks

Punching check



This check can be executed based on required (theoretical), provided and user reinforcement.

2D SLS crack width check



This check can be executed based on required (theoretical), provided and user reinforcement.

Code dependent deflections



This check can be executed based on required (theoretical), provided and user reinforcement.

2D ULS capacity check

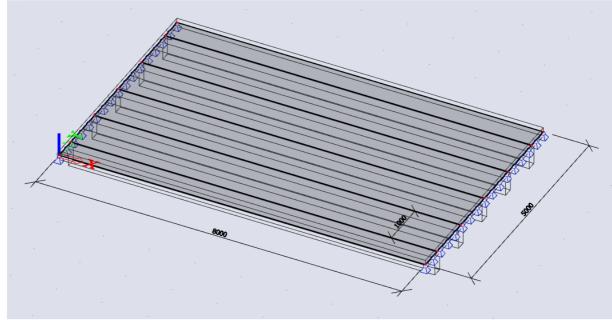


This check can only be executed when there is user reinforcement present in the concrete.

Example 10: Slab with ribs

1. Input of the geometry

 Project data: General XYZ – Steel S 235 – Concrete C20/25 – thickness of slab 200 mm – Ribs rectangular 400mm x 200mm



• Input of slab: method 1

Filter the Input Panel workstation to "Structure" and choose for "Plate" under "2D members". When doing so under the Scia Spotlight some options for the shape appear, choose for rectangle:



And enter coordinates "0 0 0" and "8 5 0" in the Scia Spotlight.

To input the ribs filter the Input Panel workstation to "Structure" and choose for "Rib" under "1D members":

🚺 Plate rib				×		
	Name	B1				
	Type rib	platerib (92)				
	Analysis model	Standard		v		
	Cross-section	CS1 - Rectan	gle (400; 300)	×		
	Shape of rib	T symmetric				
w IZ	Effective width	width		*		
	for int. forces [mm]	1000				
	for check [mm]	1000				
	FEM type	standard		*		
	Layer	Layer1		۷		
1	 Buckling 					
X	System lengths and buckling sett	Default				
	Secondary member					
	Structural model					
			OK Ca	ancel		

Then simply draw the ribs onto the slab using the following coordinates: "0 0,5 0"-"8 0,5 0", "0 1,5 0"-"8 1,5 0", "0 2,5 0"-"8 2,5 0", "0 3,5 0"-"8 3,5 0", "0 4,5 0"-"8 4,5 0".

• Input of slab: method 2

Filter the Input Panel workstation to "Structure" and choose for "Ribbed slab" under "2D members". First fill in the properties of the slab:

IZ	Name	6.2	
			100
	Element type		Y
	Element behaviour		
1		plate (90)	v
	Material	Service service	×
		Isotropic with beam	*
	Thickness [mm]		
	Member system-plane at	Centre	Y
	Eccentricity z [mm]	0	
	LCS type	Standard	~
	Swap orientation	no	
	LCS angle [deg]	0,00	
	Layer	Layer1	×
	 Beam layout 		
	Position	Distance	*
	First offset [m]	0,500	
	Last offset [m]	0,000	
	Switch offsets	no	
	Distance [m]	1,000	
	Number	0	
	First beam	🔽 yes	
	Last beam	yes	
	Alignment	Bottom	×
	Generate subregions	no	

Draw a rectangular slab from coordinates "0 0 0" to "8 5 0", end with ESC and fill in the properties of the rib:

🛾 Plate rib			
	Name	B6	
	Type rib	plate rib (92)	
	Analysis model	Standard	~
	Cross-section	CS1 - Rectangle (40	0; 300) *
	Alignment	Bottom	
	Shape of rib	T symmetric	*
	Effective width	width	*
	for int. forces [mm]	1000	
	for check [mm]	1000	
W IZ	FEM type	standard	*
	Layer	Layer1	×
xy		OK	Cance

• Input of supports

Filter the Input Panel workstation to "Structure" and choose for "Hinged support in node" under "Boundary conditions".

2. Input of the geometry

LC1: Self weight

LC2: Service load (Var.) \rightarrow Input a surface load of 5 kN/m²

3. Refine the mesh

Refine mesh via Main menu > Tool > Calculation, Mesh > Mesh setting; average size of 2D mesh element = 0,20 m

4. Results

1D members

Go to the results branch of the Process Toolbar and choose for "1D internal forces".

Note the below option in the result properties:

RESULT CASE		
Type of load	Load cases	\sim
Load case	LC1 - Self weight	\sim
Rib		

When disabled you look at the results of the beam only, when enabled you look at the results of the whole T-section in this case (slab + rib).

• 2D members

Go to the results branch of the Process Toolbar and choose for "2D internal forces". Note again the below option in the result properties:

▼ RESULT CASE		
Type of load	Load cases	\sim
Load case	LC1 - Self weight	\sim
Rib		

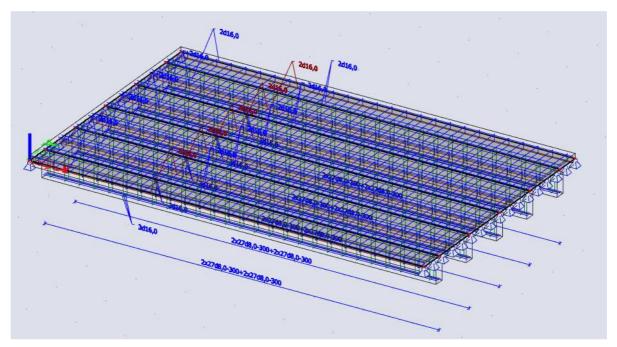
When disabled you look at the results of the slab only, when enabled the part of the results that was already summed up in the T-sections is left out of the result.

5. Reinforcement in T-sections

The effective width is an approximation from the norm, where the connection beam-plate is replaced by a Tbeam for the design of the reinforcement. By selecting the option Rib, the internal forces in the beam are adapted. These adapted forces represent the forces in the T-section, so they can be used to design the reinforcement in the T-beam.

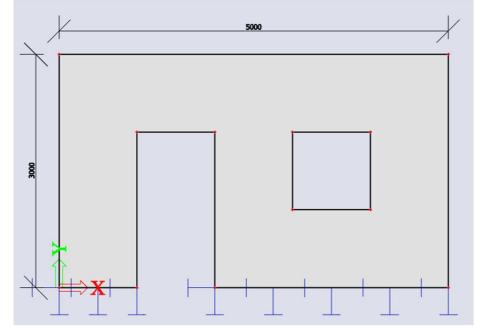
Suppose: effective width = distance between the ribs

Filter the input panel workstation to "Concrete" and under "Concrete reinforcement" choose for "Add reinforcement on whole beam". Select the ribs to insert user reinforcement:



Example 11: Precast wall

- 1. Input of the geometry
- Project data: Wall XY, Concrete C25/30, thickness 150 mm



• Input of wall

Filter the input panel workstation to "Structure" and choose for "Plate" under "2D members". When doing so under the Scia Spotlight some options for the shape appear, choose for rectangle:

Nouvelle plaque - Nou	veau	polyg	one - I	Premie	r poir	nt>							
	14	0	¢,	\odot	0	đ	1	1	5	\wedge	\cap	S	5

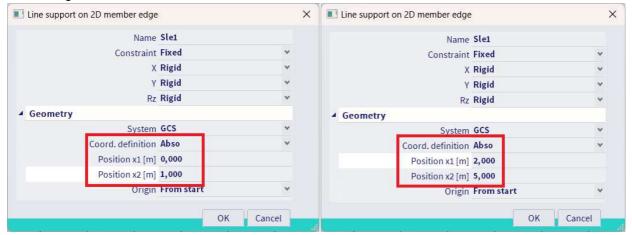
Enter a wall using the coordinates "0 0 0" and "3 5 0".

Filter the input panel workstation to "Structure" and choose for "Opening on 2D" under "2D members". Enter two rectangular openings for the door and window using the coordinates "1 0 0"-"2 2 0" and "3 1 0"-"4 2 0".

• Input of supports

Filter the input panel workstation to "Structure" and choose for "Line support on 2D edge" under "Boundary conditions".

We want to separate the parts left and right of the door opening, therefor do the above action twice with the below settings:

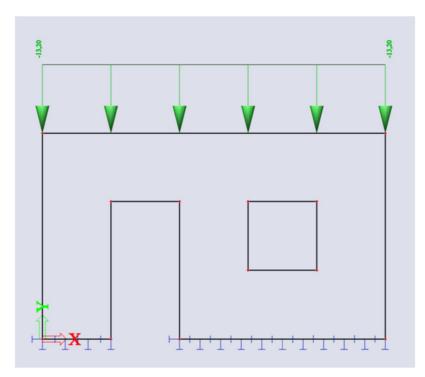


2. Loads

LC1: Self weight

LC2: Slab (Perm.) → Insert a line load of 13,2 KN/m

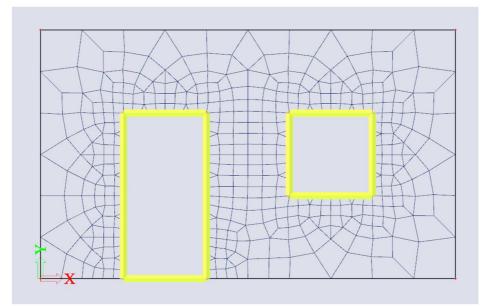
Note: because we choose "Wall XY" in the project settings, the self weight is in the direction of the Y-axis by default.



3. Refine the mesh

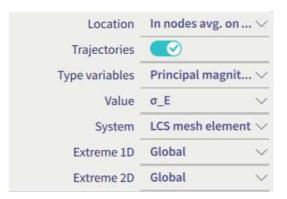
Refine mesh via Main menu > Tool > Calculation, Mesh > Mesh setting; average size of 2D mesh element = 0,30 m

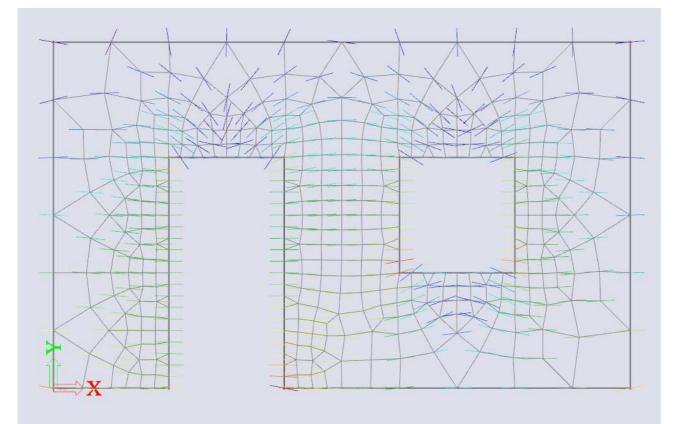
To locally refine around the openings, filter the input panel workstation to "Calculation & Results" and choose for "Edge mesh refinement" under "Result tools". Use a setting of 0.1m and then select all the 2D edges around the openings:



4. Results

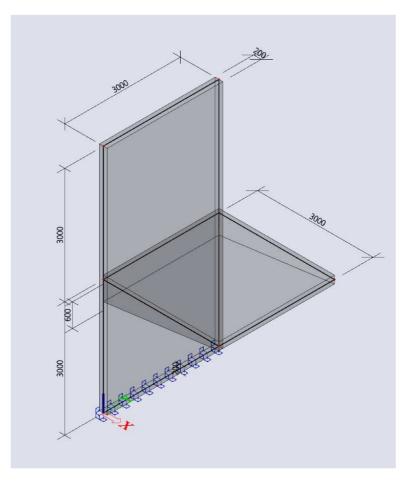
Go to the result branch of the process toolbar and choose for "2D stresses/strains". Set the result properties as shown below to show the trajectories of the main principal stresses:





Example 12: Balcony

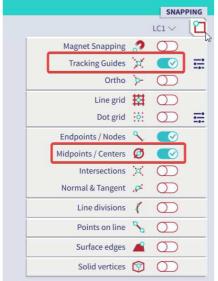
- 1. Input of the geometry
- Project data: General XYZ, Concrete C30/37



• Input of the balcony

Filter the input panel workstation to "Structure" and choose for "Wall" under "2D elements". Insert a wall of height 6m on coordinates "0 0 0" and "0 3 0".

Filter the input panel workstation to "Structure" and choose for "Plate" under "2D elements". Insert a slab of 3x3m using the below snap settings:



Adjust the thickness of the slab with the below settings in the property panel to get a varying height:

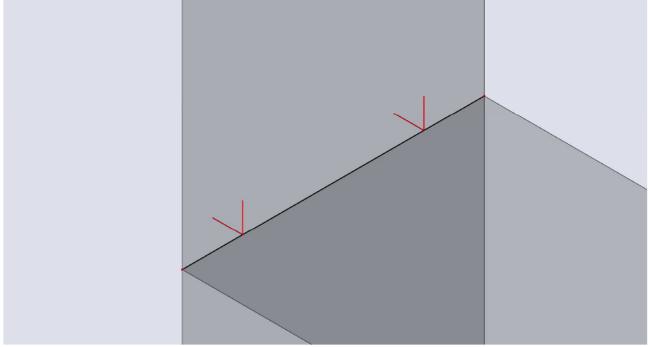
▼ VARIABLE THICKNE	SS	
Thickness type	variable	\sim
Direction	Global X	\sim
Point 1	N5	\sim
Thickness Th. [mm]	600	
Point 2	NG	\sim
Thickness Th. [mm]	200	
Member system-pla	Тор	\sim
Eccentricity z [mm]	100	
LCS type	Standard	\sim
Swap orientation	\bigcirc	
LCS angle [deg]	0,00	=
3D Wind	0	

The node numbers might differ depending on the direction you used to input the slab.

2. Actions after the input of the geometry

- Tools > Check structure
- Edit > Modify > Connect members/nodes Or you can simply type "Connect members/nodes" in the SCIA Spotlight. Note that you can do this action both for a selection or the whole structure (select nothing to do the latter).

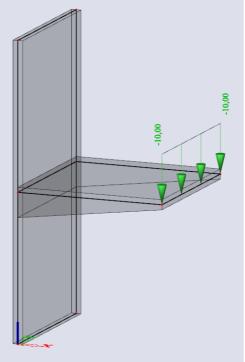
Note that the connection between the wall and slab is now marked with red lines to indicate the two elements are properly connected:



3. Loads

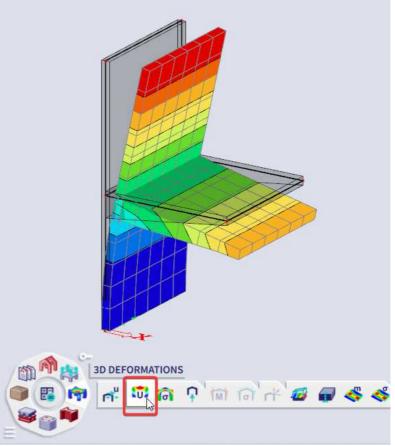
LC1: Self weight

LC2: Balcony (perm) → Insert a line load of 10 KN/m on the outer edge of the balcony



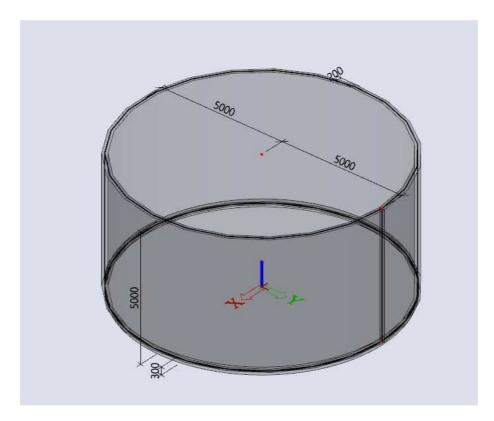
4. Results

Check if the displacements are as expected. Go to the results branch of the Process Toolbar and choose for 3D deformations:



Example 13: Tank

- 1. Input of geometry
- Project data: General XYZ, concrete C25/30



Input of tank

Filter the Input Panel workstation to "Structure" and choose for "Plate" under "2D members". When doing so under the Scia Spotlight some options for the shape appear, choose for circle by radius:



Enter a circular slab by using the coordinates "0 0 0" and "5 0 0"

Filter the Input Panel workstation to "Structure" and choose for "Wall" under "2D members". When doing so under the Scia Spotlight some options for the shape appear, choose for "select line":



Then you can simply select the edge of the slab you already created to insert a cylindrical wall of height 5m.

• Input of supports

Look for "Surface support on 2D" under "Boundary conditions" in the Input Panel. Copy a predefined subsoil to the project library, click "OK" and assign the support to the bottom slab of the tank.

2. Loads

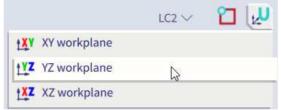
LC1: Self Weight

LC2: Variable pressure (var.) \rightarrow Variable surface load of 0 to" 50 KN/m²

Input of free surface load:

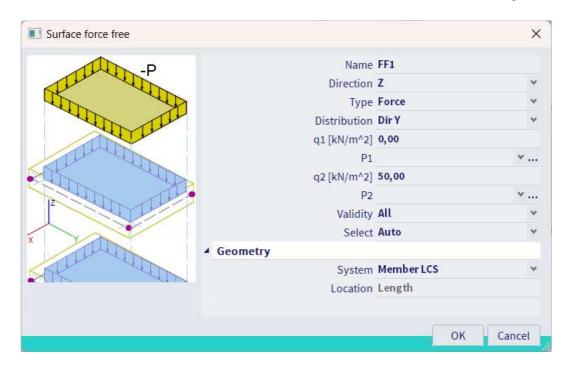
• UCS

A free load is always inputted on the UCS workplane and generated on some elements afterwards. Therefor first we are going to set the UCS vertically. Choose for YZ workplane under the UCS settings:

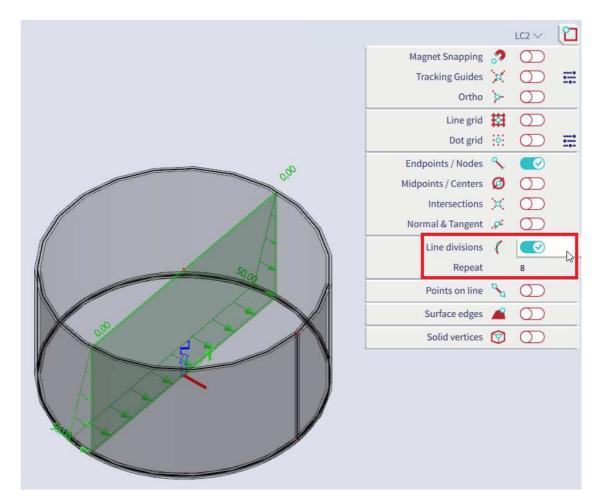


• Free load

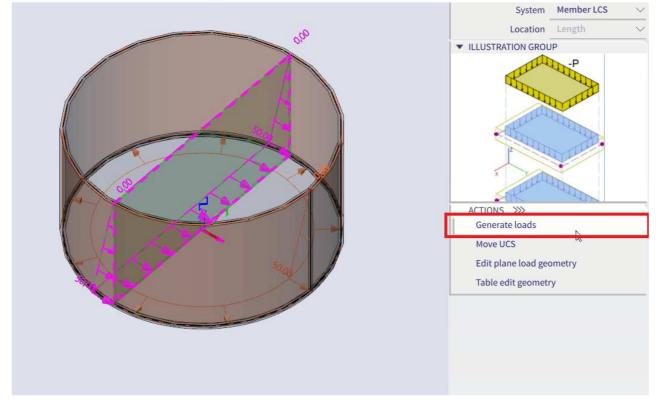
We are going to draw a free load over the whole projection of the tank and afterwards let Scia generate the loads according the LCS system of the tank. Filter the Input Panel workstation to "Loads" and choose for "Free surface loads" under "Surface loads". Use the below settings:



Draw a free surface load using the below snap settings:



Note: if you want to check how the load will be generated in the calculation, select the free load and click on "Generate loads" on the bottom of the property panel:



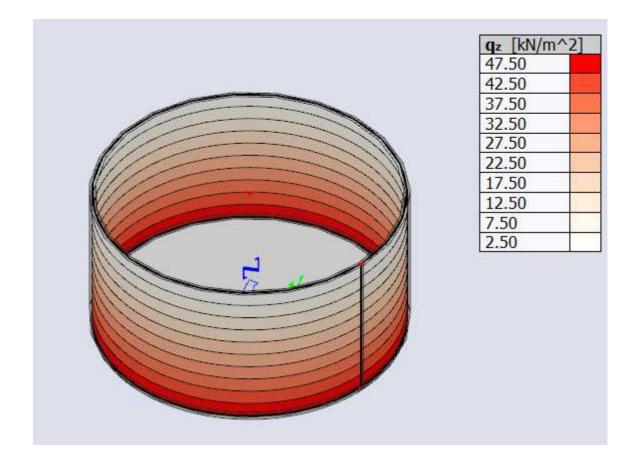
3. Refine the mesh

Refine mesh; size of mesh elements = 0,2m

4. Check of loads

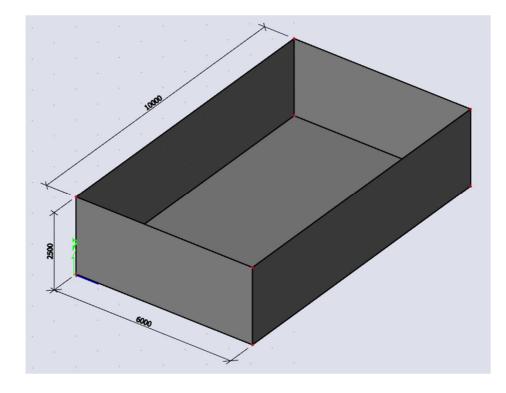
- Run the linear calculation
- Go to the results branch of the Process Toolbar and choose for "Surface loads" to see how the loads are taken into account:





Example 14: Swimming pool

- 1. Input of the geometry
 - Project data: General XYZ, Concrete C25/30, Thickness of elements: 300 mm



• Input of pool

-Filter the Input Panel workstation to "Structure" and choose for "Plate" under "2D members". When doing so under the Scia Spotlight some options for the shape appear, choose for rectangle:



And enter coordinates "0 0 0" and "6 10 0" in the Scia Spotlight.

-Filter the Input Panel workstation to "Structure" and choose for "Wall" under "2D members". When doing so under the Scia Spotlight some options for the shape appear, choose for "select line":



Then you can simply select the edges of the slab you already created to insert 4 walls of height 2,5m.

• Input of supports

Look for "Surface support on 2D" under "Boundary conditions" in the Input Panel. Copy a predefined subsoil to the project library, click "OK" and assign the support to the bottom slab of the tank.

2. Loads

LC1: Self Weight

LC2: Water pressure (var.) → Free surface load from 0 till 25 KN/m²

LC3: Ground pressure \rightarrow Soil pressure load from borehole data

Input of free surface load:

• UCS

A free load is always inputted on the UCS workplane and generated on some elements afterwards. Therefor first we are going to set the UCS vertically. Choose for YZ workplane under the UCS settings:

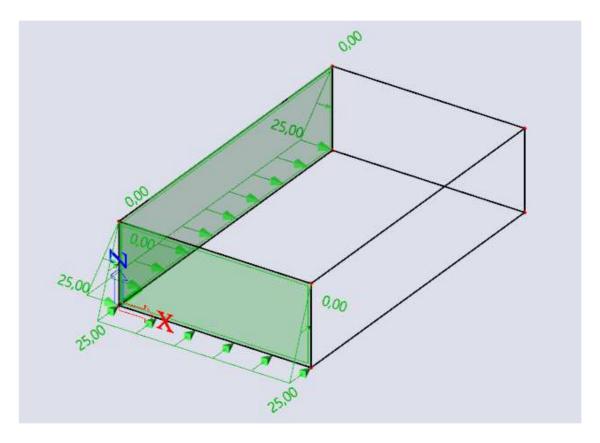
	LC2 \smallsetminus	2
XY workplane		
YZ workplane	6	
XZ workplane		

• Free load

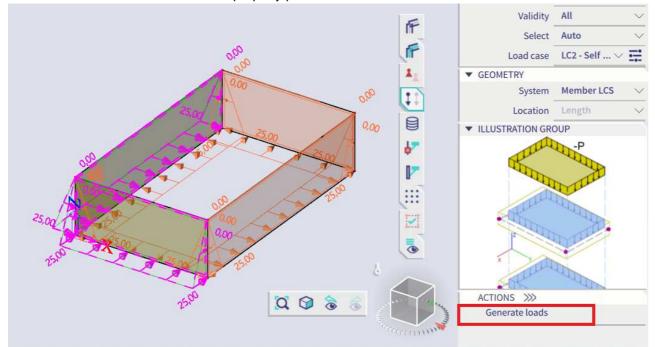
We are going to draw a free load over the whole projection of the tank and afterwards let Scia generate the loads according the LCS system of the tank. Filter the Input Panel workstation to "Loads" and choose for "Free surface loads" under "Surface loads". Use the below settings:

Surface force free				×
-P	Name	FF1		
The way	Direction	Z		*
	Туре	Force		*
	Distribution	Dir Y		*
ALC ALC	q1[kN/m^2]	0,00 🗟		
auth	P1			۰
ATT -	q2 [kN/m^2]	25,00		
	P2			×
	Validity	All		*
x	Select	Auto		*
	Geometry			
Ster and	System	Member LCS		~
COLING STIN	Location	Length		
		_	OK	Cancel

Draw a free surface and do so for UCS YZ as well as XZ to have loads in both directions:



Note: if you want to check how the load will be generated in the calculation, select the free load and click on "Generate loads" on the bottom of the property panel:



Note: remember we selected element LCS to generate the loads, if the load would be in the wrong direction you would have to invert the 2D element with this option in the property panel:



Input of soil pressure load:

• Borehole data

It is possible in Scia to automatically let generate the soil pressure on 2D elements based on a ground profile. Filter the Input Panel workstation to "Structure" and choose for "Borehole data" under boundary conditions. Fill in the below properties in the borehole data:

Geologic profile						×
		Thickness =	2.50[m],	Edef = 15.00[MN/	m^2], Weight = 18.	00[kN.
Name	-	Edef[MN/m^2]	Poisson		Wetweight[kN/m^3]	m
1	2,50	15,00	0,200	18,00	20,00	0,20
*	0,00	0,00	0,000	0,00	15,00	0,20
Water level	2,500 m	Name	e GP	1		
Non-compre	essible subsoil belo	ow the last inputte	d layer		ОК С	ancel

The top of the profile should represent the zero level of the ground, therefore input the borehole on top of the pool.

Soil pressure load

Filter Input Panel workstation to "Loads" and choose for "Surface load on 2D" under "Surface loads":

Surface force			×
	Name	SF1	
	Direction	Z	*
	Туре	Soil pressure	¥
-P	Coeff	0,330	
The way	Borehole profile	BH1	*
	▲ Geometry		
The states	System	LCS	*
ALL P	Location	Length	
x Y		ОК	Cancel

The coefficient can be used to define the active ground coefficient. Click on ok and select the walls of the pool that should be loaded with a horizontal ground load.

3. Refine mesh

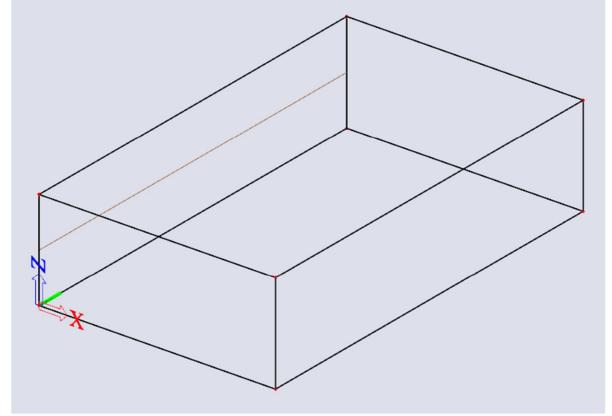
Refine mesh; size of mesh elements = 0,3m

4. Results

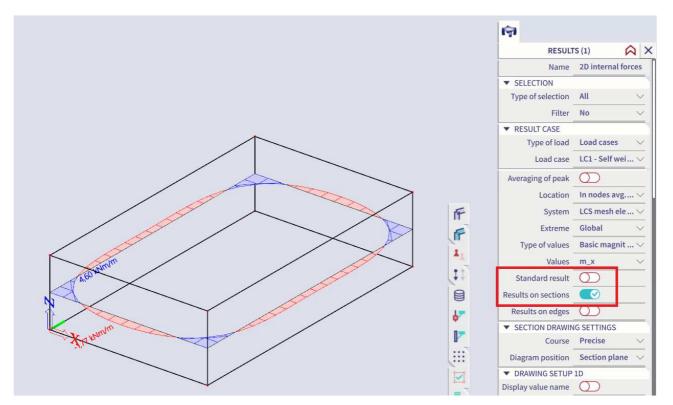
Search for "Section on 2D" in the Scia Spotlight. Make a section in X-direction:

Section on 2D member	×
Name	SE1
Draw	Z direction 👻
Direction of cut [m]	1,000/0,000/0,000
Layer	Layer1 Y
	OK Cancel

Draw a section in the middle of one of the pool walls:

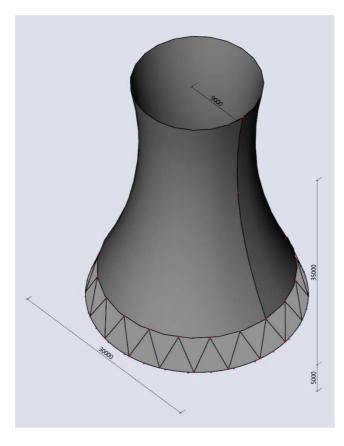


Go to the result branch of the Process Toolbar and choose for "2D internal forces". Enable section results and disable standard results to look at internal forces on the exact location of the section:



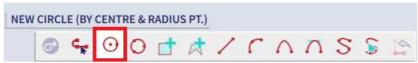
Example 15: Cooling Tower

- 1. Input of geometry
- Project data: General XYZ Concrete C30/37 Shell thickness 200mm



Input of base slab

Filter the Input Panel workstation to "Structure" and under "2D members" choose for "Plate". Under the Scia Spotlight choose for "New circle (by center & radius Pt.)" and enter coordinates "0 0 0" and "15 0 0".



• Input of tower

-Insert the line for revolution

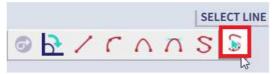
Type "Line" in the Scia Spotlight and choose for "New parabolic arc" from the shape options:



Enter the following coordinates:"13,5 0 5" - "8 0 25" - "9 0 40"

-Filter the Input Panel workstation to "Structure" and under "2D members" choose for "Shell – surface of revolution".

Under the Scia Spotlight choose for "Select line" and select the previously create parabolic arc, end with "ESC":



Rotation ang	le and axis	\times				
ROTATION						
Angle <u>360,00</u> deg						
AXIS VECTO	R					
Working plane axis X Working plane axis Y Working plane axis Z Define axis by cursor Enter custom axis vector						
CUSTOM A	XIS VECTOR					
x	0,000	m				
y 0,000 m						
z 0,000 m						
OK Cancel						

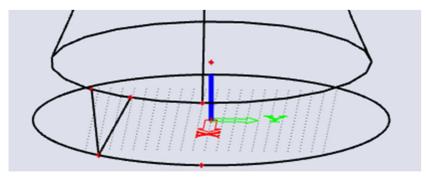
Click on "OK" and enter the origin's coordinates: "0 0 0".

• Input of pillars

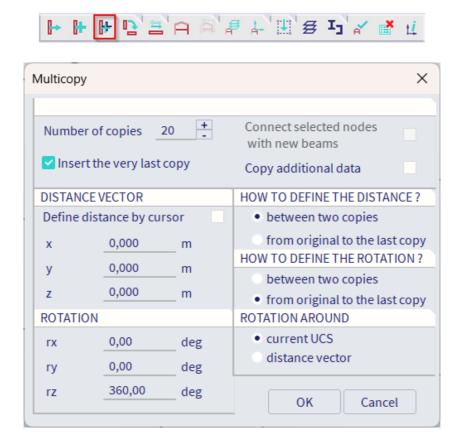
-Go to the snap settings and insert a value of "40" under "Line divisions"

		LC1 \sim	2
Magnet Snapping	2	\bigcirc	
Tracking Guides	, pr		=
Ortho	` p -	\bigcirc	
Line grid	斟	\bigcirc	
Dot grid	Ó	\bigcirc	=
Endpoints / Nodes	\$		
Midpoints / Centers	Ø		
Intersections	Ø	\bigcirc	
Normal & Tangent	. ç ÷	\bigcirc	
Line divisions	(
Repeat		40	
Points on line	S	0 [°]	3
Surface edges		0	
Solid vertices	1	0	

Filter the Input Panel workstation to "Structure" and under "1D members" choose for "1D member". Choose a circular cross-section of diameter 500mm and draw 2 V-shaped columns:



Select the two columns and in the structure branch of the process toolbar choose for multicopy:



• Input of supports

-Filter the Input Panel workstation to "Structure" and under "Boundary conditions" choose for "Line support on 2D edge".

2. Actions after input

- Check structure data (main menu > tools > check structure)
- Connect members and nodes (main menu > edit > modify > Connect members and nodes)

The latter is automatically done when any calculation is run.

3. Loads

- Load cases
- LC 1: Self weight
- LC 2: Temperature load (Var.) > Thermal on 2D member, Delta = 40 K
- LC 3: Wind load (Var.) > Surface load 0 to 1,4 kN/m²
 - Load groups
- LG 1: Permanent
- LG 2: Variable, EC1 Load type = Temperature
- LG 3: Variable, EC1 Load type = Wind

• Temperature load

Switch the load cases dropdown menu to "LC2", filter the Input Panel workstation to "Loads" and choose for "Temperature load on 2D " under "Temperature loads":

I Thermal on surface		×
	Name	ST1
	Distribution	Constant 🗸
	Delta [K]	40,00
		OK Cancel

Click on "OK" and select the shell.

Free surface load

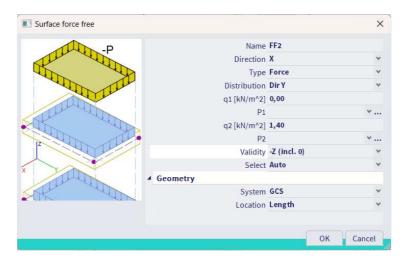
Input a free surface load to simulate the wind pressure.

-UCS

A free load is always inputted on the UCS workplane and generated on some elements afterwards. Therefor first we are going to set the UCS vertically. Choose for YZ workplane under the UCS settings:

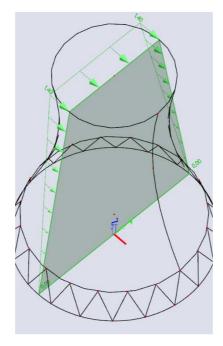
	LC2 🗸	2
XY workplane		
YZ workplane	5	
XZ workplane		

We are going to draw a free load over the whole projection of the tower and afterwards let Scia generate the loads according the GCS system. Filter the Input Panel workstation to "Loads" and choose for "Free surface loads" under "Surface loads". Use the below settings:

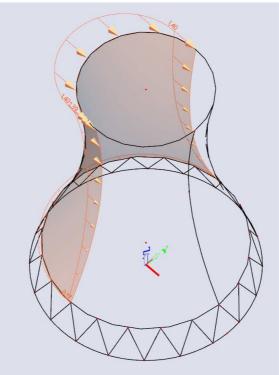


⁻Free load

Draw a free surface load and make sure you cover the whole projection of the building:



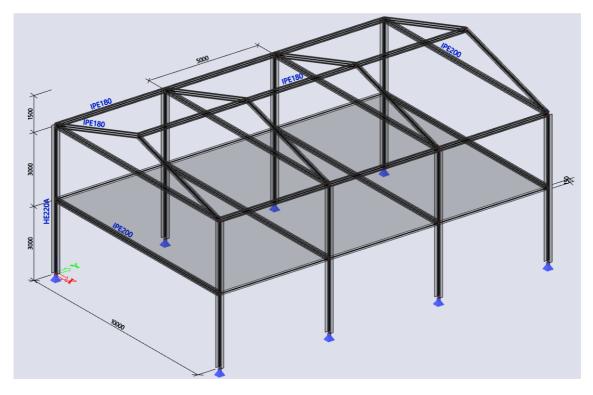
Note: if you want to check how the load will be generated in the calculation, select the free load and click on "Generate loads" on the bottom of the property panel:



Example 16: Steel hall with concrete plate

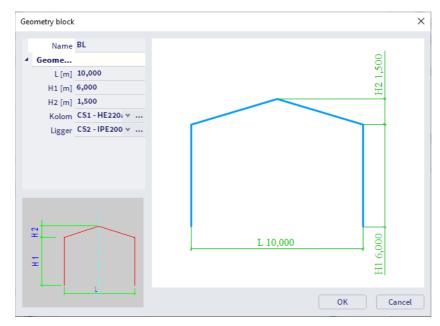
1. Input of geometry

• Project data: General XYZ – Concrete C25/30 – Steel S 235



• Input hall

- Provide the correct types of cross-sections in the cross-section library and then input one frame first. In the Input Panel set the category filter to 'Import & Blocks' and choose for 'Catalog blocks'. Choose for frame 2D and fill in values as shown below:



Insert the block on the origin by typing "0 0 0" in the command line.

Then provide one additional beam between the midpoints of the columns and one additional beam between the end points of the column.

You can do that by using the marking menu (ALT+RMB):



- Select all the 1D elements of the frame and also the top 3 nodes. From the structure branch of the process toolbar, choose for "Multicopy".

Aulticopy			×
Number	r of copies	3 +	Connect selected nodes vith new beams
🗹 Inser	t the very last	сору	Copy additional data 🛛 🗹
DISTANO	CE VECTOR		HOW TO DEFINE THE DISTANCE ?
Define o	listance by cu	rsor	 between two copies
x	0,000	m	• from original to the last copy
v	5	m	HOW TO DEFINE THE ROTATION?
z	0,000	m	 between two copies from original to the last copy
ROTATIO	DN		ROTATION AROUND
rx	0,00	deg	current UCS
ry	0	deg	o distance vector
rz	0,00	deg	OK Cancel

Choose a cross-section for the connecting beams in the next window.

- Input the slab using the marking menu (ALT + RMB):



Choose for a rectangular shape and click on two outer middle nodes of the columns:



2. Connect elements

• Automatic

In the main menu go to "Edit" > "Modify" > "Connect members/nodes". Enable "Check structure data" in the next window if you want to check the structure in one go.

• Manual

2D elements in Scia are by default not connect with in plane beams you added to the model. You can do two things if you want them to be connected:

- Create an internal edge on the 2D element on the exact same location of the beam so that Scia can create mesh nodes along this edge to connect the elements.

Internal ed <mark>ge</mark>		$\mathbf{?}$
🔼 Internal edge	0 🖻	
	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	

On the boundary edges of the slab the above is not necessary because there obviously already exists an edge.

-Convert the 1D elements or beams into ribs. There is an additional option in "Connect members/nodes" to do this automatically:

	Align structural entities to planes (moving n			
	Align			
-	Geometrical tolerance			
Z X X	Min. distance of two nodes, node to curve [m] 0,001			
	Max. distance of node to 2D member plane [m] 0,000			
	Connect (generate linked nodes, intersectio			
	Connect 🗹			
	Autorestore buckling group when reconnecting 🔽			
	Connect 1D members as ribs 💌			
	Connect 1D members with rigid links			
	Max. length of rigid link [m] 0,100			
	Create new linked node for master node 🔽			
	Check structure data			
	Check (merge duplicate nodes, erase invalid entities) 🔽			

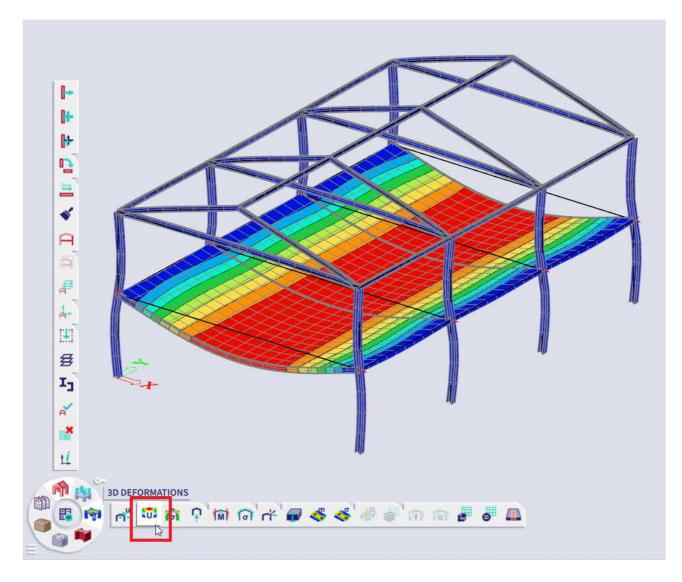
## 3. Load cases

LC1: Self weight

LC2: Service load (var.)  $\rightarrow$  Insert a surface load of 2 KN/m²

## 4. Results

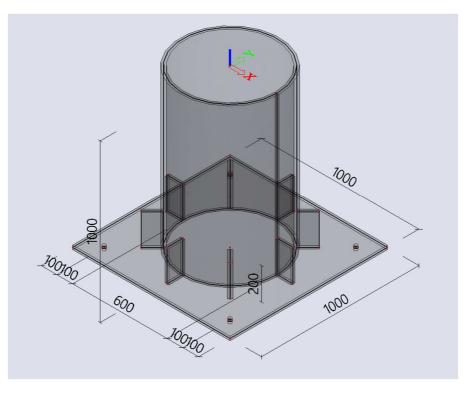
After the calculation verify if the beams and the slab are moving together in the 3D deformations:

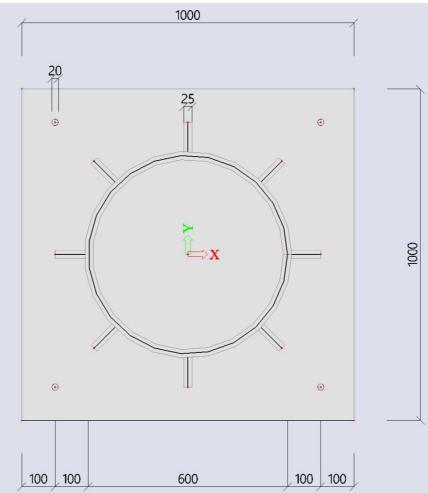


# Example 18: Detailed study of a column base

## 1. Input of geometry

• Project data: General XYZ, steel S 235





### • Input column base:

You can narrow down the content of the Input Panel by setting the category filter to '2D Members' and choose for "Plate":



Set the properties, click "OK" and choose for a rectangular shape of plate definition:



Enter coordinates "0 0 0" and "1 1 0".

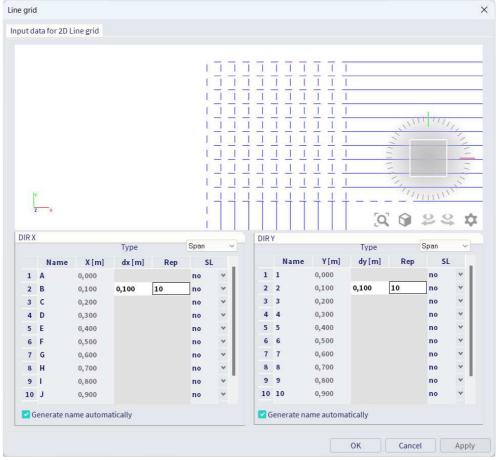
Use the same Input Panel filtering and select "Wall". Set the properties, click "OK" and choose for "new circle (by center & radius PT.)"

Enter coordinates "0,5 0,5 0" and "0,8 0,8 0"

### • Input bolt holes:

Diameter of the holes is 20 mm.

Filter the Input Panel category to "Grids & stories" and choose for "Rectangular grid". Fill in the values as shown below, press "OK" and enter the coordinates "0 0 0" followed by enter in the SCIA Spotlight:



### Enable the snap to line grid:



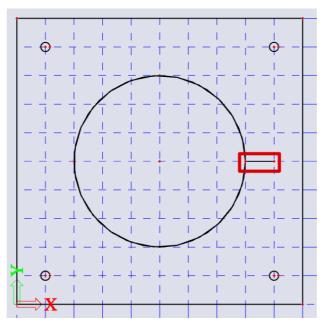
Search for "Opening on 2D" in SCIA Spotlight and create the first opening by choosing "New circle (By center & radius pt.)" underneath the SCIA spotlight, snapping on the grid lines and typing "@0,01 0 0" in the SCIA Spotlight to define the radius.

Afterwards copy the opening to the correct locations with the below icon from the Process Toolbar:

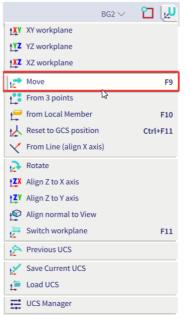
₽

• Input of stiffeners:

Filter the Input Panel workstation to "Structure" and under 2D members choose for "Wall". In the properties choose a height of 0,2m and use the grid lines to define the wall:



We are going to use multicopy to define the remaining part of the stiffners. But a rotational copy is always defined around the UCS sytem. So we are going to move it to the center of the column:



Afterwards use "Multicopy" from the Process Toolbar and use the below settings:

Multicopy X						
Number of copies 7 +			Connect selected with new beams	nodes		
Insert the very last copy			Copy additional d	lata 🔽		
DISTANCE VECTOR			HOW TO DEFINE THE DISTANCE ?			
Define distance by cursor 🛛 🔽		between two	copies			
x	0,000	m	from original	to the last copy		
v	0,000	m	HOW TO DEFINE T	HE ROTATION ?		
y		- "	<ul> <li>between two</li> </ul>	copies		
Z	0,000	m	from original	to the last copy		
ROTATION		ROTATION AROUND				
rx	0,00	deg	current UCS			
ry	0,00	deg	distance vector			
rz	45	deg	ОК	Cancel		

### • Input supports

Filter the Input Panel category filter to "Boundary conditions" and: -Apply surface supports on the base plate. -Apply line support on 2d edge of opening.

### 2. Actions after the input of the geometry

### • Tools > Check structure

Edit > Modify > Connect members/nodes
 Or you can simply type "Connect members/nodes" in the SCIA Spotlight. Note that you can do this action both for a selection or the whole structure (select nothing to do the latter).

#### Note: connecting the elements is done by default when you perform a calculation:

## 3. Charges

### Load cases

LC1: Self weight

LC2: Normal force: -60 kN/m at the top edge of the column LC3: Moment: 20 kNm/m at the top edge of the column in global Y direction **p** 

### Combinations

Linear – ULS: 1,00*LC1 + 1,00*LC2 + 1,00*LC3

### 4. Finite element mesh

Global mesh refinement: "Main menu" > "Tool" > "Calculation & Mesh" > "Calculate" or click in the center of the Process Toolbar wheel. Size of the mesh elements = 0,025 m.

Local mesh refinement around the bolt openings: "Input Panel" > "Local mesh refinement" > "Node mesh refinement". Set radius to 0,05 m, ratio to 0,01 and select centers of openings.

Mesh generation: "Main menu" > "Tool" > "Calculation & Mesh" > "Generate mesh". This is automatically done when you run the calculation.

Display the mesh: right click on screen > "View parameters for all entities":

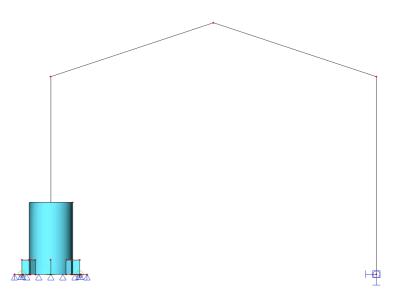
Vie	ew parameters setting	
C	heck / Uncheck gro	Lock position
٩		T 🔯 🧐 🌌 🔍 🕨
	Check / Uncheck all	
+	Service	
+	Structure	
+	Effective width of plate ribs	
+	Panel	
+	Tributary areas	
+	Structure nodes	
+	Member parameters	
Ξ	Mesh	
	Draw mesh	
	Free edges	
	Display mode	wired 🔹
+	Local axes	
	Show names in tab	OK Apply Cancel

### 5. Results

Go to the "Results" branch of the Process Toolbar and choose for "3D deformations". Go to the "Results" branch of the Process Toolbar and choose for "3D stresses".

## 6. Link 2D (connection detail) to 1D (global structure)

Create a frame around the detail as shown in example 2.



Filter the Input Panel category filter to "Boundary condition" and choose for "Line rigid link". Select the end node of the column and afterwards the upper edge of the detail:

