\ SCIA ENGINEER

TUTORIAL

Load generators

Table of Contents

1. Introduction

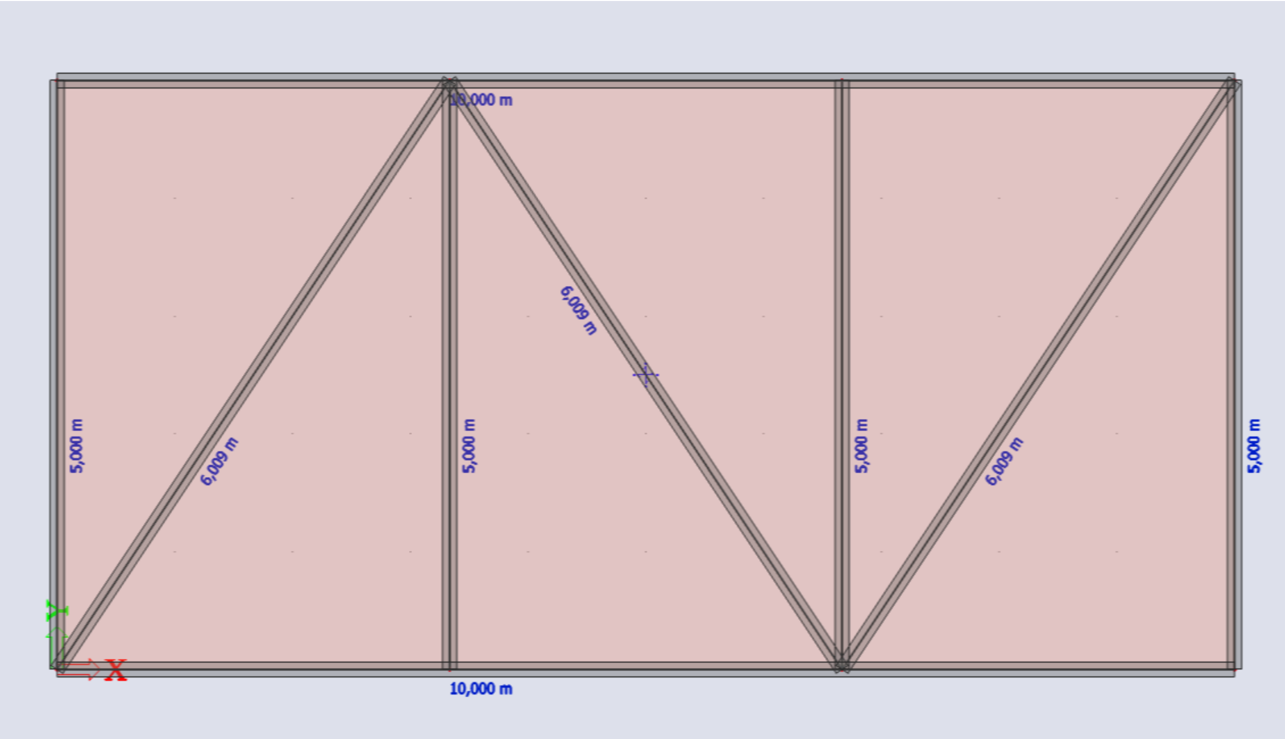
This tutorial will explain the principles and the use of all different load generators. Most of the options in the course can be calculated in SCIA engineer with the concept edition. For some functionalities an extra module (or edition) is required, this will always be indicated in those paragraphs.

This tutorial assumes that basic the modelling of a structure is already understood.

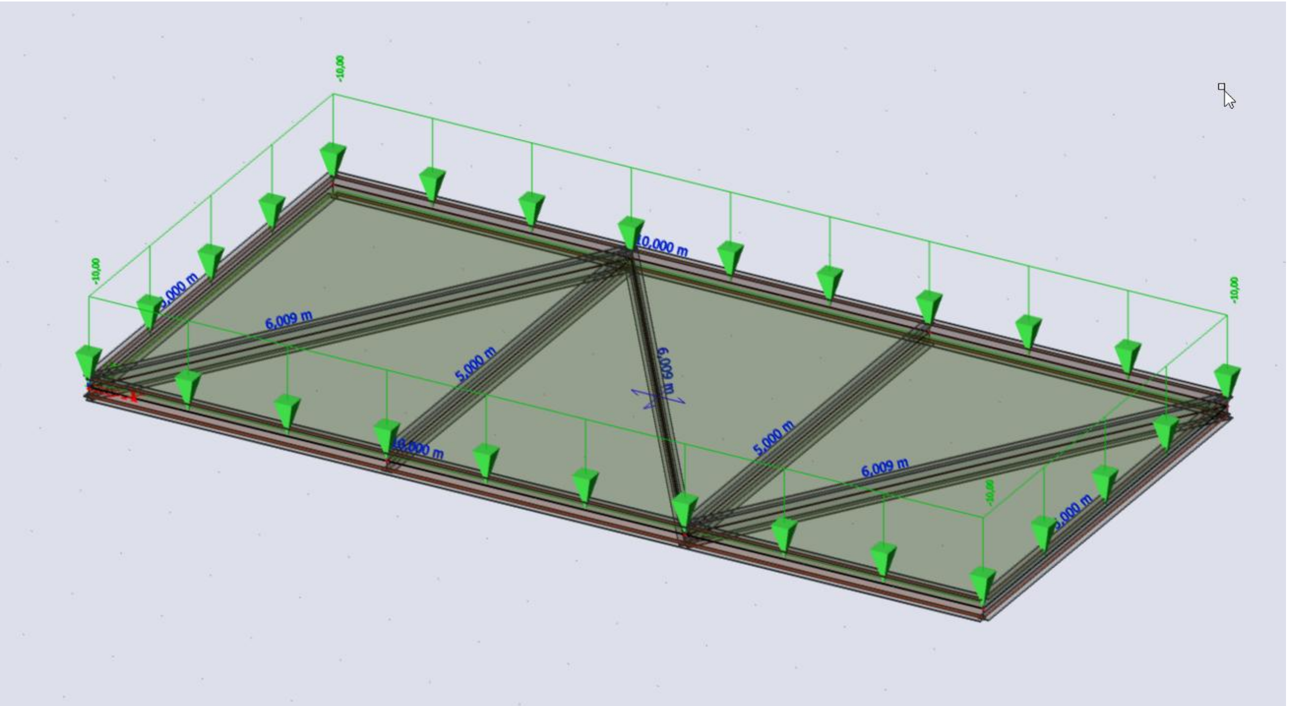
2. Load panels

Load panels are entities which are not taken into account in the FEM calculation (Finite Element Method Calculation). Load panels have no self-weight, have a certain stiffness to distribute loads to underlying members but this stiffness is not taken into account in the stiffness of the structure.
All types of loads can be applied to the load panels and will be distributed to the underlying members of the load panel.

This chapter will use an example to show the different properties of the load panels. This is the file **loadpanels.esa**.



In all the examples a surface load of -10 kN/m² is used.



2.1.General properties

You can input load panels via the input panel, the general properties are listed below.

2.1.1. Panel type

There are 3 types of load panels:

- Load to panel nodes
- Load to panel edges
- Load to panel edges and beams

Each of these types will be demonstrated in paragraph 1.2.

2.1.2. Load transfer direction

You can choose in which direction the loads should be distributed. This direction will always follow the local coordinate system of the load panel.

- In the X-direction of the load panel
- In the Y-direction of the load panel
- In both directions of the load panel

2.1.3. LCS type

This property adapts the local coordinate system of the load panel, this section is completely optional, because you will be able to turn the LCS with the property 'LCS angle'.
Note: The load panel does not have mesh elements, therefore most of the types will behave similar to each other. More information for 2D members and the LCS type can be found here.

2.1.4. Swap orientation

This function controls the direction of the local Z-axis of the panel. This direction is important when generating loads because they will follow the direction of the local axis' instead of the global coordinate system.

2.1.5. LCS angle

This function is used to rotate the x-axis of the local coordinate system and thus also the load transfer direction because this direction uses the local coordinate system.

2.1.6. Selection of entities

When this property is set on 'All', all the nodes/edges/beams in the transfer direction will be used to distribute the loads. If this option is set to 'user selection' you can use the function 'update edge/beam selection' to select the nodes/edges/beams where the load can be distributed.

- Set the value on 'user selection'
- Click 'update edge/beam selection'
- Deselect the nodes/edges/beams where the load should not be distributed to
- End the function by pressing 'esc'

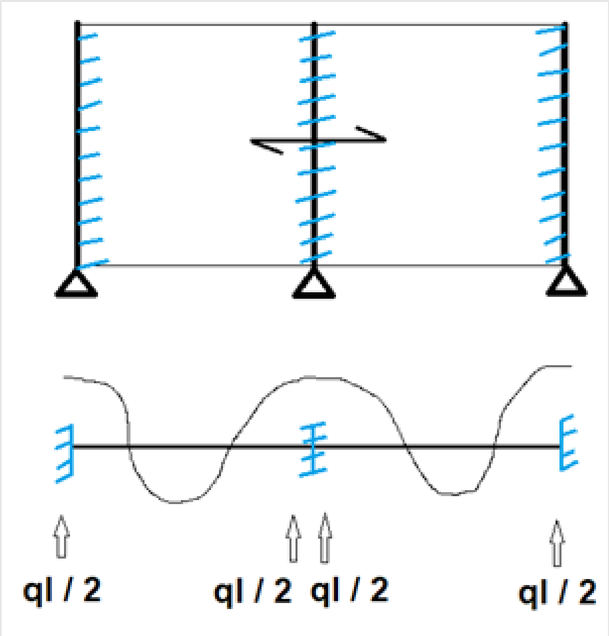
2.1.7. Load transfer method

Standard:
The sum of the load is transferred to beams according to the length of individual beams and supported edges. The user can set the weight factor for individual beams or exclude some of them (using the Beam Selection Action Button) from load transfer.

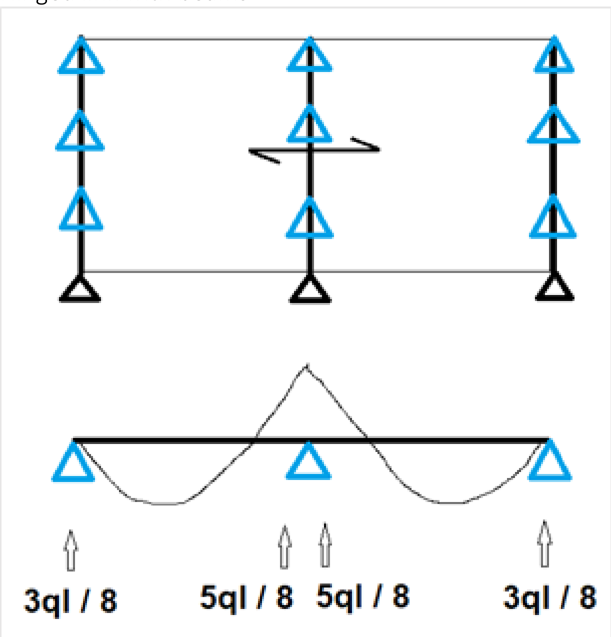
Tributary area:
The beams are loaded based on the tributary area of the particular beam.
Tributary areas are found using Voronoi diagrams based on a member's position to other neighbouring members. The influence area of a member is determined, then, by scaling up the distances from the member to the boundary of the tributary area by a factor of 2. Once scaled, any area outside the boundary of the floor to which the member is connected will automatically be trimmed and excluded from the influence area.

Accurate (fixed link with beams/hinged link with beams):
The finite element method is used to recalculate the applied load to individual beams.

- Fixed link with beams will be similar to the tributary area method.



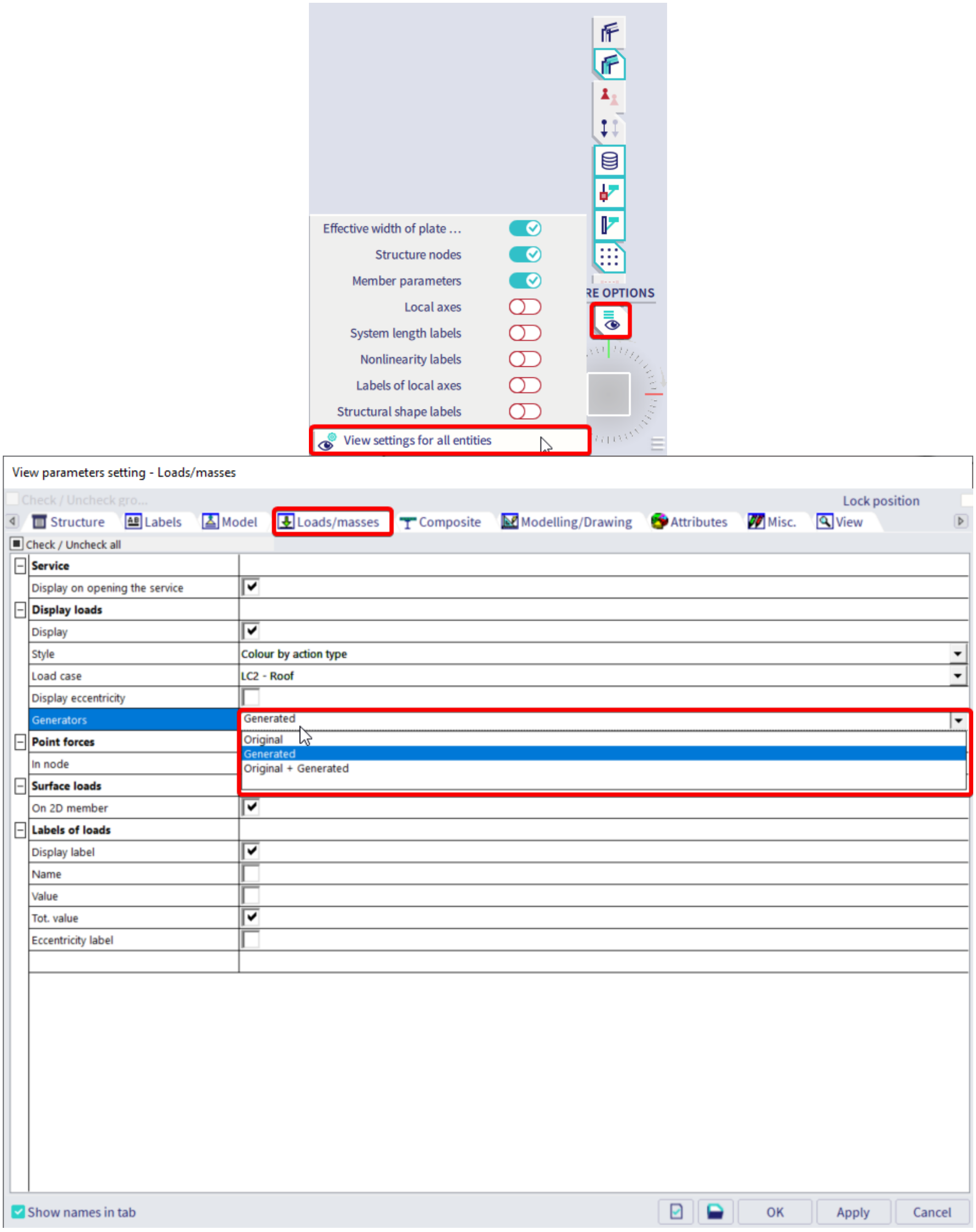
- Hinged link with beams



2.1.8. Generating loads

A load applied to a load panel will be transferred to the selected elements. These transferred loads can be generated with the action button ‘generate load’ in the properties of the load panel. The loads will be generated automatically when calculating the project.

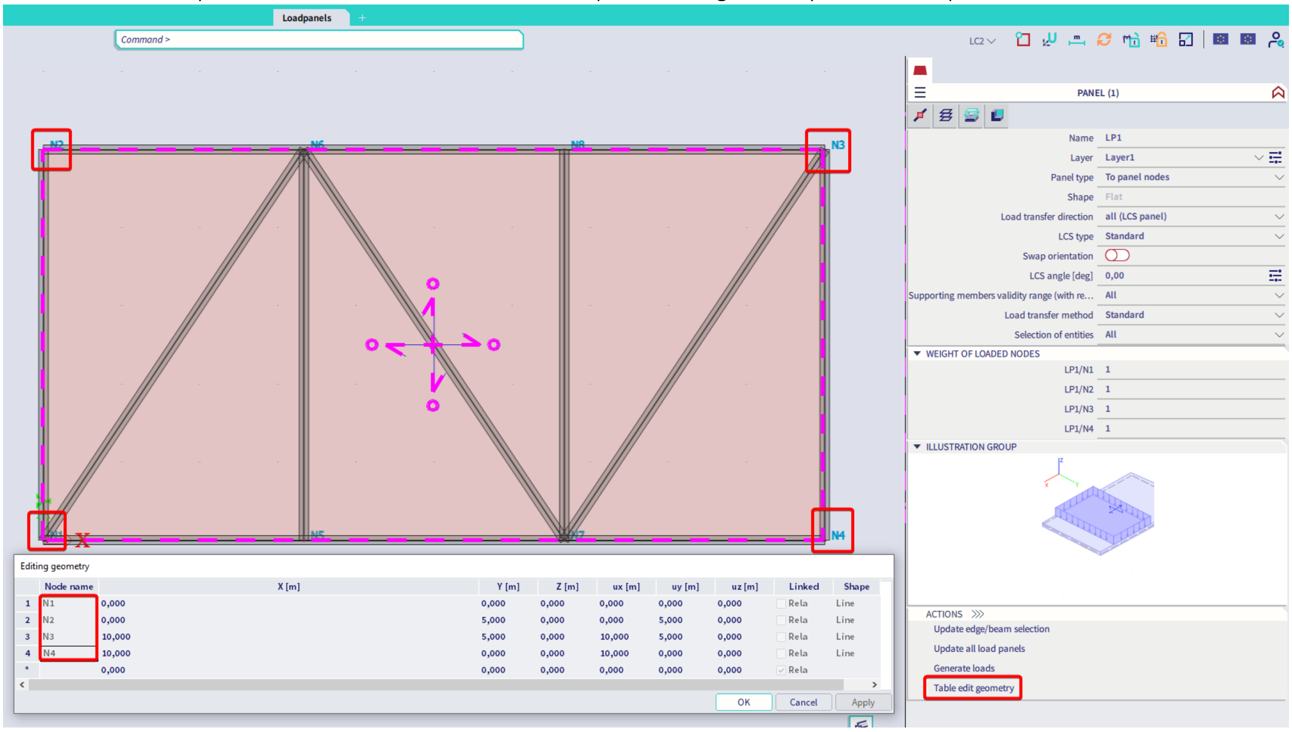
After generating, the original loads will be hidden and that the generated loads are shown. This is easily changed by changing the view parameters. If you want the original loads to stay visible, you should set the generator on ‘original + generated’.



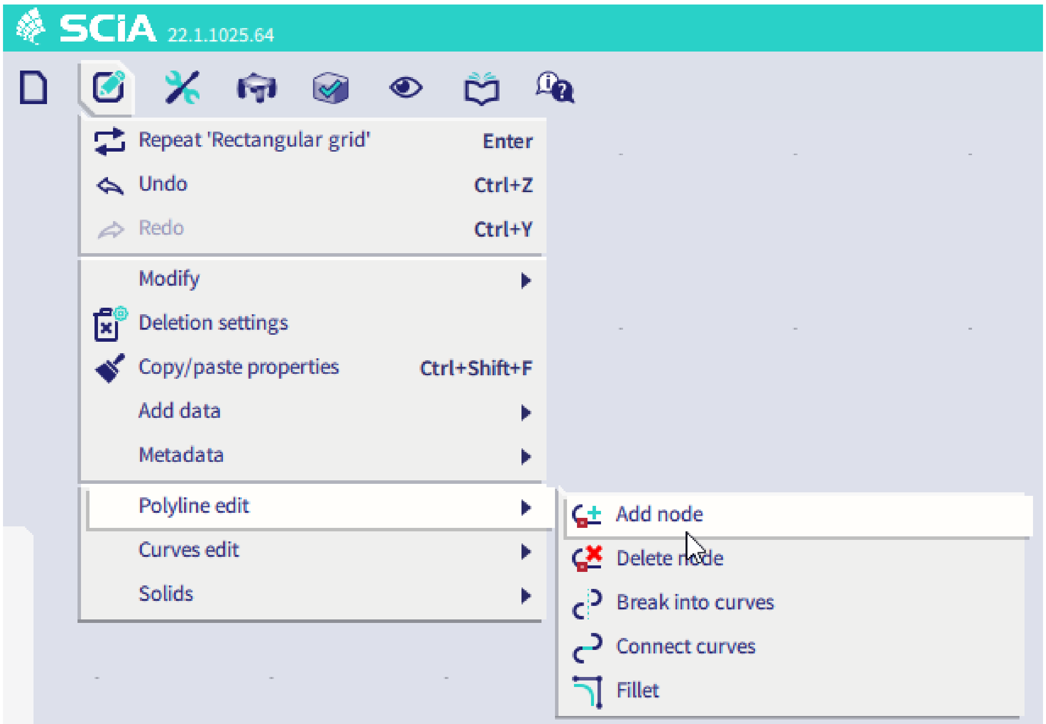
2.2. Panel types

2.2.1. Load to panel nodes

A load panel with type load to panel nodes will distribute the load to the nodes of the load panel and generate point loads. You can only transfer loads to nodes which are a part of the geometry of the load panel.



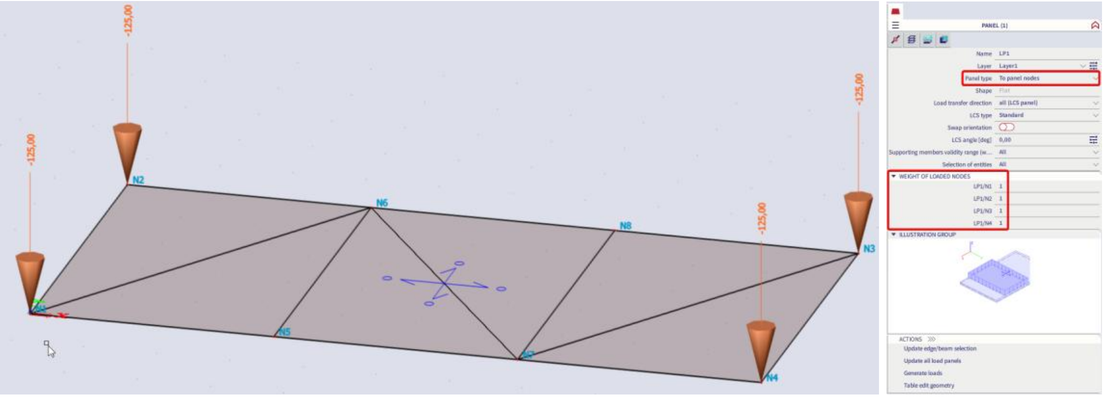
Adding nodes to the geometry of a loadpanel can be done via **edit > polyline edit > add node.** In this example the load will only be transferred to the 4 corner nodes.



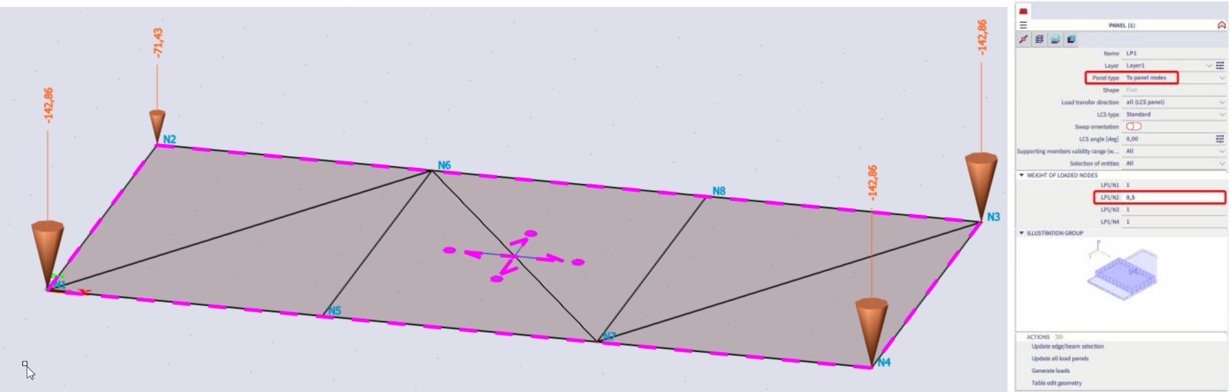
2.2.2. Load transfer method 'standard'

The **load transfer method 'standard'** allows the user to define the weight distribution factor in the property panel. The load will be transferred accordingly to that chosen weight.

In the image below, the weight distribution factor is set on 1 for each node, which means that each node will have the exact same point load: -10 kN/m² x (10m * 5m) = -500kN and -500kN / 4 nodes = -125 kN / node.



In the image below, the weight distribution factor is set to 0.5 for N2 while the remaining factors are kept at 1. The point load in node N1, N3 and N4 will be twice as large as the point load in N2.



Excluding a node can be done by changing the weight factor to zero or by changing the selection of entities as explained in paragraph 1.1.

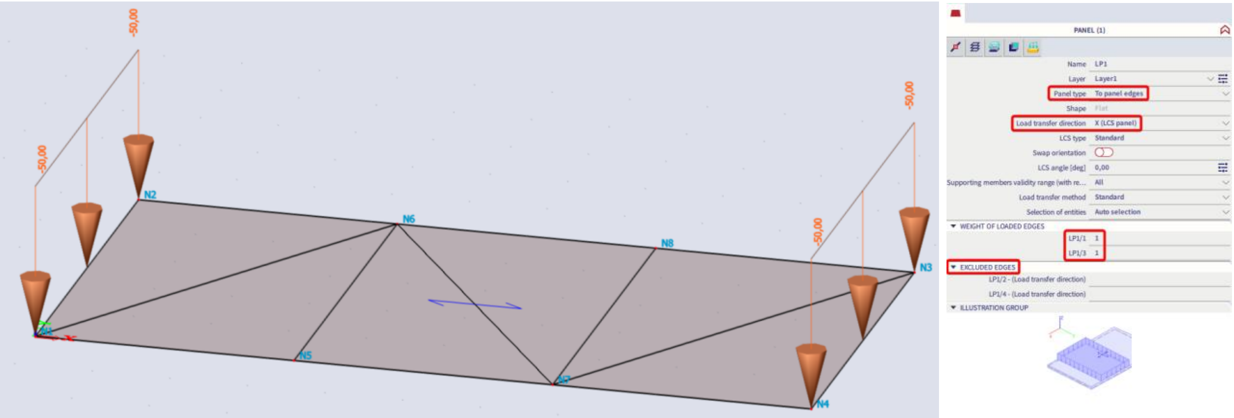
2.2.3. Load to panel edges

A load panel with type load to panel edges will distribute the load to the edges of the load panel and generate line loads. You can only transfer a load to an edge which is (partly) supported by a beam or a 2D member edge.

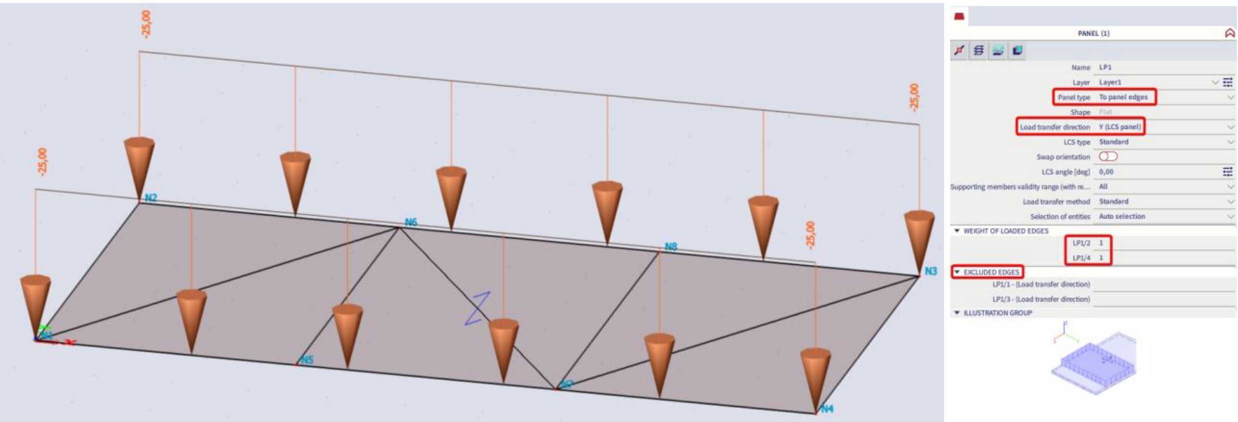
2.2.4. Load transfer method 'standard'

The **load transfer method 'standard'** allows the user to define the weight distribution factor in the property panel. The load will be transferred accordingly to that chosen weight. Also the **load transfer direction** can be modified to exclude certain edges. The weight factor for each example below is set to 1 for all edges.

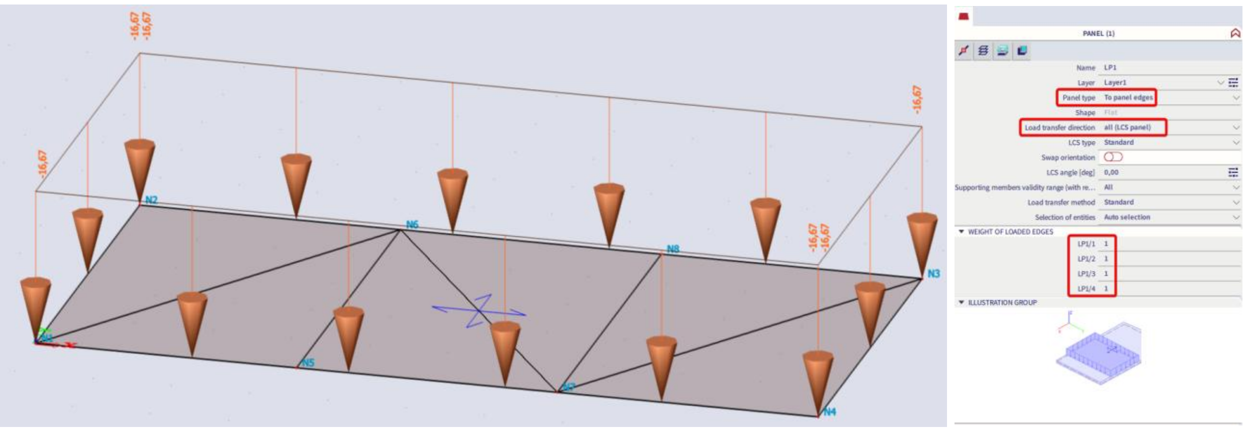
Load transfer direction X:



Load transfer direction Y:



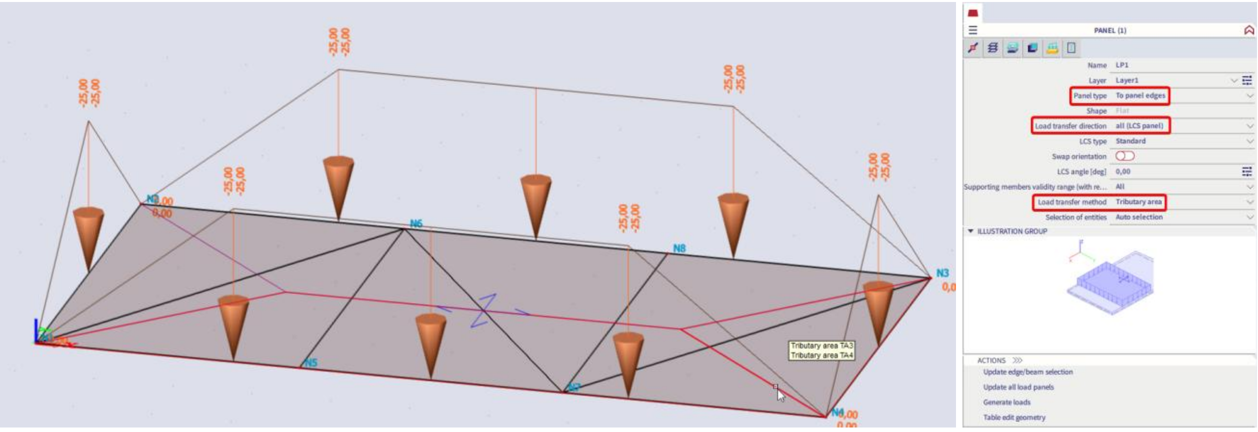
Load transfer direction All:



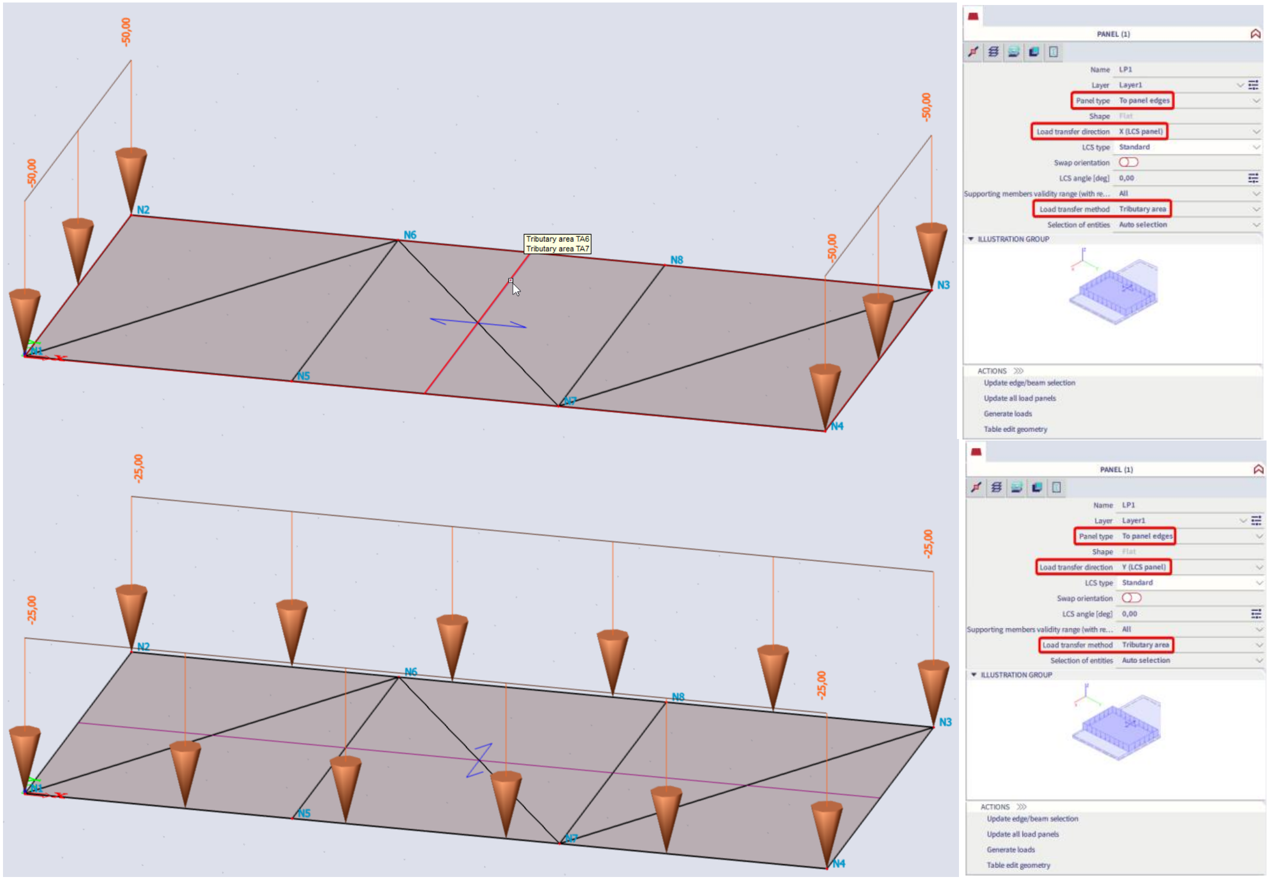
2.2.5. Load transfer method 'Tributary area'

The **load transfer method is changed to 'tributary area'** there is no possibility to change the weight factor anymore. The weight will be defined with the tributary area which is generated when updating the loadpanel. will transfer the load based on the geometry of the structure. The tributary area is visible in the graphical scene as well after updating all the loadpanels.

Load transfer direction All:



The load transfer direction X or Y will result in the same line loads as for the standard method with the weight factor set to 1 since the tributary areas for this geometry are equal.



Note: For this type of load panel the load transfer method 'FEM' is not available.

2.2.6. Load to panel edges and beams

A load panel with type load to panel edges and beams will distribute the load to the edges of the load panel and to the beams or 2D member edges in the plane of the load panel. This will again generate line loads.

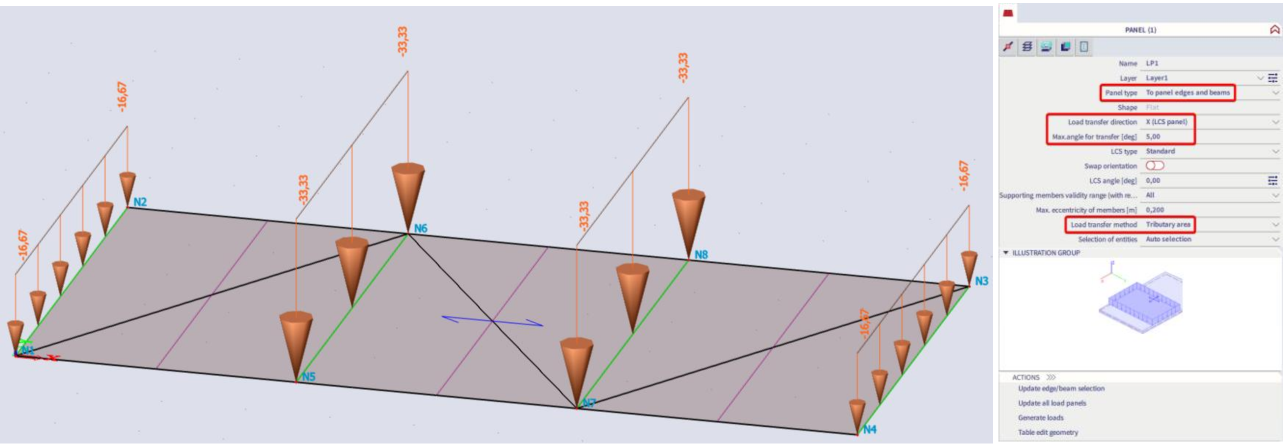
2.2.7. Max. angle for transfer

For this type of load panel only the load transfer direction X will be demonstrated since this direction will take the beams in the plane of the load panel into account. The load transfer method 'tributary area' is used in this example.

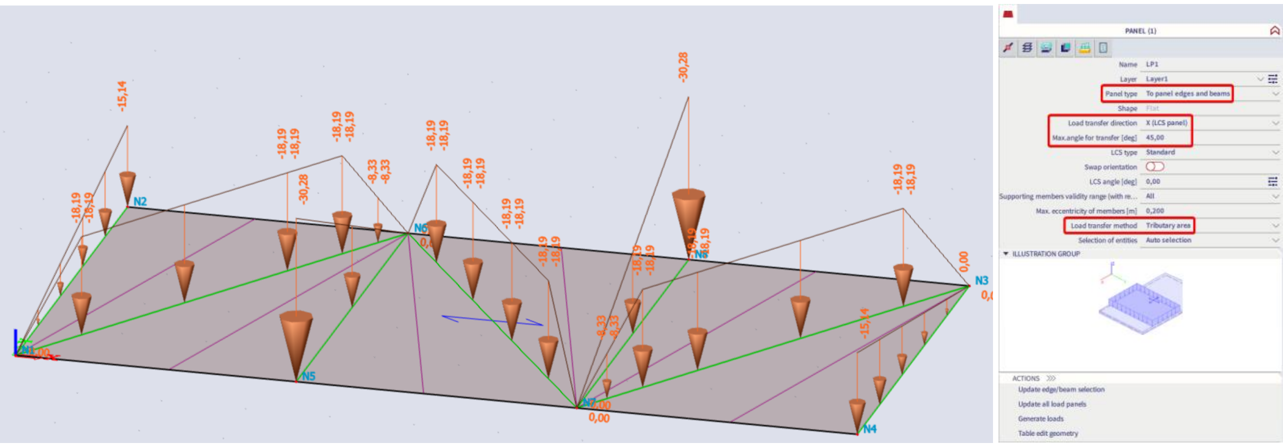
Note: For this type of load panel the load transfer method 'standard' and 'FEM' are available as well.

In the property panel a new property 'Max. angle for transfer' will be visible. This angle will be used to include or exclude specific beams. The default value 5 deg. will make sure that only the beams perpendicular to the load transfer direction are taken into account. To take into account the bracings as well, you check the angle between the bearing beams and the bracings and make sure that the max. angle is set to a higher value.

Max. angle for transfer = 5 deg (default value).

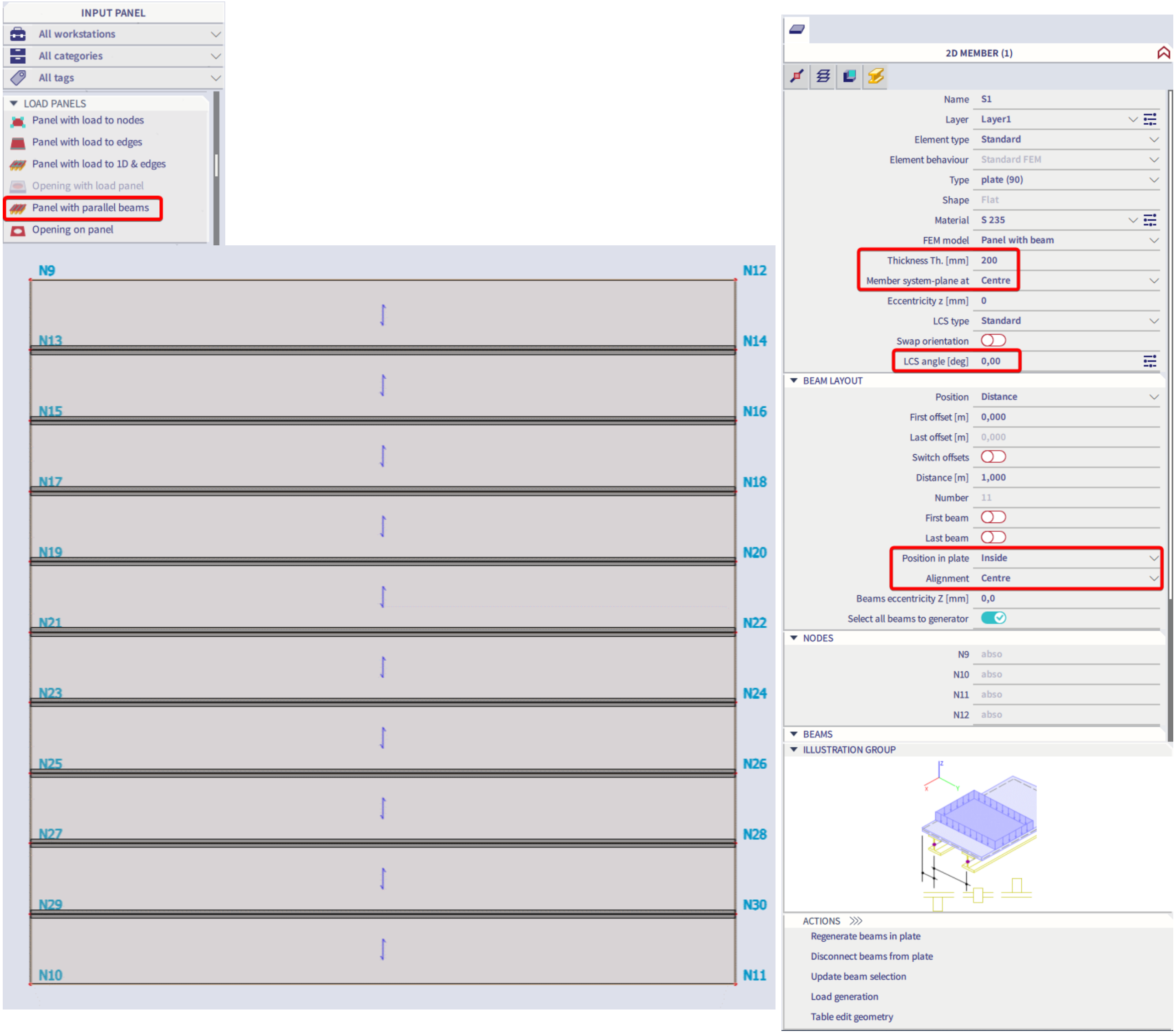


Max. angle for transfer = 45 deg.



2.2.8. Panel with parallel beams

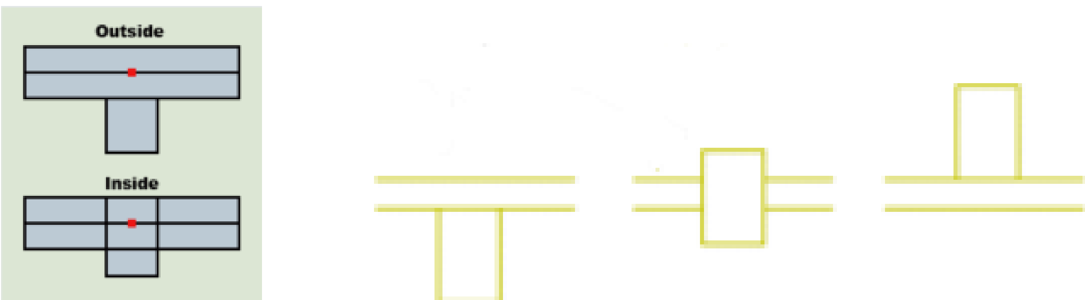
With this option you can directly insert a panel with supporting beams. The beams are placed in the x direction of the panel LCS, and the load transfer direction will be the y direction of the panel LCS. To rotate the beams, you modify the LCS angle in the properties of the panel. As an example a panel with parallel beams of the size 10m by 10m is used. A surface load of -10 kN/m² is added.



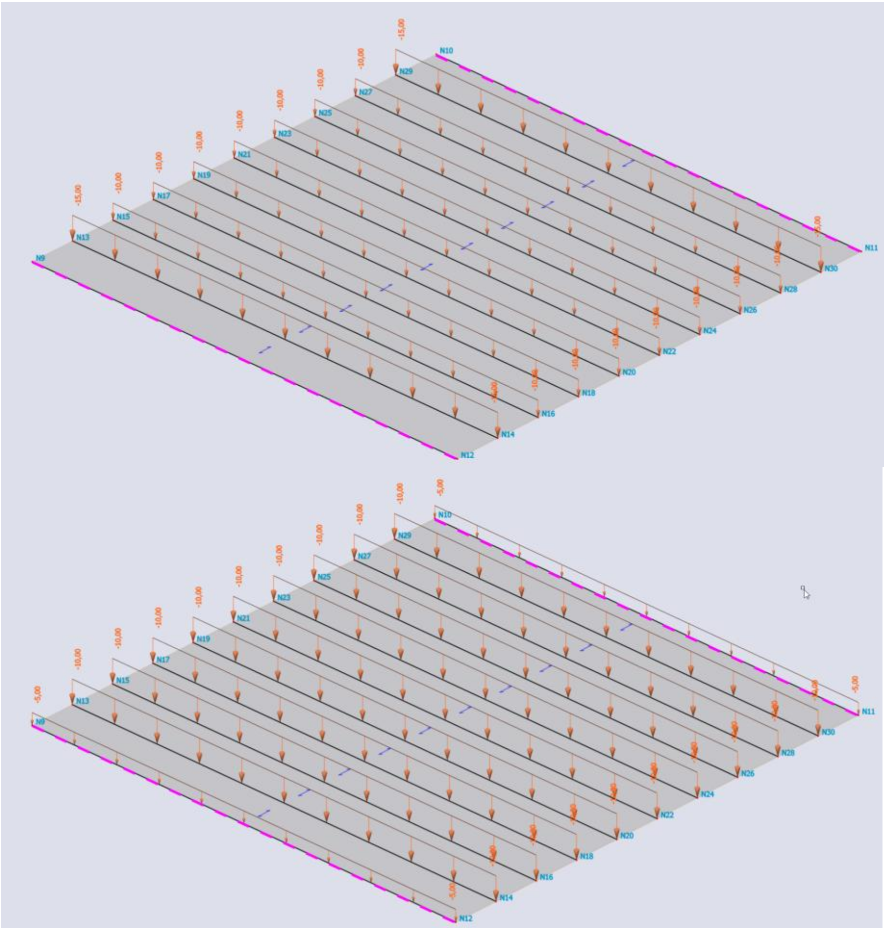
Different from other load panel types, the panel has a thickness. This has an influence on:

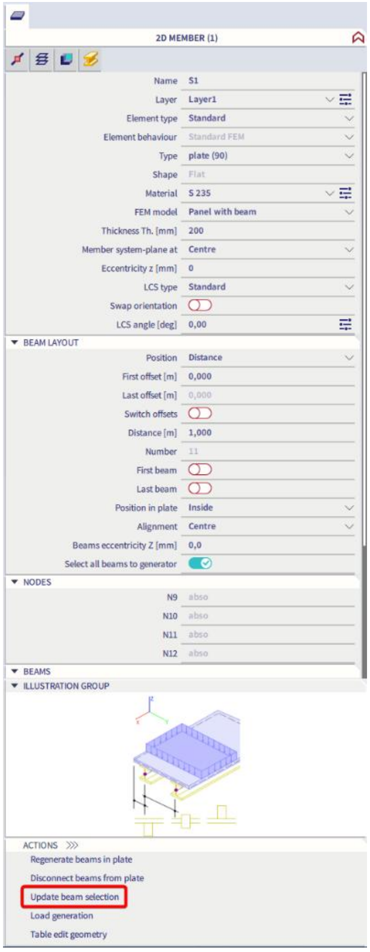
- The eccentricity of the beams
- The self-weight of the panel (this was not the case for the other discussed load panels).

The position in plate (inside or outside) and the alignment (bottom, centre or top)define the placement of the beam.



The load will be transferred to the beams which are a part of the panel itself. Other beams modelled in the same plane will not be taken into account automatically. To add other beams to the plane you use 'update selection'. In this example, the first and last beam are modelled separately, the loads are only transferred to these beams once 'update beam selection' has been used.





You can generate the loads by clicking on 'load generation' or by calculating the project.

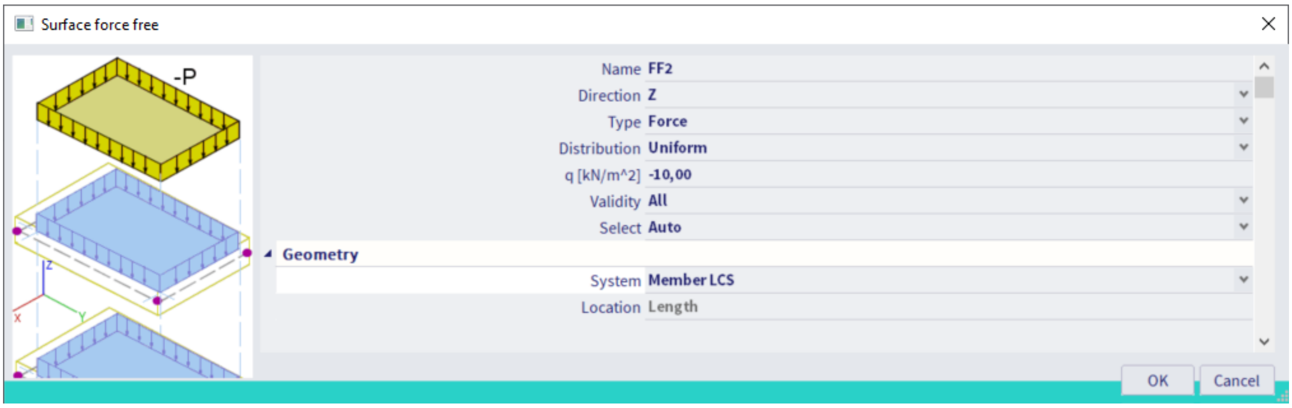
3. Free loads

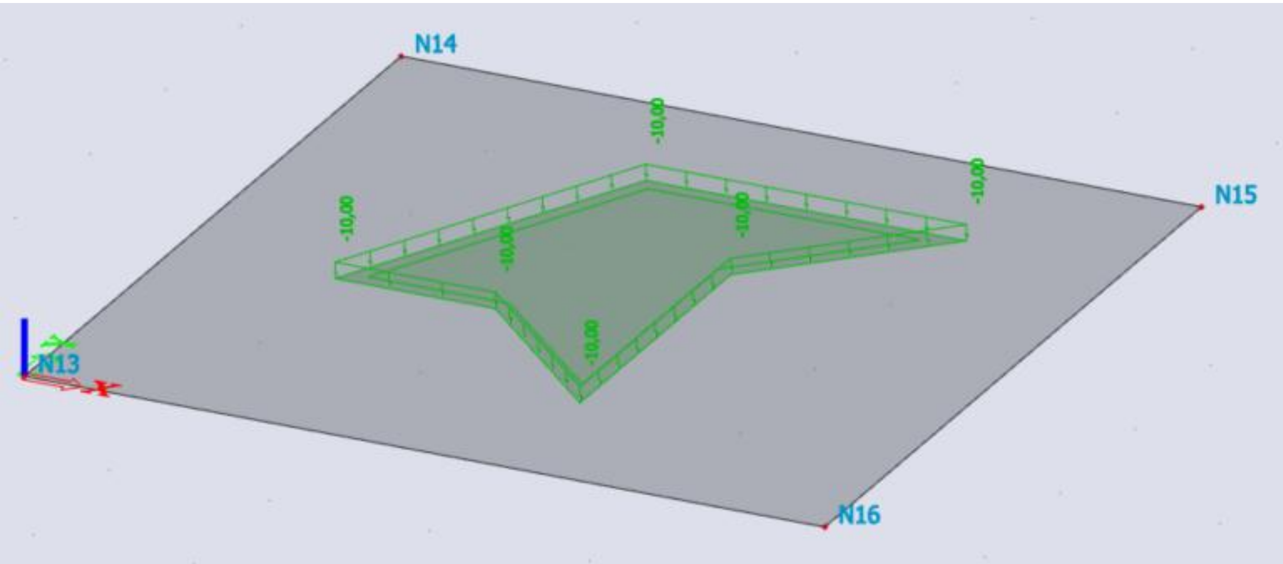
Free loads are related to 2D members, flat or curved (plates, walls, shells, load panels …).

The definition of free loads is composed of their geometry, which is independent on geometry of structural members, direction of load effect and a list of 2D members which are influenced by the free loads. Free loads are in fact easy load generators. A free load differs from a 'regular load' by the fact that it is not attributed as an additional data to a specific 2D member. Since the free load is not linked to one particular member, more loads can be generated with one free load.

3.1. Properties of a free load

Start this chapter with modelling a plate and add a load case. Click  in the input panel to start modelling a free load on the 2D member. This free load will always be modelled as a projection in the active workplane (XY, XZ, YZ), draw an arbitrary geometry and press escape. It can be necessary to move the UCS and change the active workplane.





3.1.1. Direction

This is the direction in which the load should act. This is always according to X, Y or Z.

3.1.2. Type

In this case, the load will be introduced as a force. Depending on the loadcase also selfweight and wind are available.

3.1.3. Distribution

Both a uniform load or a variable load can be used.

Uniform: one constant value for the complete surface load.

Dir X: a variable course of the free surface load in the direction X of the member LCS.

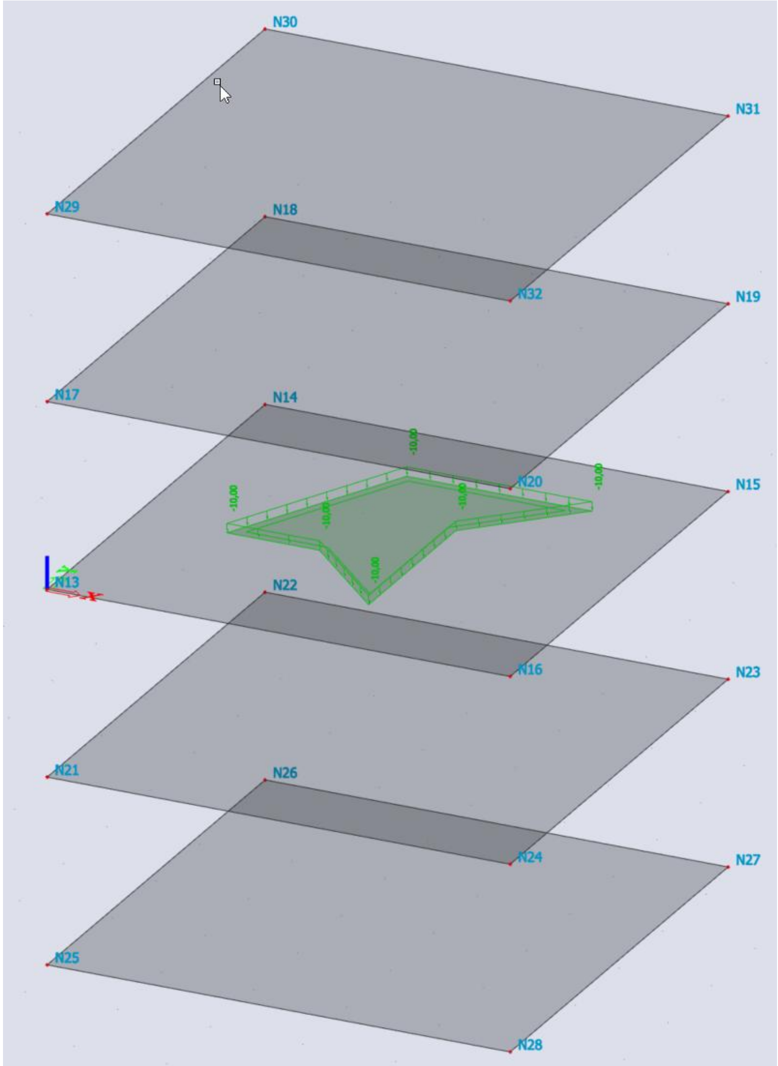
Dir Y: a variable course of the free surface load in the direction Y of the member LCS.

3 points: a variable course of the free surface, according to 3 points chosen by the user.

Note: When a variable load is modelled you will need to give two different values for the applied load. The first load will be applied in the first node that is drawn, the second load will be applied in the second node that is drawn.

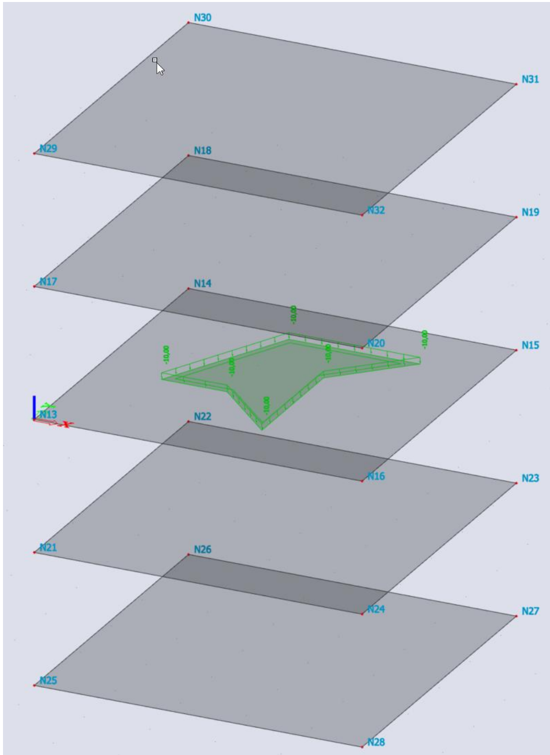
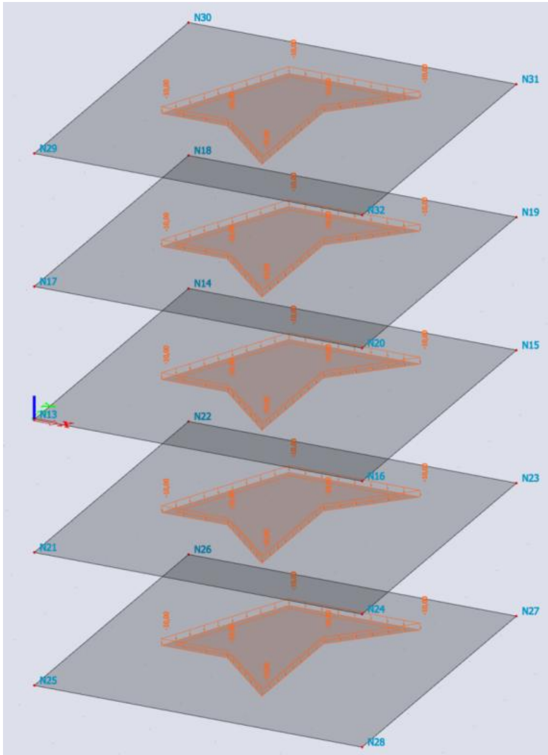
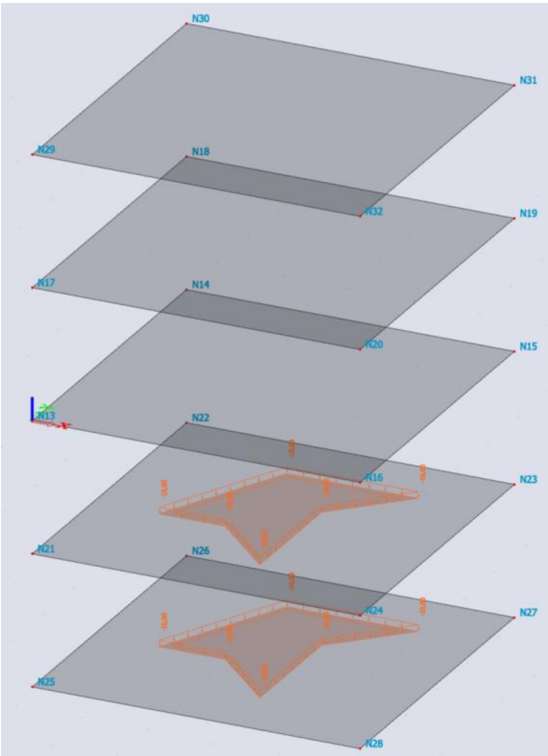
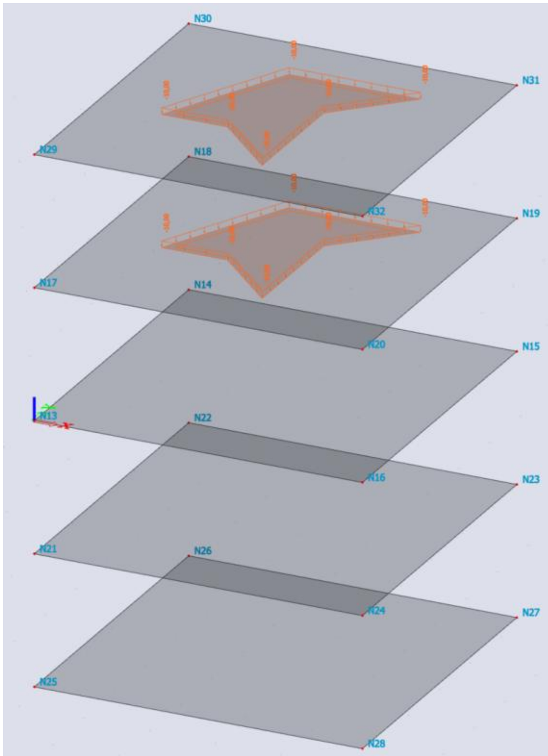
3.1.4. Validity

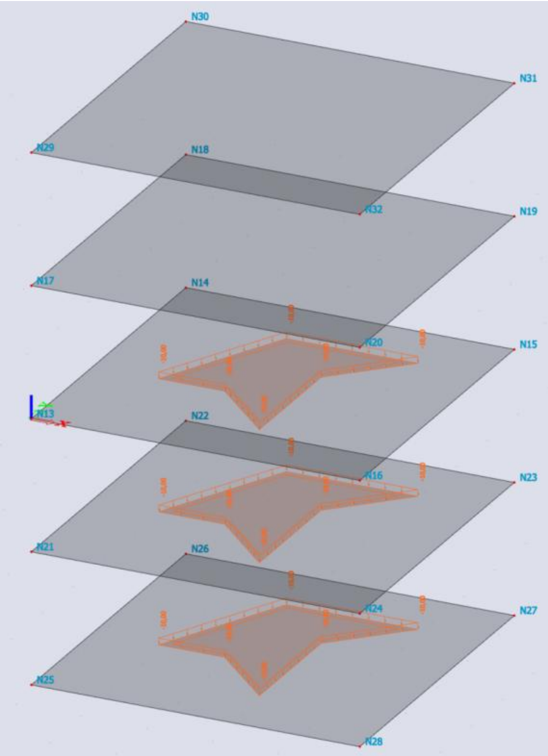
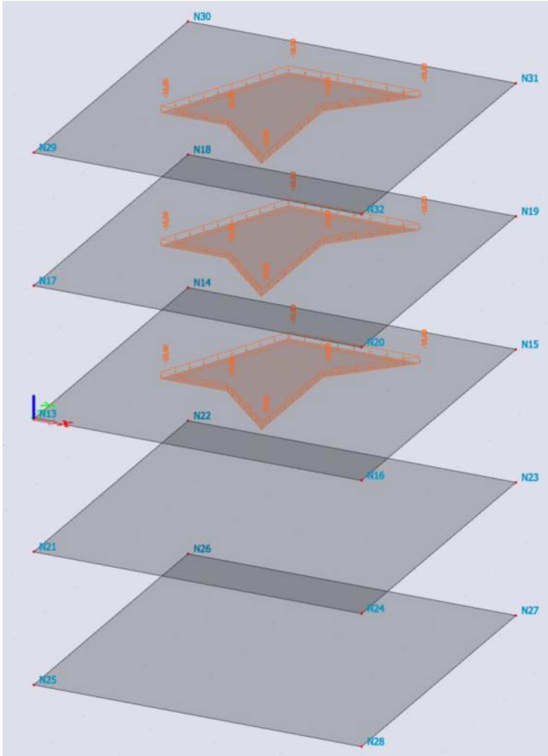
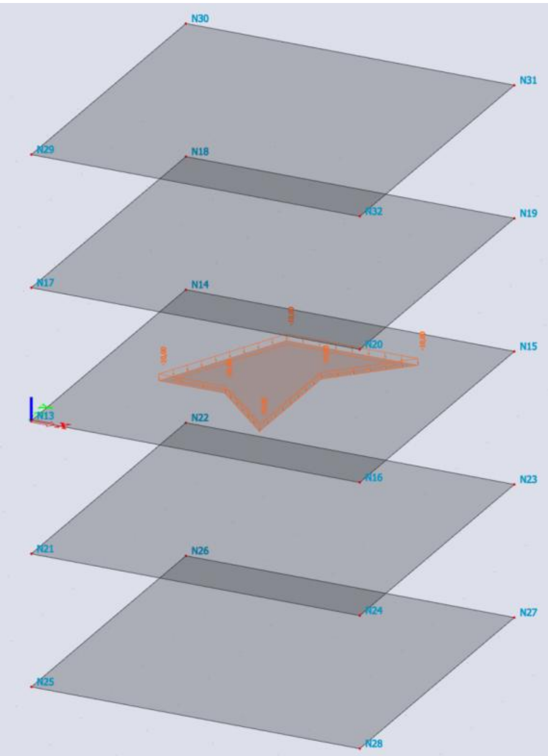
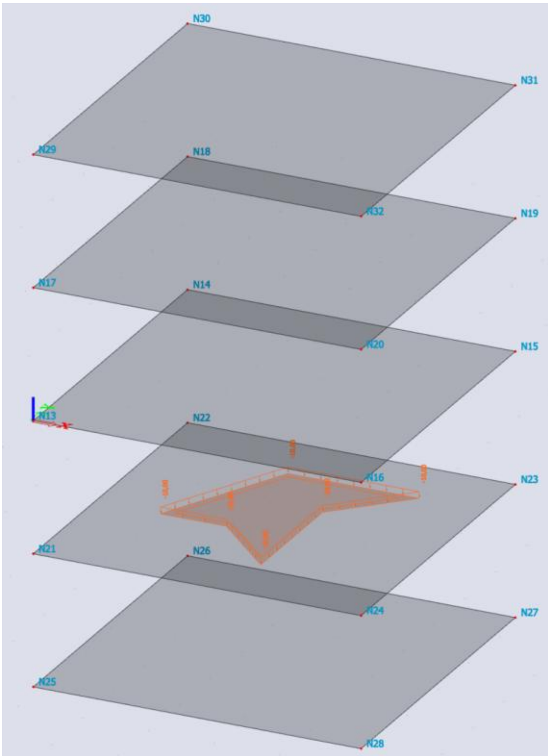
Now copy the 2D member two times above and below the original 2D member.



A free load can generate loads on different 2D members at once. To define where the load should be generated you need to set the right validity.

When a free load is generated, it will use a projection to apply the loads to all existing 2D members. The validity defines where the load should be generated.

Original load	All
	
-Z	+Z
	

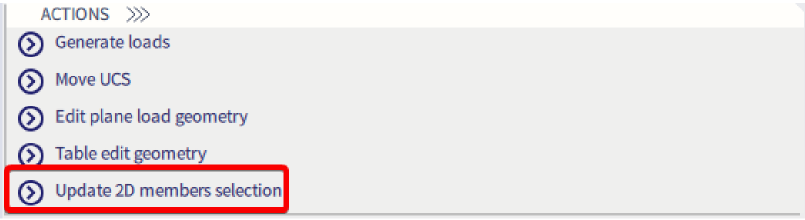
-Z (incl. 0)	+Z (incl.0)
	
Z=0	From -3m to 0m
	

Note: When the validity is set on ‘from .. to ..’ , this height is relative to the original load. Also be aware that a member at exactly the given height will not be taken into account, the slab at exactly 0m is not taken into account.

3.1.5. Select

It is also possible to manually select the members on which the load should be generated. There are two options:

- Auto: all the elements, which correspond with the validity, will be loaded.
- Select: The user can select the elements, which correspond with the validity, to be loaded.
  The selection can be modified by using the action 'update 2D members selection'.

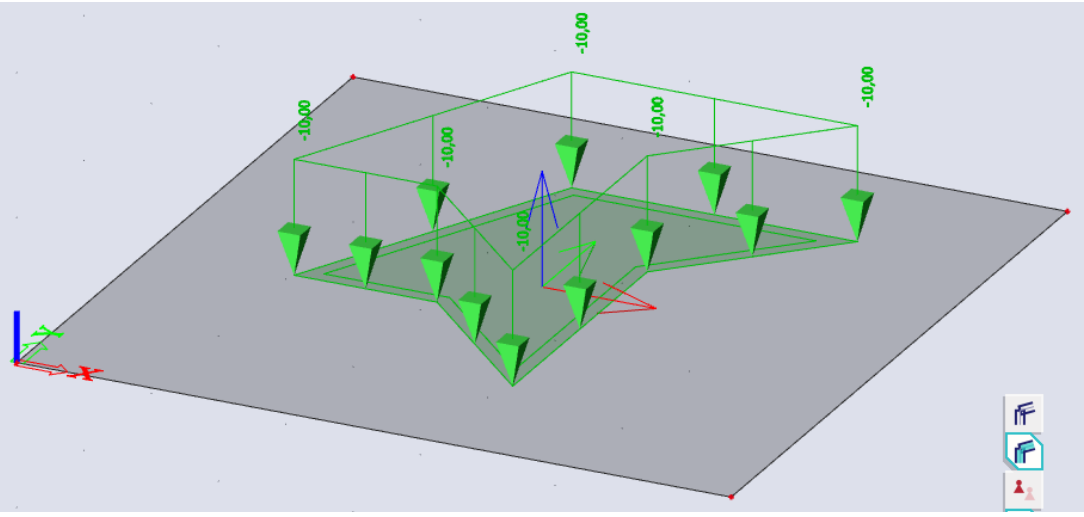


3.1.6. System

Depending on the set coordinate system the load reacts differently when generated.

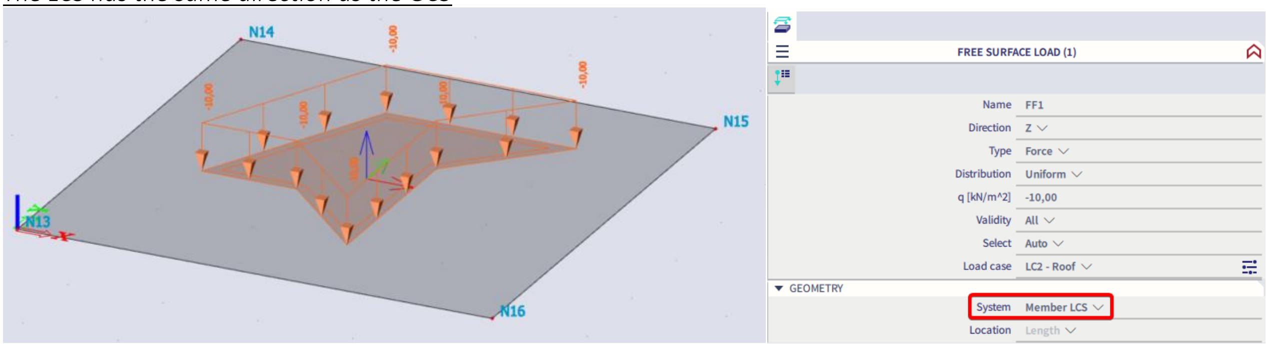
- GCS: the direction of the load according to the GCS (Global Coordinate System)
- Member LCS: the direction of the load according to the member LCS (Local Coordinate System)
- Load LCS: the direction of the load according to the load LCS. The load LCS is defined based on the active UCS when modelling the free surface load.

To demonstrate this you can change the view parameters and show the local coordinate system of the 2D member. Set the system of the original free load to member LCS. The original load is shown downwards with a negative value.



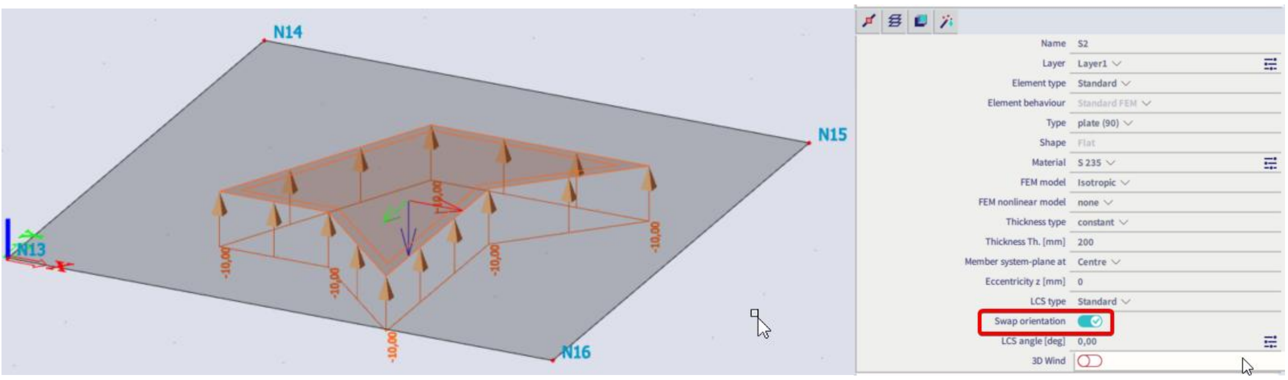
Now you can generate the load and you will see that the generated load has the same direction as the original free load. This is because the LCS of the member has the same direction as the GCS.

The LCS has the same direction as the GCS



Now when you select the 2D member and toggle on 'swap orientation', this will change the direction of the local Z axis. The generated load will be deleted, the original load will be shown and you can generate the load again. Now you will notice that the generated load will be in the opposite direction as the original free load.

The LCS has a different direction as the GCS

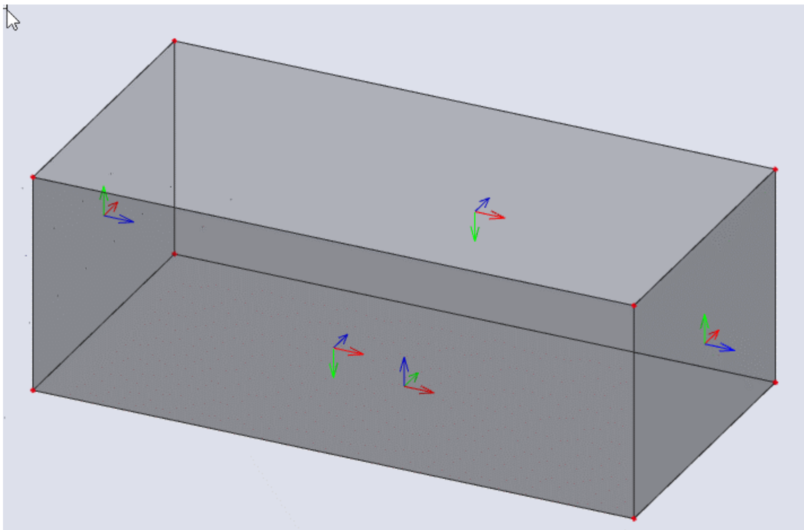


This property becomes very useful when modelling water pressure inside a tank as seen in the next example.

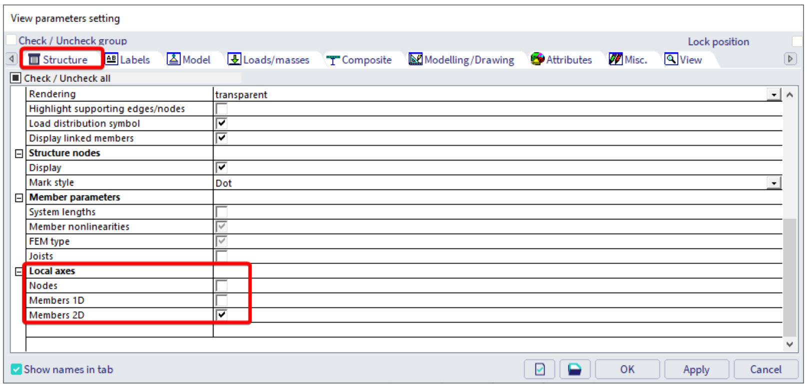
Note: Here you can find a video tutorial about this as well.

3.2. Example: Rectangular swimming pool

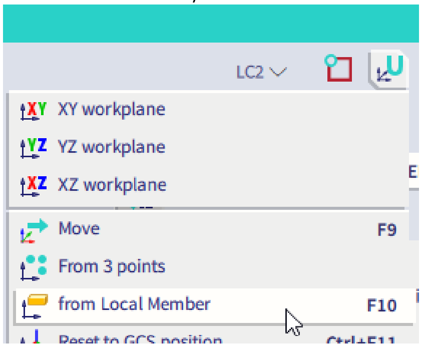
Open the project **rectangular_swimmingpool.esa**, in this project you find a the following model consisting of a rectangular floor of 5m by 10m and 4 walls with a height of 3.6m. To model the water pressure, we will create a triangular load on each wall while making use of the free surface load.



If the LCS is not visible yet, change the view parameters so the local axis of the walls and the floor are visible.



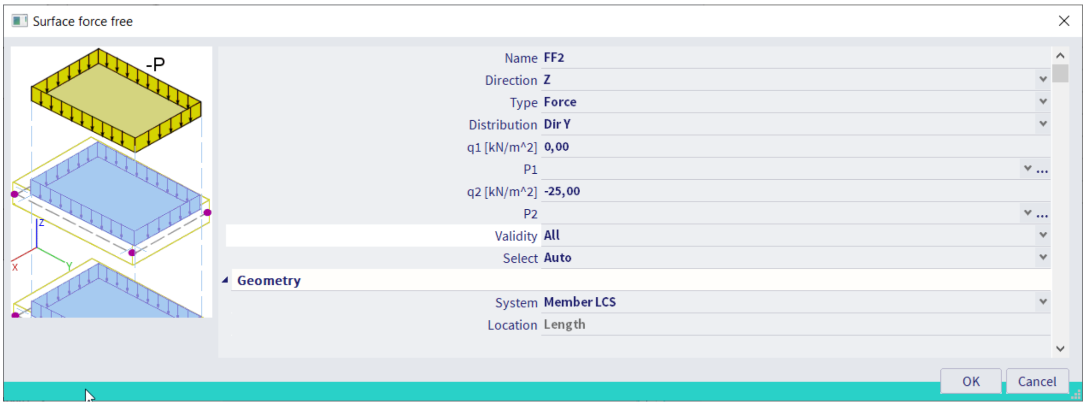
First of all, the active workplane needs to be modified. Since the load needs to be modelled and projected on the walls it is best to modify the UCS based on the LCS of the 2D member.



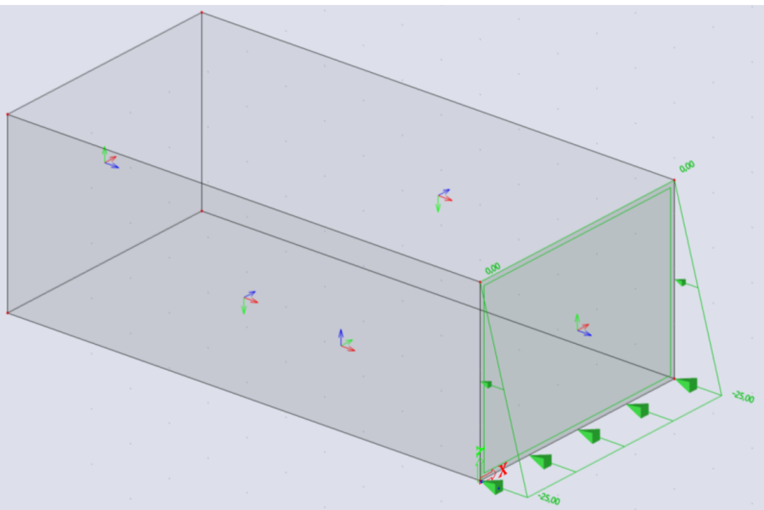
Start modelling a free load. Change the distribution, the values and the system accordingly. Instead of modelling a free load for each wall, one original load will be used and generated on all walls at the same time. The following properties are modified to accomplisch this

- The system needs to be set on Member LCS, so the direction of the load will always be the local Z-axis of each wall.
- Direction is set to Z.
- Validity is set to 'all'.
- Select is set to 'Auto'.

To create a triangular load, the distribution is modified from 'uniform' to Dir Y and the values are set. The value is 0 kN/m² on the top and for example -25kN/m² on the bottom of the wall.

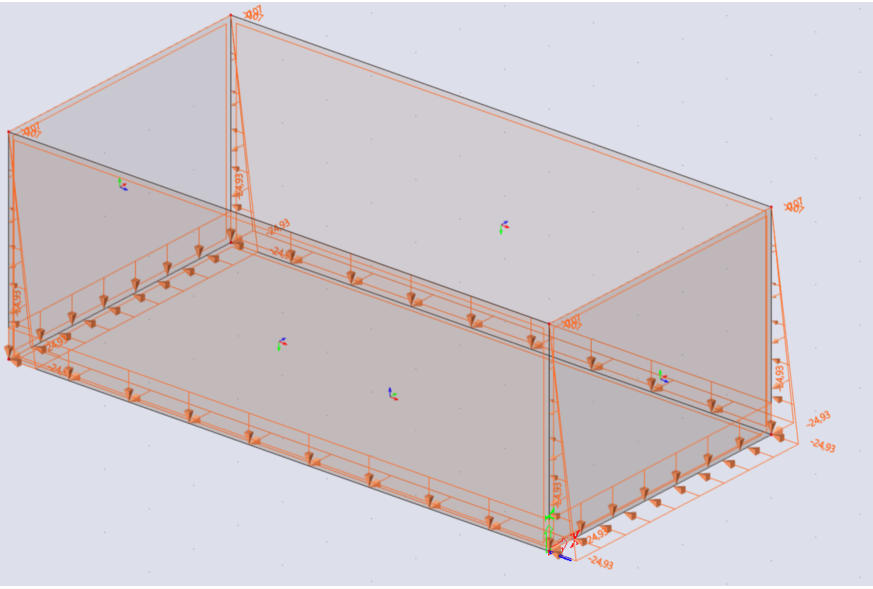


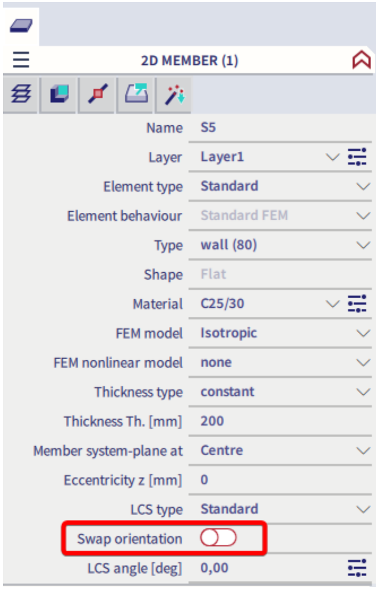
The free load will be projected in the active workplane. The application of q1 and q2 is based on which node you model first. Therefor, start modelling on top of the wall.



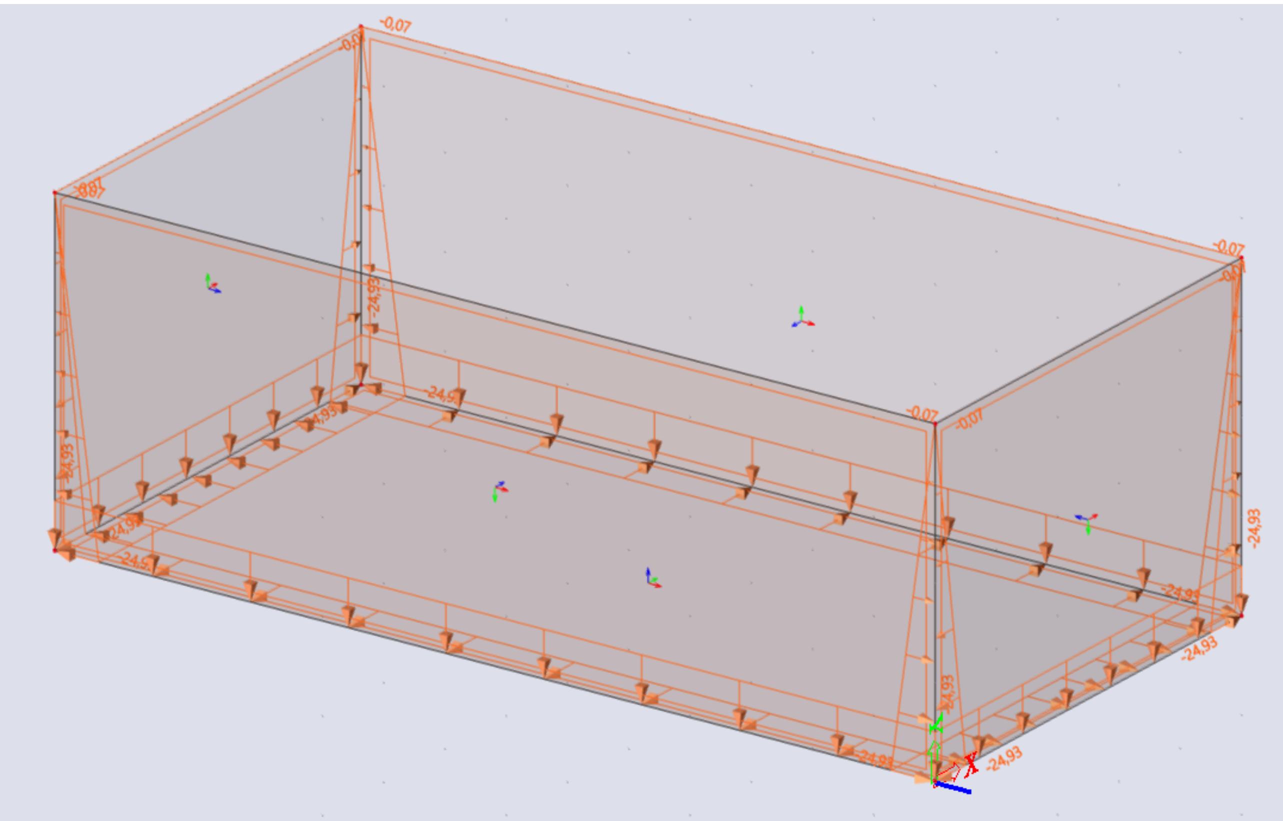
Generate the free load. The loads will be generated on each wall AND the floor based on the LCS. The load on the floor is a uniform surface load.
Notice that some loads are pointed inwards, this is caused by the direction of the LCS of the 2D element. You can simply modify the LCS of these 2D elements by selecting them and toggle on 'swap orientation' in the 2D properties.

Note: When modifying the orientation, keep in mind that surface supports are always modelled on the negative side of a 2D element.





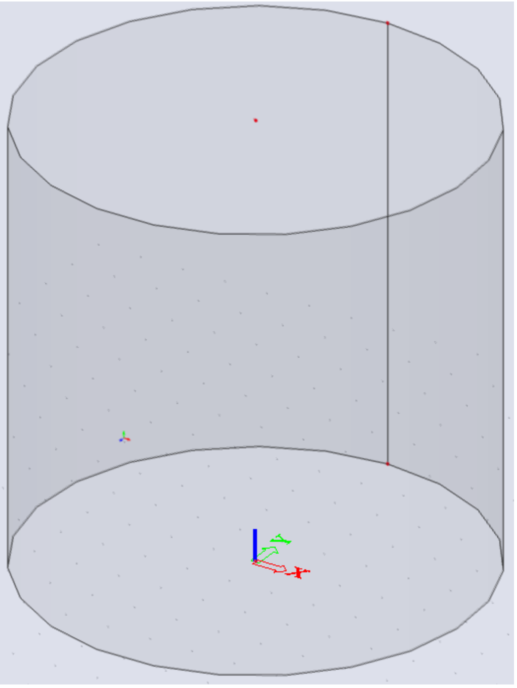
After generating the loads again, this is the final result.



3.3. Example: Cylindrical tank

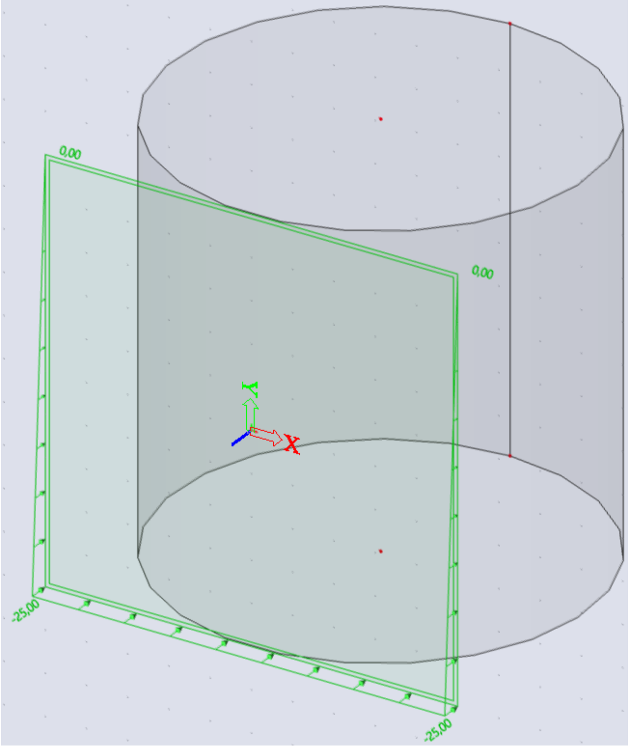
Similar to the previous example, a free load can also be generated on a cylindrical element. Open **cylindrical_tank.esa** to find this example.

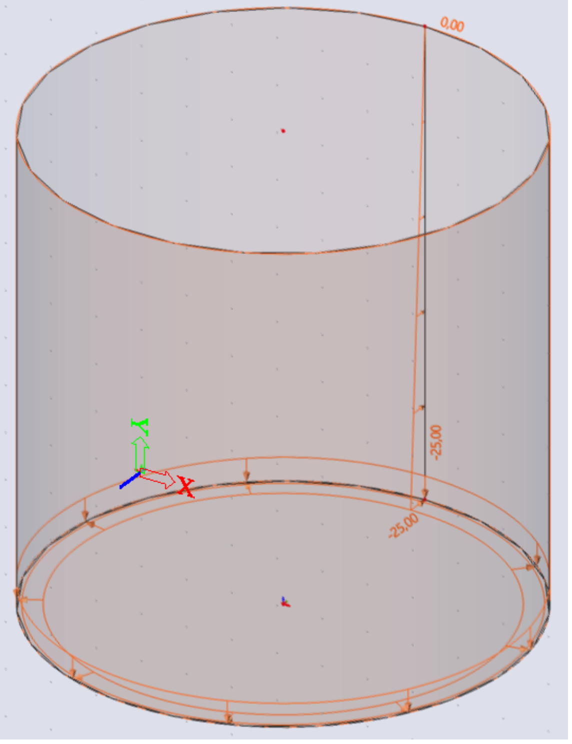
The model consists of a cylindrical element with a diameter and a height of 10m.



The needed steps are similar to the steps in the previous example:

- Move the UCS based on the LCS of the cylindrical element
- Model the free load with the same properties, use the edges of the cylindrical element
- Generate the free load, when necessary modify the LCS of the cylindrical element with 'swap orientation' or modify the value of the free load.

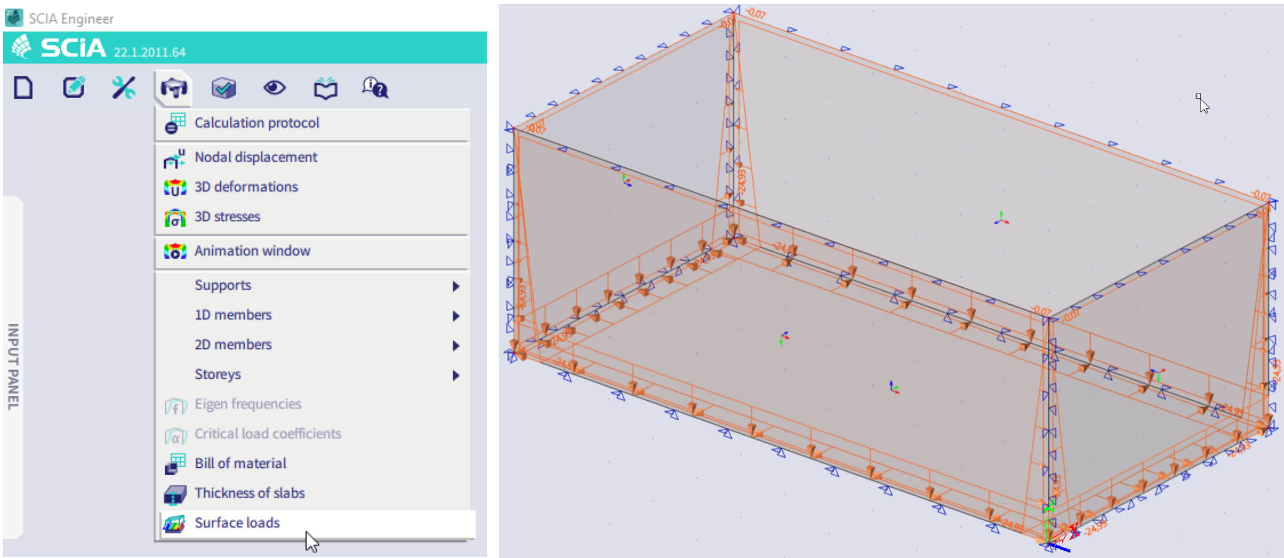




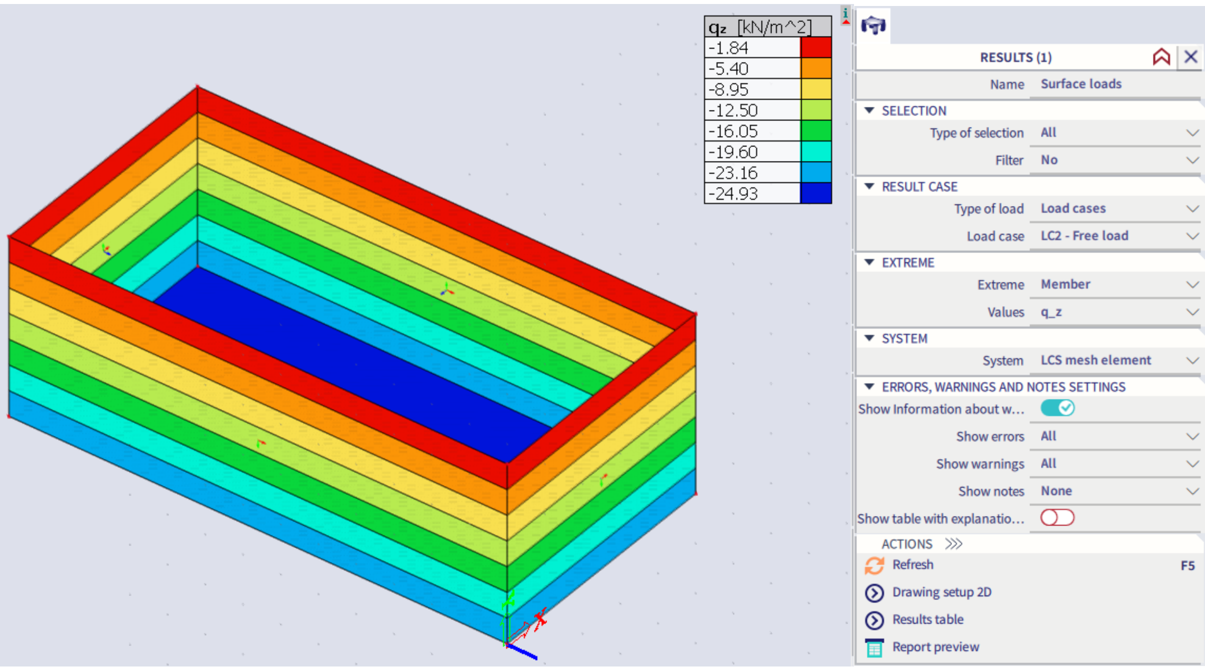
Note: Here you can find a video tutorial for the rectangular pool and the cylindrical tank as well.

3.4. Result: surface loads

The command ‘surface loads’ which you can find in the results menu will allow you to visualize the applied surface loads. A similar example is used as in paragraph 2.2. This model contains a rectangular swimming pool with 2D surface supports. The water pressure inside the swimming pool is modelled with free loads.

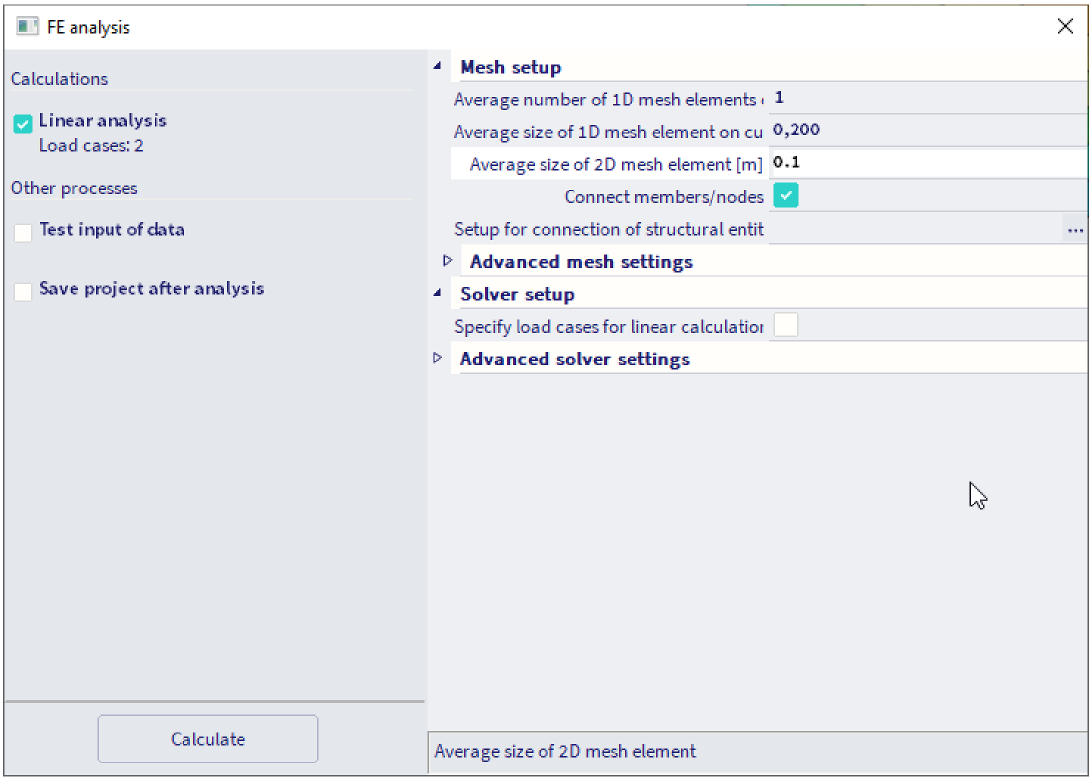


Set the result properties. In this example we will check the surface loads applied from the free load in LC2. By default, the LCS of each mesh element is used.

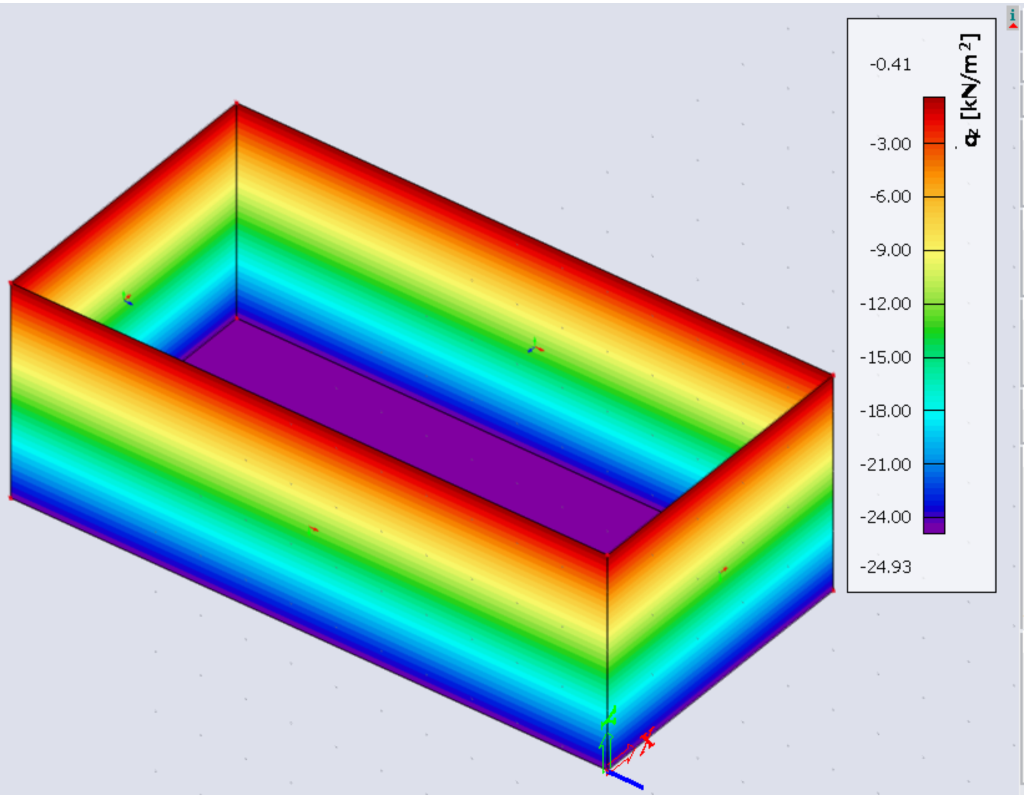


Keep in mind that the mesh size has an influence on how the surface load is applied. In the example above, the 2D mesh size was set to 0.5m. In this selection, there are less 2D mesh elements and less than 20 different values to be displayed, so the legenda is shown with separate values.

When the mesh size is reduced, this will be reflected in the surface load results. The example is recalculated with a 2D mesh size of 0,1m.



Since there are more 2D mesh elements and more than 20 different values that need to be displayed, the legenda is shown as a continuous legend.



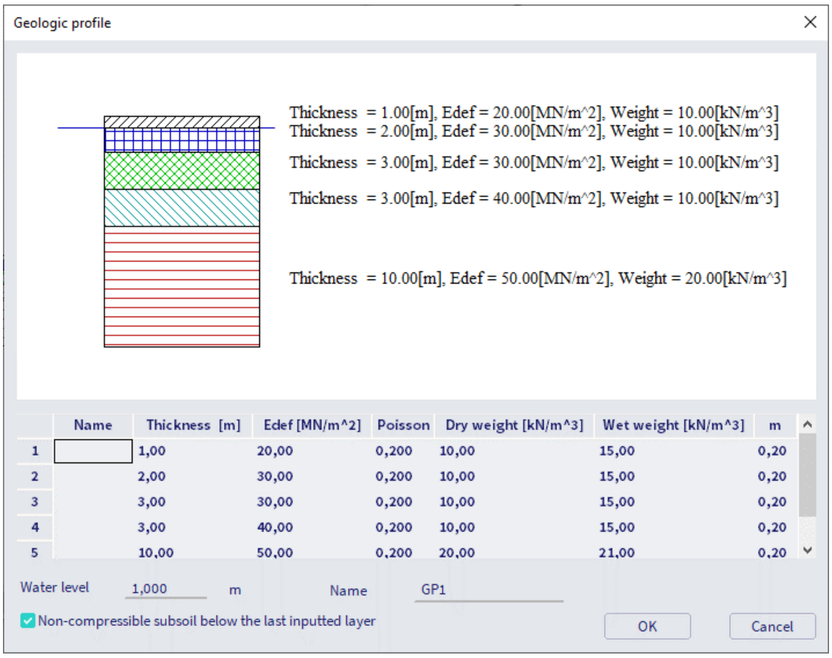
Note: This tool is only available for 2D elements, surface loads on load panels are not available in this result.

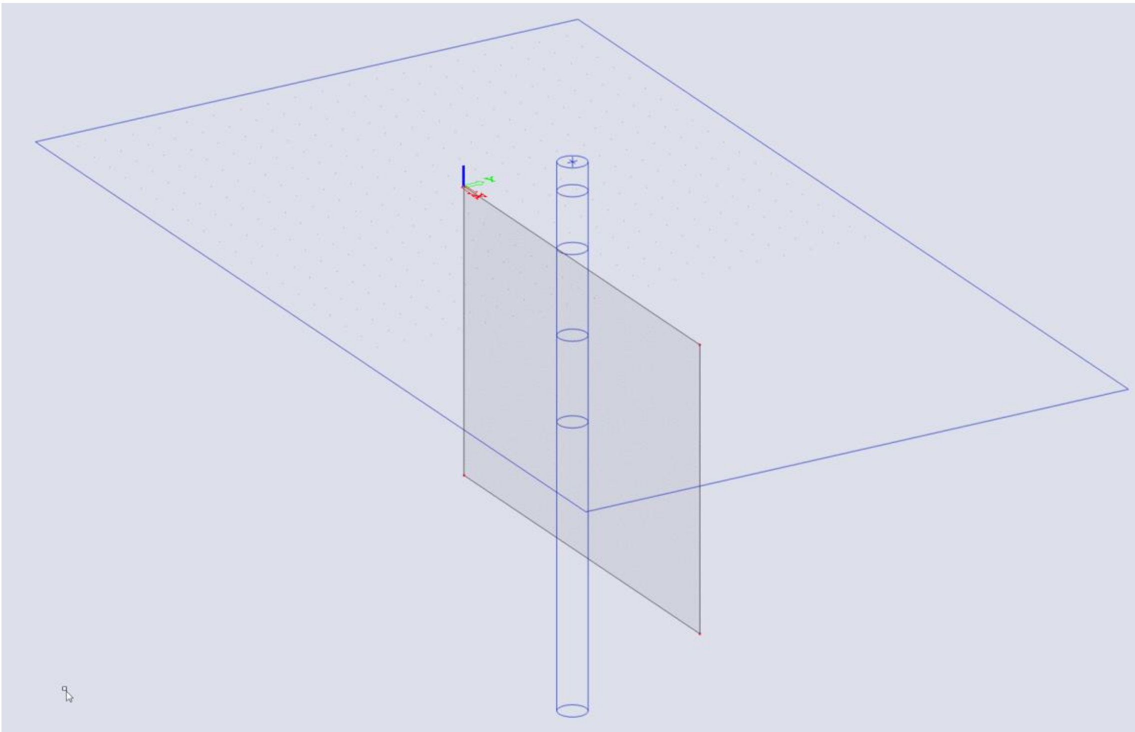
3.5.Soil and Water pressure based on borehole

Several types of load (point force, line load and surface load) can be defined as what is called "soil pressure" or "water pressure " based on the input of boreholes.

Note: The soil-in module and the creation of boreholes is explained in the foundations manual.

In the example **borehole_soil_pressure.esa** a small example is created. It contains a wall modelled below Z=0 and a borehole with the following geologic profile.



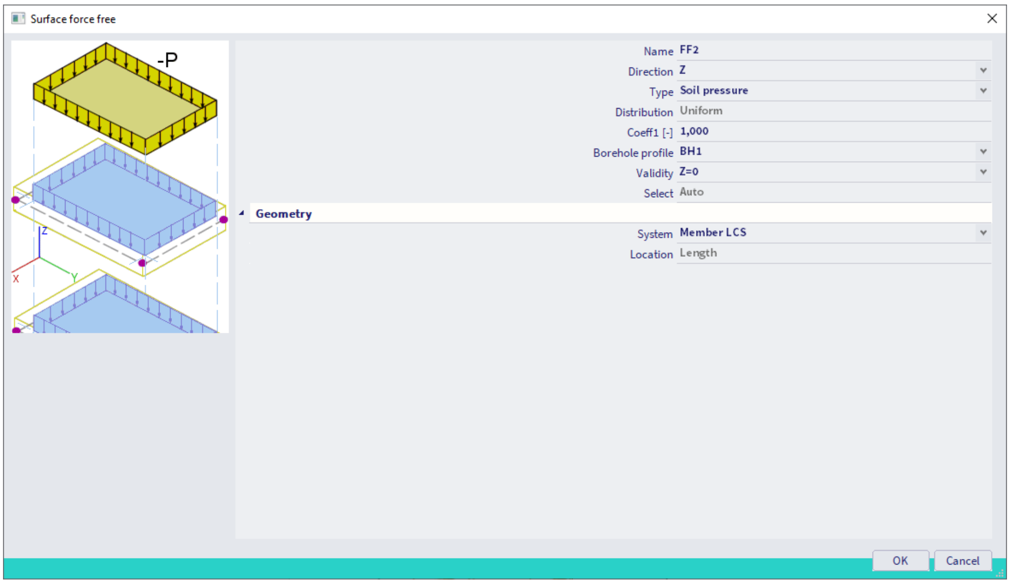


Water and soil loads can be input for the following load cases:

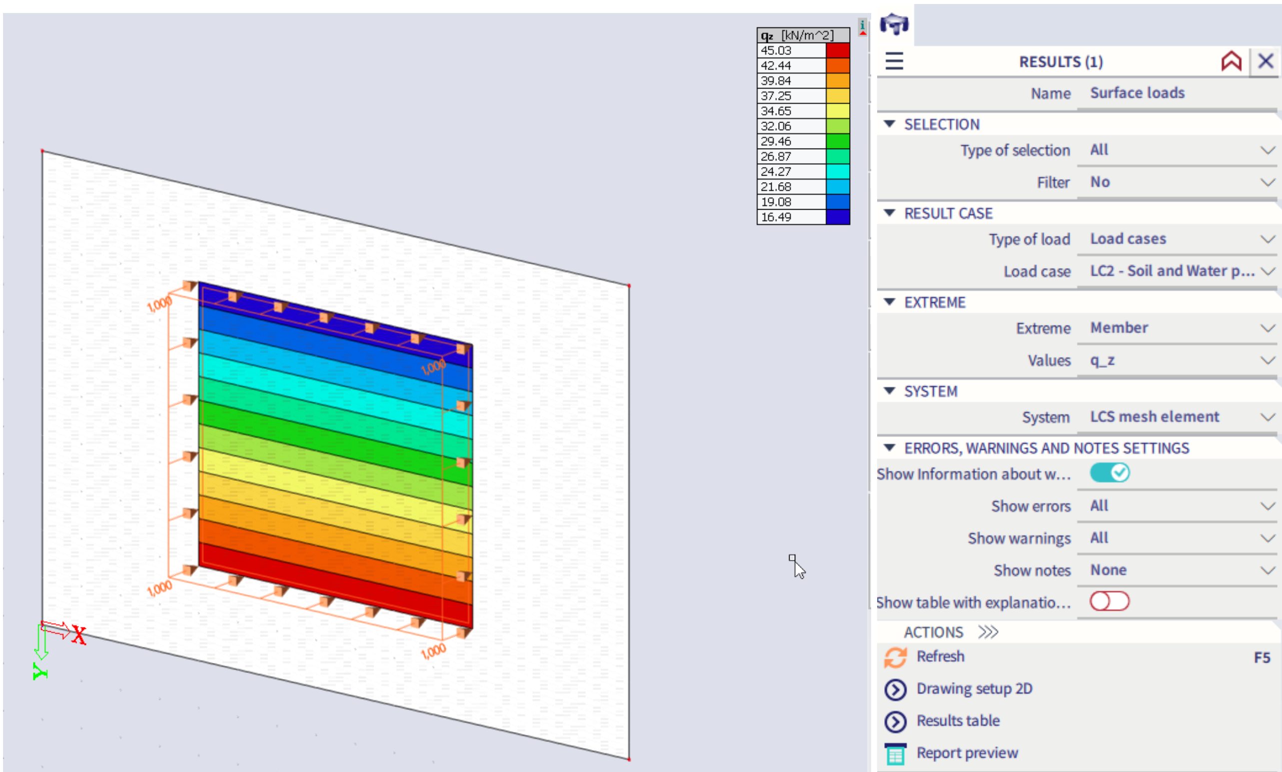
- action type = "permanent" and load type = "standard",
- action type = "variable" and load = "static".

A general surface load or a free surface load can be created with the following properties:

- Type is set to soil pressure
- The distribution is set to 'uniform' automatically
- The LCS of the member is used
- Borehole BH1 is used to define the soil pressure

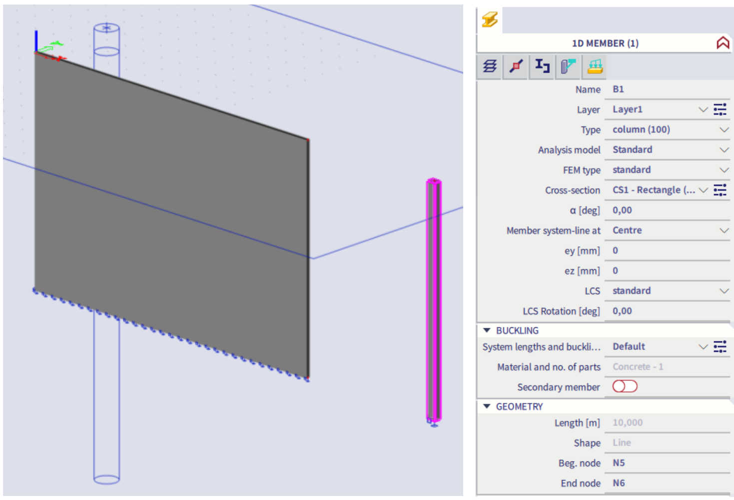


the 2D surface load applied on a 2D surface of a certain shape (rectangle in case below) is visible only as offset of this shape (inner rectangular frame in case below), because the unit of this surface load is the coefficient value. The generated load is not visible directly. In order to view the generated load, use the result command 'surface loads'.



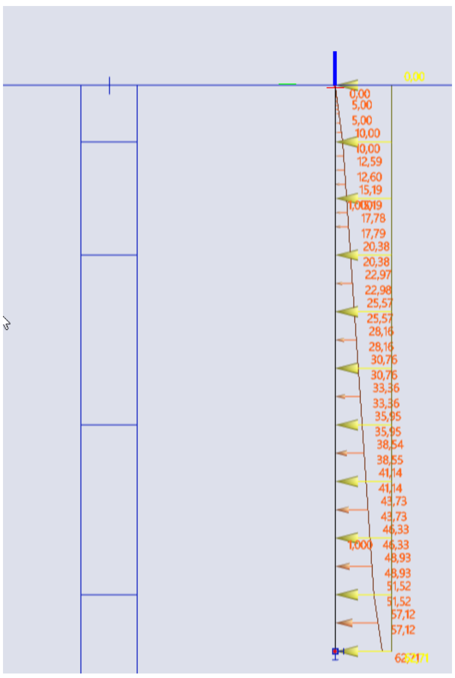
Note: The same can be done for water pressure.

Soil or water pressure on a 1D element can be modelled with 1D Line loads. As an example a column with a height of 10m is modelled right next to the wall.



A line load is added to this column with the following properties. BH1 is used to define the soil pressure.

In yellow the original line load is displayed, in orange the generated values are shown directly.



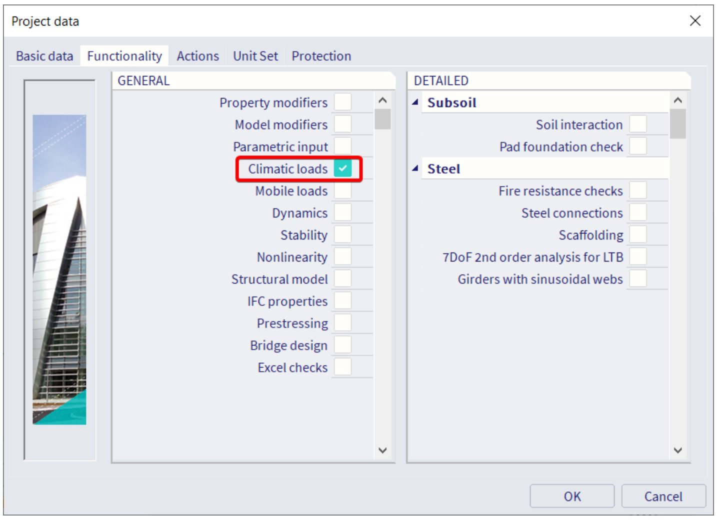
4. 3D wind generator

In this chapter, the 3D wind generator will be explained. This generator is used to generate the wind loads in all directions according to the code on <u>closed structures</u>. Since version 22, the possibilities of the 3D Wind-Load Generator have been expanded in order to quickly and automatically generate wind load on the structure of Awning type and Canopy type. The load generation is done in accordance with EN 1991-1-4:2005 (E), chapter 7.3. This automatic generation of load is available for EC-EN (Eurocode).

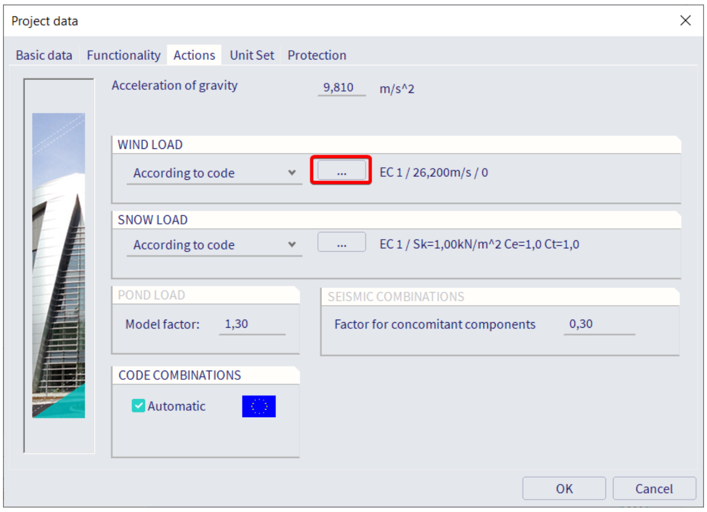
Note: Here you can find the general documentation about the 3D wind generator.

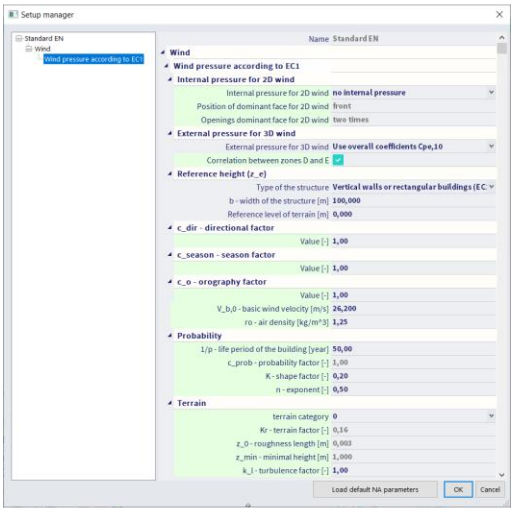
Note: The 2D wind- and snow generator is not available in post processing environment 'default'. To use these functionalities you should use the 32-bit version of SCIA engineer. You can find more information about this in chapter 6 of this manual.

Open the example **'3D Wind generator.esa'**. The structure consists of a steel hall and is closed using load panels. To be able to use the generator, the functionality 'Climatic loads' is used with the wind load according to the code (EN1991-1-4). This functionality is only available if you have the module sens.15.en.

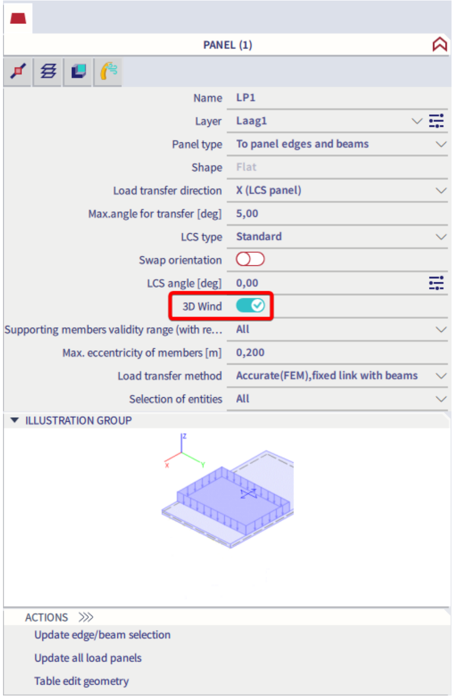


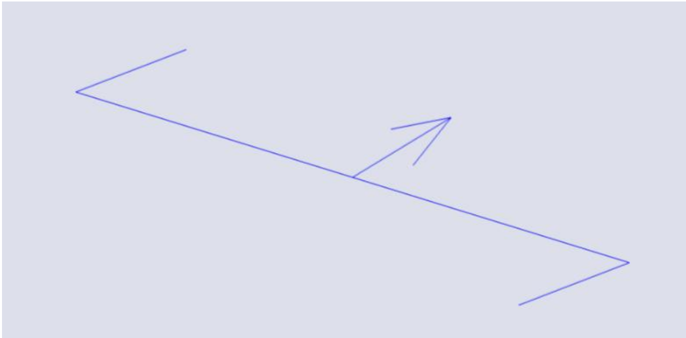
In the tab 'actions' you can choose the wind load to be according to the code or user defined. The used parameters can be viewed and changed if necessary by clicking the three dots.

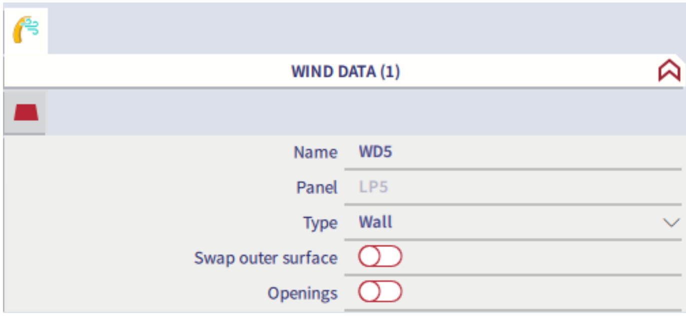




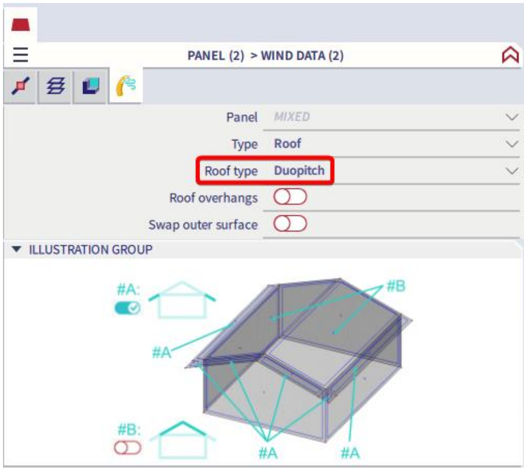
The load panels are of the type To panel edges and beams. Since these load panels will be used for the distribution of the wind load into line loads on the columns/beams, the option 3D Wind will be marked in the properties of the panels. This will add 3D wind data to each loadpanel.



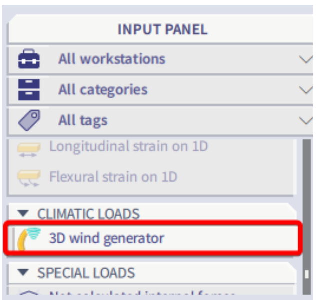


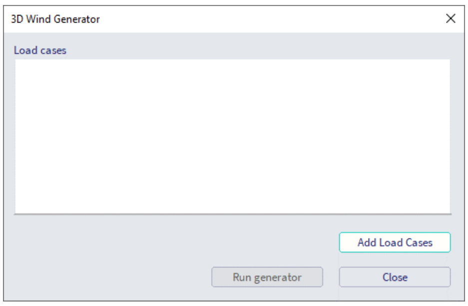


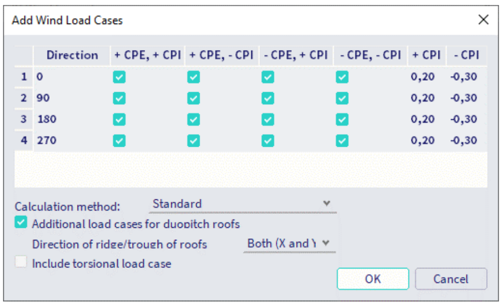
The arrow that represents the wind data needs to be pointed <u>**outwards**</u>. For the panels where the wind data is pointing inwards, the orientation can be modified in the Wind data itself by enabling ‘swap outer surface’. The Wind data for the load panels on the roof should be modified as well. By default the **Roof type** is set to Monopitch. This needs to be changed to **Duopitch**.



Afterwards, the 3D Wind Generator can be used in the load menu. Use the button ‘Add Load Cases’ to choose which load cases have to be generated.







By default, 16 load cases are generated. Four cases for each wind direction. For the Load Coefficients, the Cpe values are taken from the code (EN 1991-1-4):

For the vertical walls, table 7.1 of EN 1991-1-4 is used:

Table 7.1 — Recommended values of external pressure coefficients for vertical walls of rectangular plan buildings

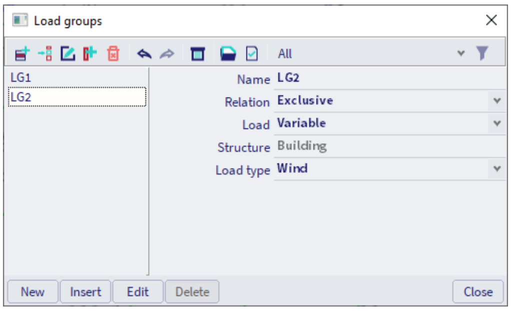
Zone	A		B		C		D		E	
h/d	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$
5	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,7	
1	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,5	
≤ 0,25	-1,2	-1,4	-0,8	-1,1	-0,5		+0,7	+1,0	-0,3	

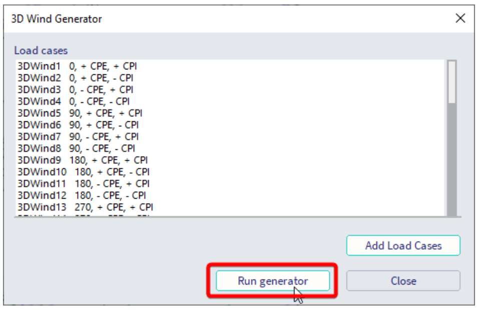
For the roof, table 7.4a of EN 1991-1-4 is used:

Table 7.4a — External pressure coefficients for duopitch roofs

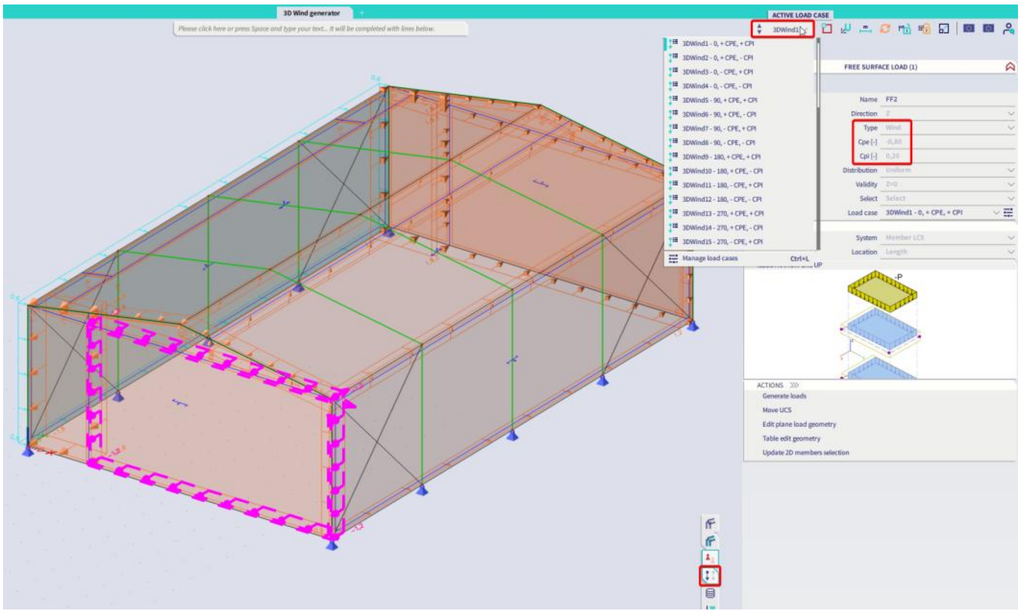
Pitch Angle α	Zone for wind direction $\theta = 0°$									
	F		G		H		I		J	
	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$
-45°	-0,6		-0,6		-0,8		-0,7		-1,0	-1,5
-30°	-1,1	-2,0	-0,8	-1,5	-0,8		-0,6		-0,8	-1,4
-15°	-2,5	-2,8	-1,3	-2,0	-0,9	-1,2	-0,5		-0,7	-1,2
-5°	-2,3	-2,5	-1,2	-2,0	-0,8	-1,2	+0,2		+0,2	
							-0,6		-0,6	
5°	-1,7	-2,5	-1,2	-2,0	-0,6	-1,2	-0,6		+0,2	
	+0,0		+0,0		+0,0				-0,6	
15°	-0,9	-2,0	-0,8	-1,5	-0,3		-0,4		-1,0	-1,5
	+0,2		+0,2		+0,2		+0,0		+0,0	+0,0
30°	-0,5	-1,5	-0,5	-1,5	-0,2		-0,4		-0,5	
	+0,7		+0,7		+0,4		+0,0		+0,0	
45°	-0,0		-0,0		-0,0		-0,2		-0,3	
	+0,7		+0,7		+0,6		+0,0		+0,0	
60°	+0,7		+0,7		+0,7		-0,2		-0,3	
75°	+0,8		+0,8		+0,8		-0,2		-0,3	

After clicking on OK, all the load cases that are marked will be created. These load cases will be gathered in a new automatic created variable load group. Generate the loads by clicking on 'run generator'.

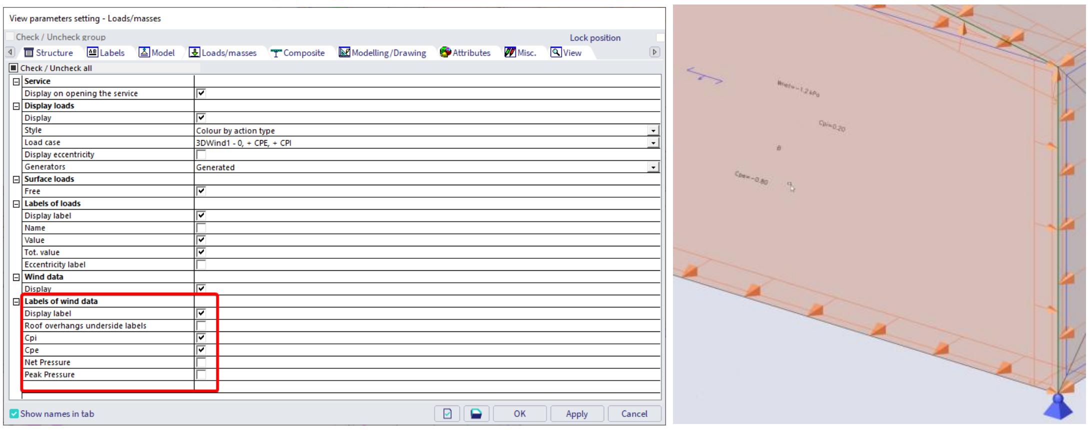




The loads can be viewed by turning the visibility on and changing the active load case accordingly. The loads are generated as surface loads for each zone. By selecting such a surface load, the calculated coefficient can be viewed in the property window:

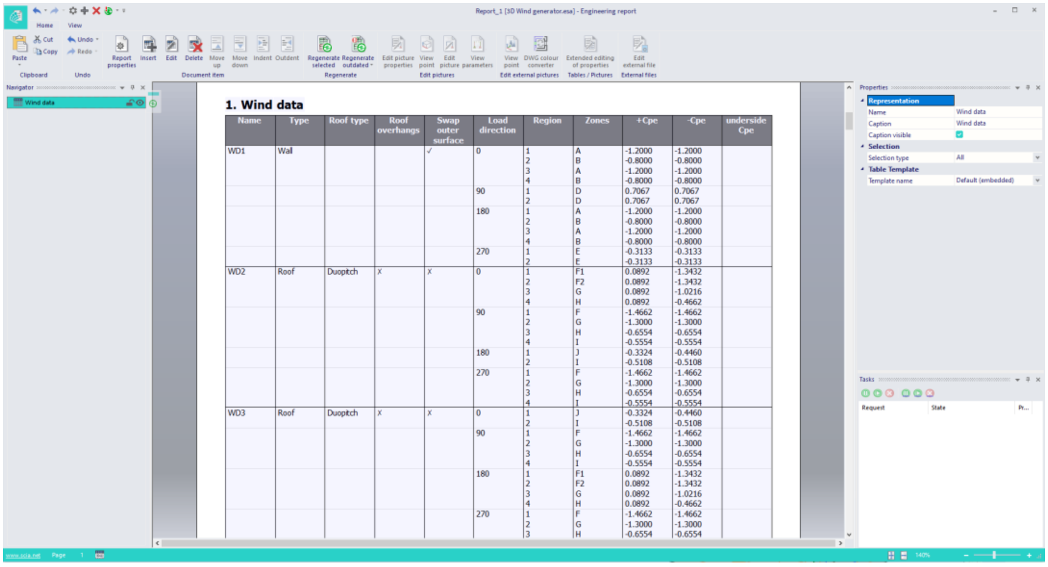


The Cpe and Cpi coefficients can showed graphically by marking this option in the View parameters. The Net and Peak Pressure can also be visualized like this.



All Cpe and Cpi values for each zone and wind direction, can be viewed in the **Engineering report**

.



5. Traffic loads 2D

With traffic loads, you can model mobile load patterns on 2D elements. In this example, a bridge deck will be modelled as a concrete plate on three line supports.

Note: Traffic loads are not compatible with 1D elements. Traffic lanes created in versions older than v26.0 will be removed when the project is opened in v26.0 due to the improvements to the traffic and mobile loads.

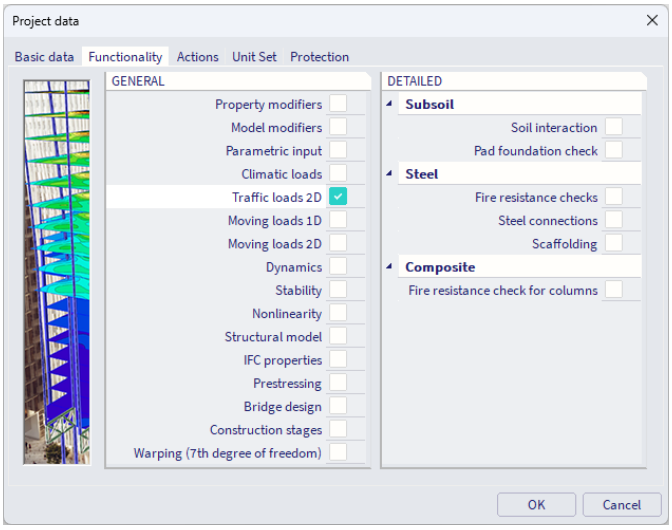
5.1. Example model

5.1.1. Project data

A new project is created with the following parameters:

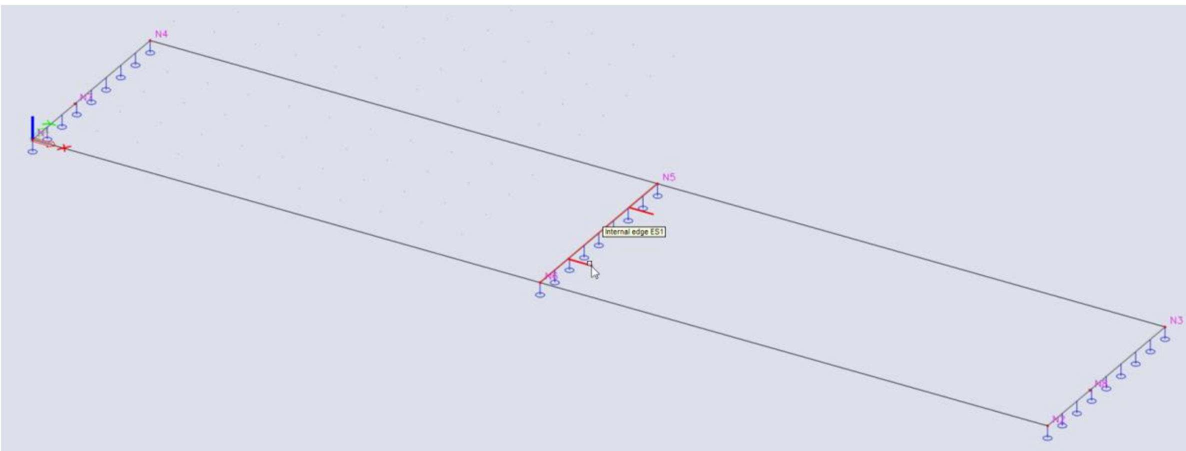
- Code: Eurocode
- Material: Concrete C25/30
- Structure: Plate XZ

To be able to use traffic loads in a project, this functionality needs to be activated in the **Project Data**:



5.1.2. Construction

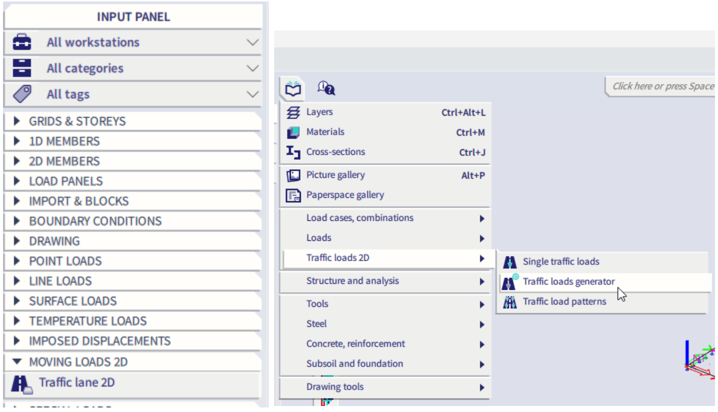
The bridge deck can be entered as a Plate with thickness **500mm**. The length of the bridge deck is **25m**, the width is **5m**. In the middle of the bridge deck, an internal edge is created with the option **Internal Edge**. Afterwards a line support is added to the short edges and the internal edge. Only the translations in the z-direction is prevented. You can also open this project **Trafficloads.esa**.



5.2. Input traffic loads

5.2.1. Traffic lane

Via the input panel, the libraries or the SCIA Spotlight, the Traffic lane can be inserted.



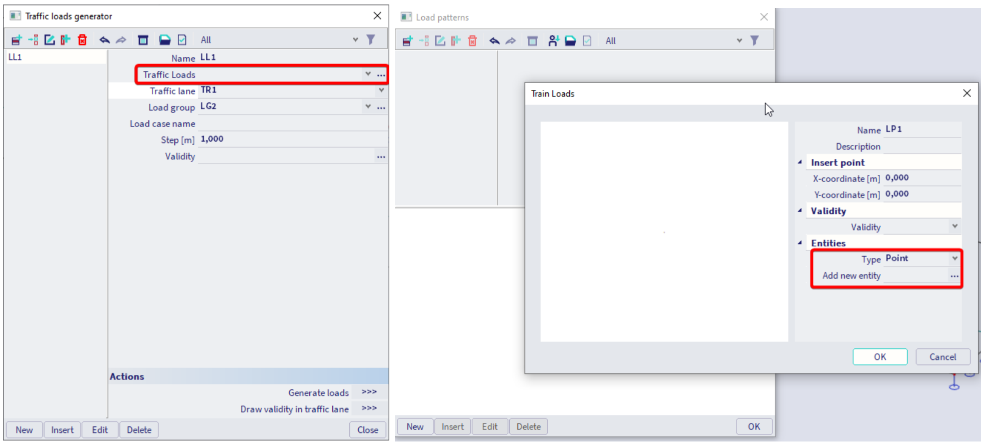
This traffic lane will consist of two rails with a distance of 1,4m between them. To make sure that the train drives on two rails at the same time, 1 traffic lane is entered. The traffic lane has to be entered on 1.8m from the edge to be able to place the train track in the middle of the bridge. The coordinates can be entered in the Command line (0;1,8 and 25;1,8).

5.2.2. Traffic loads generator

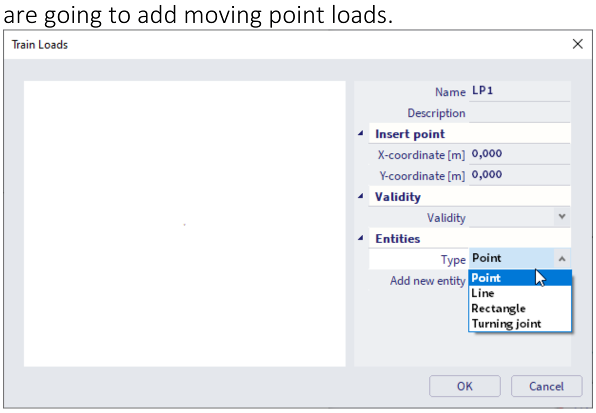
The Traffic loads generator is used to generate any inputted traffic loads onto the traffic lane. The load cases will be created and the distance between each step is given.

5.2.3. Traffic loads

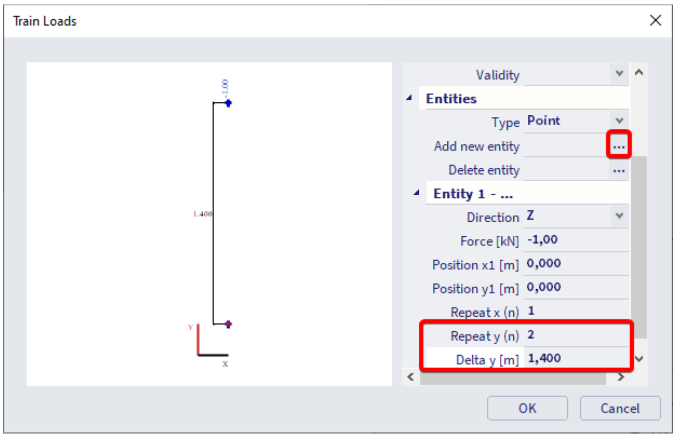
In the traffic loads generator you define the traffic loads.



Add an entity in this dialog. You can add point loads, line loads, surface loads and a turning joint. For this example we are going to add moving point loads.



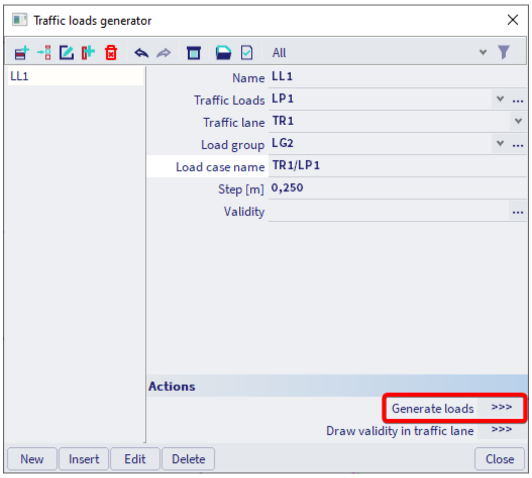
Add an entity, define the Force and the Repetition of the point load. In this case, there will be two train tracks with a distance in between of 1.4m so we repeat the point load twice according to the y-direction and set the Delty y to 1.4m.

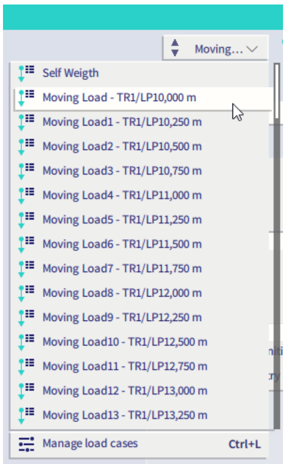


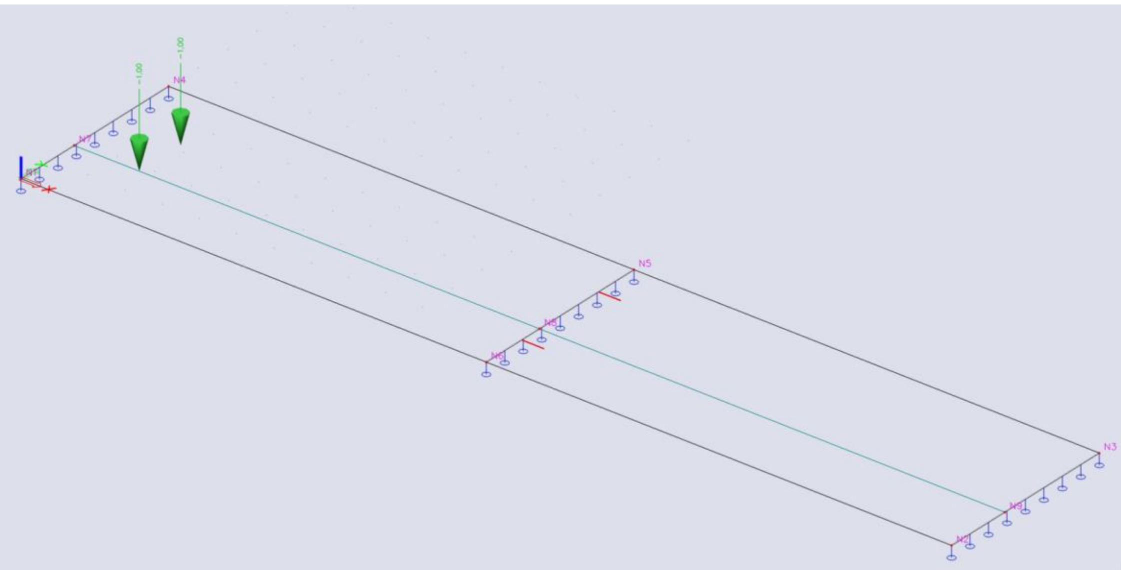
Note: There is a system database for Traffic load patterns as well.

5.2.4. Generate loads

The selected train load pattern will move along the specified traffic lane with the here-defined step of 0.25m. A separate load case is generated for each position of the moving load, the description of the load case is modified based on the load case name and the position. You also need to define the load group in which the load cases are added. Create a variable load group LG2 and set the relationship accordingly.







6. Moving loads 1D

In this chapter the 'Moving loads' functionality will be examined in detail. With this functionality, moving load systems, connected to a traffic lane, can be placed and calculated on a structure.

These load systems represent e.g. the following physical systems:

- A crane on a crane track
- A train on a bridge
- A vehicle on a viaduct
- People on a bridge

There can also be multiple load systems:

- Trains with various types of wagons
- Trains on parallel tracks or one after the other
- Different vehicles on a bridge in combination with pedestrians

Through SCIA Engineer it is possible to look for extreme design components such as extreme moments, reaction forces, and deformations … for these load systems.
In the first part of the course, the principles are explained, in the second part they are illustrated by means of projects.

6.1.Principle

The principle of the module Mobile Loads is based on the theory of the influence lines. An influence line represents a diagram that shows the effect of a unit load on a variable position in a given point of the structure.

This is illustrated on the picture below:

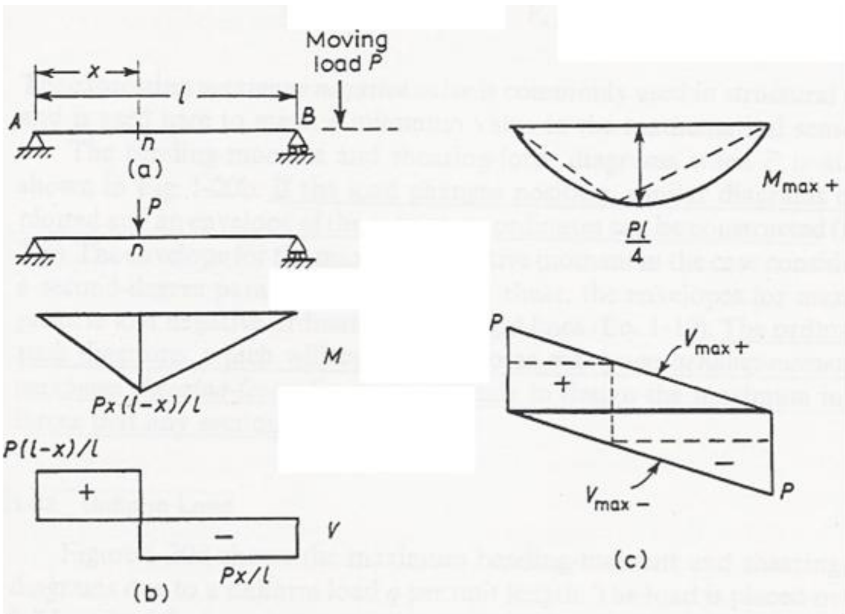


Figure (a) represents a simple beam on 2 supports, across which a concentrated load P can move.

In every section "n" the moment and the shear force are maximal if the load P is exactly above "n". This is shown on figure (b).

When the position of the load is changed, similar diagrams can be made. Finally the envelopes can be drawn as shown on figure (c). As expected, the maximal moment appears in the middle of the beam and the extreme shear forces in the supports.

Using these influence lines, the effect of more loads on the structures, the so-called load system, can be determined. The goal is to find the position of the load system, for which the effect on the structure in a certain point is maximal.

This is illustrated on the following figure.

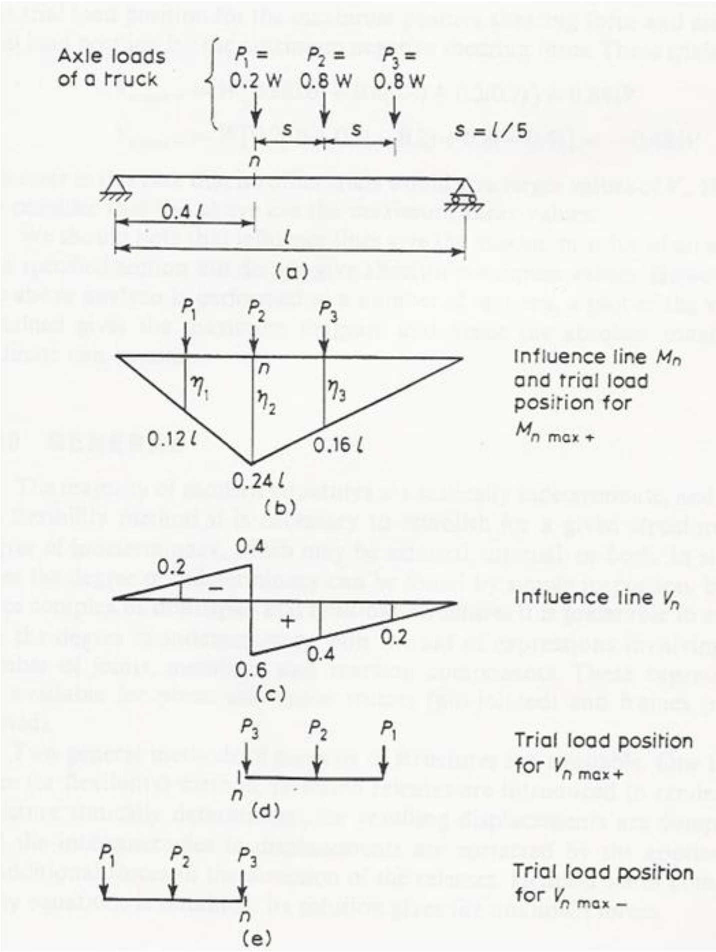


Figure (a) represents a simple beam on two supports again. Across the beam, a system of three point loads can move which represent e.g. the axis loads of a lorry. We look for the position of the load system for which the moment and the shear force are maximal in the section “n”.

The influence line for M_n, the moment in n, is shown on figure (b). The moment resulting from the load system can now be determined as follows:

$$M_n = \sum_{i=1}^{3} P_i \eta_i$$

At which h_i represents the location of the influence line exactly below P_i.
The maximum of M_n is found by trial and error so the sum of the products of an axis load and the influence location below is as large as possible.
This maximum is shown on figure (b) at which the moment M_n can be determined as follows:

$$M_n = Wl\left[0{,}2(0{,}12) + 0{,}8(0{,}24) + 0{,}8(0{,}16)\right] = 0{,}344Wl$$

For every other position of the load system, a lower maximum in n is obtained.
In an analogous way this is illustrated for V_n, the shear force at the place of the section “n”. Figure (c) shows the influence line for the shear force V_n.

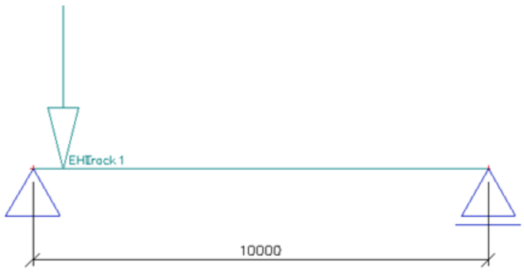
Figures (d) and (e) show the positions of the load system for the maximal positive shear force and the maximal negative shear force.

In SCIA Engineer these various steps appear as follows:

- Create a load case of the type ‘influence lines’
- Input a unit deformation on the position for which the influence line needs to be generated
- Create a load system
- Input Traffic lane across which a load system can move
- Create a ‘moving load generator’ to link the load system to a traffic lane and the load case with the unit load to find the most extreme results on that position for the moving load
- Generate enveloping load cases to gain insight in the global behaviour of the structure with a moving load.

6.2. Influence lines

In this first project **Influence lines.esa** a simple beam is modelled on 2 supports. By means of the module Moving Loads, a traffic lane and a unit deformation in the middle of the beam are defined so the influence lines of the various design components can be reviewed.



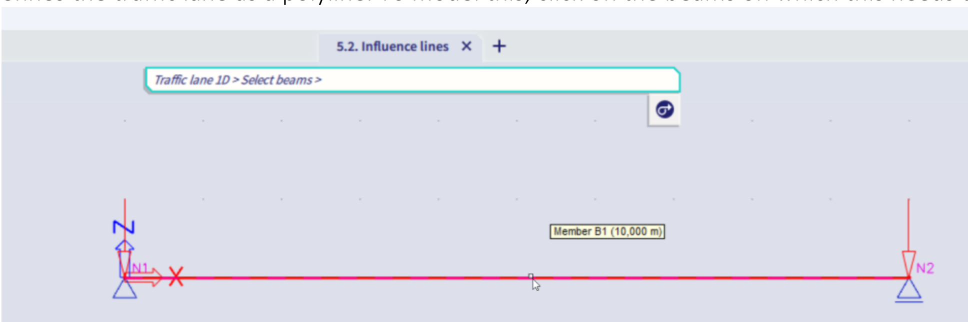
As shown at the Principles, an influence line represents a diagram that shows the effect of a unit load on a variable position in a given point of the structure. To be able to meet this definition, a **traffic lane** has to be defined first, together with a **load case** of the type **‘influence lines’** and a **unit deformation**.

6.2.1. Traffic lane

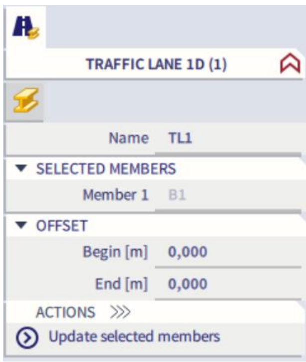
You can import this traffic lane through 

.

The program defines the traffic lane as a polyline. To model this, click on the beams on which this needs to be created.



If the traffic lane consists of several members, you select all beams in the correct order. The property window shows the offset of the traffic lane.

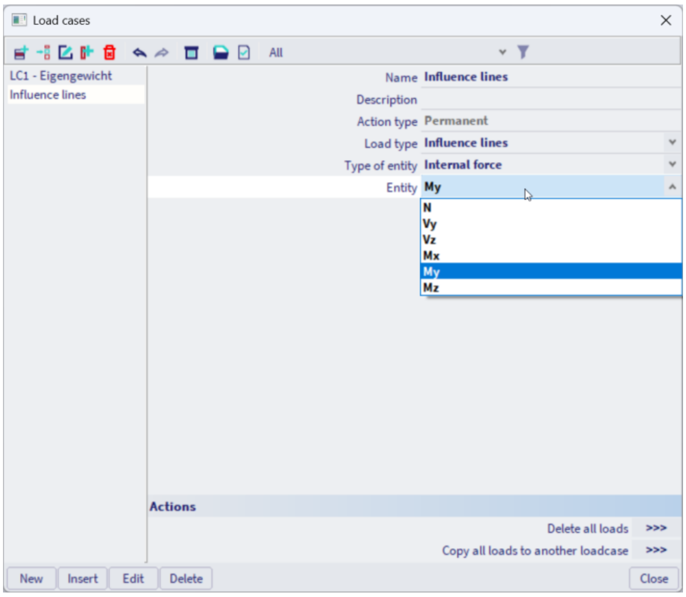


The action ‘Update selected members’ allows regenerating the traffic lane when anything changes to the model.

6.2.2. Unit deformation

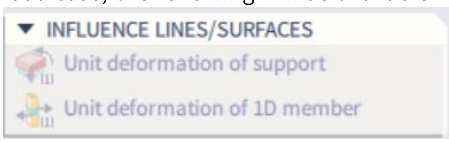
A loadcase needs to be created with the following info. Based on these settings, it is defined which influence line is to be provided.

- Load type: influence lines
- Type of entity: Internal force
- Entity: My

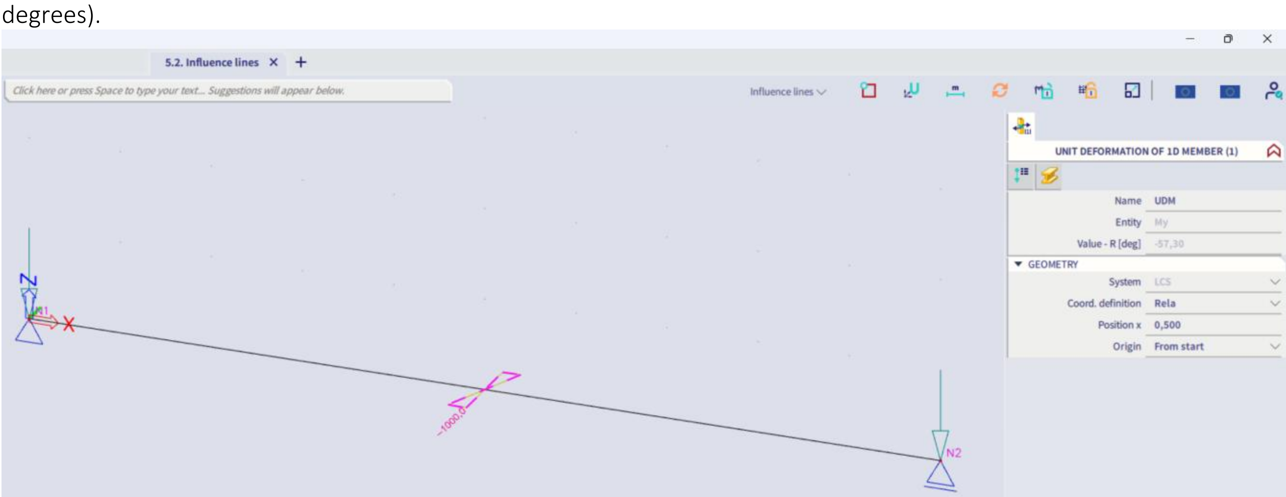


Other components can also be selected (Deformation, reaction force). If you want to review multiple components, you can create multiple load cases.

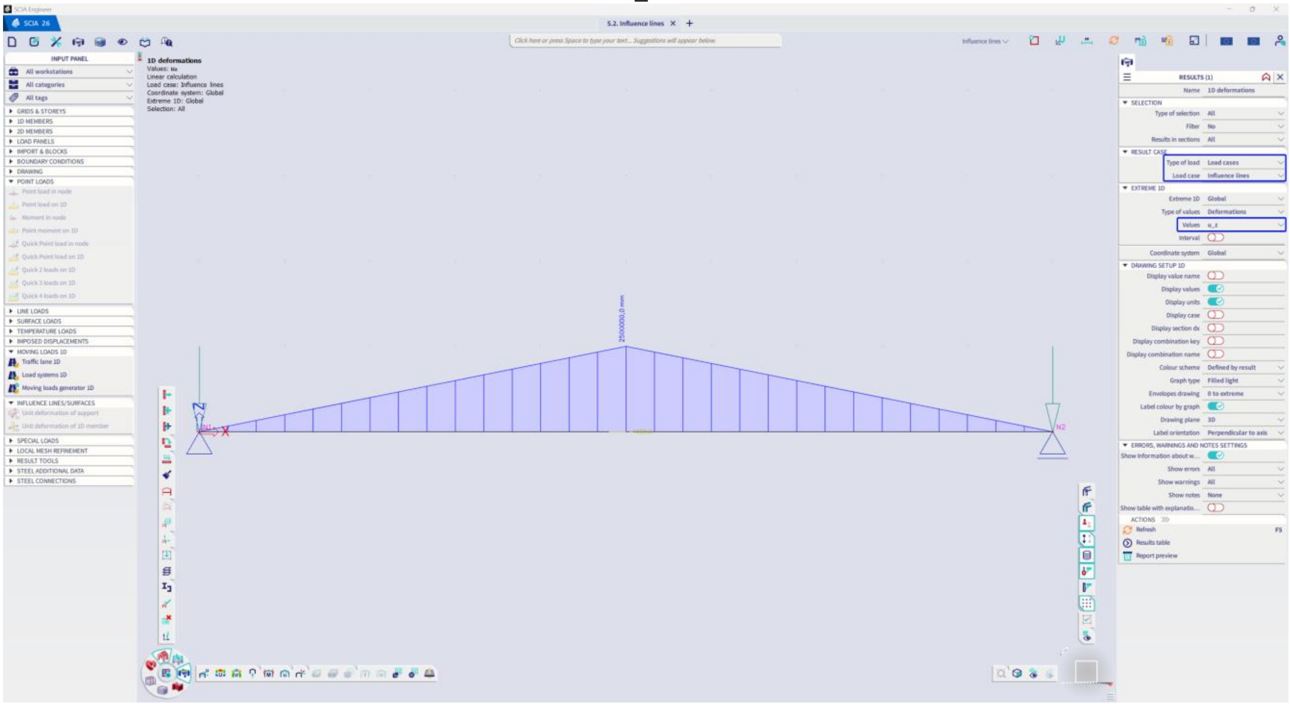
Now the corresponding unit loads can be defined. These are found in the input panel. Depending on the settings of the load case, the following will be available. You will only be able to add one unit deformation to each load case.



Add 'unit deformation of 1D member' to this load case. You define one exact position on the whole structure. This is the position the influence line should be considered. This load itself is adjusted according to the load case settings. The selected value will be 'one unit' and is only informative, it can not be edited. For example: in case of an influence line for the bending moment, the unit deformation is one radian (depending on the unit settings, this could be depicted in degrees).



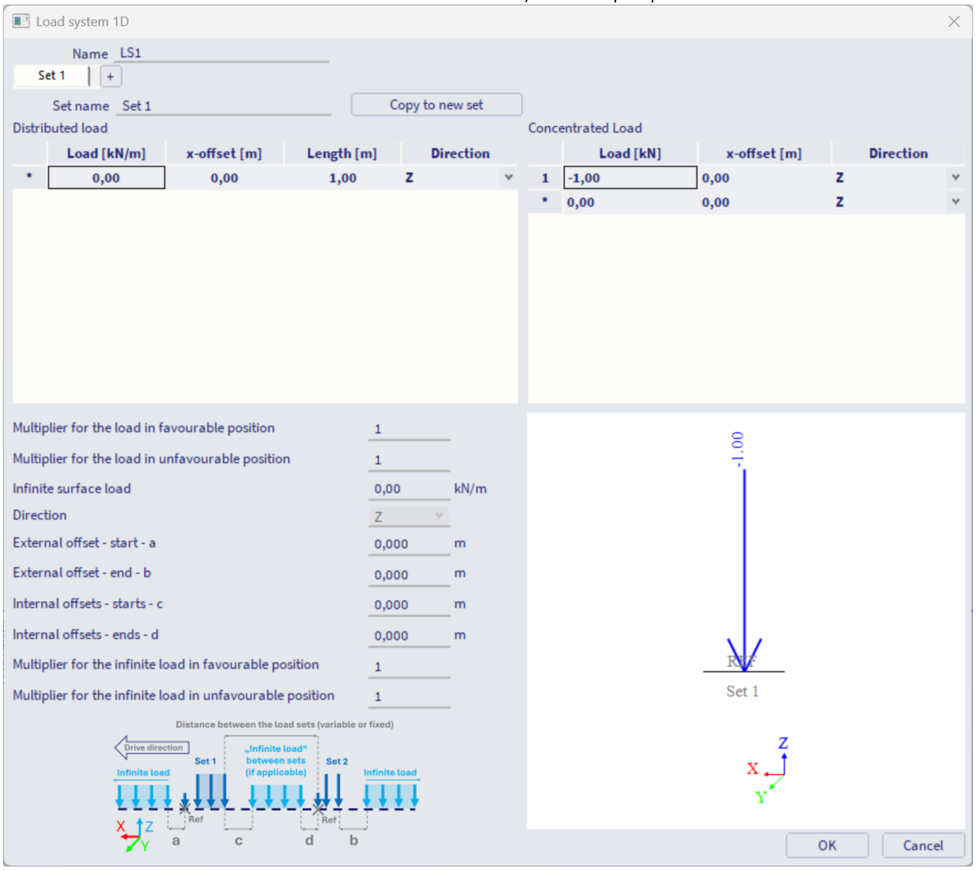
After performing the linear analysis, the influence line can be visualized with the '1D deformations'. You use the created load case as the result case. In this case we use the result u_z.



To review the extreme results of a moving load in this exact position (position where the unit deformation is added), a **load system** needs to be created together with a **moving load generator 1D** which will link the load system with the traffic lane and allow you to define the steps of the moving load.

6.2.3. Load system

With the command 'load system 1D' a load system can be created. For this example, we create a system with one concentrated load, named LS1. The loads and their offset or length are created with the two tables on top (one for a distributed load and one for a concentrated load). Other properties will be discussed later on in this manual.

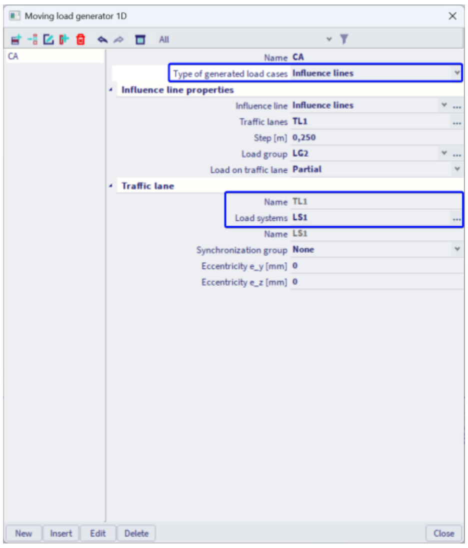


6.2.4. Moving load generator 1D – influence lines

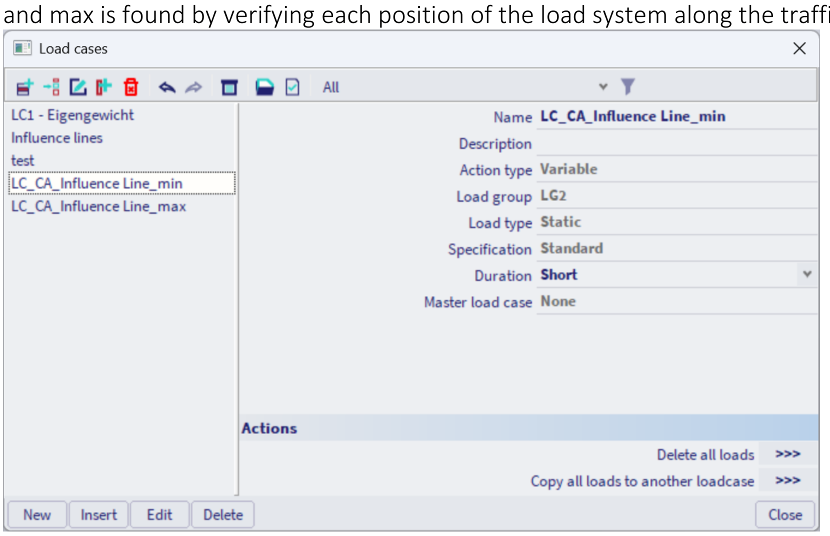
Open the ‘moving load generator 1D’ and create a new entity. This entity will get the name CA by default, standing for combination assembly. This load generator is actually a library of CA entities. It will create new load cases during the linear analysis containing the requested result. These load cases will be added to a selected load group for which you can define the properties. The relation of this load group is set to ‘exclusive’

For this generator we use the following properties:

- Type of generated load cases: Influence lines
- Traffic lane: TL1 (you can select multiple traffic lanes at once)
- Step: 0.250 (defines the different positions of the moving load)
- Load group: LG2 (relation: exclusive)
- Load on traffic lane: Partial or complete
  This property defines if a load system can be partially on the traffic lane when consisting of multiple concentrated and/or distributed loads.
- Load system: LS1
- Eccentricity e_y(z): 0
  This property allows you to define an eccentricity to the load system. For this example we will keep this on 0mm

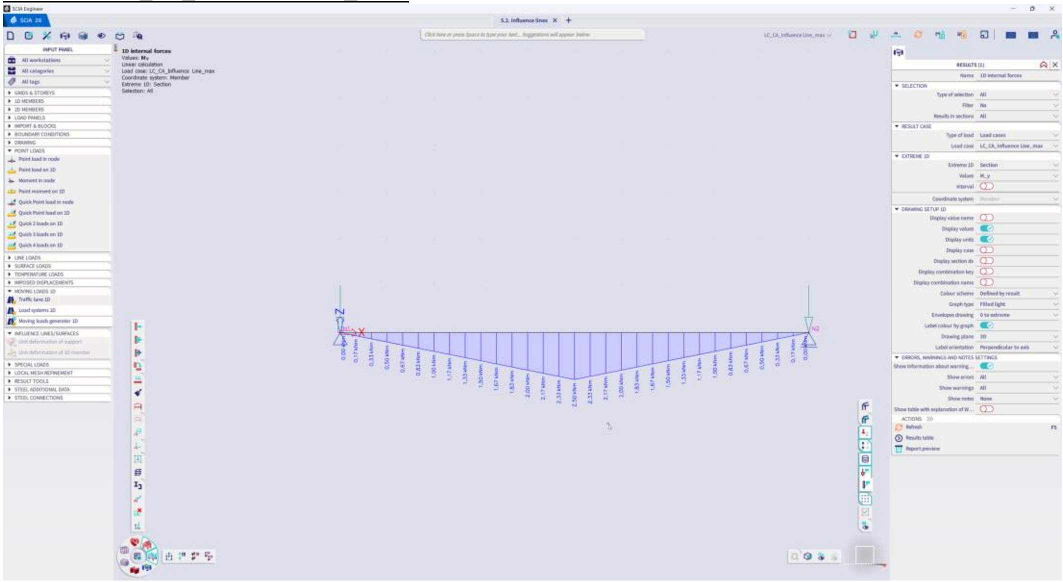


Run the linear analysis. After this analysis we expect to have 2 new load cases created by the generator. One for the most maximum value of My and one of the most minimum value of My on the exact position of the unit load. This min and max is found by verifying each position of the load system along the traffic lane.

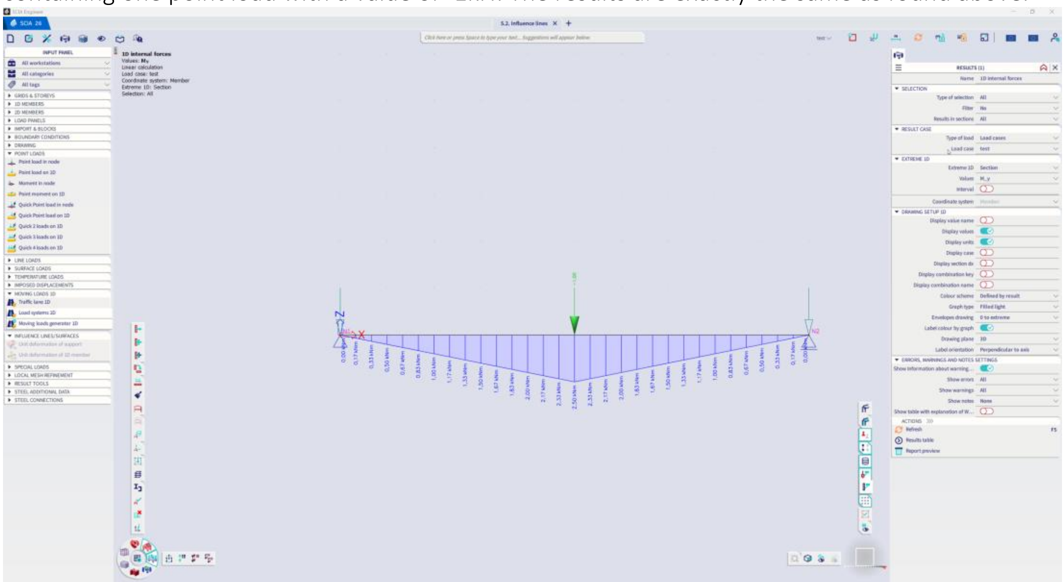


The results of these type of load cases are found in the general results. Since 'type of entity' was set to My for the load case, we review the internal forces on 1D and choose the value M_y.

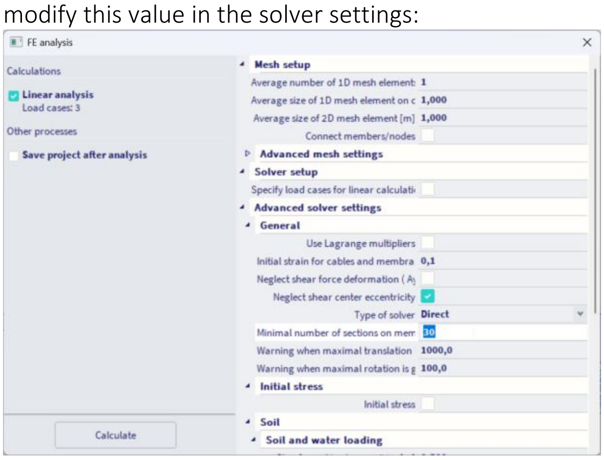
<u>Load case LC_CA_influence Line_max.</u>



This result will correspond to the position of the load system in the middle of the beam. We can manually verify this by creating a test load case with a point load in the middle of the beam. Below you find the result for a test load case containing one point load with a value of -1kN. The results are exactly the same as found above.

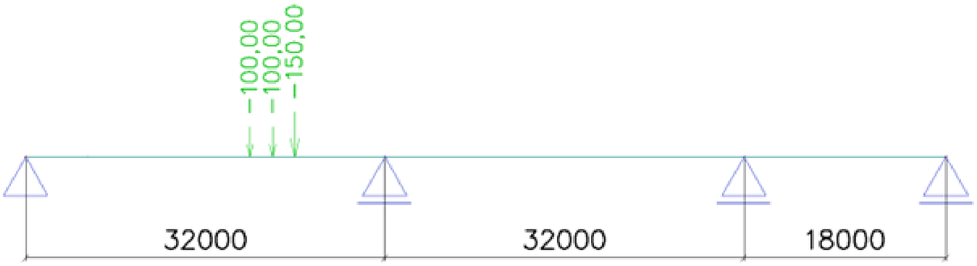


<u>Note</u>: The minimal amount of sections can have an influence on the results. To generate more precise results, you can modify this value in the solver settings:



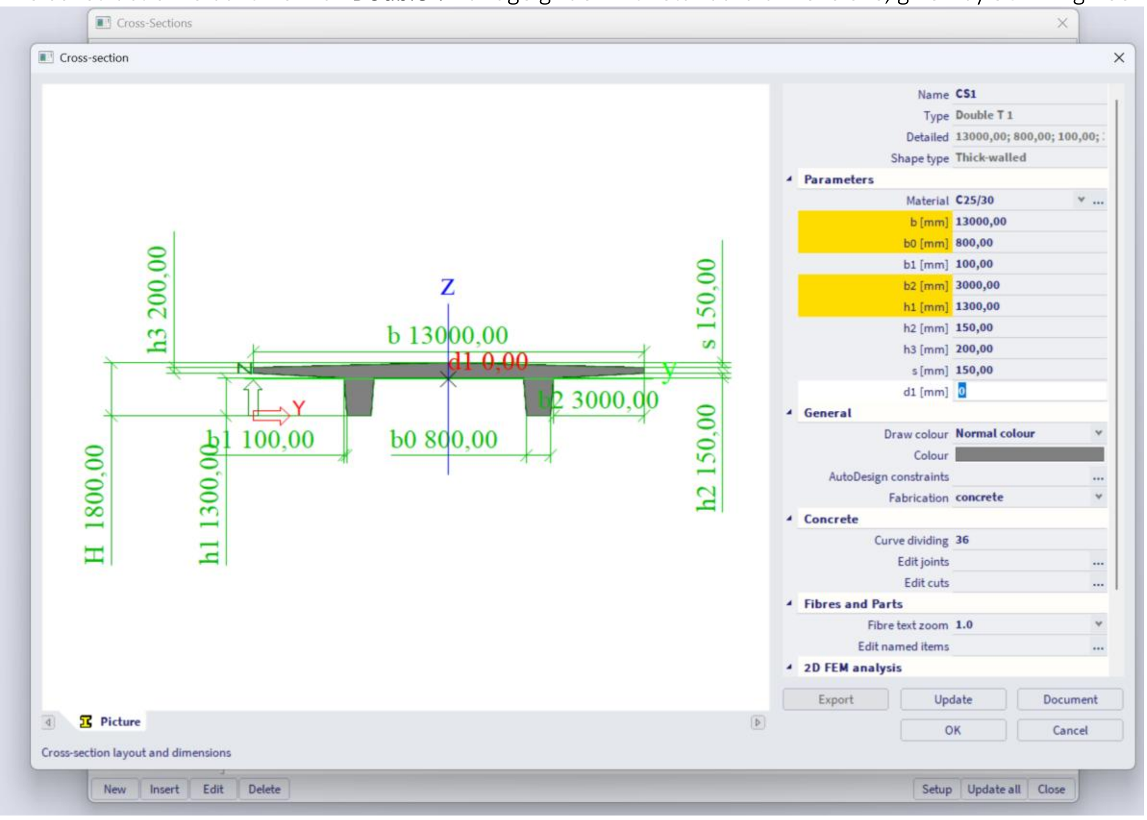
6.3. Envelope load case

Open the project **Load generator.esa**. In this project a bridge deck is modelled on several supports. After defining a **traffic lane** and a **load systems** these are linked via the **moving loads generator 1D**.
Through a selective exploitation, the load cases are automatically calculated for various positions of the load systems. In a last step, the envelope load cases are generated for various design components to gain insight in the global behaviour of the structure.



6.3.1. The construction

The construction is built from a "**Double T**" bridge girder with standard dimensions, given by SCIA Engineer.



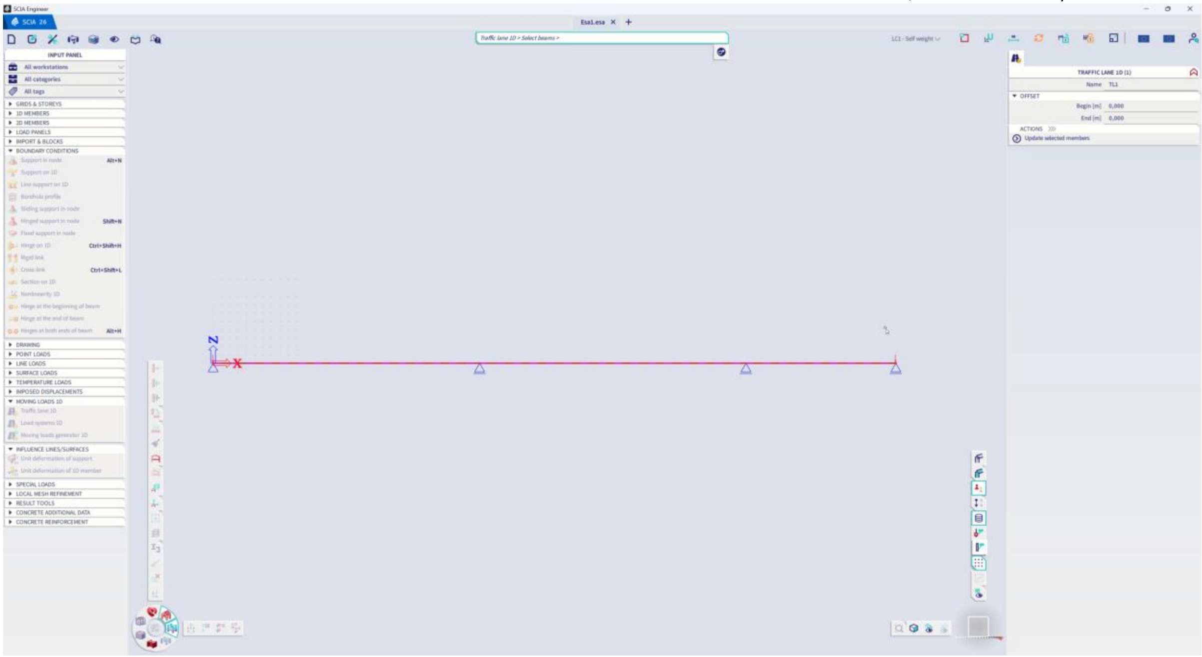
The construction is inserted as 3 horizontal beams through 

, at which the begin node is imposed hinged and the other nodes are rolled.

To be able to calculate the construction, one load case is created automatically; the Self Weight.

6.3.2. traffic lane

After entering the construction, use **Traffic lane 1D** from the input panel to create a traffic lane.
Select all beams to create a traffic lane over all members. As **Name** of the traffic lane, TL1 is used by default.



6.3.3. Load systems

After defining the traffic lane, a load system can be created via **Load systems 1D** found in the input panel. Both Single and Multiple Load systems can be defined.

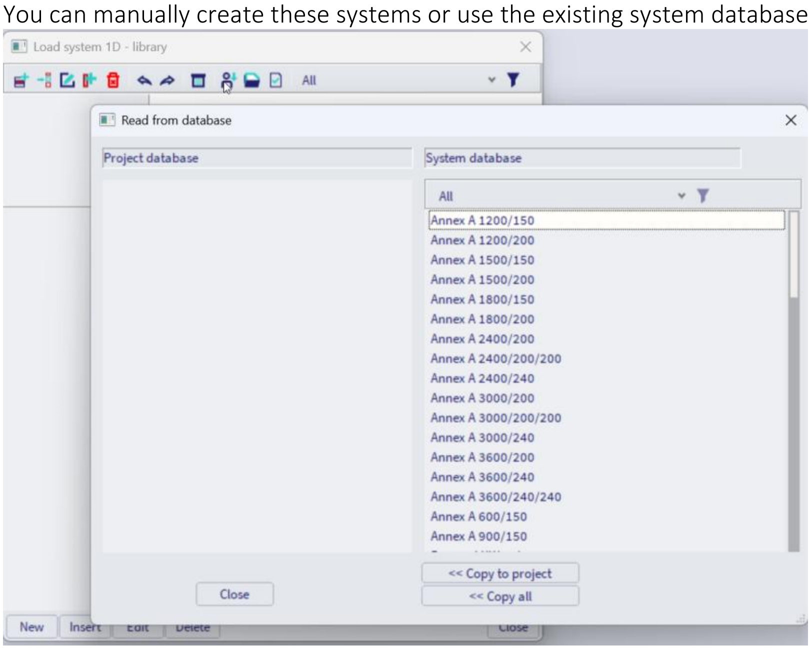
Possibilities with Single Load systems:

- A coherent combination of concentrated loads
- A coherent combination of distributed loads
- A combination of both concentrated and distributed loads

Possibilities with Multiple Load systems:

- Multiple independent systems of concentrated loads and distributed loads with variable interval in combination with an infinite distributed load of an indefinite length.

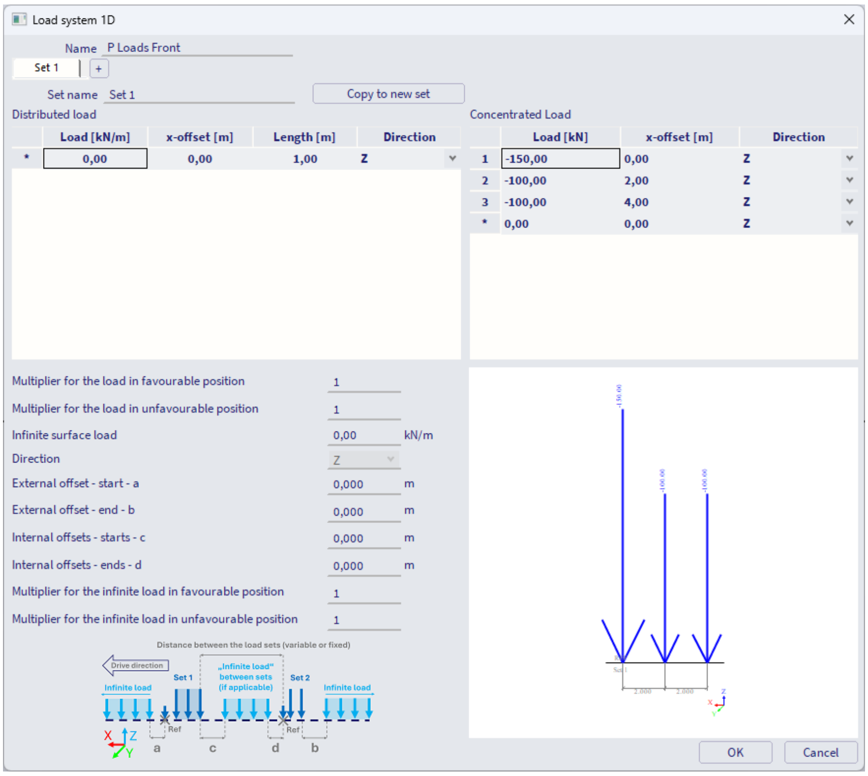
You can manually create these systems or use the existing system database:



In this project we will discuss the following single load systems:

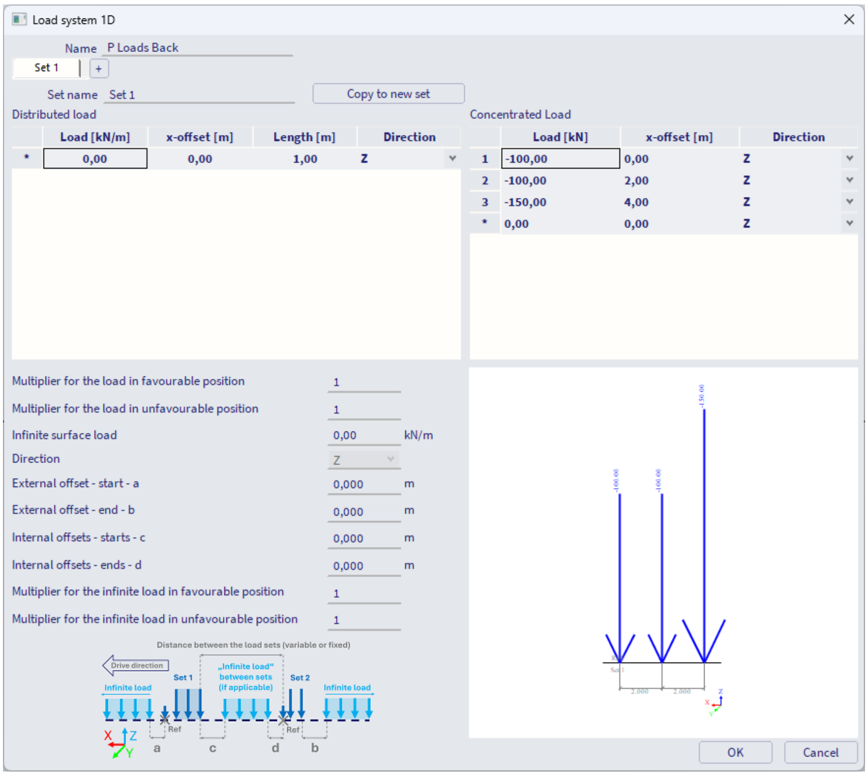
*<u>1) Single Load system **P Loads Front**</u>*

This load system consists of a point load of 150 kN and 2 point loads of 100 kN with a mutual distance of 2m. The point load of 150 kN is at the front and will be the first to run on the traffic lane.



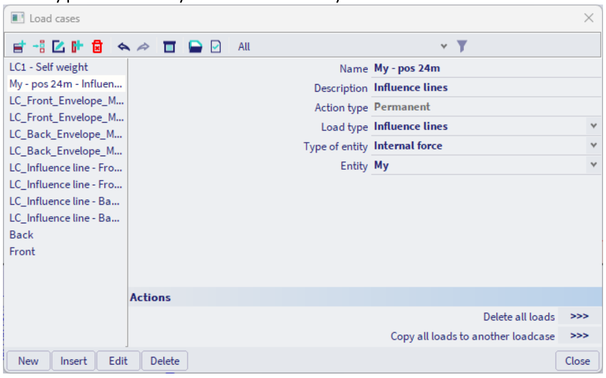
*<u>2) Single Load system **P Loads Back**</u>*

This load system consists of a point load of 150 kN and 2 point loads of 100 kN with a mutual distance of 2m. The point load of 150 kN is at the back and will be the last to run on the traffic lane.

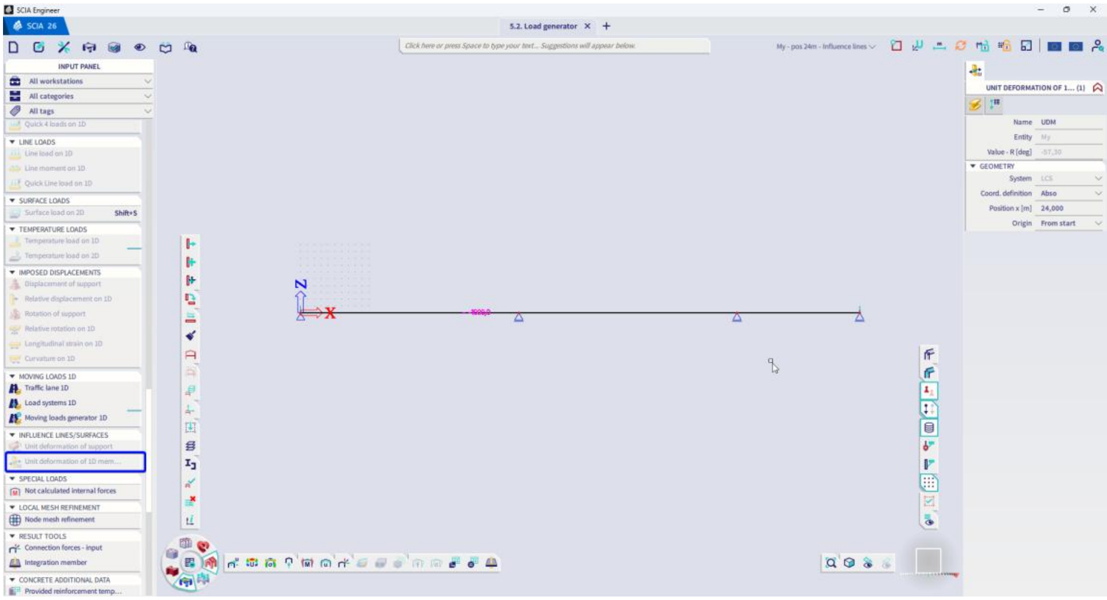


6.3.4. Moving load generator 1D – Influence lines

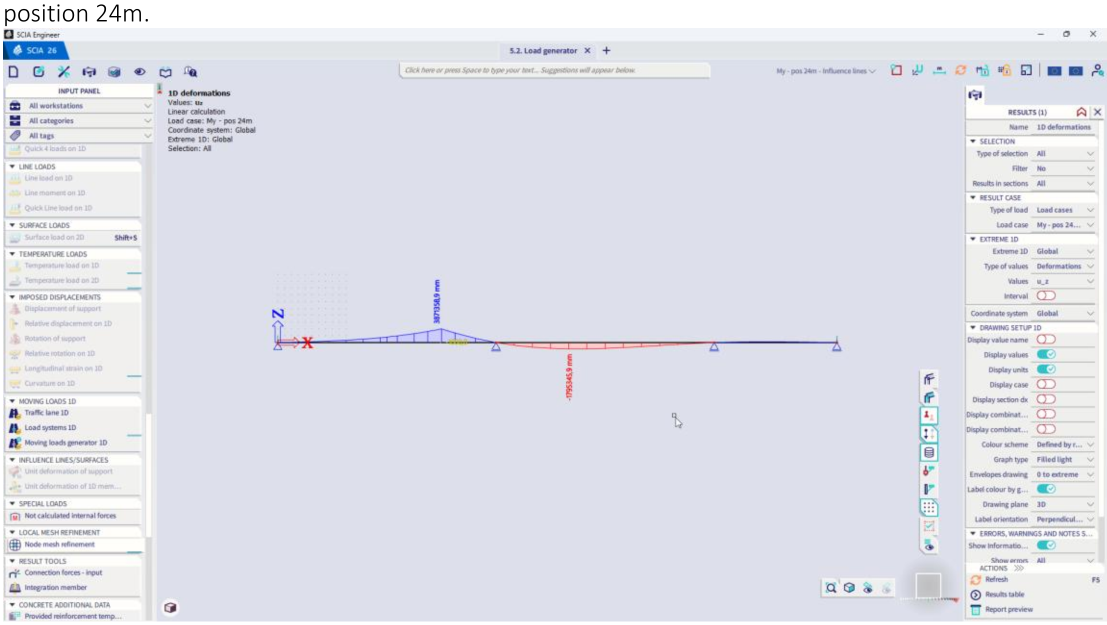
After defining the traffic lane and the load system, a load case can be created of the type ‘influence lines’. You define the type of entity and the entity to define which influence line needs to be generated. We choose the component My.



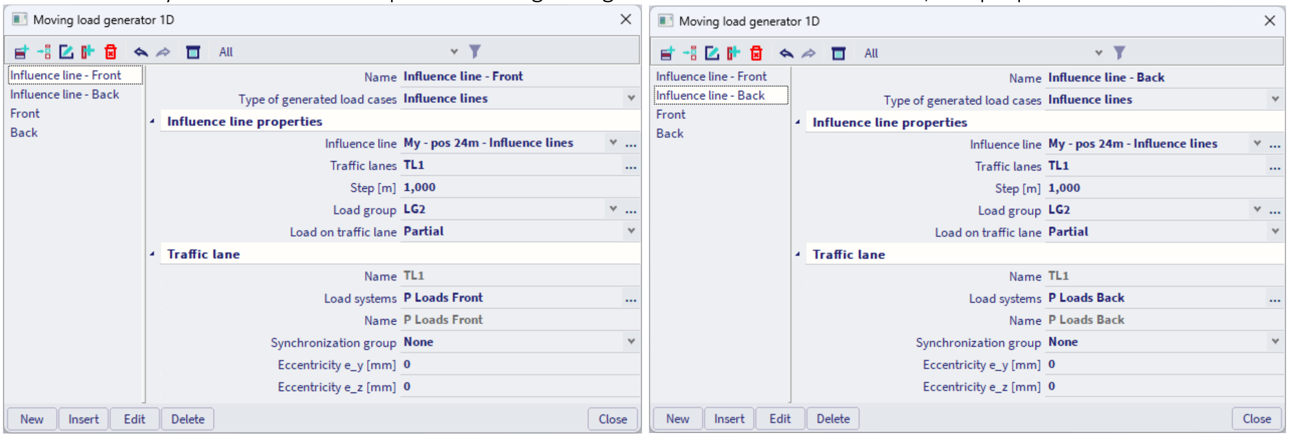
Inside this load case, you add a ‘unit deformation’ in the position you would like to find the extreme results of the mobile load. In this case a unit deformation is added to beam B1 at a position of 24m. The entity and value is defined based on the selected load case.



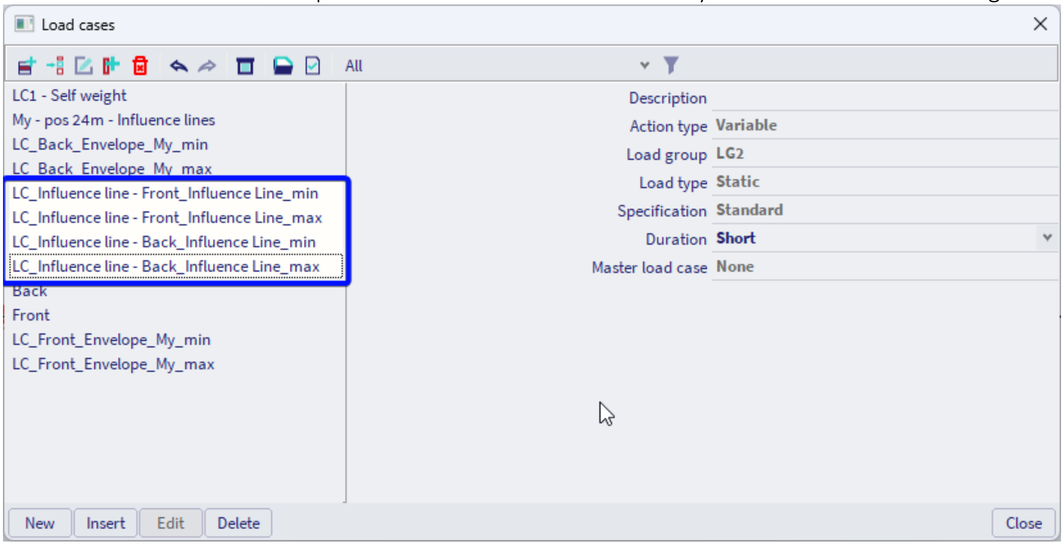
Perform the linear analysis. The influence line for the unit deformation can be checked through ‘1D deformations’. Review the result u_z for the created load case ‘My- pos 24m – Influence lines’. This shows the influence line for position 24m.



With the moving load generator 1D, an exploitation is performed for the moment **My** on a position **24m** on the first beam **B1**. We will do so for the two load systems separately.
For each load system we create a separate moving load generator 1D. Other than that, the properties are the same.

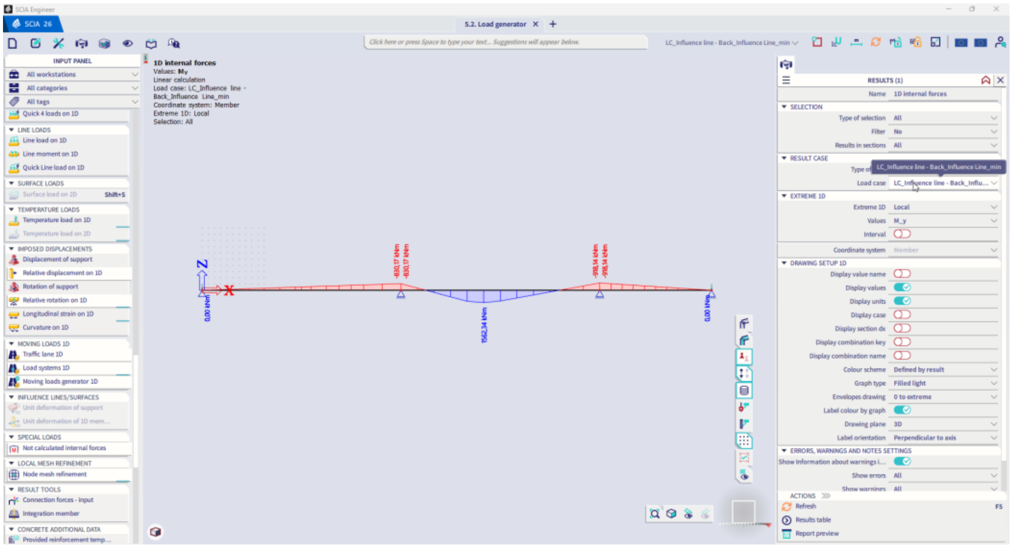


Run the linear analysis. Each generator will create two load cases containing a maximum and a minimum result of bending moment My found on position 24m. This maximum and minimum is found by running the load system over the traffic lane with a defined step distance. This load case will show you the result of the moving load in one exact position.

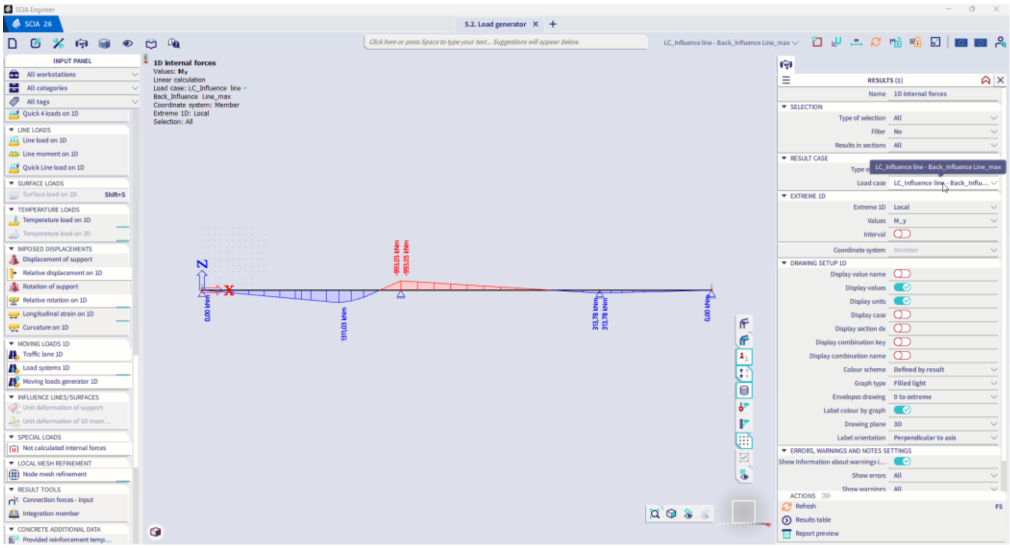


Since the type of entity of the load case is set to My, the results of these load cases, can be found via '1D internal forces'.

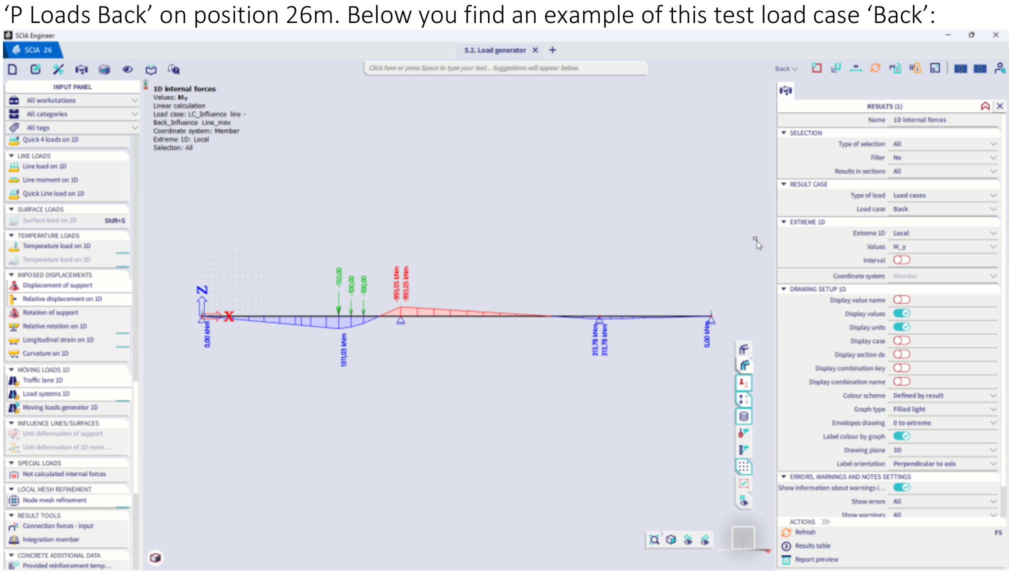
- Load system 'P loads Back'
  max:



min:

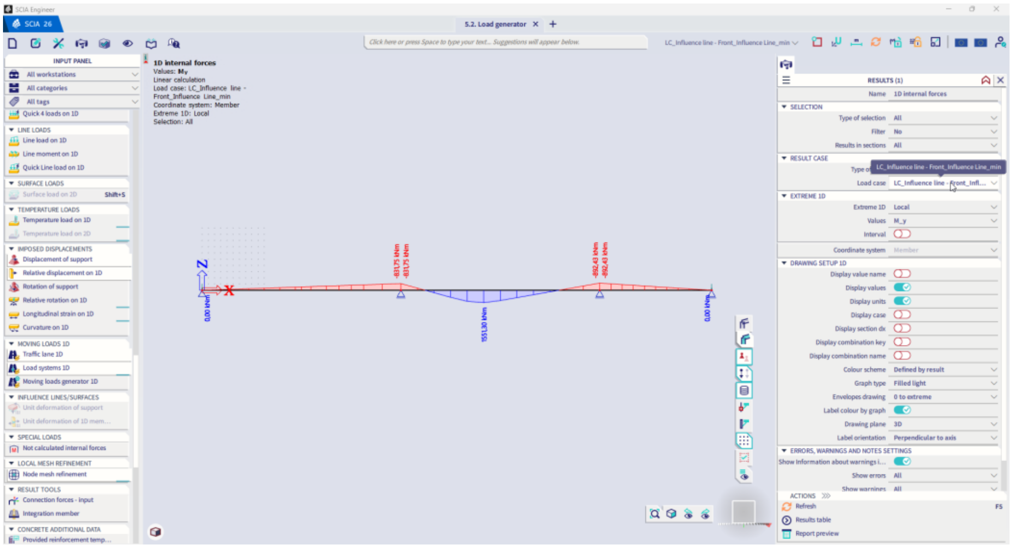


You can manually verify these result by manually creating a load case representing the load system on the position causing the maximum result. The maximum result My on position 24m is found when load system 'P Loads Back' on position 26m. Below you find an example of this test load case 'Back':

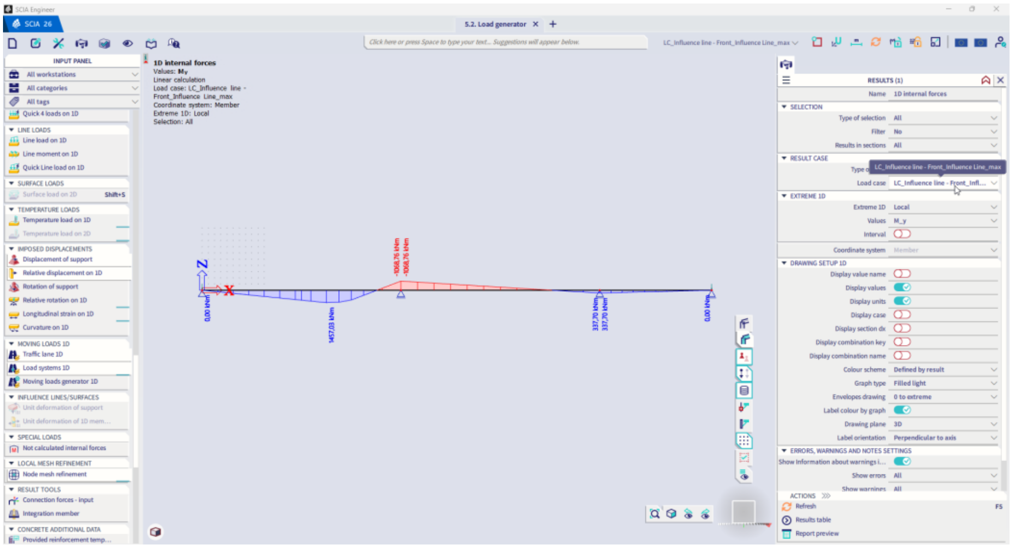


- Load system ‘P loads Front’

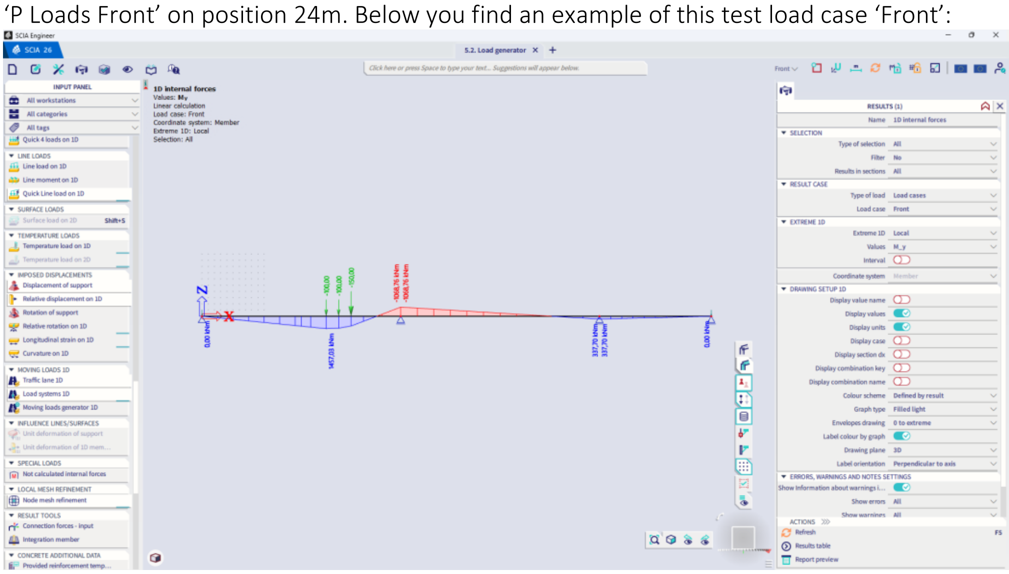
  Max:



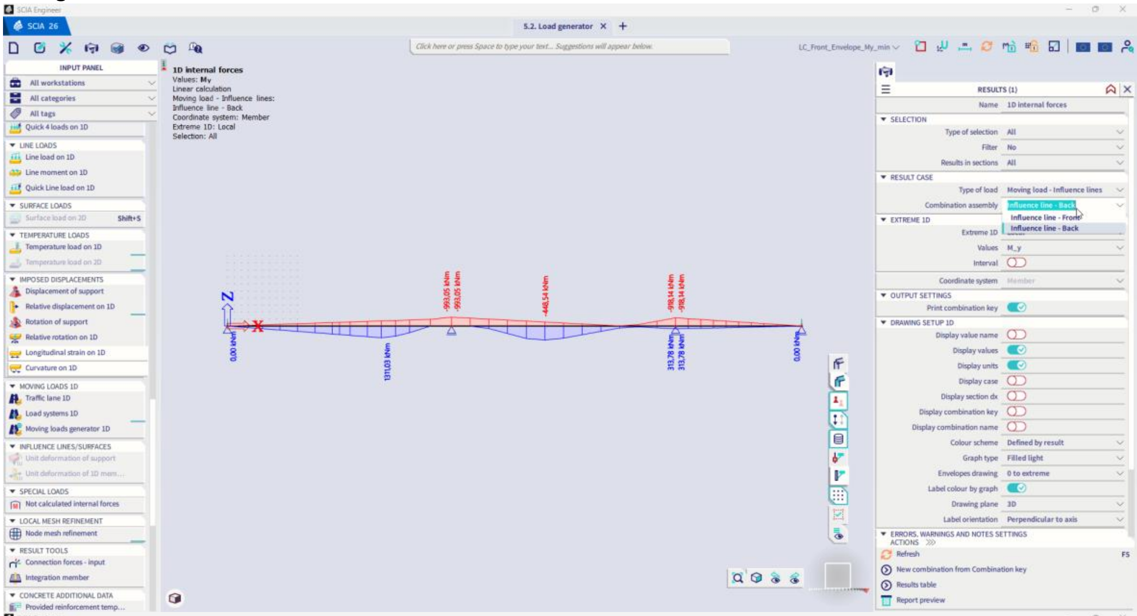
Min:

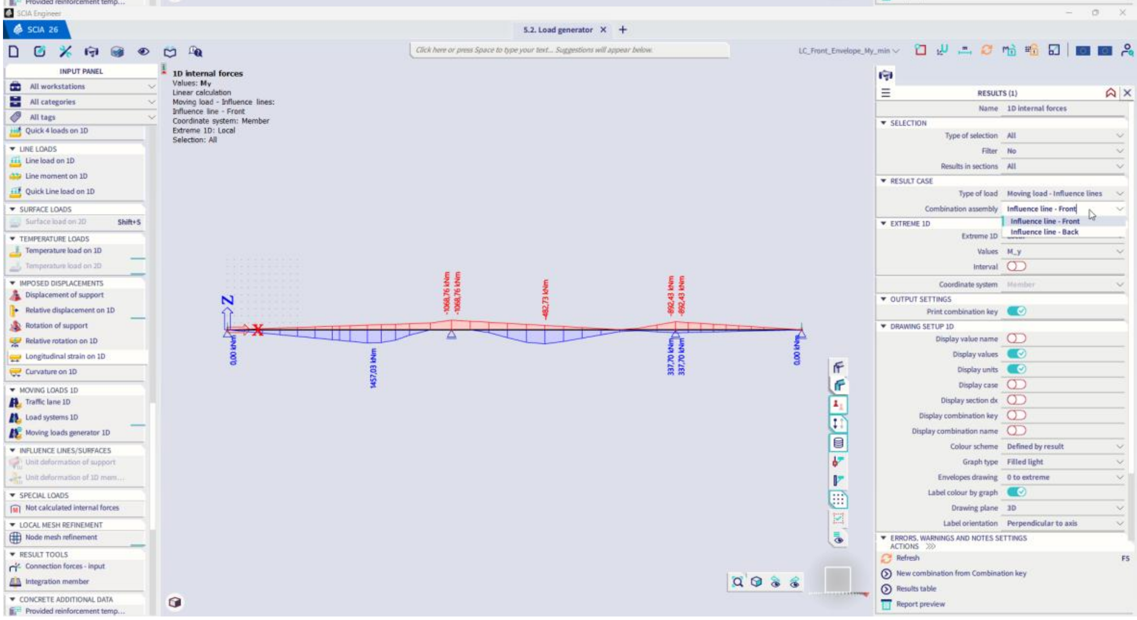


You can manually verify these result by manually creating a load case representing the load system on the position causing the maximum result. The maximum result My on position 24m is found when load system ‘P Loads Front’ on position 24m. Below you find an example of this test load case ‘Front’:



An envelope of this maximum and minimum can be found as well by changing the type of load in the result properties to 'moving load – influence lines'. Here you find the 'combination assembly' based on the name you used in the moving load generator 1D.

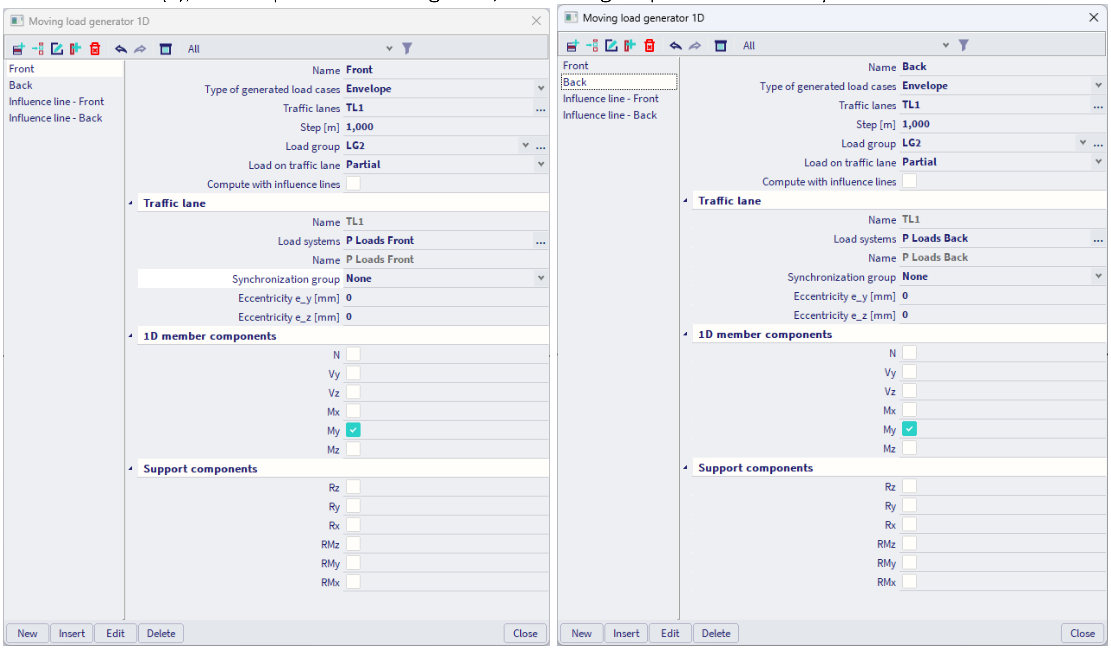




6.3.5. Moving load generator 1D – Envelope load case

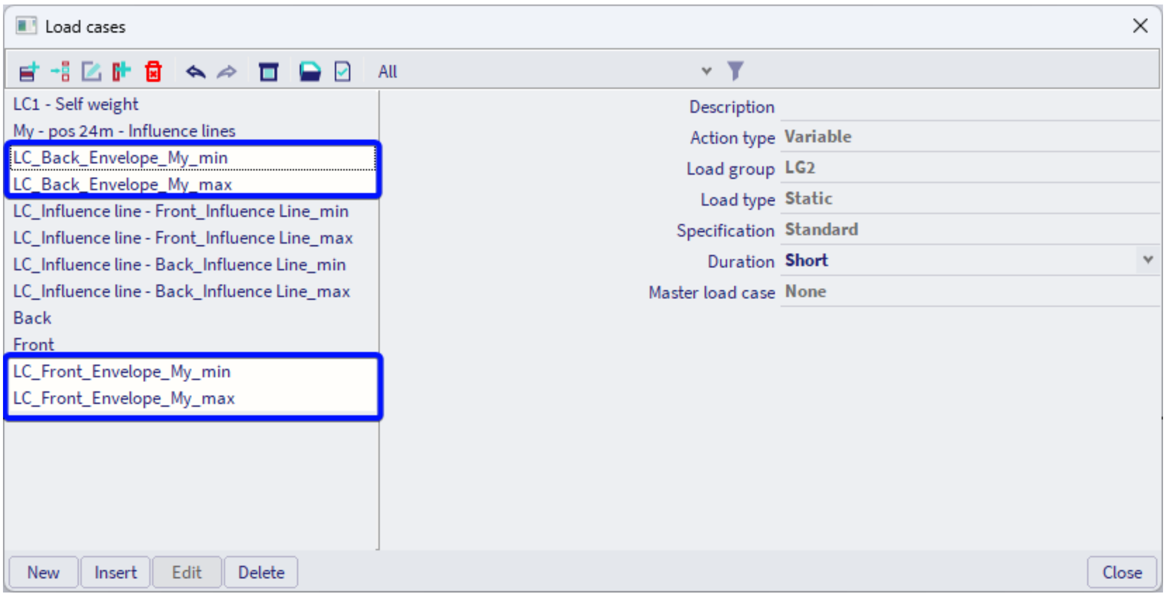
Instead of generating 'influence lines', envelope load cases can be generated. For these envelope load cases, internally several linear load cases will be analysed, respecting varying positions of the load sets within the load system assigned to one or more traffic lanes. Out of this vast number of load cases, a resulting envelope will be provided. For this it is required to select which internal forces and/or which support reactions you want to investigate through the whole structure. At least one component needs to be selected.

To investigate both load systems individually, we create two separate moving load generators. The 'type of generated load cases' is set to 'Envelope'. We select the component My in this example, but multiple components can be selected to create multiple envelopes. Other properties are similar to generating 'influence lines' as discussed before, you define the traffic lane(s), the step of the moving load, the load group and the load system.

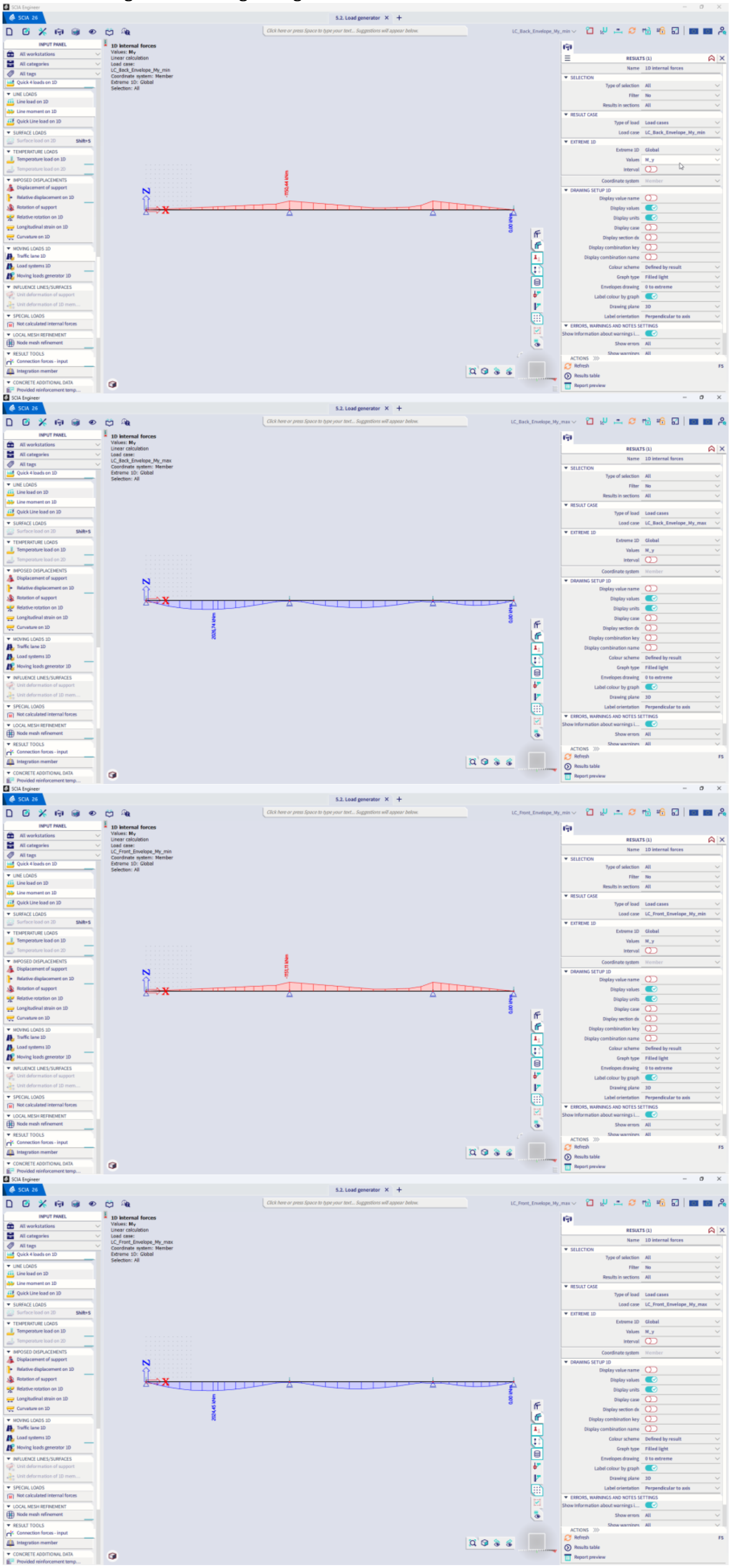


Note: the properties 'Synchronization group' and 'compute with influence lines' will be discussed later on.

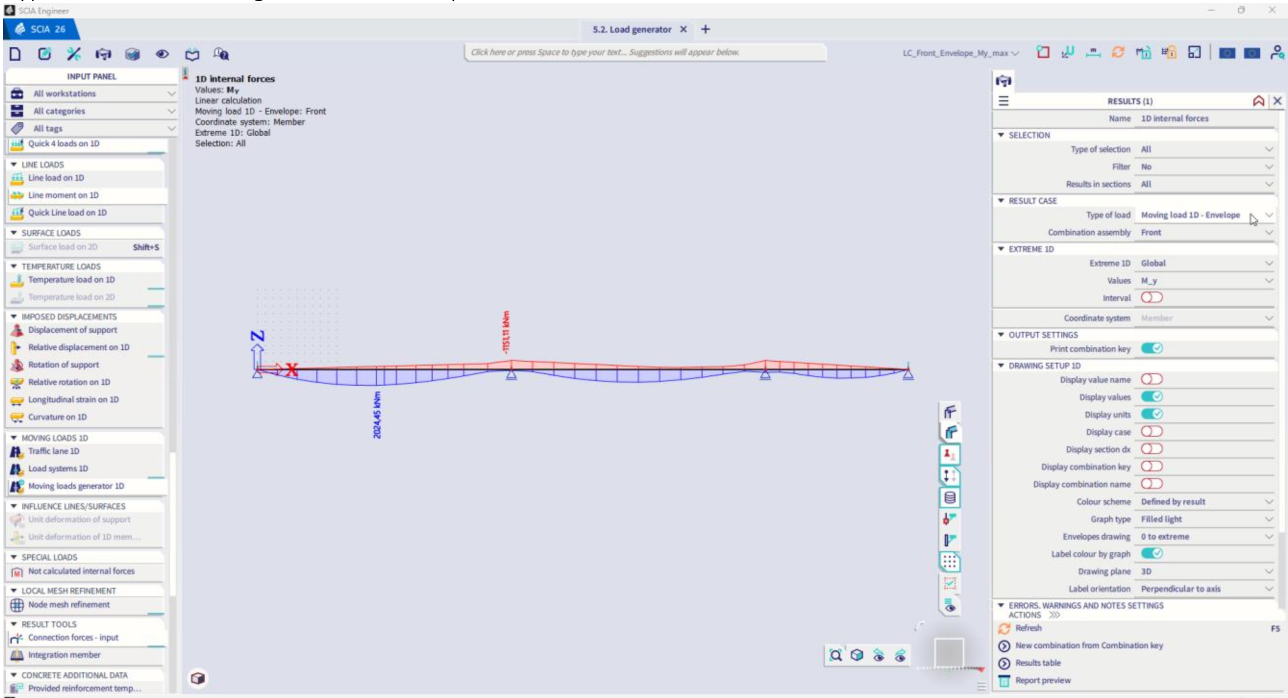
After performing the linear analysis, these envelope load cases are added. For each component of each generator a minimum and maximum will be created.

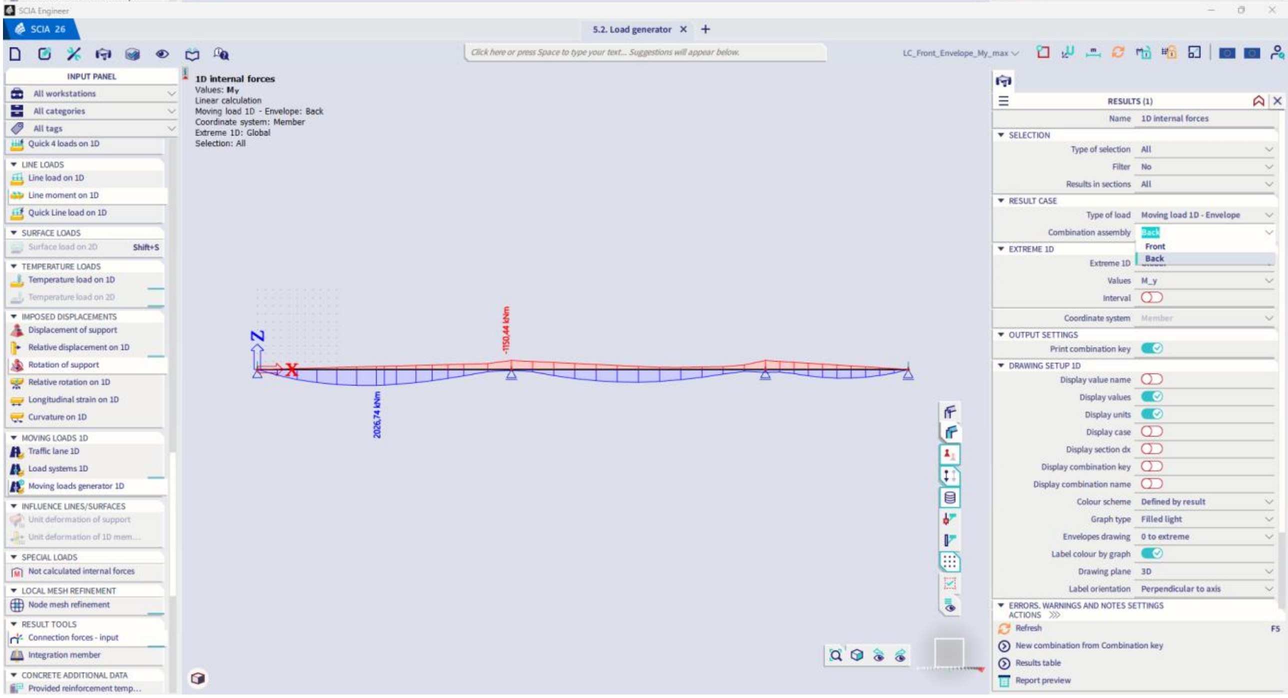


These results can be viewed via ‘Internal forces 1D’. These will show you either the maximum or minimum results of My for the moving load moving along the traffic lane.



Similar as found for influence lines, there will also be an envelope to review the min and max at the same time. Set the 'type of load' to 'Moving load 1D – Envelope'.



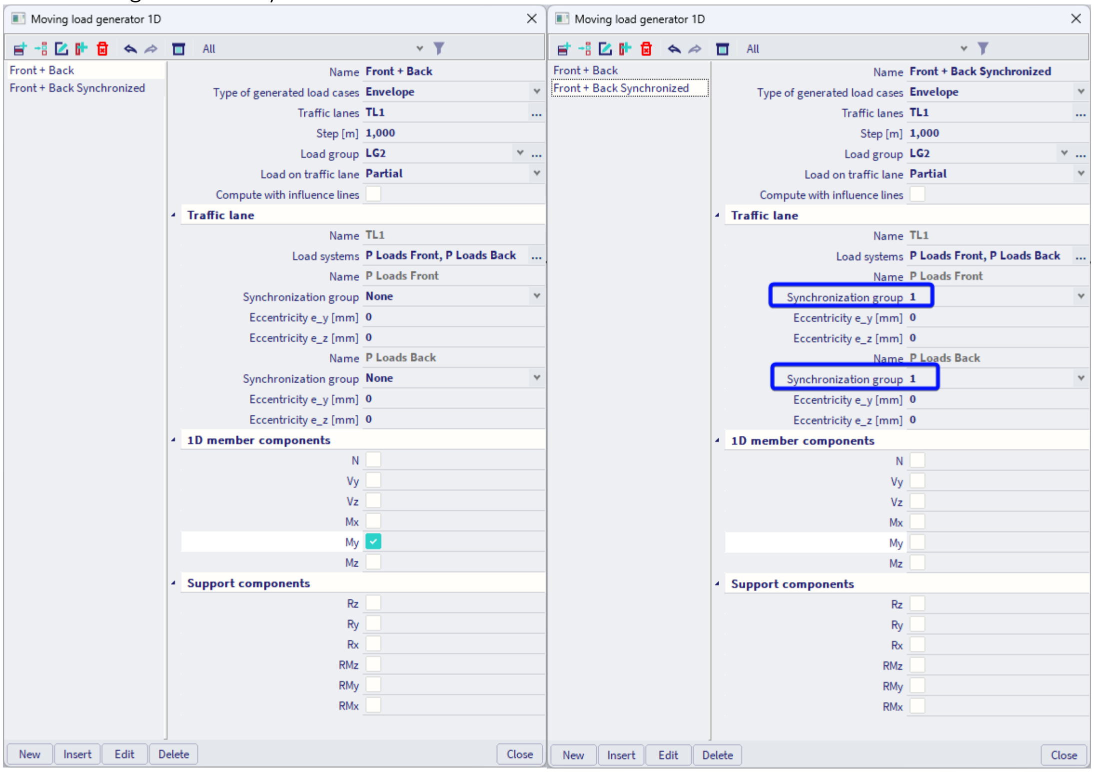


6.3.6. Multiple load systems

In the previous example, we created different singular load systems. It is also possible to let multiple load systems run on one or multiple traffic lanes or to create a load system with multiple sets and variable distances.

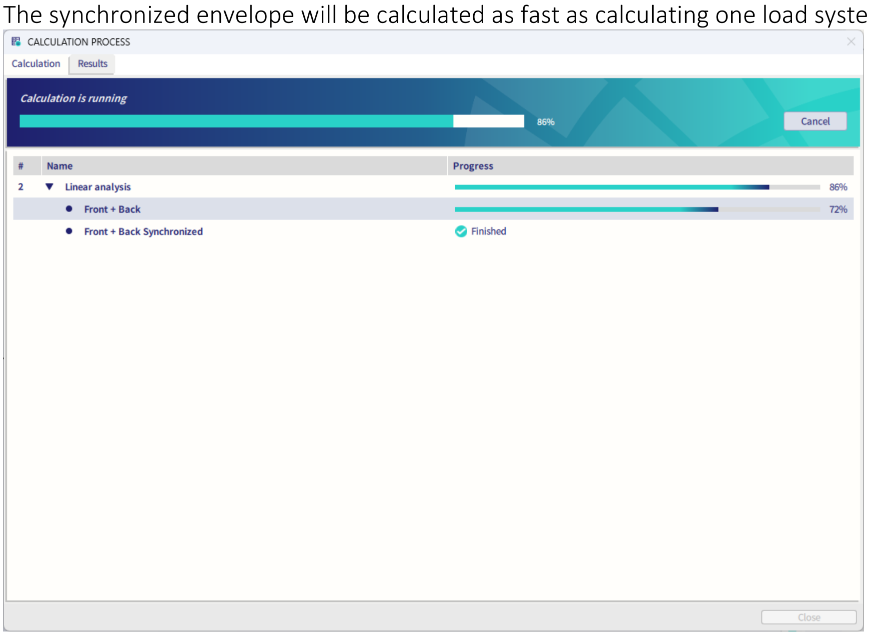
Different load systems on one traffic lane

In the example **Synchronized Load Systems.esa** the same load systems 'P Loads Front' and P Loads 'Back' are created. To let the different load systems run on the same traffic lane at the same time, we can create a 'mobile load generator 1D' including both load systems.



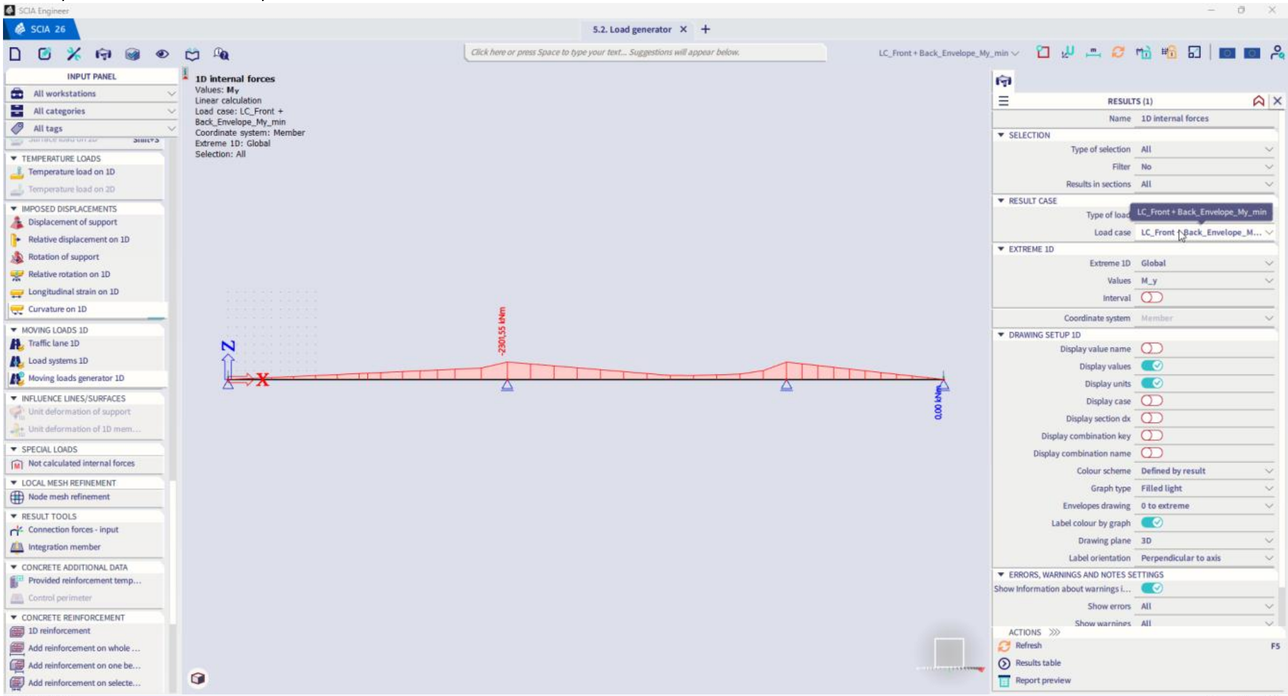
Now both load systems are added, you can select a synchronization group to indicate if different load systems need to be moving at the same time. We created two generators, one 'Front + Back' which will not be synchronized, one 'Front + Back synchronized' which will be synchronized by selecting Synchronization group '1'.

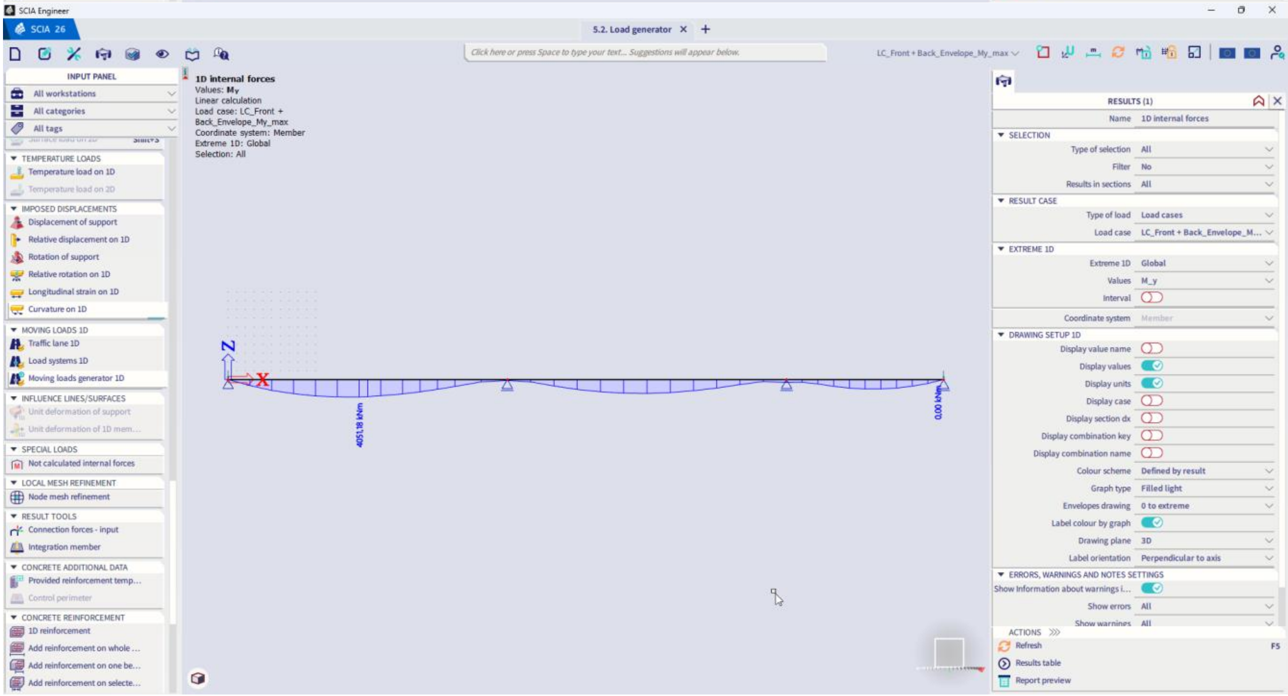
After performing the linear analysis, the envelope load cases are created. You will notice that the calculation for the unsynchronized envelope will take longer since both load systems on different positions need to be taken into account. The synchronized envelope will be calculated as fast as calculating one load system on a traffic lane.



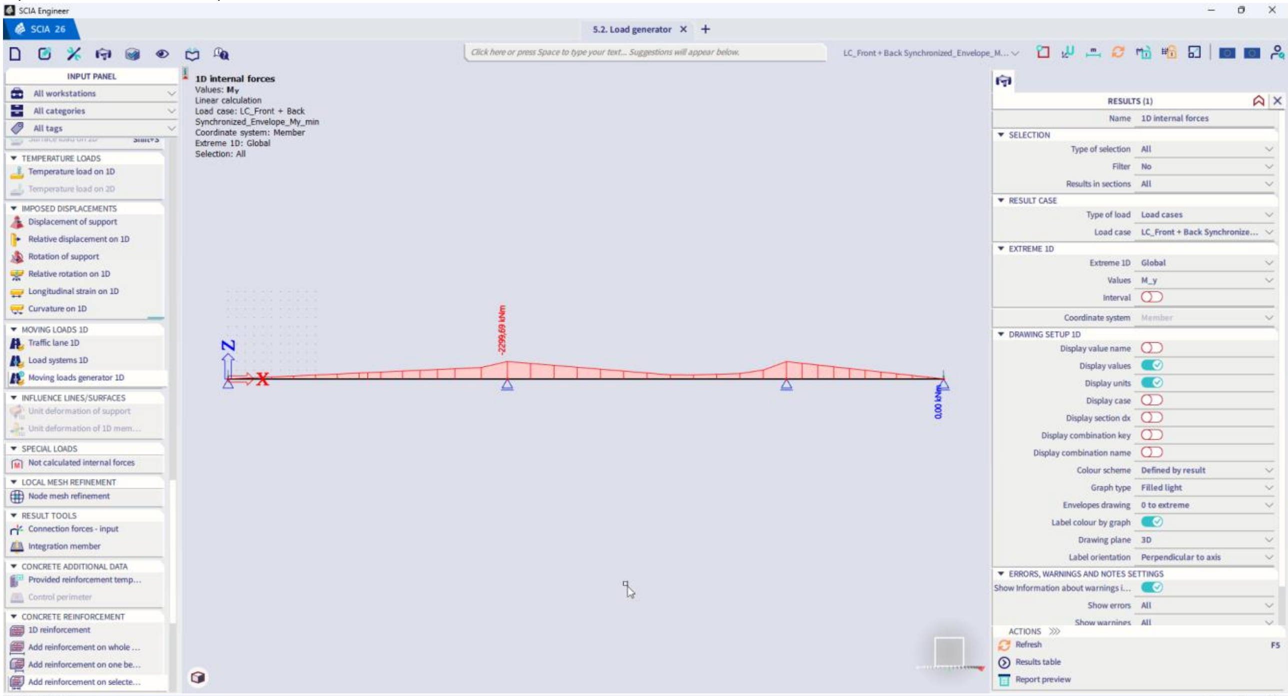
For both generators, a min and max envelope will be created and the results can be viewed again via '1D internal forces'. You will also find an envelope containing both min and max values.

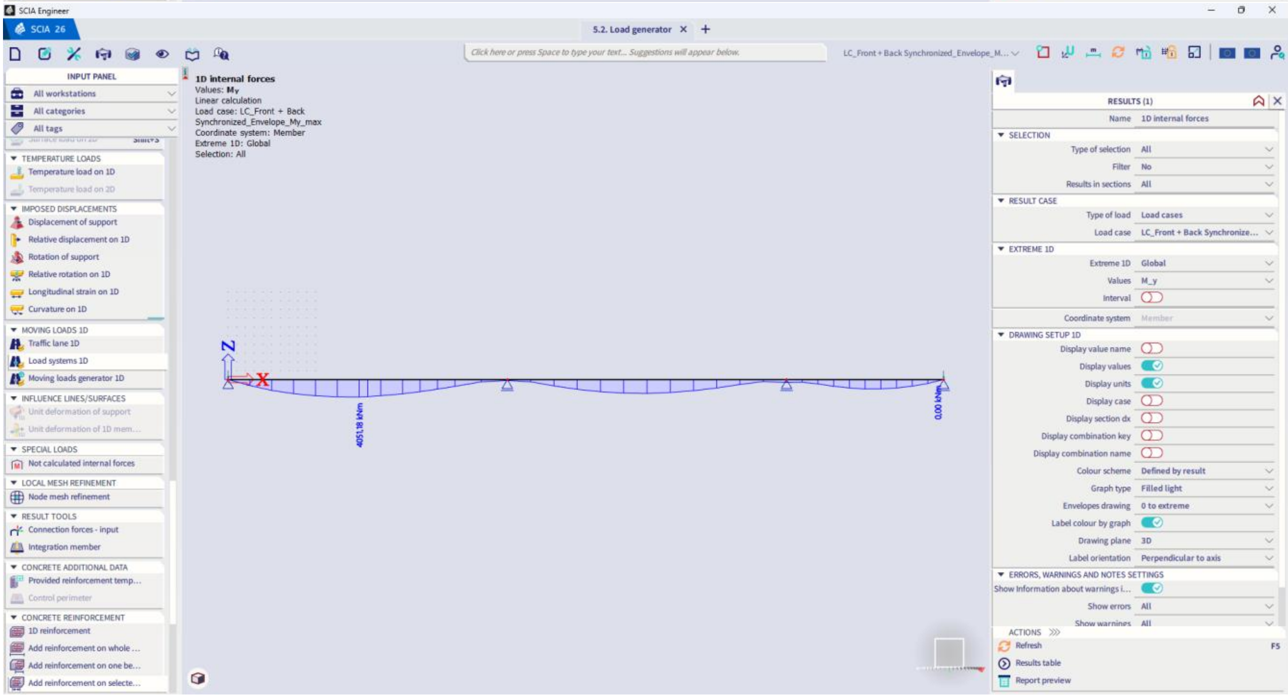
- Not synchronized load systems:





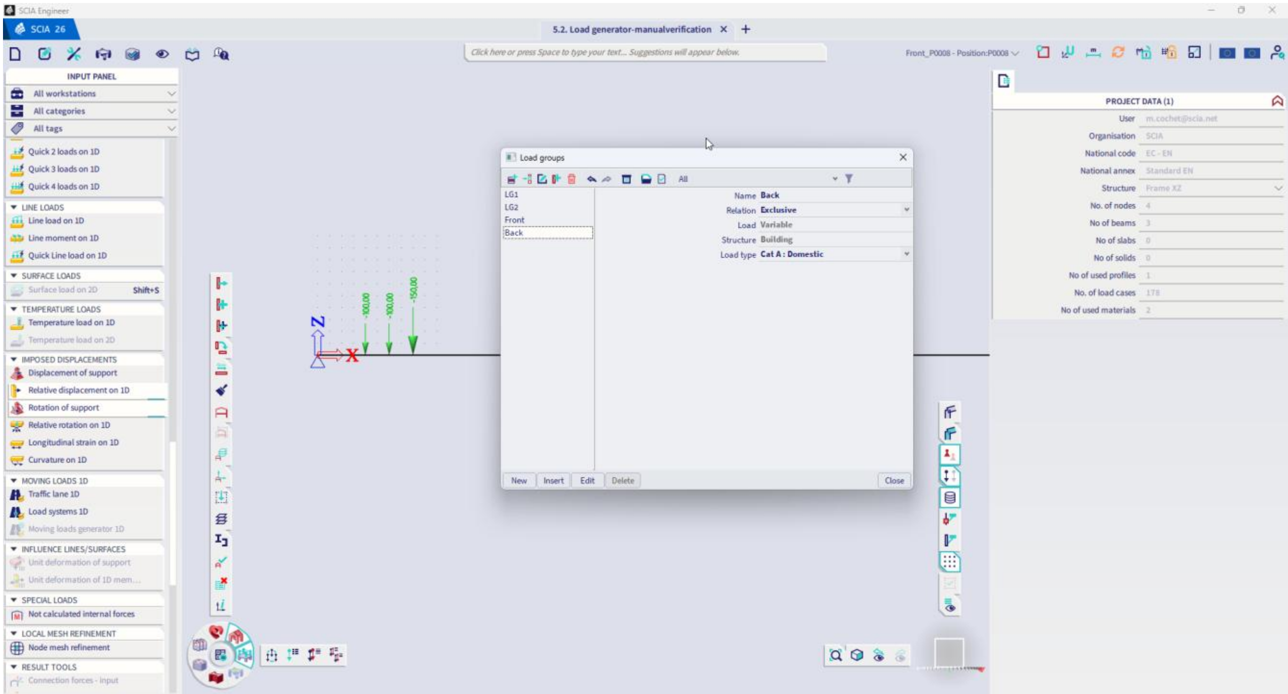
- Synchronized load systems:

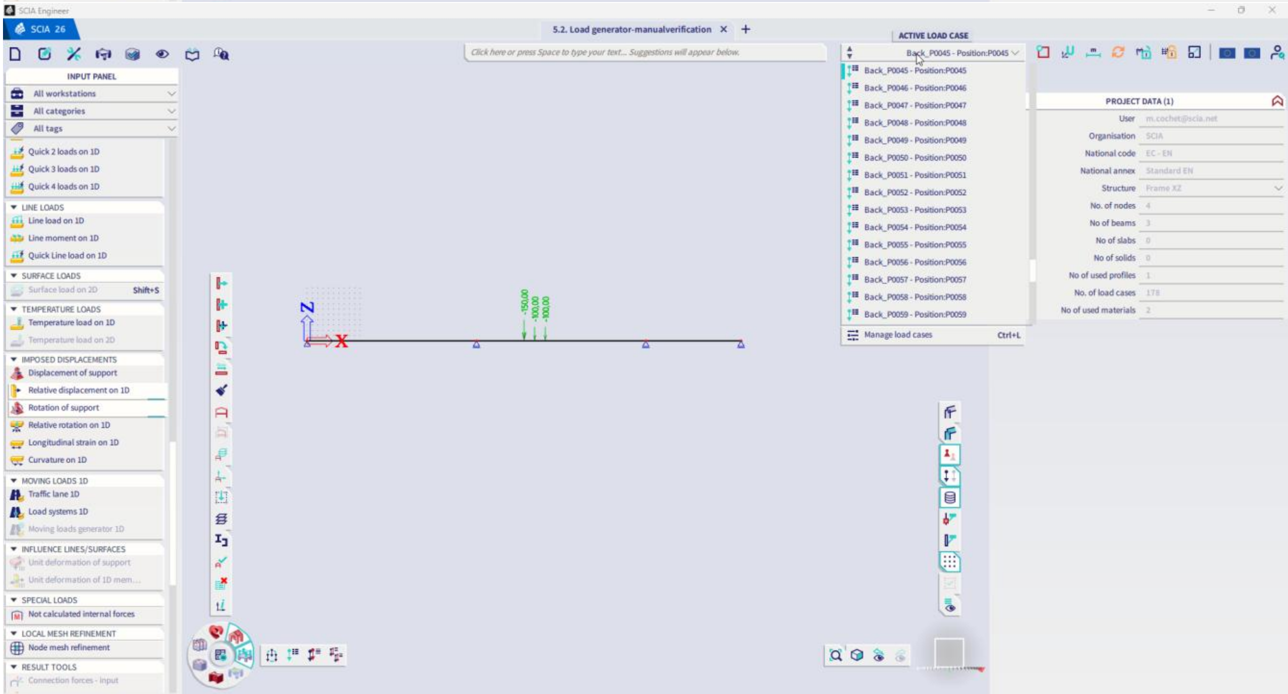




The maximum result of the synchronized and unsynchronized load systems are the same since the max result is found for the load system in exactly the same position. You will notice a small difference in the section results for which the synchronized load system finds a smaller result compared to the unsynchronized load systems.
The minimum result of the synchronized and unsynchronized load system is different. The unsynchronized load systems will find the minimum result for both load systems on a different position.

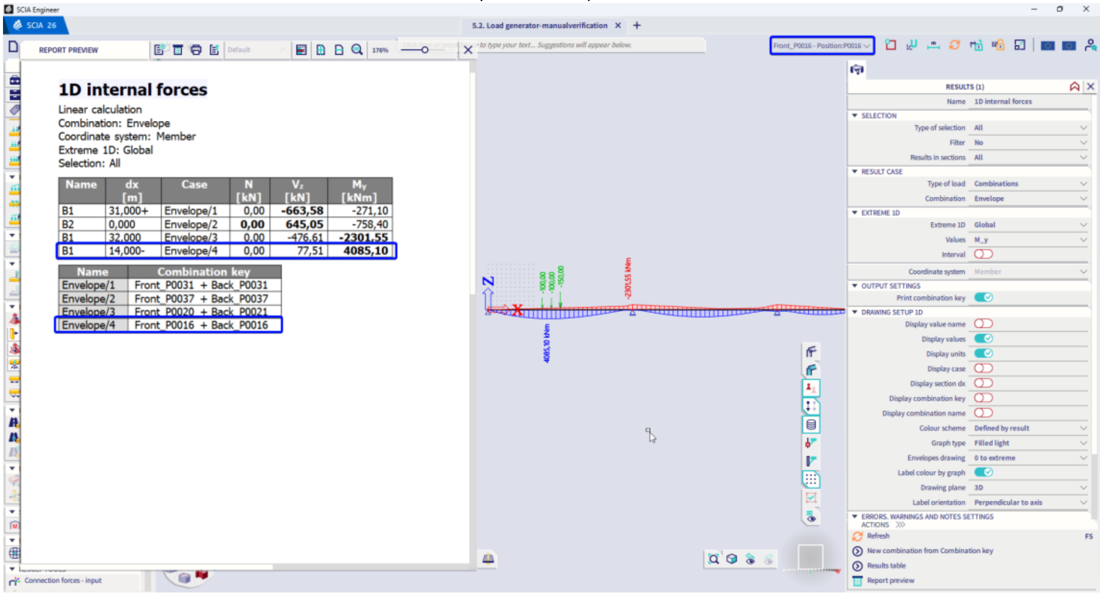
With some test load cases, we can verify which position is causing the most extreme result. To verify this, you can find **Load generator-manualverification.esa**. In this model, the moving load is modelled with separate load cases for each load position of a load system. All load cases of one load system are located in the same load group. For each load system, a separate group is created with the relation set to 'exclusive'. To create these different load cases, an add-on from our SCIA website was used: https://www.scia.net/en/support/addons/train-loads-1d-members



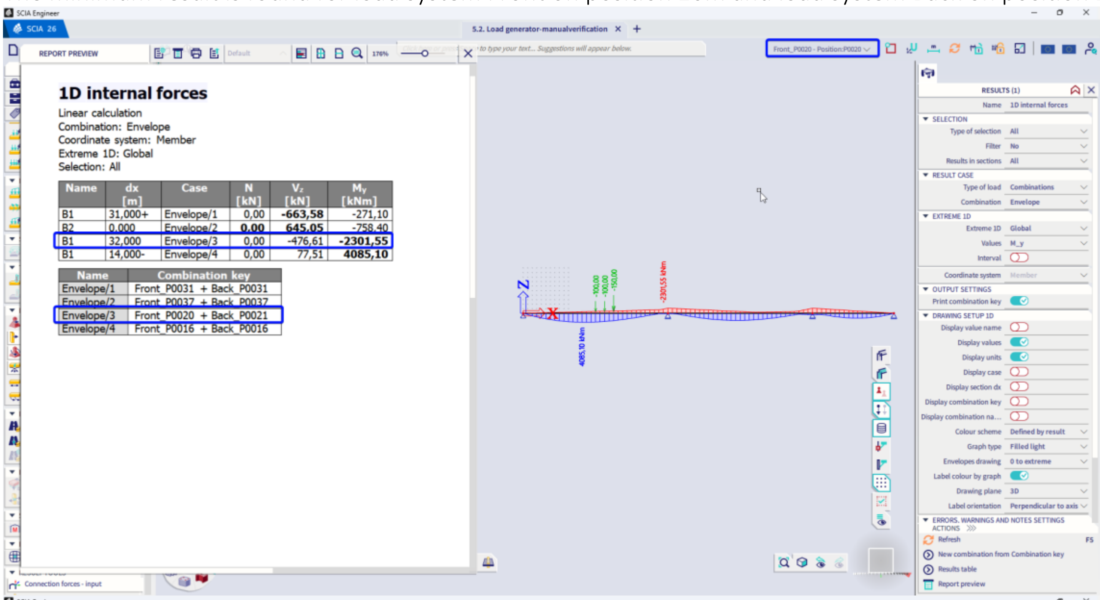


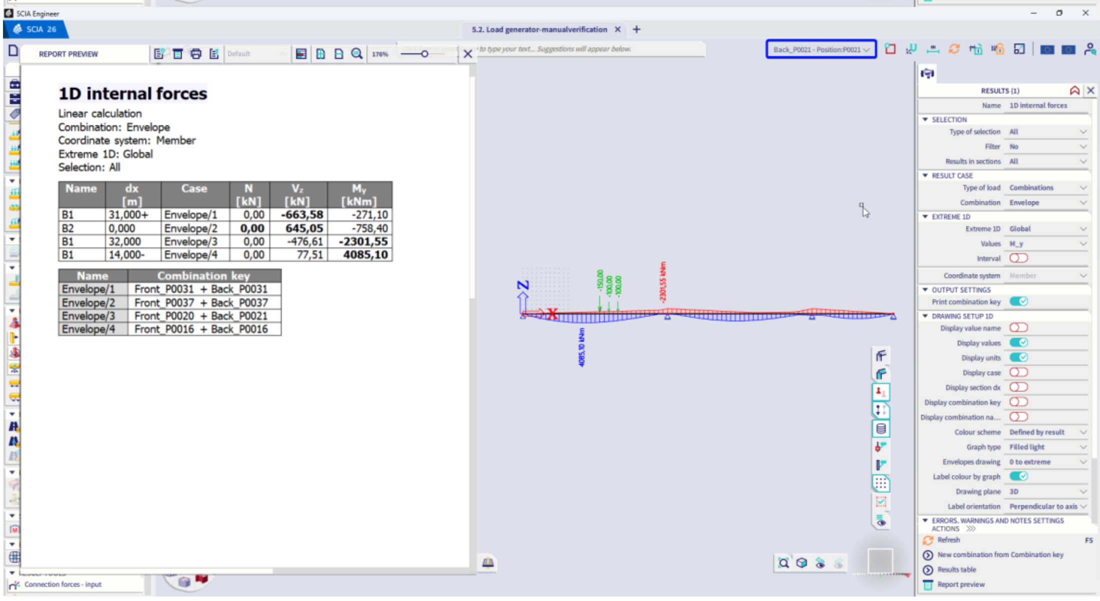
With an envelope combination, we can combine the different positions of the two different load systems and find the most minimum and maximum result:

- The maximum result is found for both load systems at position 16m:



- The minimum result is found for load system Front on position 20m and load system Back on position 21m:

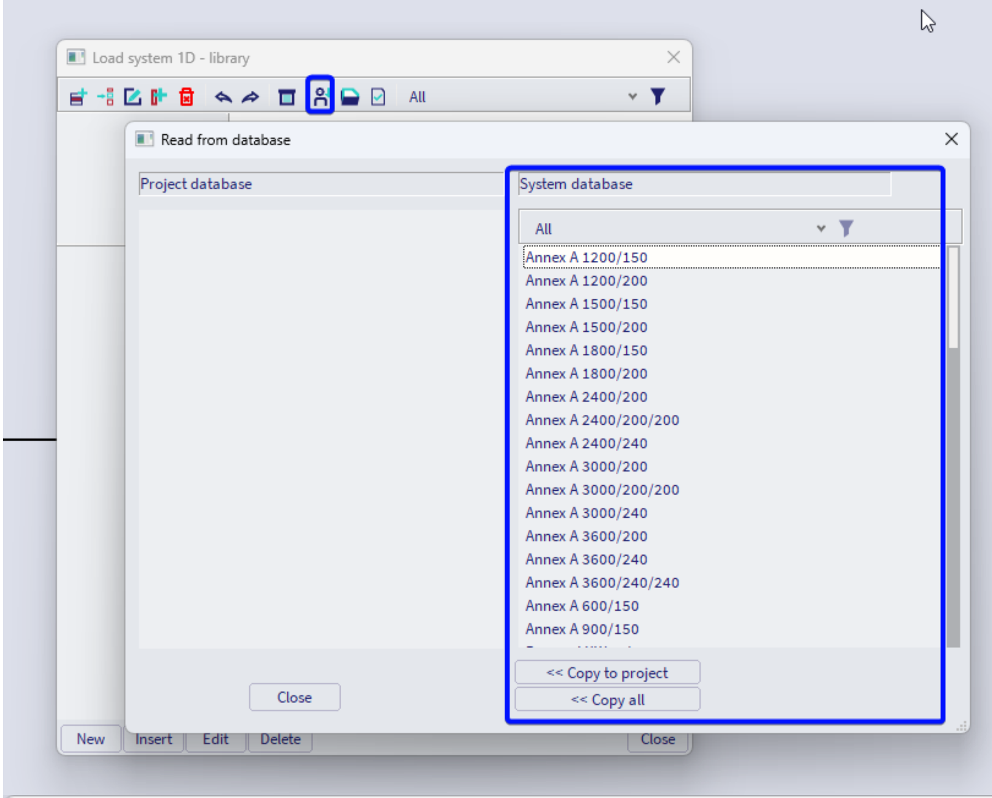


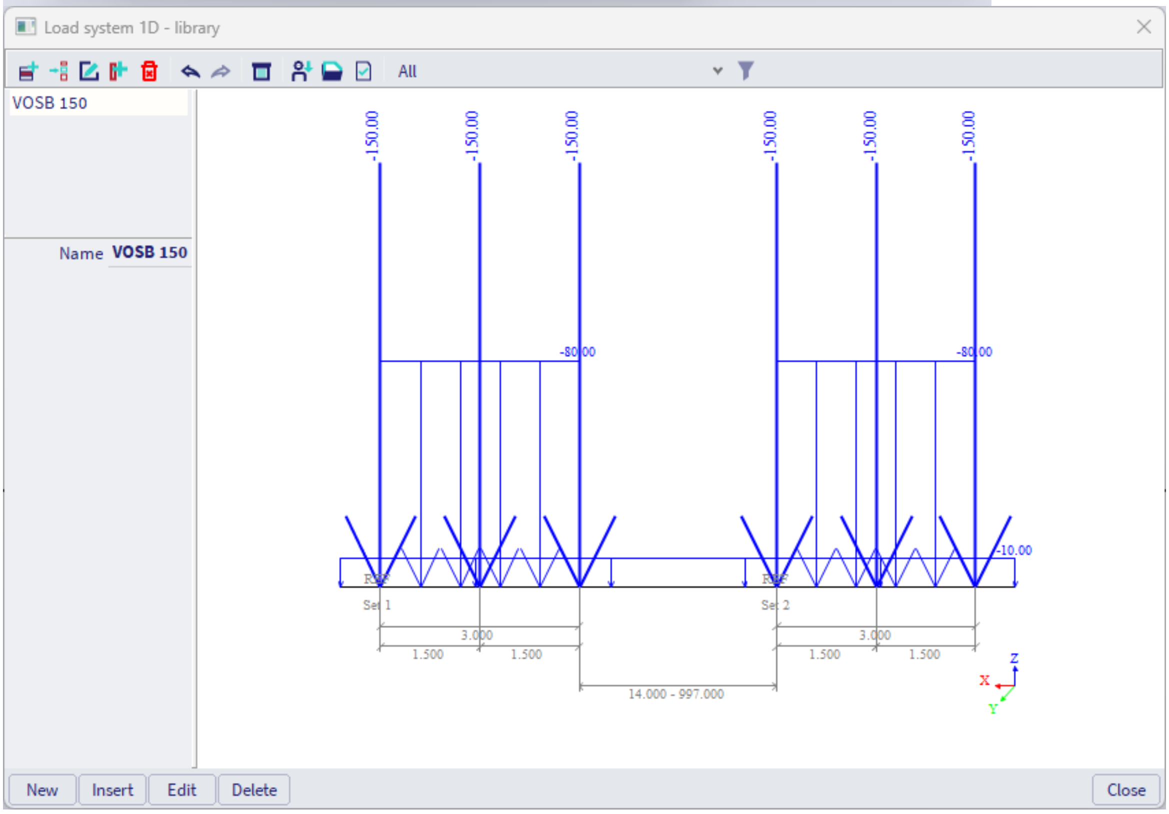


Load system with multiple sets

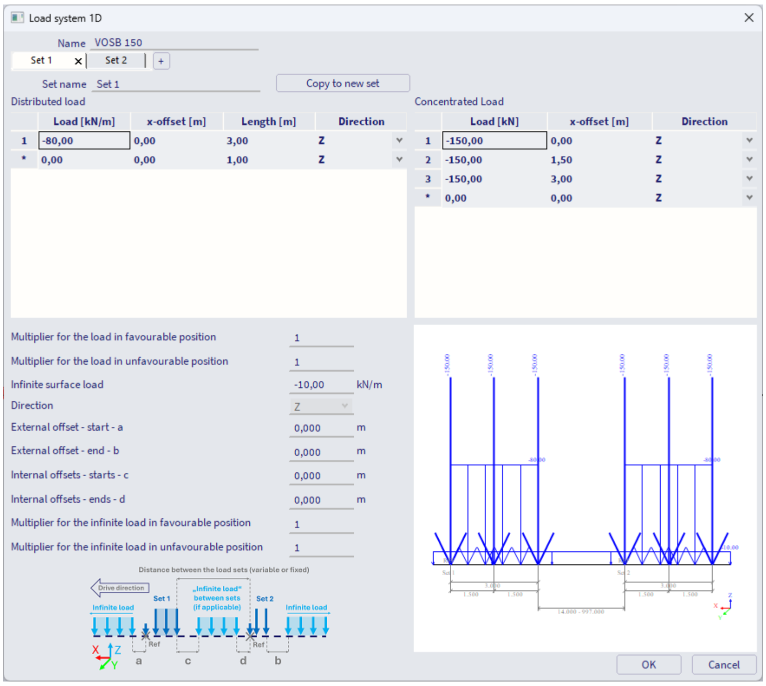
In the previous example, we discussed different load systems moving synchronized or not. It is also possible to create one load system with multiple sets of loads moving together. The distances in between these sets can be fixed or variable. In between these sets, an indefinite line load can be added as well. You can find this example in **load generator-multiplesets.esa**

For these type of load systems, you will find a system database. These are preprogrammed. We will use VOSB 150 from the system database.

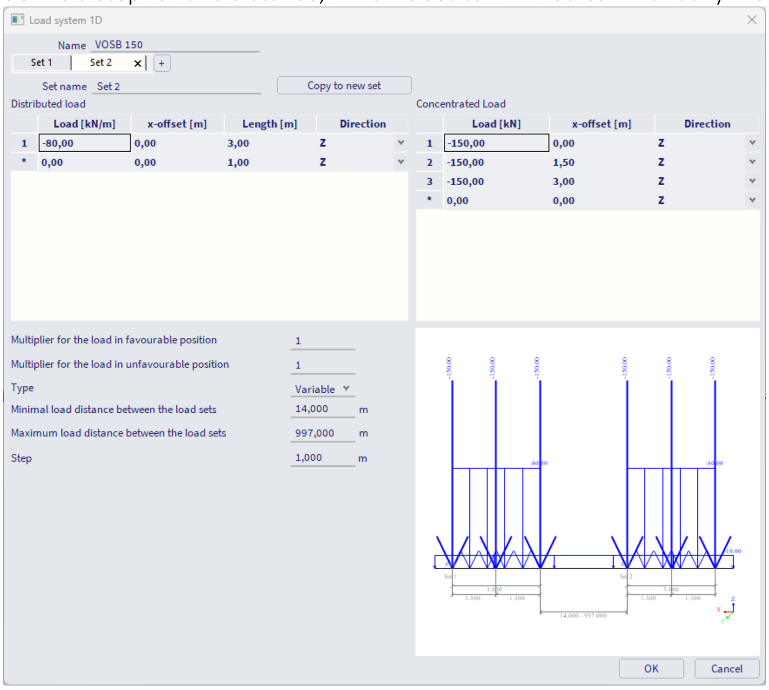




In this load system, two sets are created. Both sets have the exact same nodal loads and line loads defined in the top table. In the property set on the bottom left of this dialog, the infinite surface load is defined as -10kN/m. This infinite load is applied in before, in between and after the sets. You can define an offset creating a distance between the set and the infinite load.

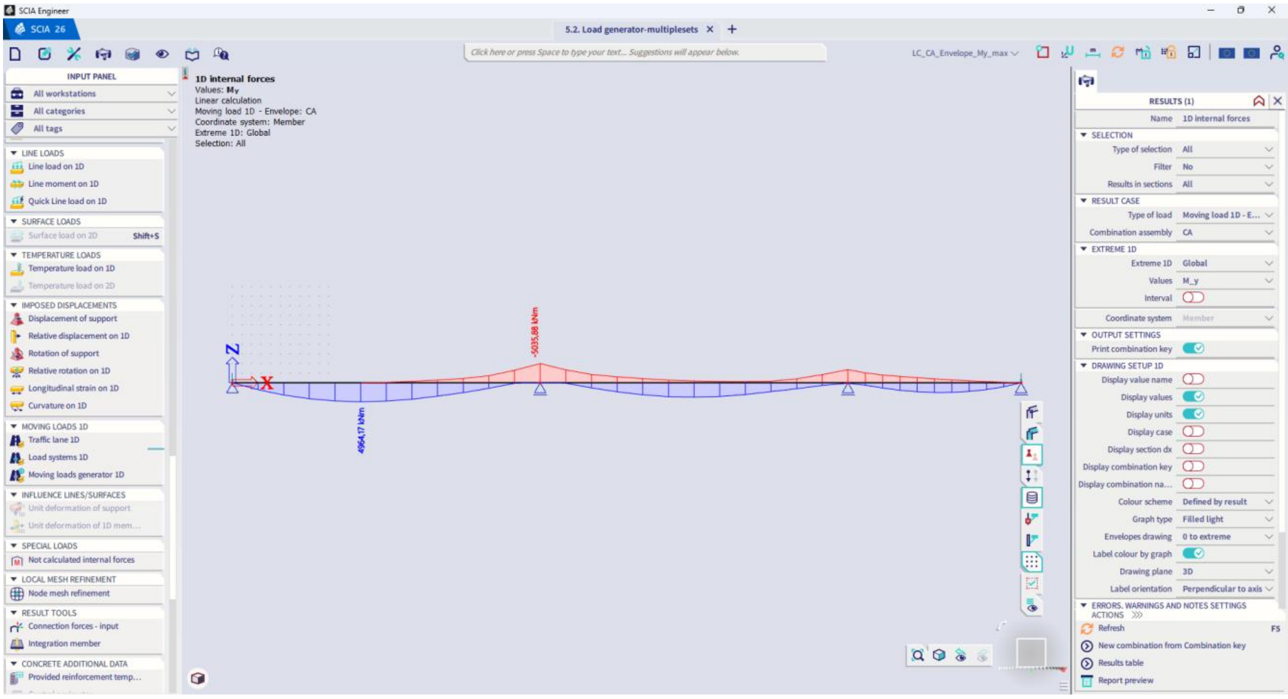


The second set is defined in a new tab. In this tab you also define the distance in between the first set and the second. In this case the distance in between is variable. The minimum is set (by default) to 14m the maximum to 997m. You also define a step for this distance, which is set to 1m. You can manually modify these distances.



For this example, the minimum distance is modified to 25m and the maximum distance is modified to 40m.

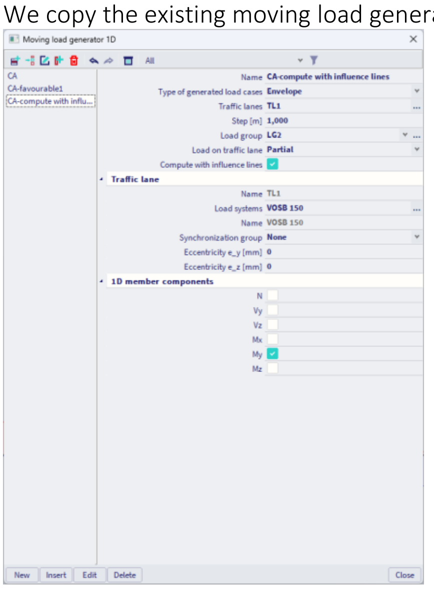
Similarly to previous examples, you can find the envelope results either via a minimum and maximum load case or 'moving load 1D – envelope' which will show you both the minimum and maximum results of the moving load in each section.



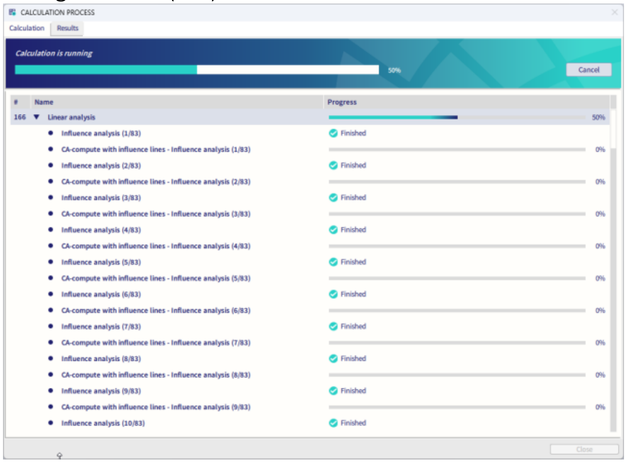
6.3.7. Compute influence lines

In all examples where an envelope load case is generated, we deactivated 'compute with influence lines'.
The option 'compute with influence lines' automatically creates vast number of calculations using influence lines in the background and merges the results into the envelope. If this check box is activated, along the traffic lane in each position (respecting the defined step size, e.g. with 1m span), influence line for each selected 1D internal force result will be calculated. Hence, in each position along the traffic lane, the corresponding linear load cases using the Unit deformation of 1D member will be analysed firstly (in the background). For each such influence line, two linear load cases with extreme values (min and max) for the corresponding 1D internal force will be firstly determined (to find the position of these extremes based on the influence line) and subsequently analysed. The envelope will be provided out of the results of these internally calculated load cases. This solution offers possibility to use the reduction of the load in places when the loads acts in favourable direction (decreasing the corresponding internal force at the considered position) - hence the load multipliers for favourable and unfavourable positions defined within the Load Systems 1D are valid for the "envelope" solution only when this check box is activated (when the influence lines are used).

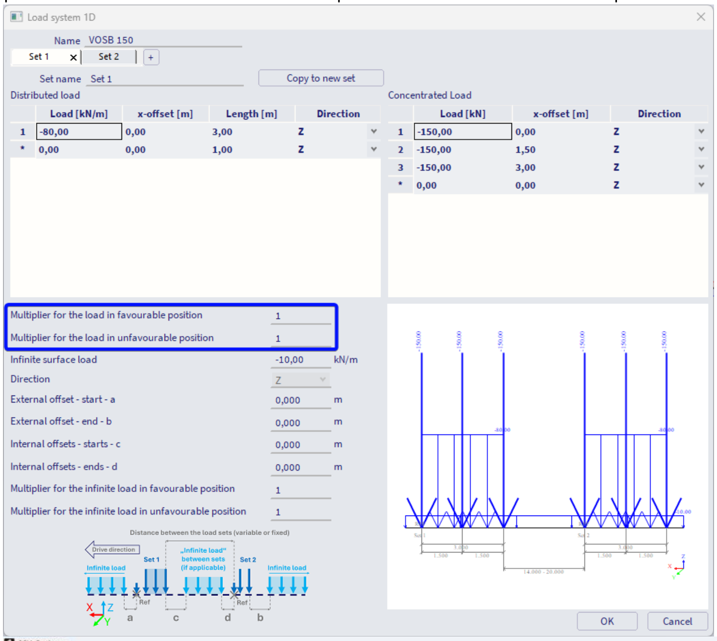
We copy the existing moving load generator and activate this option.

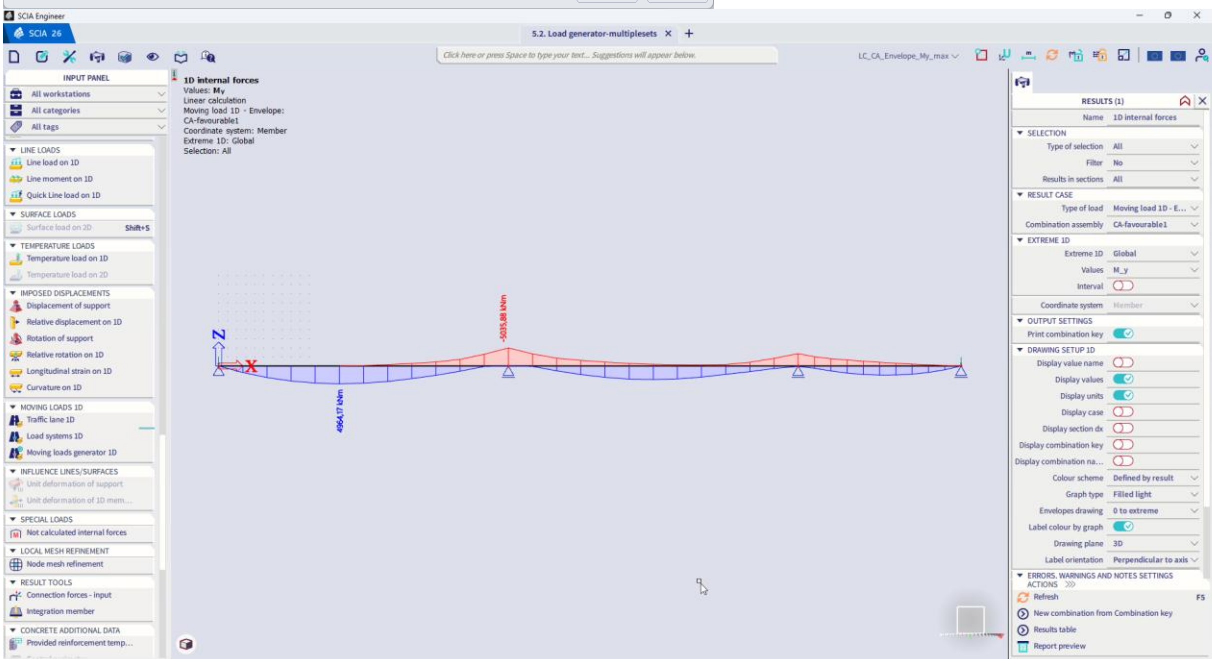


Now during the linear analysis, you will find that the influence analysis is performed at every step defined in the moving load generator (1m).

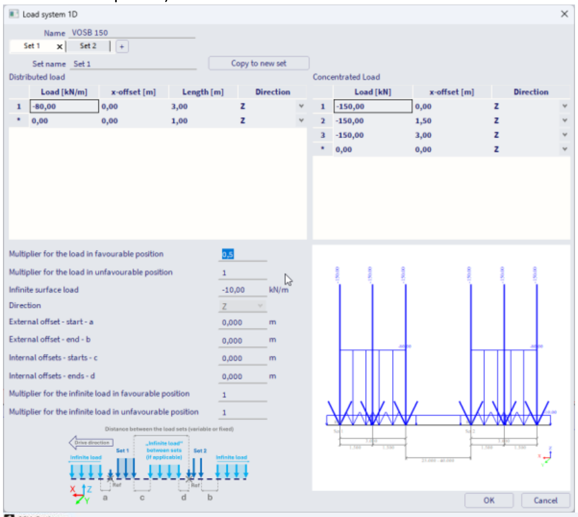


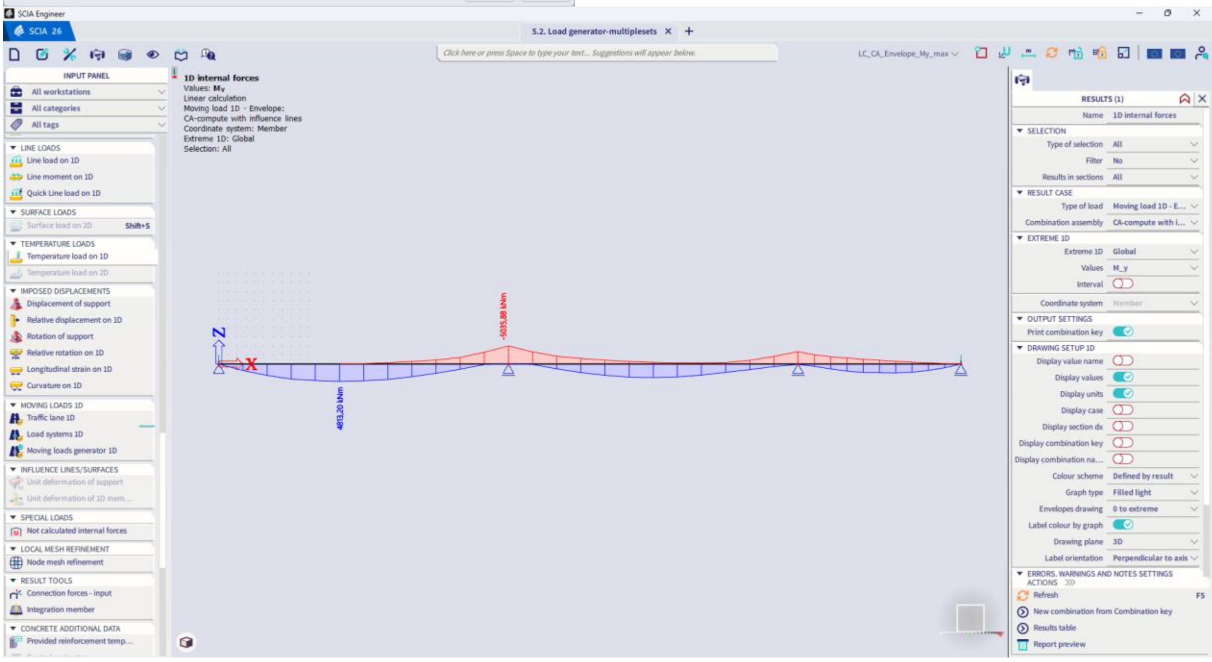
The result of this new envelope will be exactly the same as before since the multiplier for unfavourable an favourable positions are both set to 1. Keep in mind that these multipliers is defined for each set. In this case I only modify Set 1.





If we reduce the multiplier for favourable positions of ‘set 1’ to 0.5, we will notice some changes in the maximum results (due to the minimum distance in between de sets, the maximum is found when both sets are positioned on different spans).



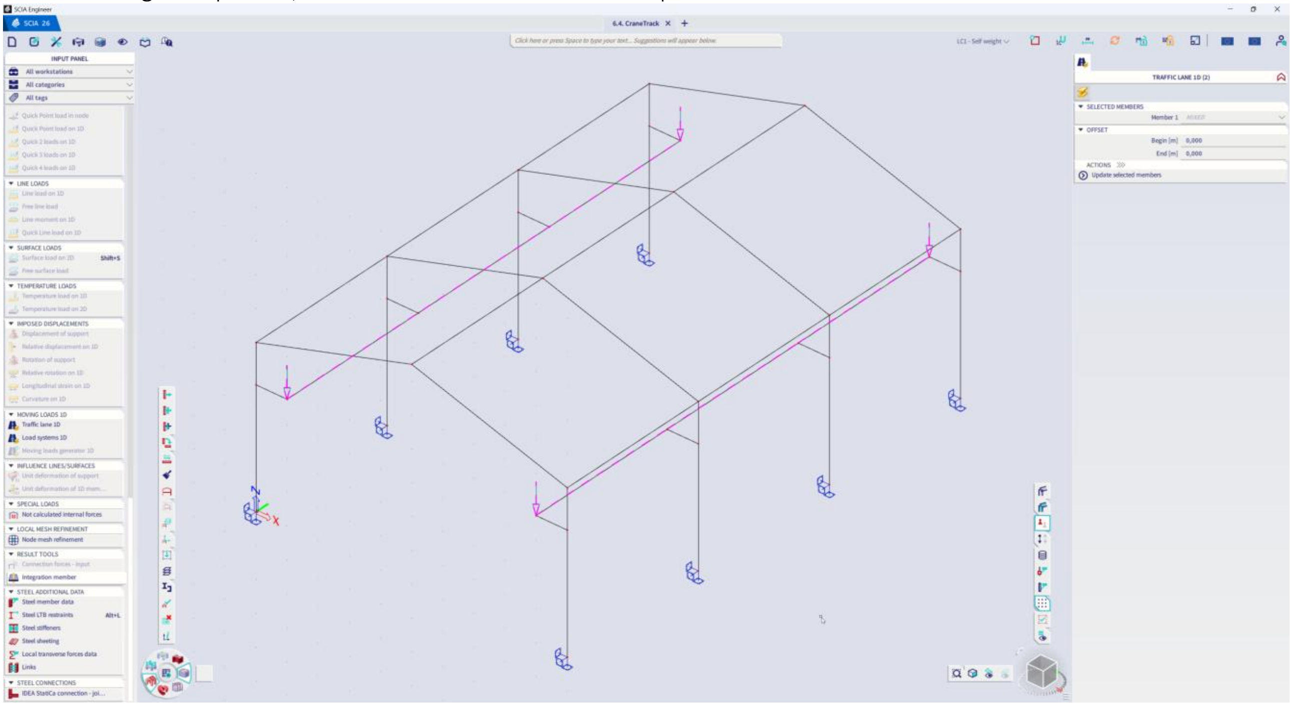


This multiplier of unfavourable and favourable positions will have no influence on the generated envelope load case if ‘compute with influence lines’ is toggled off.

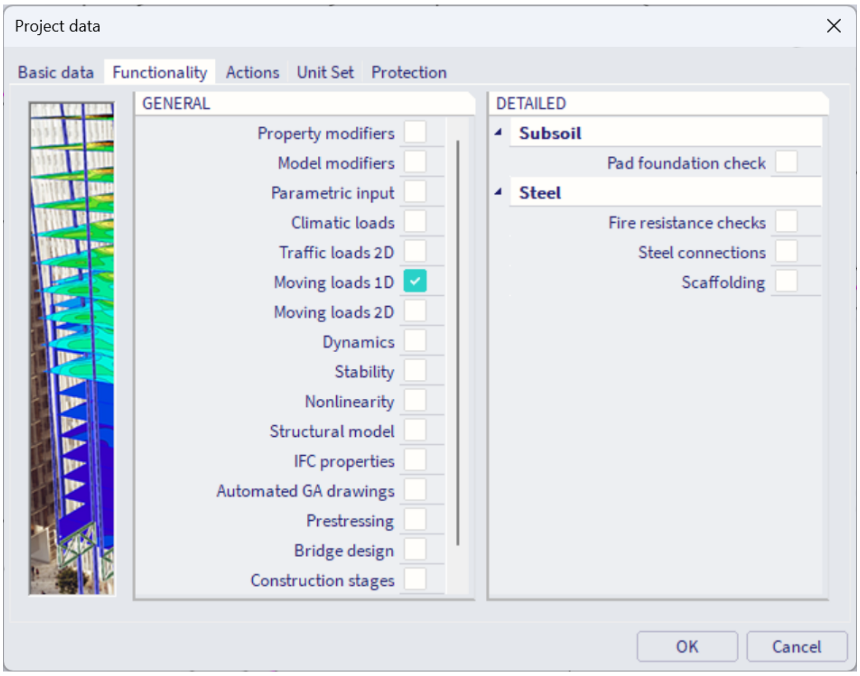
6.4. Crane track

This last example shows how the position of a load system on the structure can be adapted through various load systems. That way a crane track, which moves from left to right in a hall, can be modelled.
You can find this example in **Crane Track.esa**.

After entering a simple hall, the traffic lane is defined. Two separate lanes are created on each side of the hall.



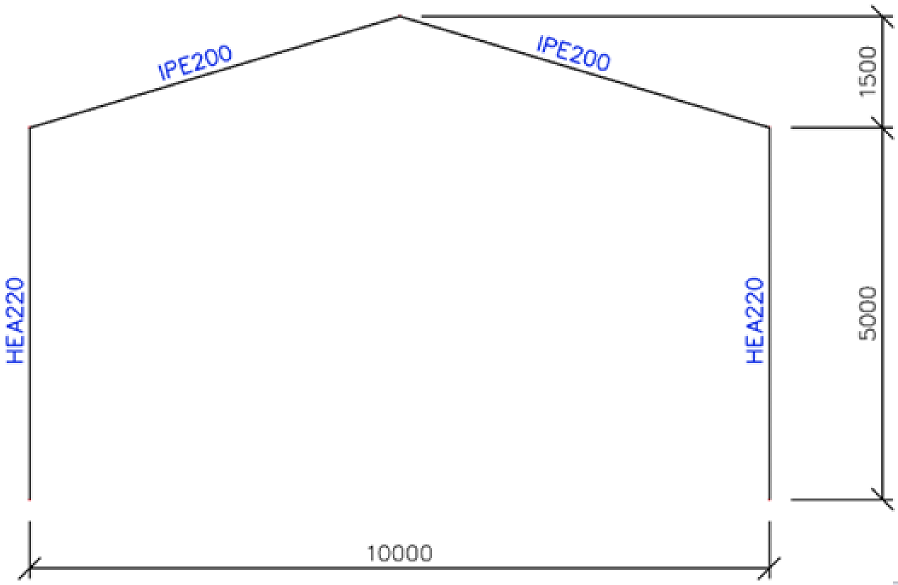
These traffic lanes represent both rails of the crane track. We will use multiple load systems with various factors to show that the crane track can also move in the transversal direction, perpendicular on the rails.
In the next step, the enveloping load cases can be generated.

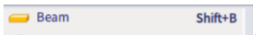


6.4.1. The construction

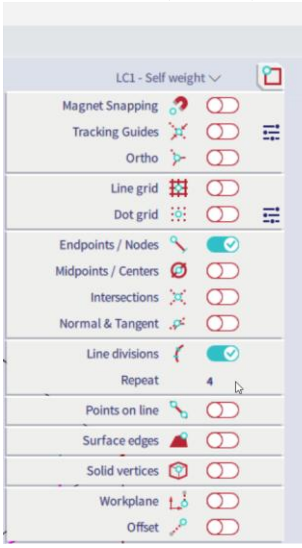
The first portal of the hall can be entered through 'catalogue blocks': 

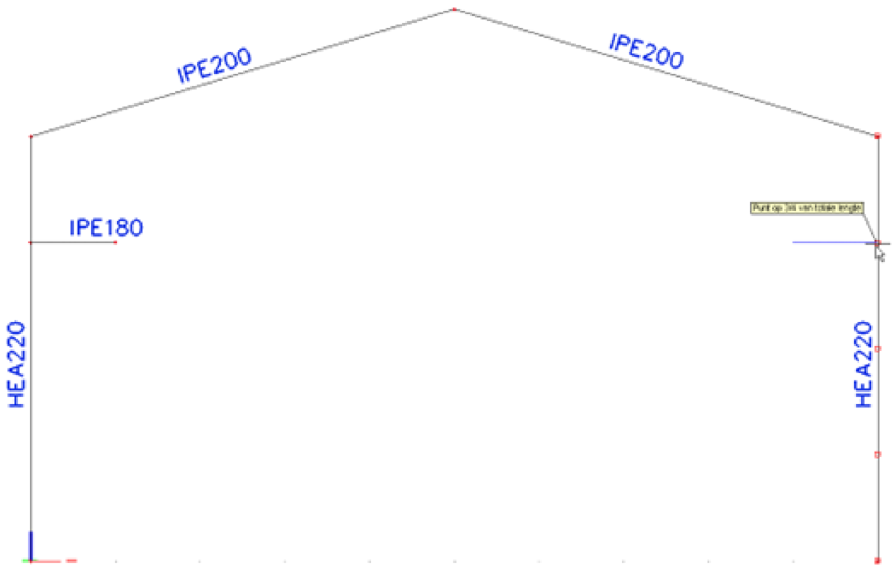
.



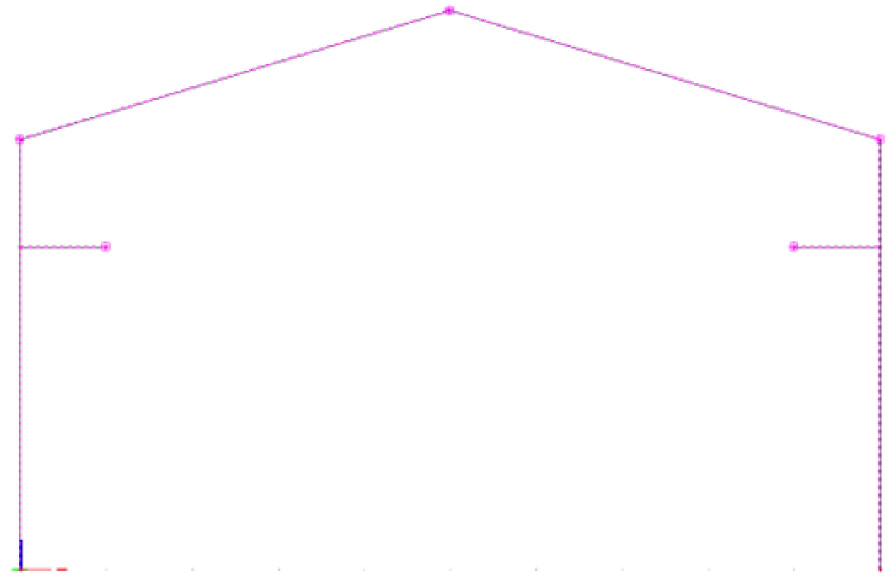
Subsequently the haunch beams on which the rail support, can be entered through 

.
The beams have a length of 1m, type IPE 180 and move across ¾ of the length of the column.
To find this Snap Point you can use the cursor Snap setting 'line divisions'.

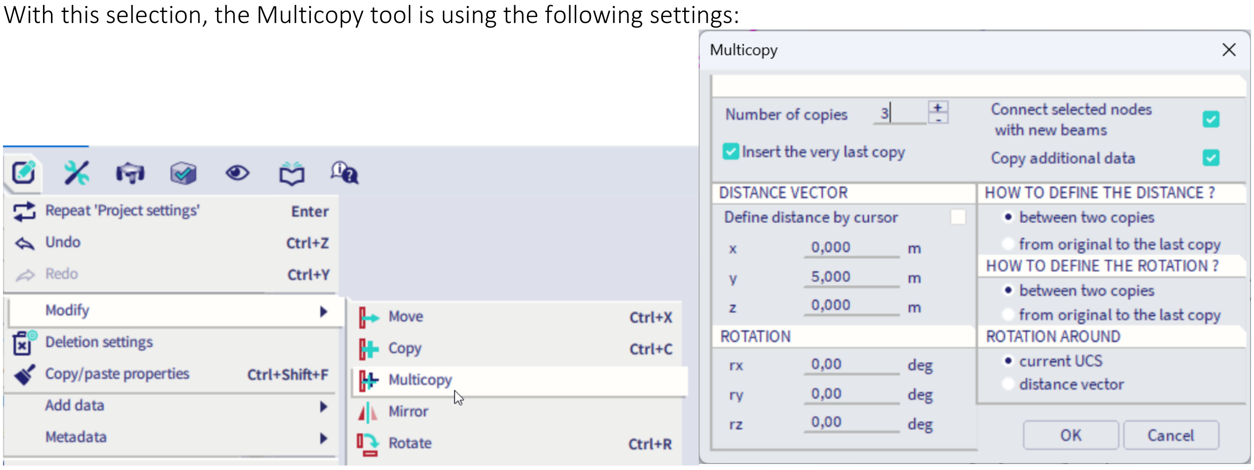




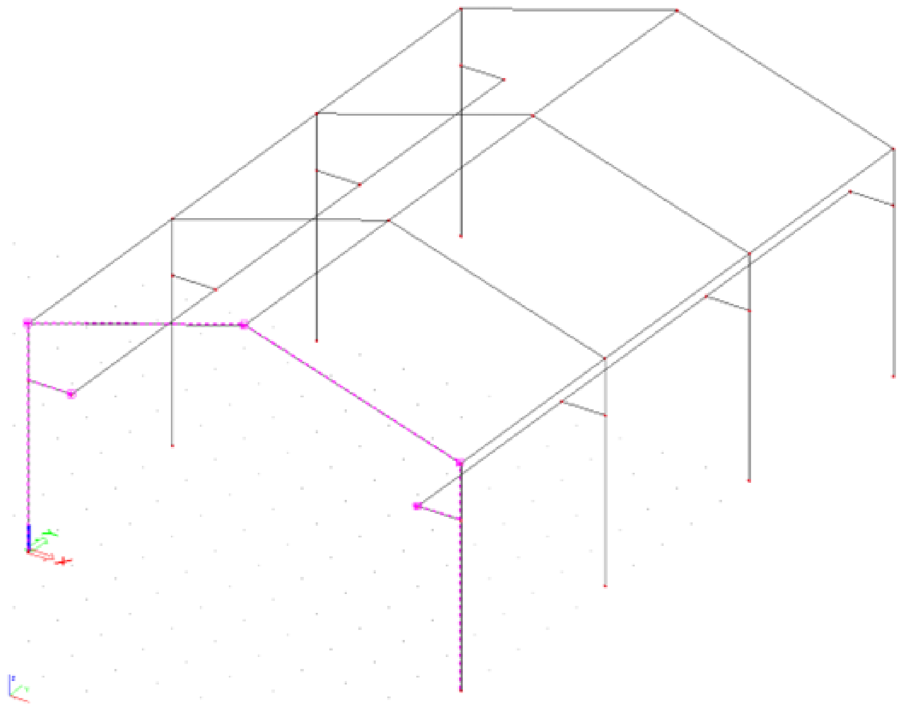
To get the full hall, the multi copy tool is used. All members, the three nodes of the roof and two nodes of the IPE 180 beams are selected.

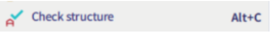


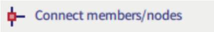
With this selection, the Multicopy tool is using the following settings:



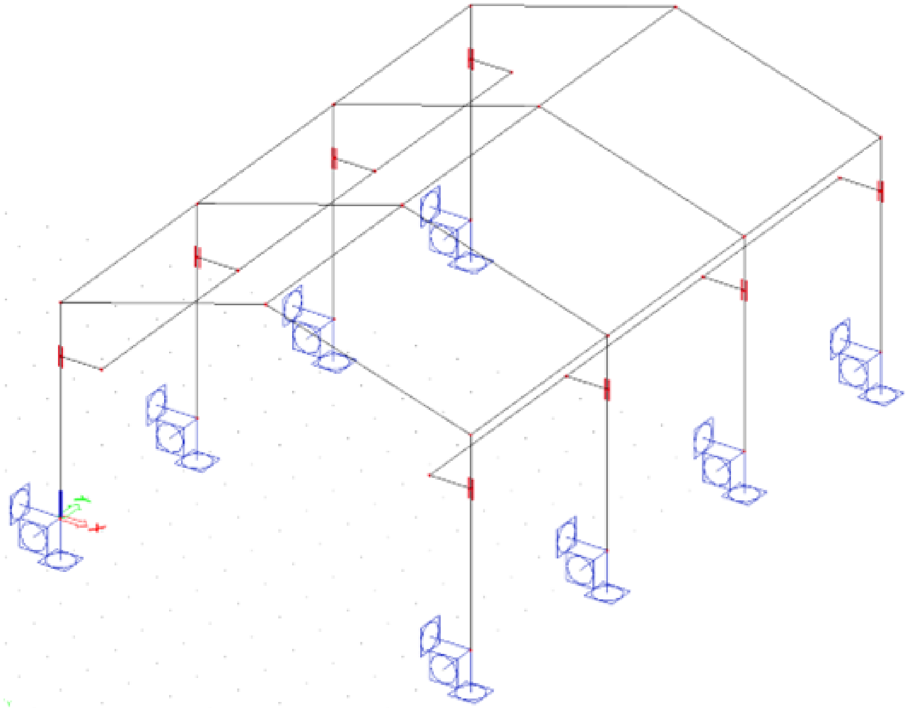
As a profile type for the connection beam between the various trusses, IPE 180 is chosen. Than we have the following structure:



The geometry input is ended by entering the rigid supports to the column bases and by executing the 

and 

to connect the various members.

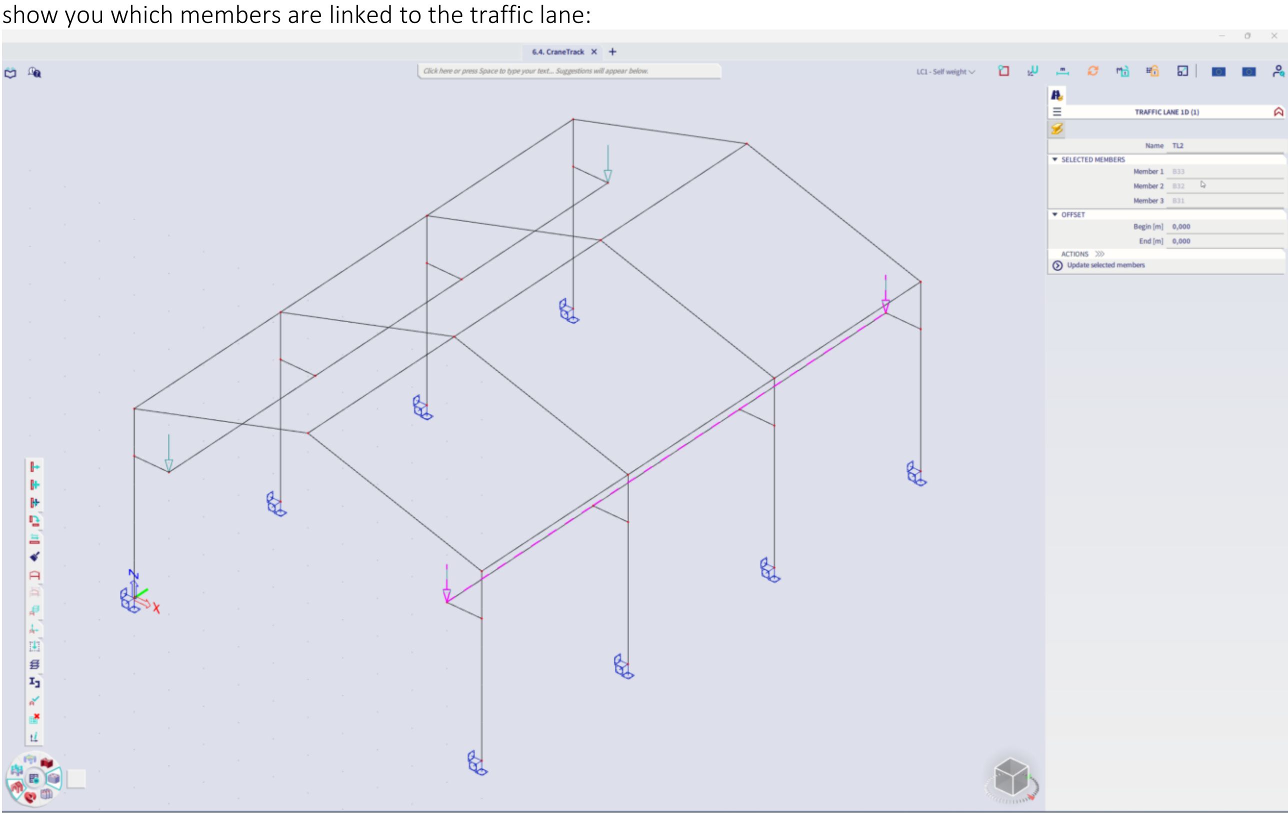


To be able to calculate the structure, one load case is created; the Self Weight.

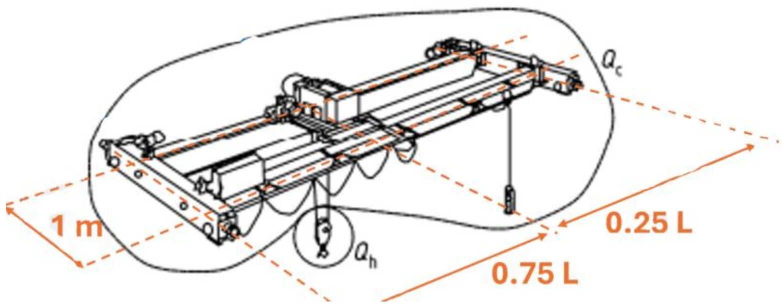
6.4.2. Traffic lanes

After entering the construction, the traffic lanes can be created. Run the command 

. To create a lane, select the members on one side of the hall. Repeat the same steps to create a second lane. The property panel will show you which members are linked to the traffic lane:



6.4.3. Load systems



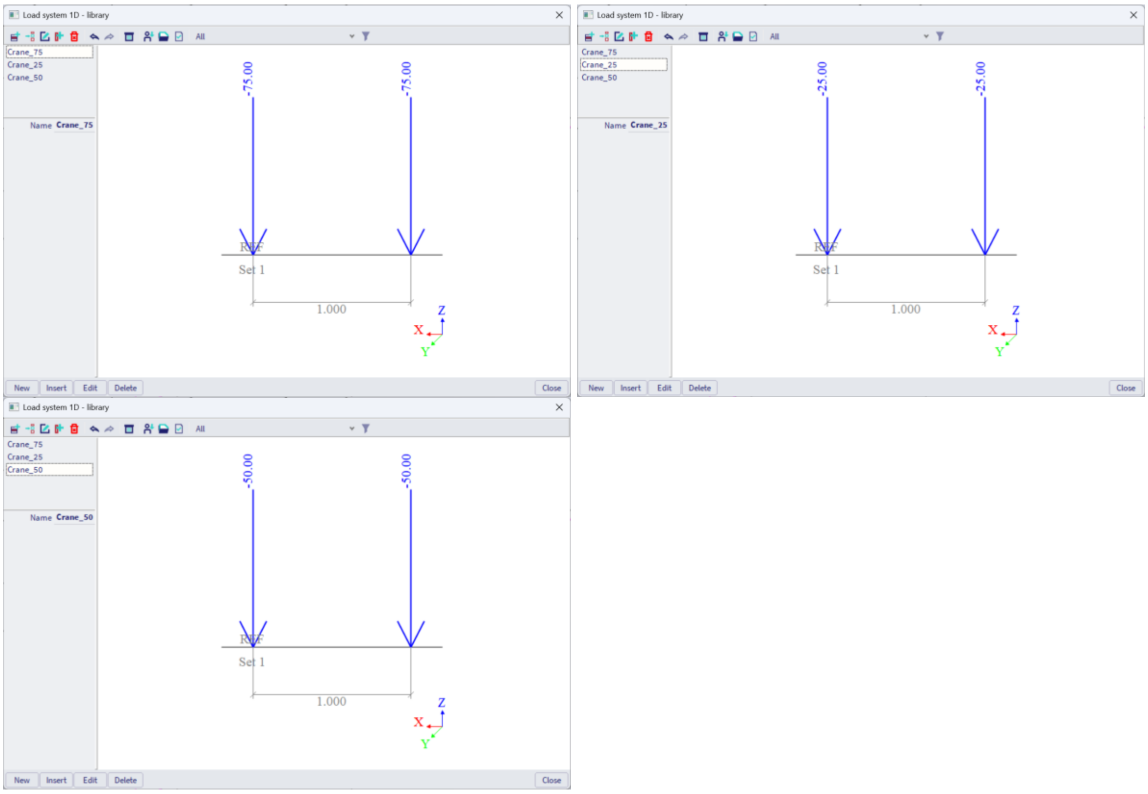
We will create different load systems to represent the different positions of the hook load Qh (in the middle, on the left, on the right). We assume the following:

- When the hook load is positioned in the middle, the weight is distributed evenly on both sides.
- When the hook load is positioned on the left or right, one side gets 25% of the weight, the other 75%
- The weight of the crane itself is neglected in this example.

Therefore we create three different load systems: **Crane_75, Crane_25, Crane_50**. 75, 25 and 50 are the values of the forces in these load systems in kN respectively.

The Crane track has two supporting nodes on each side with a distance of 1m. The weight is always evenly distributed on these two supports.

Use the command Load systems 1D to create three load systems. Each system consists of 1 set with two nodal loads. They both have the same value.



6.4.4. Moving load generator

With the generator we will combine the different load systems and let them run over the traffic lanes. By synchronizing the load systems we will make sure that both sides move along the traffic lane simultaneously. We create a new generator for each different position of the hook load.

Run the command Moving loads generator 1D to create three different generators to generate the enveloping load cases. For each generator we assign two traffic lanes, define the step as 0.5m and assume that the load system is always completely on the traffic lane. For each traffic lane a load system is assigned, a synchronisation group is selected to keep both load systems on different lanes synchronized. For this example we will generate the envelop for 1D component My and support component Rz.

CA_crane_left:

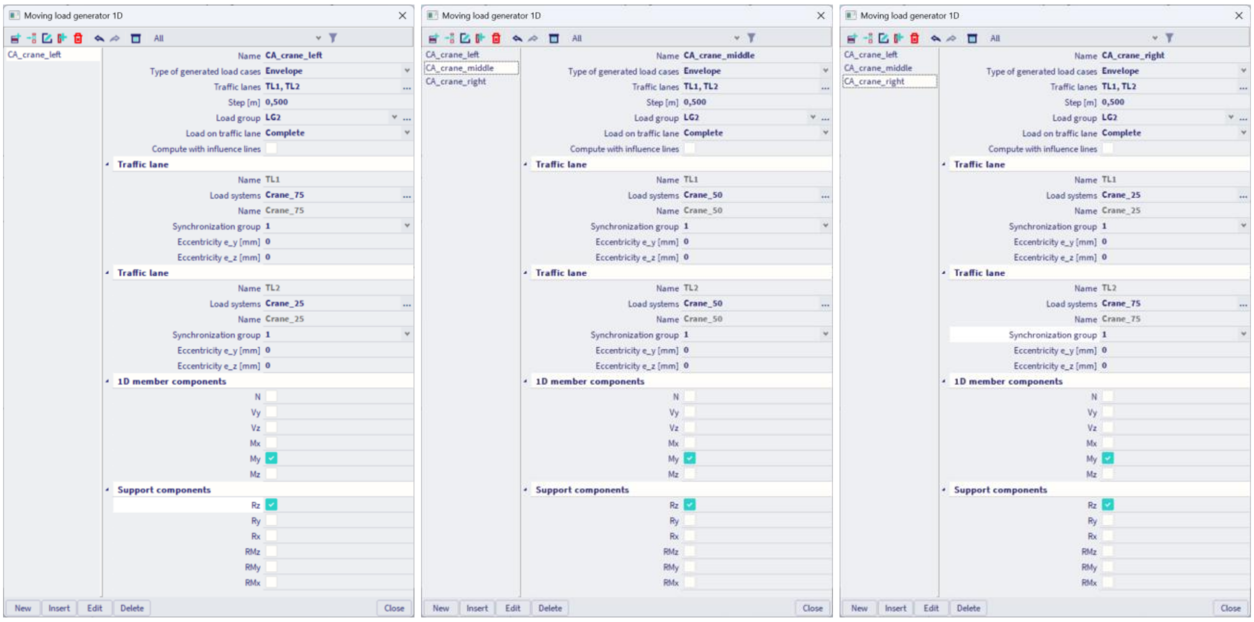
- Load system Crane_75 is moving on TL1
- Load system Crane_25 is moving on TL2

CA_crane_middle:

- Load system Crane_50 is moving on TL1
- Load system Crane_50 is moving on TL2

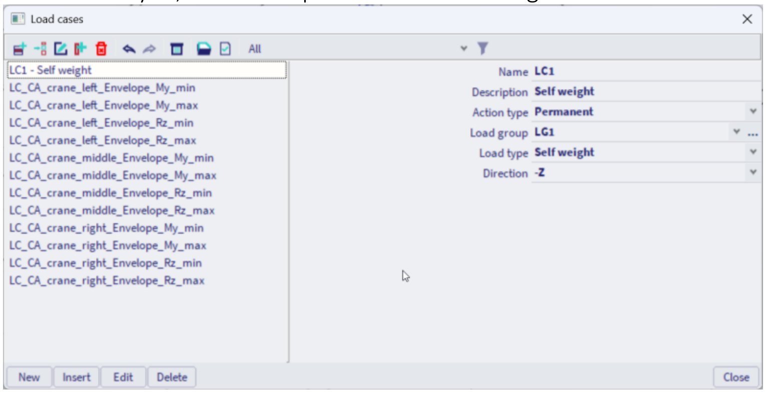
CA_crane_right:

- Load system Crane_25 is moving on TL1
- Load system Crane_75 is moving on TL2



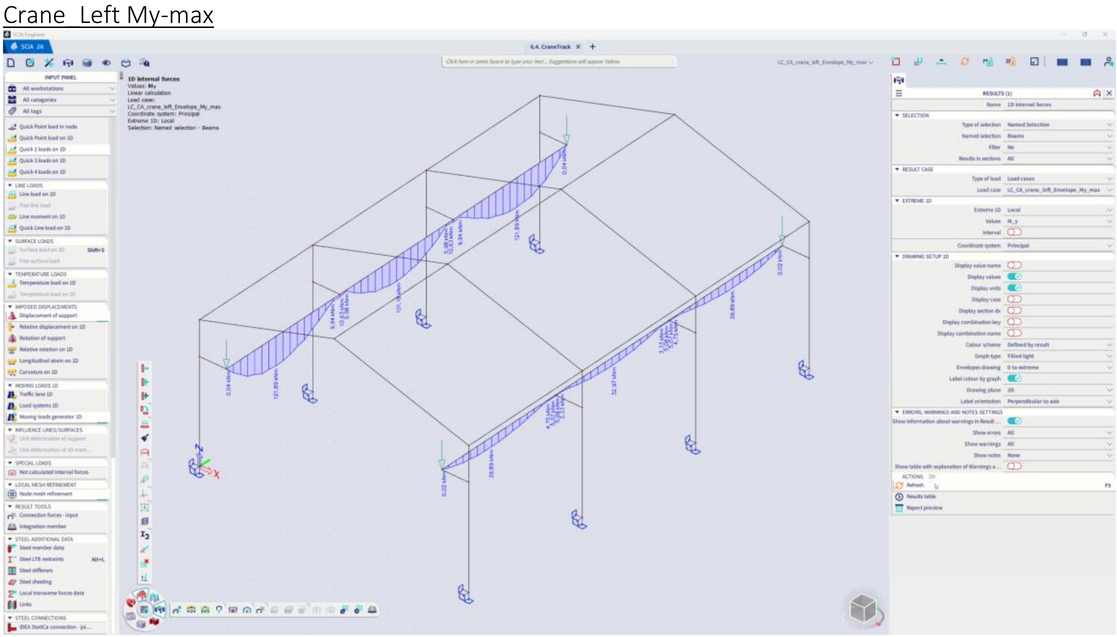
6.4.5. Envelope load cases

Run the analysis, the envelope load cases will be generated.

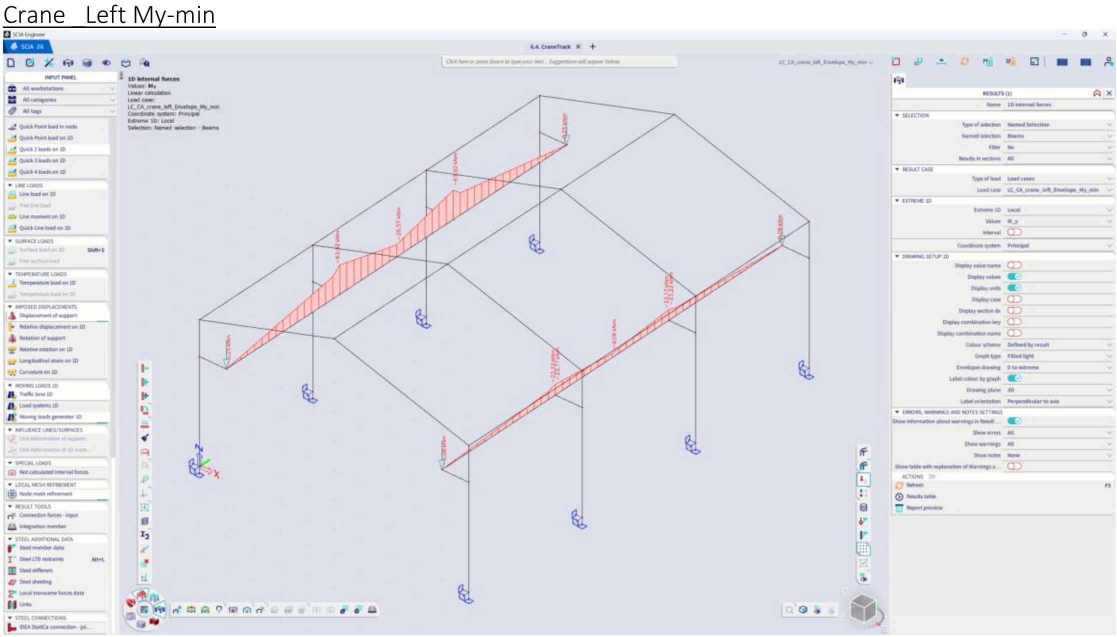


For each of these load cases, the results can be verified separately. These will show you the minimum or maximum value found in each section caused by the moving load. An envelope is created for both My and Rz since only those components were selected in the generator.

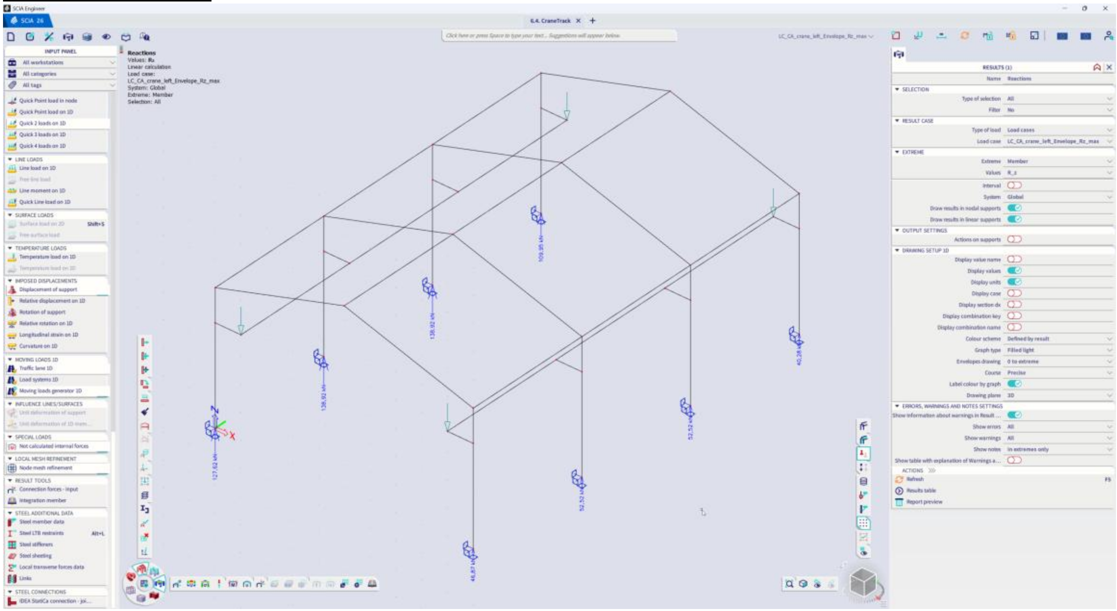
<u>Crane _Left My-max</u>



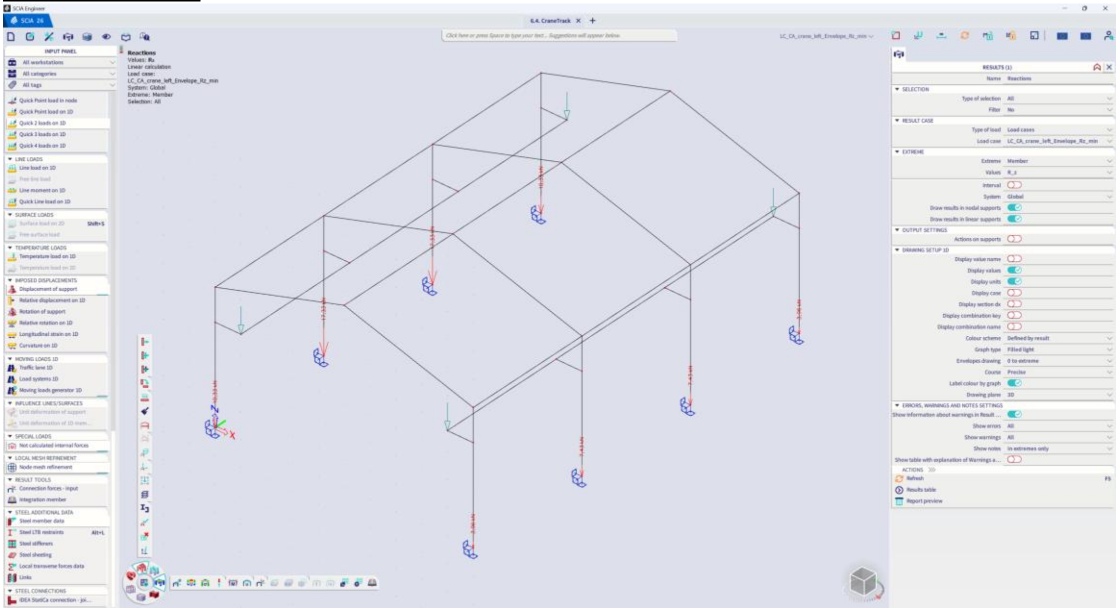
<u>Crane _ Left My-min</u>



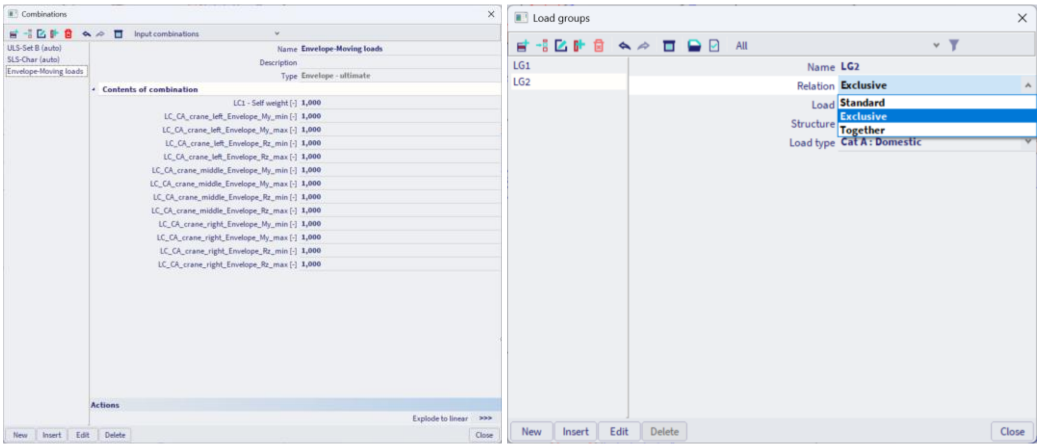
Crane_Left Rz-max:



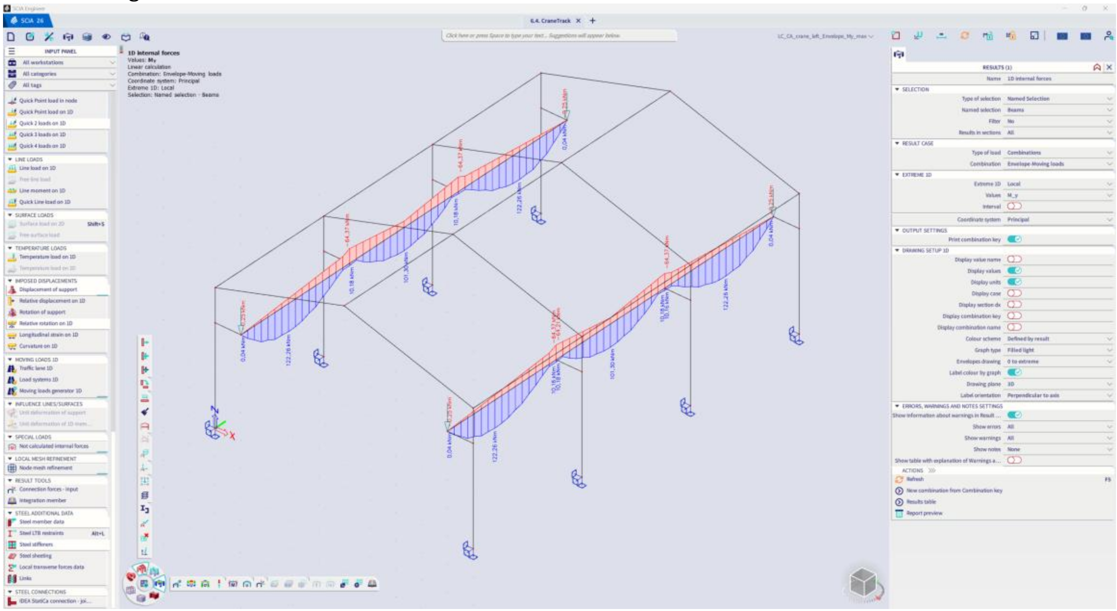
Crane_Left Rz-min



These results can be used in a combination as well. Allowing you to further process these results in combination with other load cases. Automatically generated linear load combinations can also consider these envelopes. Keep in mind that the load group of the envelope load cases is relevant. In this example the different envelopes give us information about different hook load positions. Therefore LG2 has the relation 'exclusive' meaning that different envelope load cases cannot be combined with each other.

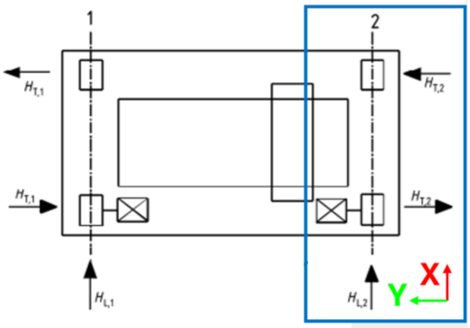


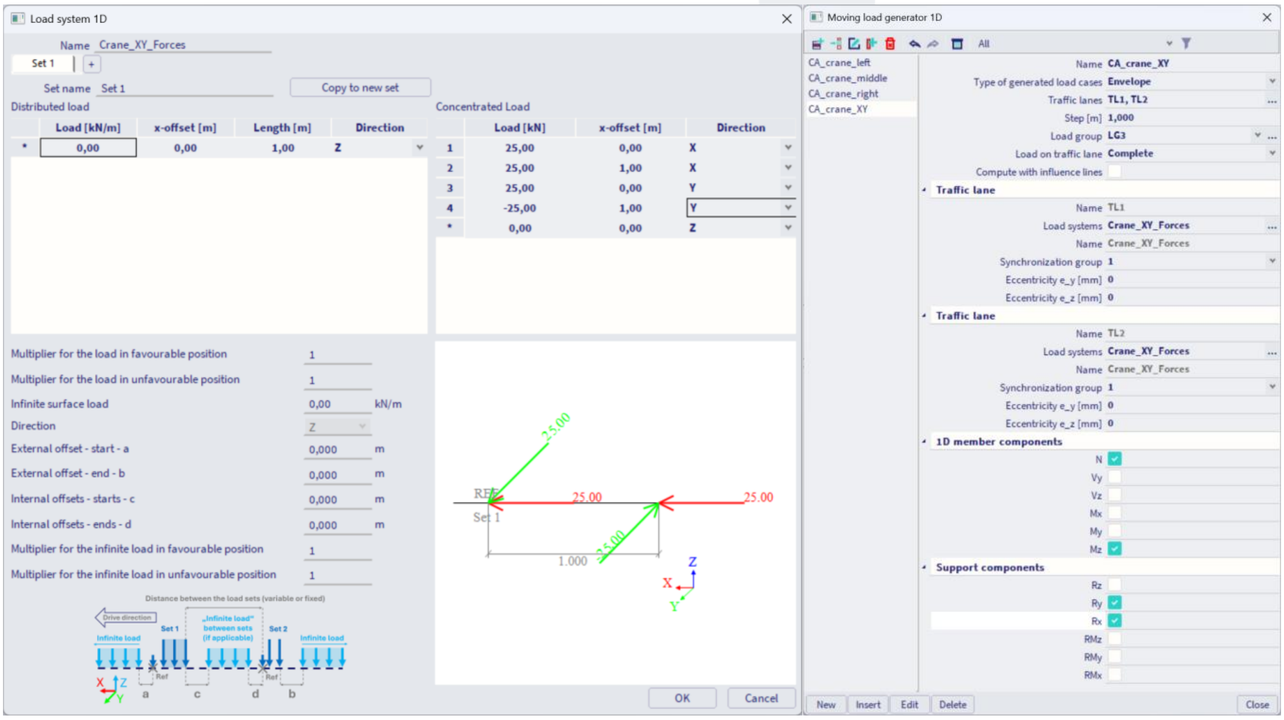
This is the result of the envelope combination. The extreme results are slightly different due to the combination with the self-weight of the structure.



6.4.6. Horizontal forces

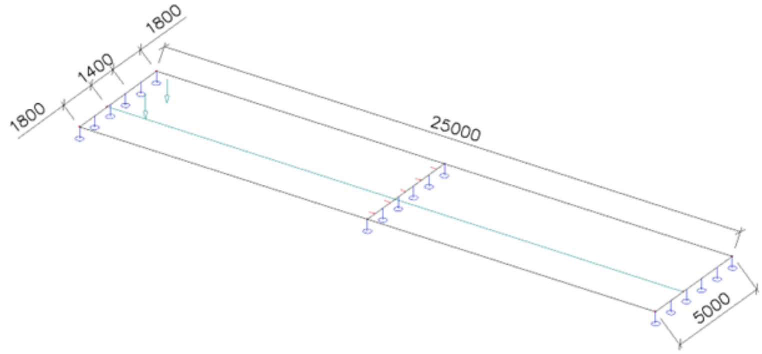
Similarly we can define Horizontal forces imposed by the crane within the load system. Horizontal loads along the X and Y axes. We assume the following: Ht,1 and Ht,2 are equal. Therefor we can create one load system and use the same system on both traffic lanes simultaneously. If these are different, you would use unique load systems on both traffic lanes.



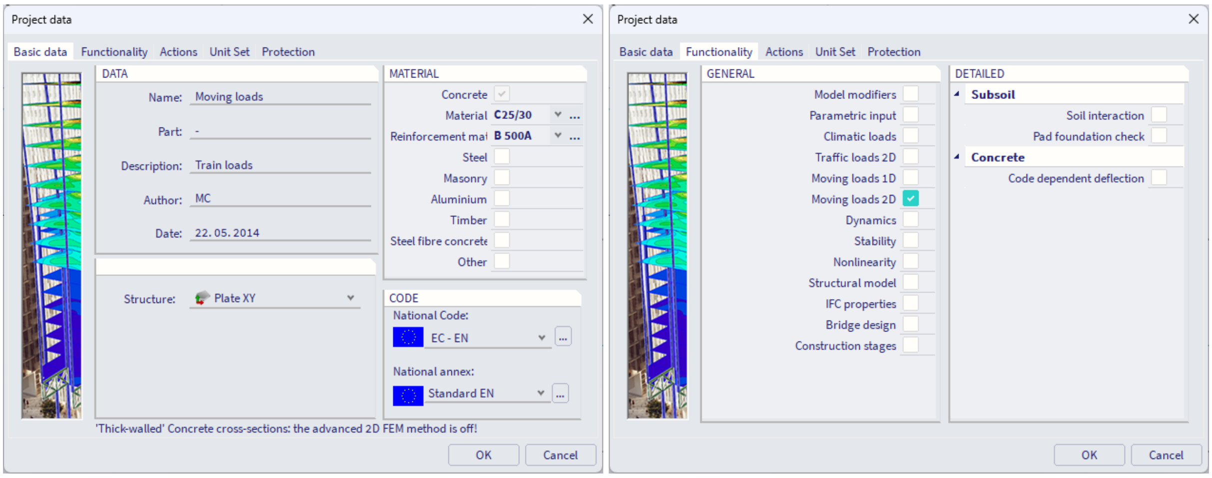


7. Moving loads 2D

In this project **Moving loads 2D.esa** a bridge deck is modelled as a concrete plate on three supports. Analogously to the previous projects, a traffic lane is defined on the bridge deck together with a load case of the type 'influence lines' so the influence lines can be determined. Other than that, in this project a load system is defined to simulate both rails of a train track. The enveloping load cases are generated.

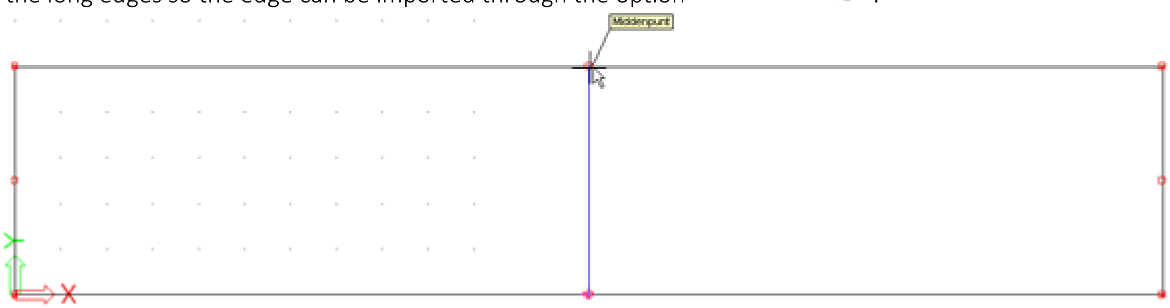


7.1. The construction

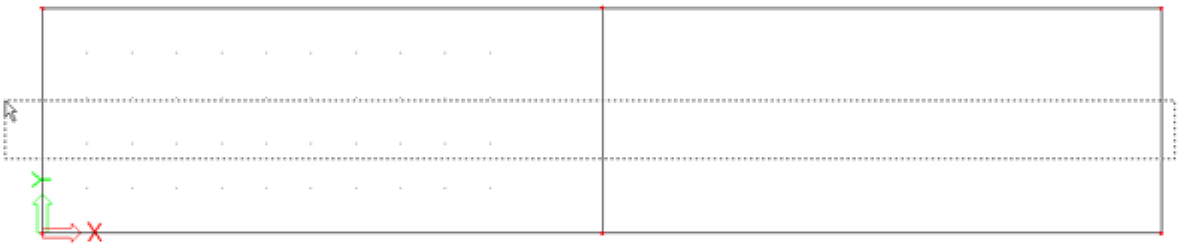


The bridge deck can be entered as Plate with thickness **500mm**.
The length of the bridge deck is **25m**, the width **5m**.

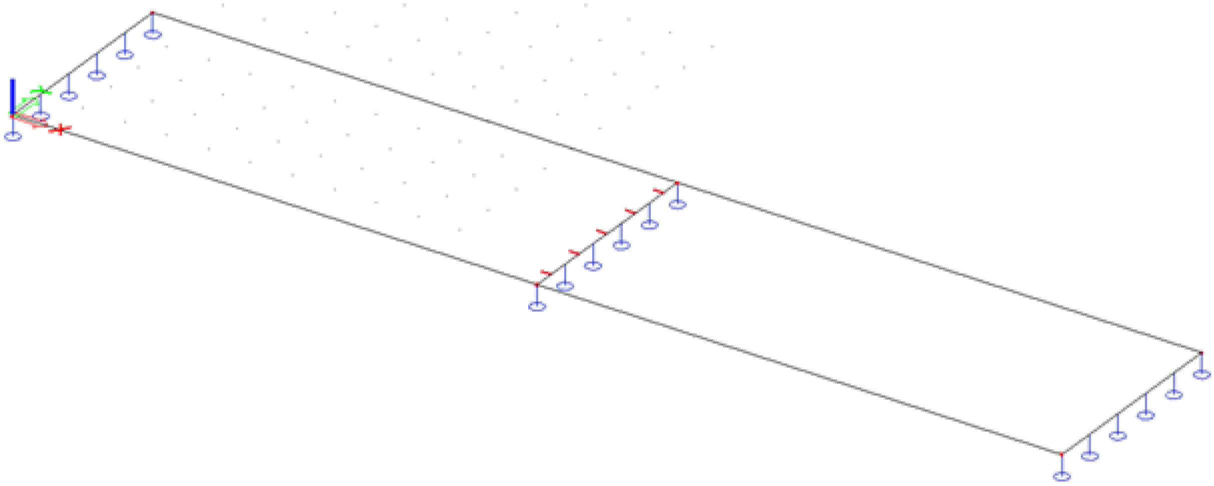
In the middle of the bridge deck an internal edge is created. Using the **Cursor Snap Settings** you can snap on midpoints of the long edges so the edge can be imported through the option Internal edge .



Using Line support on 2D edge , the translation in the Z-direction can be prevented for the three short edges. They can be simply selected by drawing a rectangle from right to left:



Then we have the following structure:



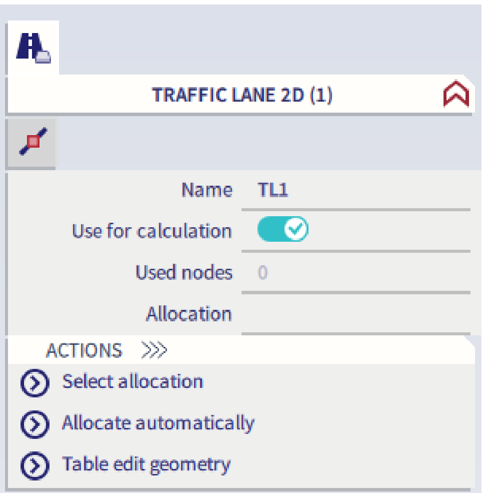
To be able to calculate the construction, one load case is made; the Self Weight.

7.1.1. Traffic lane

After entering the construction, the traffic lane can be created.
The load will move over two rails with a distance of 1.4m in between. You can choose to create multiple lanes and synchronize the loads or you can create one traffic lane. We will create one traffic lane on the edge of the 2D member. The traffic lanes have to be entered on **1.8m** from each edge to be able to place the train track in the middle of the bridge.

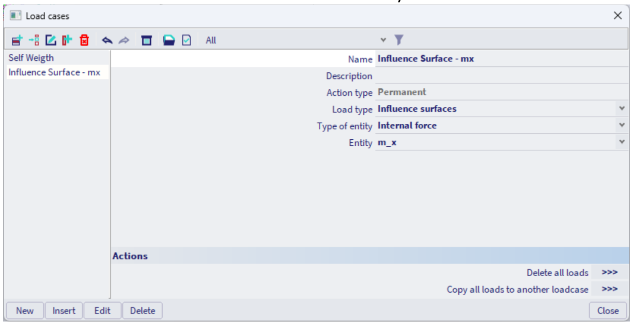
Through Traffic lane 2D the traffic lanes can be defined. We use the nodes on coordinate (0,5) and (25,5) to create the traffic lane. The load system will move on the right of the defined lane (from start to end node).

As **Name** of the traffic lane, **TL1** is entered. Make sure to allocate the correct 2D member, this can be don automatically via the action 'allocate automatically'.

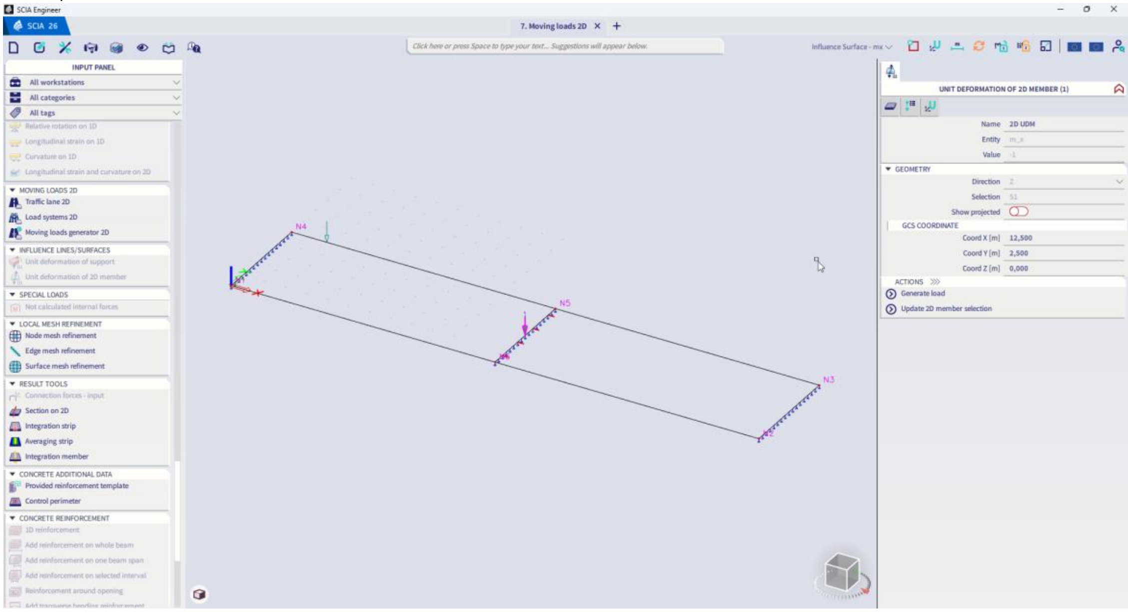


7.1.2. Unit deformation

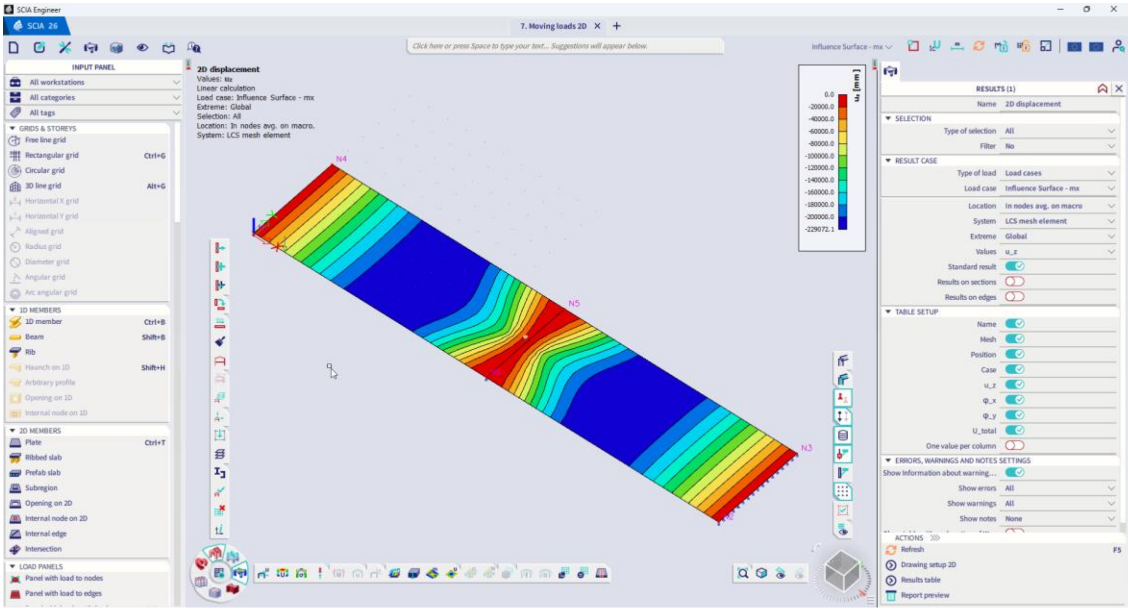
After defining the traffic lane, a load case can be created of the type influence surfaces. For this load case, you define the type of entity (Internal force, Deformation, Reaction) and you define one entity. This entity will define which influence surface will be available. If you need more influence surfaces, you will need to create multiple load cases.



Once this is created, you can add the unit deformation of 2D member. You define the exact position where this unit deformation needs to be created. For this example, we will add a unit deformation in the middle of the bridge. You can only add one unit deformation to a load case. To find the influence surface on other positions, you will need to create multiple load cases.

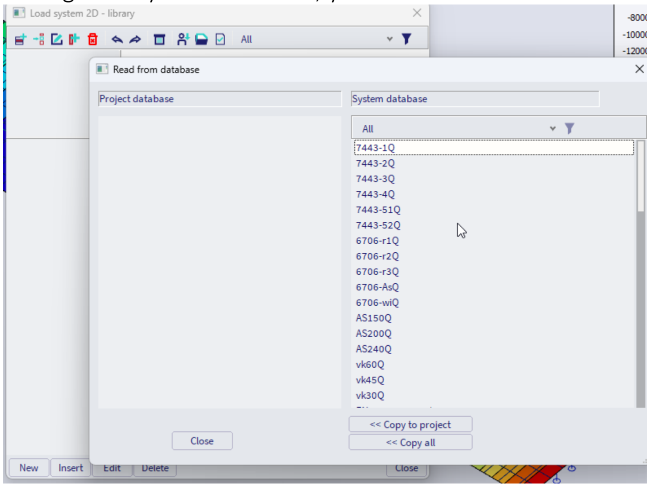


The project can already be calculated for this load case. To review the influence surface, you can check the 2D deformations uz for the created load case 'Influence Surface - mx'.

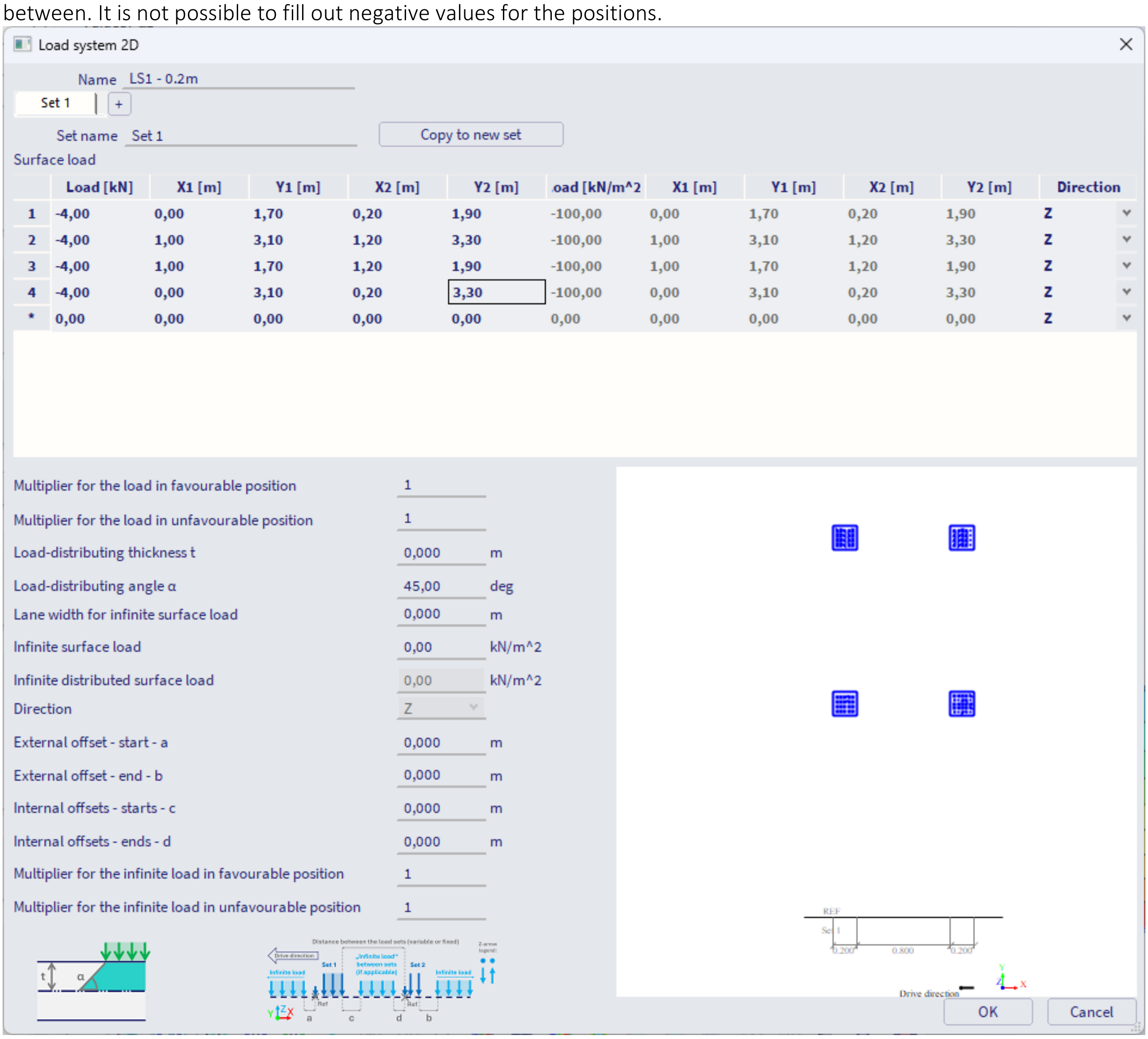


7.1.3. Load systems 2D

With the command Load systems 2D a new load system can be created. You can create any system manually or through the system database, you will also be able to select some preprogrammed systems.

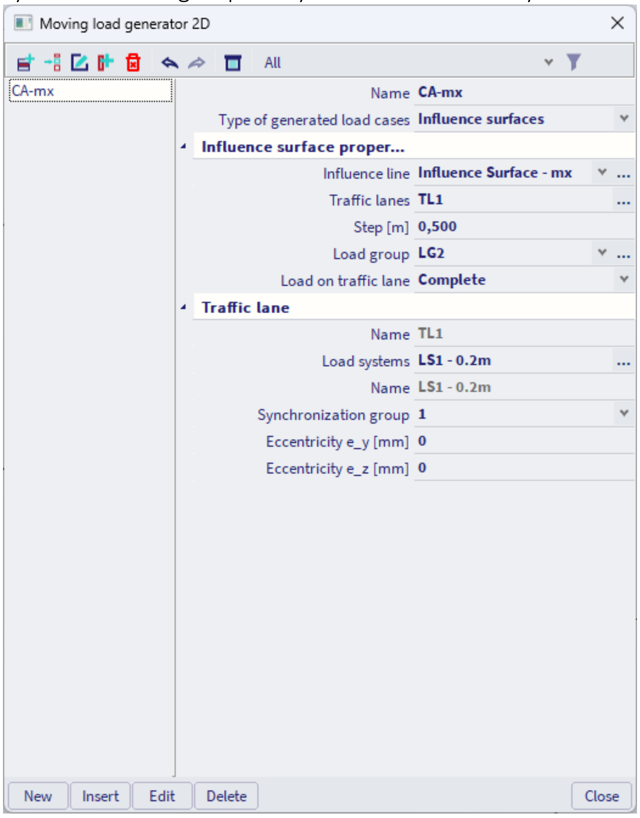


When you create a new load system, you will input different surface loads and define their size and position via X1, Y1, X2 and Y2. The value of the load and the direction can also be modified. We created four surface loads with a size of 0.2x0.2m. The distance in between is equal to 0.8m. We are not adding an infinite surface or distributed load in between. It is not possible to fill out negative values for the positions.

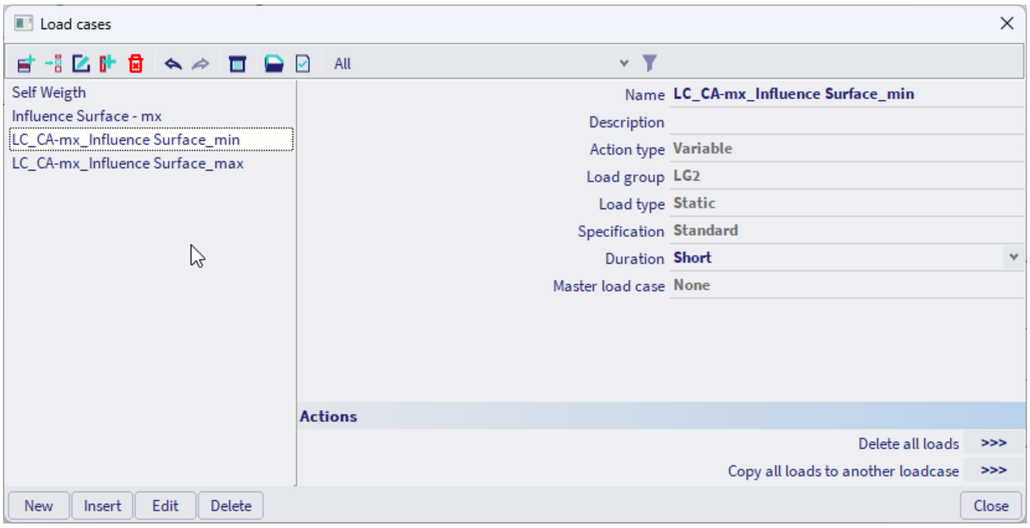


7.1.4. Moving loads generator 2D – Influence lines

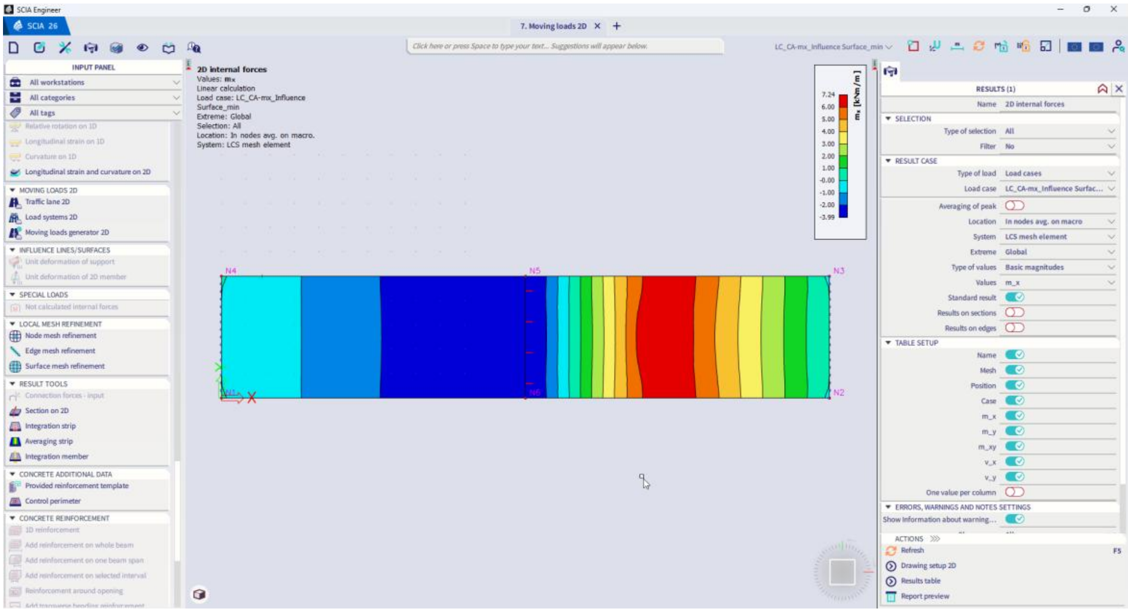
With the moving loads generator we generate load cases of the type 'influence surfaces'. In the properties you define which traffic lanes are used, what the step is of the moving loads and in which load group de load cases need to be added. For each traffic lane, you define the load system that needs to be moving along that lane. You use synchronization groups to synchronize the load systems when multiple are used on (different) traffic lanes.



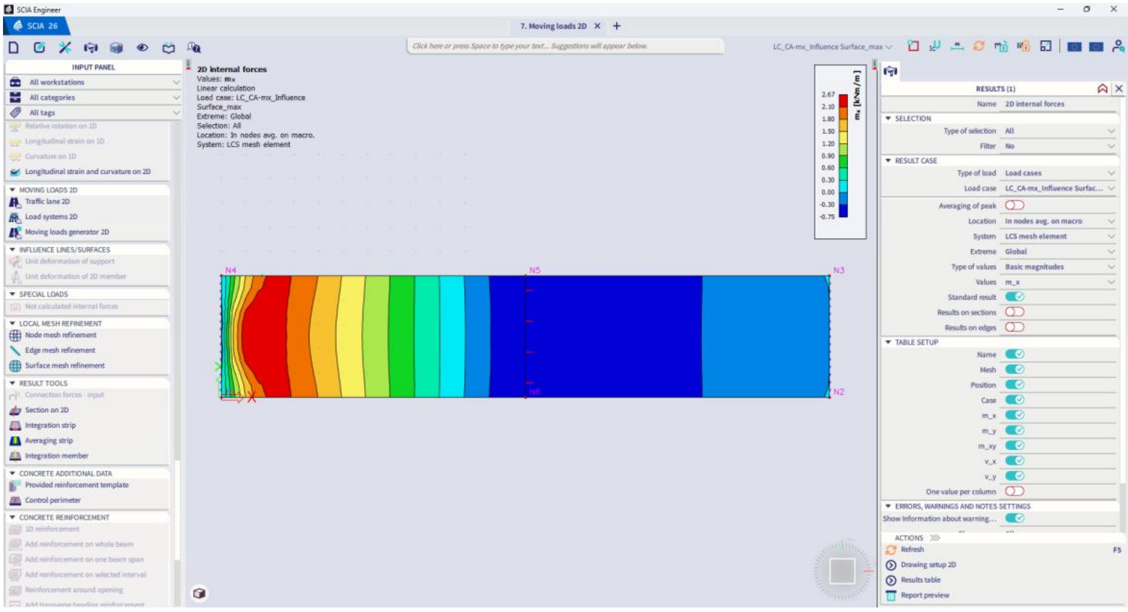
When the project is calculated, two load cases will be created. One for the maximum value of mx, one for the minimum value of mx.



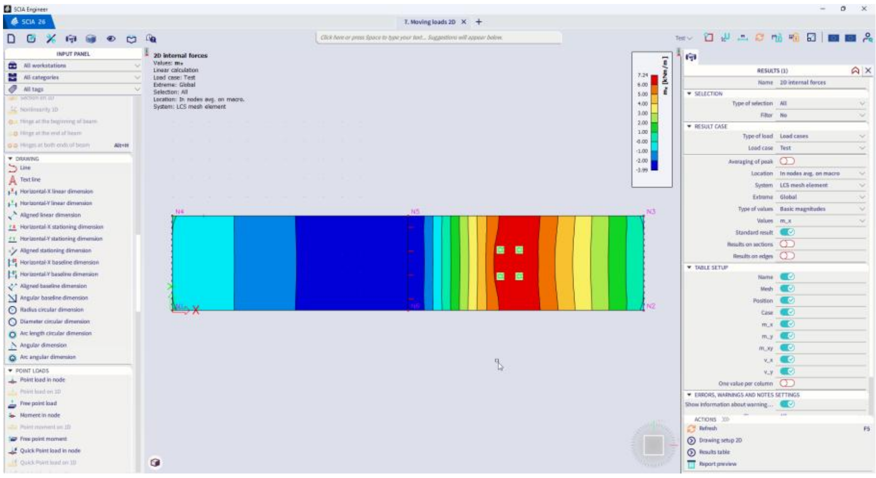
The results of these load cases can be reviewed via 2D internal forces. The results shown will represent one specific position of the moving load causing either the maximum value of mx or the minimum value of mx.
Min:



Max:

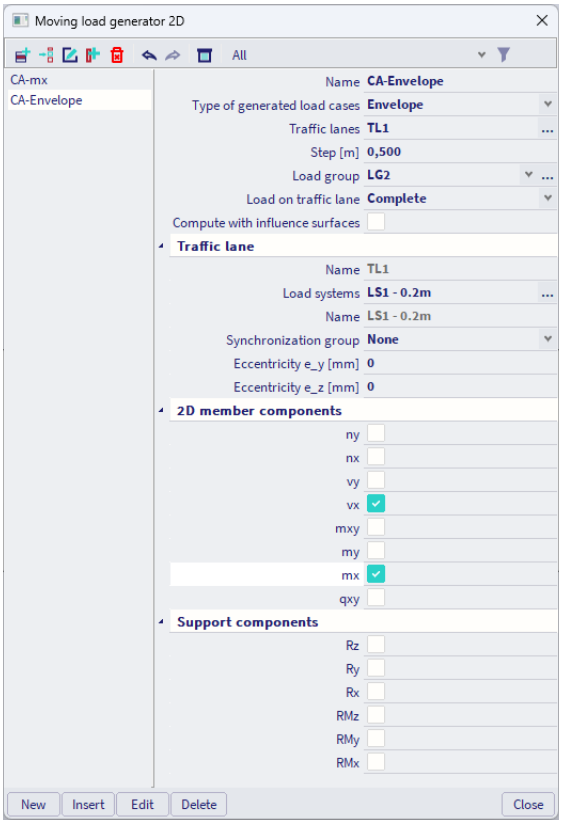


At this moment you can not visualize the exact position of the moving load, but we can easily mimic this result by creating a test load case. The minimum result in the middle of the bridge is caused by the moving load at the position x=18.5m:

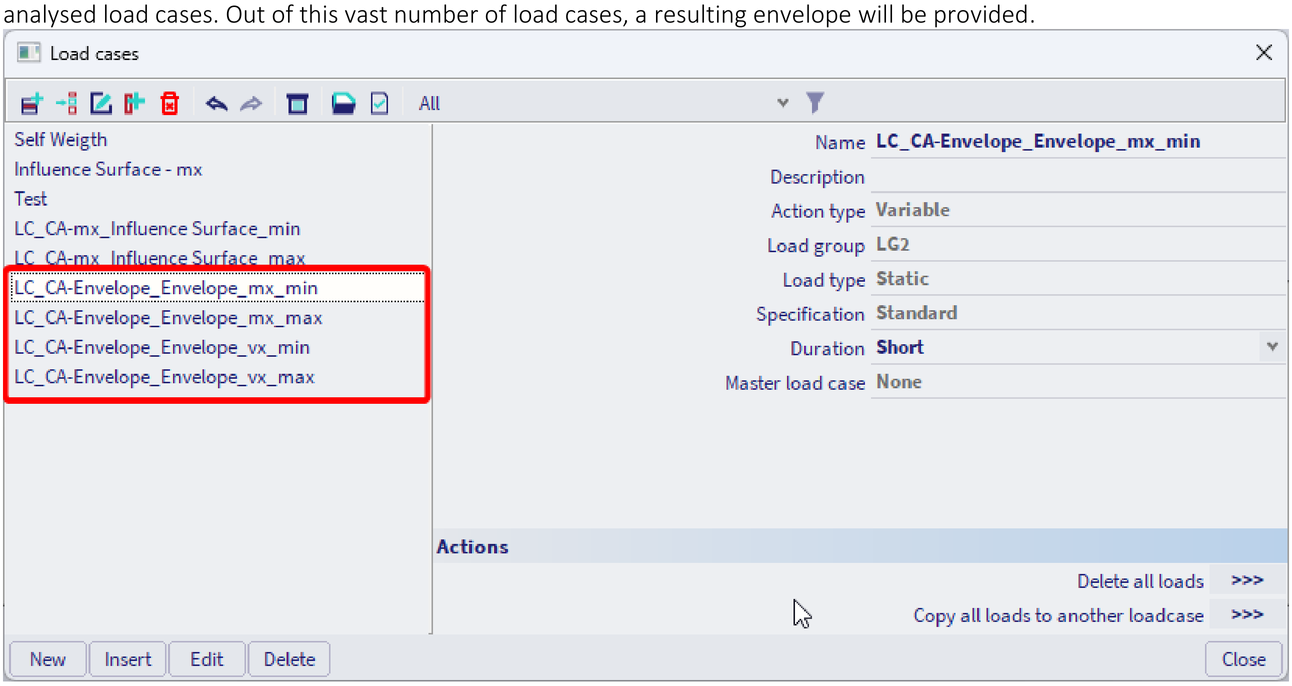


7.1.5. Moving loads generator 2D – Envelope load cases

In this project the enveloping load cases will be generated for the moment *mx* and the shear force *vx*.
The same traffic lane and load system is used. Except now, the generated load cases in the moving loads generator is set to 'envelope'. While other properties are similar, we now select the 2D member components that need to be created.

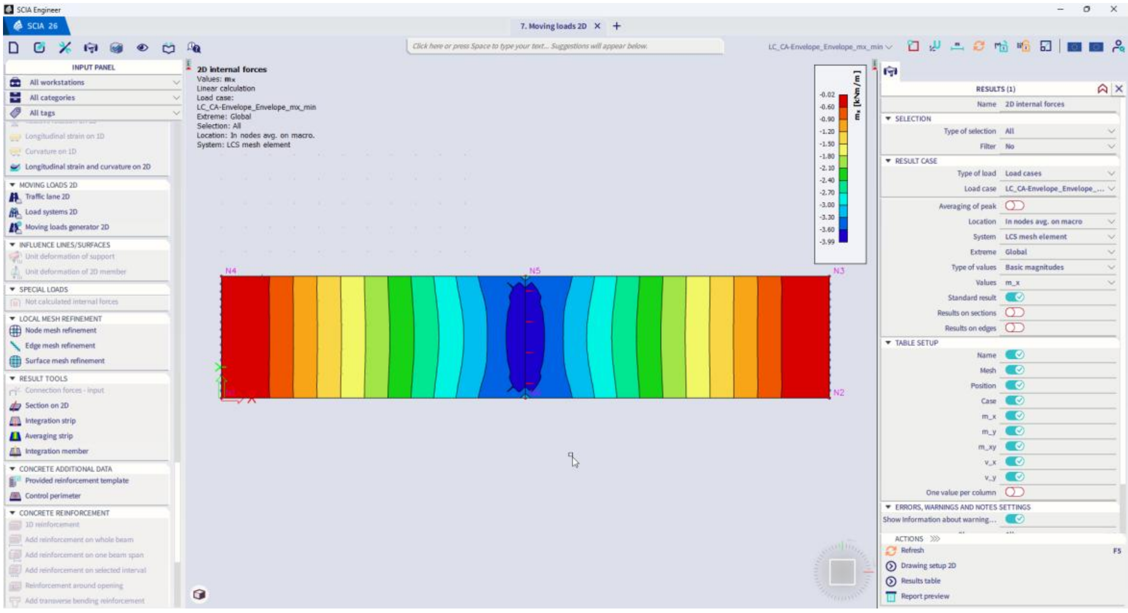


When the project is calculated, new load cases will be created. For each component selected in the generator, two load cases (a max and min) are automatically created. For these results, internally several linear load cases will be analysed, respecting varying positions of the load system, assigned to the traffic lane. This lead to vast numbers of internally analysed load cases. Out of this vast number of load cases, a resulting envelope will be provided.

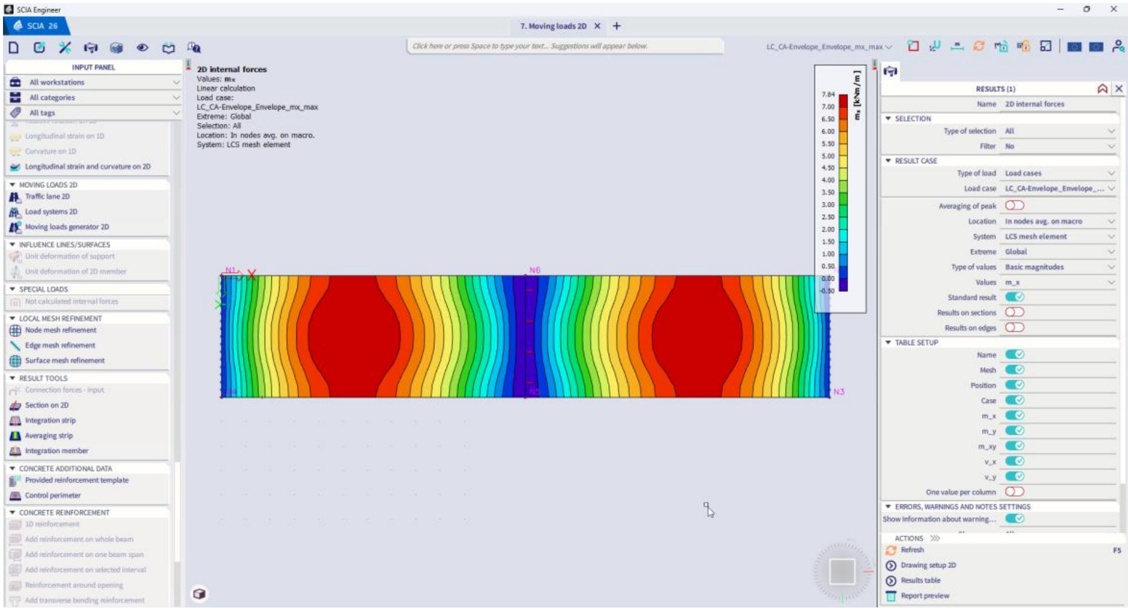


You can find the results through 2D internal forces since the selected components were mx and vx. Below is the example of mx.

Min-mx:

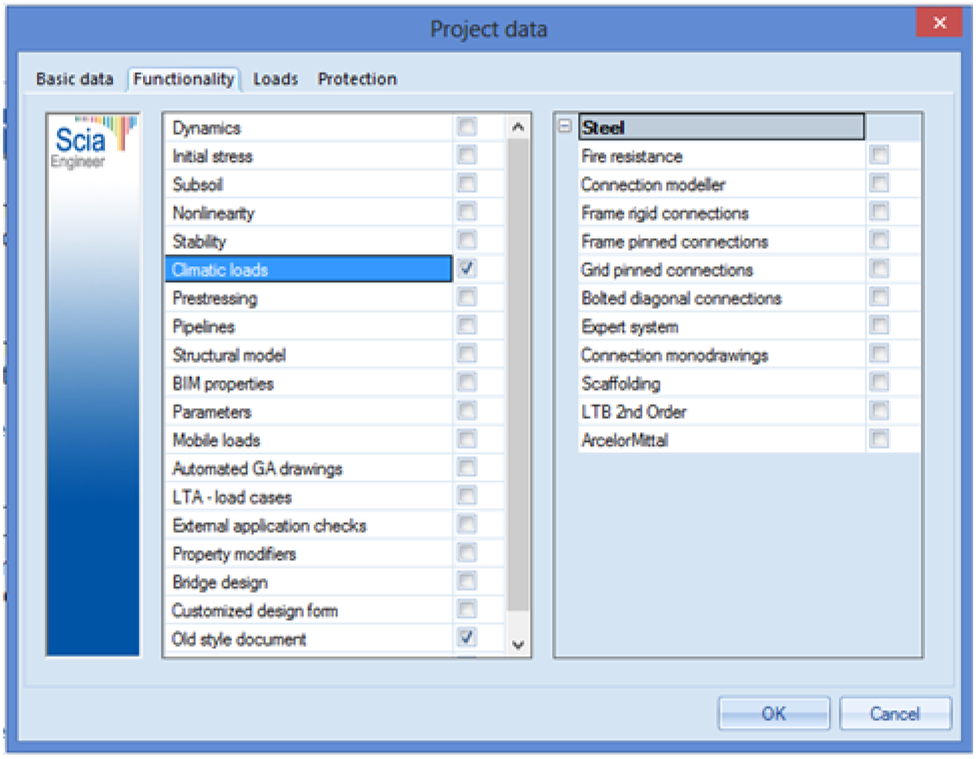


Max-mx:

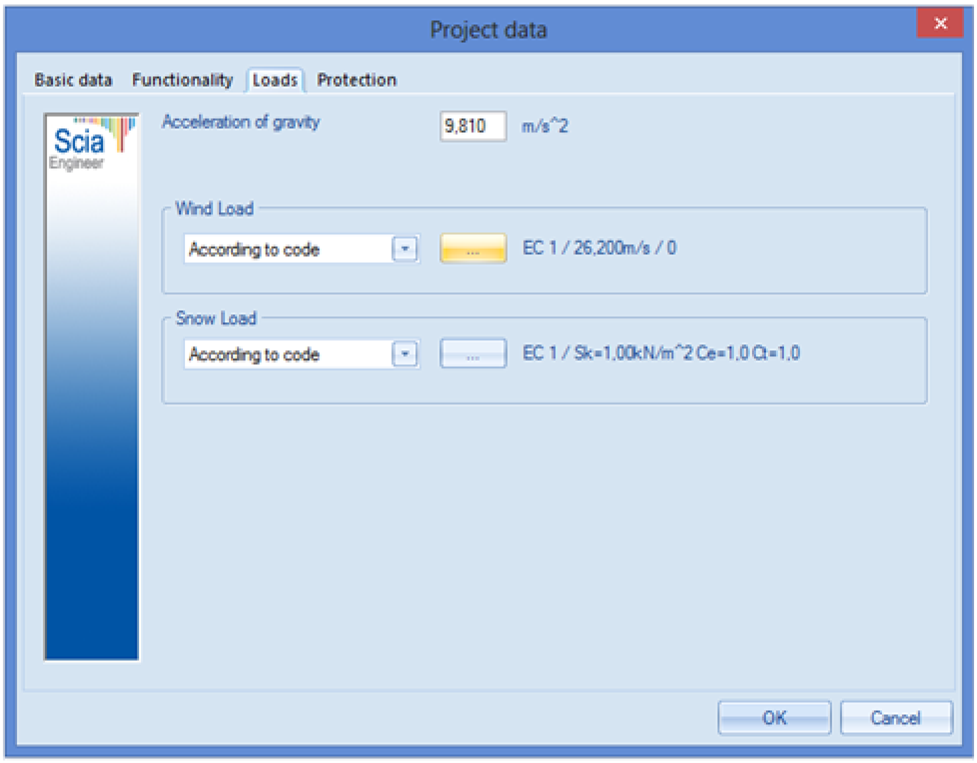


8. 2D wind- and snow generators (32-bit)

In this chapter, the wind and snow generator are explained. To be able to use this generators, the functionality 'Climatic loads' should be used.



In the next tab, the wind and snow load should be chosen (according to the code or user defined).



There are three types of climatic load generators for 2D frames in SCIA Engineer:

- Wind generator
- Snow generator
- Wind & snow generator

For all these load generators, the example '2D climatic generators' is used.
Example: 2D climatic generators.esa

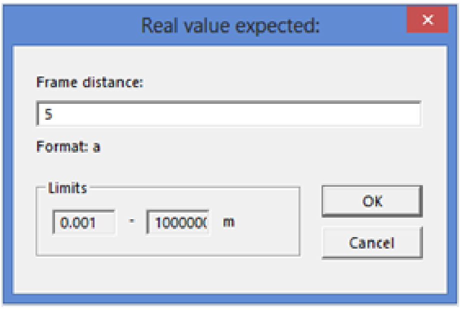
8.1. Wind generator

Since there are no load cases automatically created by the wind generator, these have to be created manually.
Load cases:

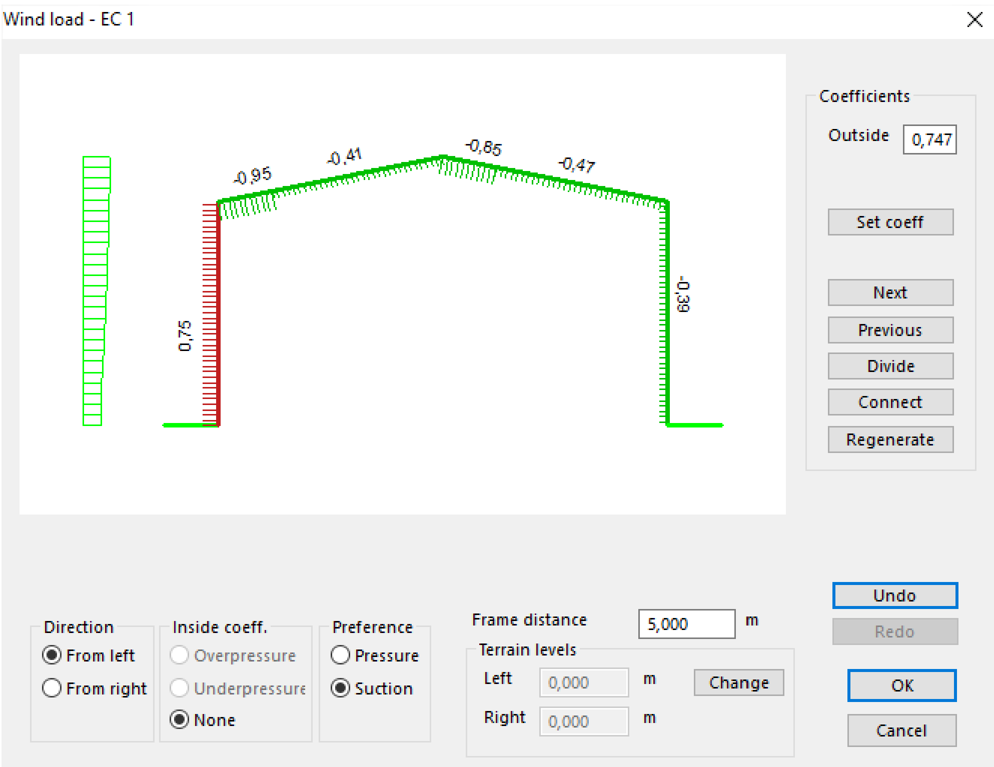
- Self Weight (Permanent)
- Wind Left (Variable, load group exclusive)
- Wind Right (Variable, load group exclusive)

After creating these load cases, the wind loads can be added in the Load menu with the option **Wind generator**.

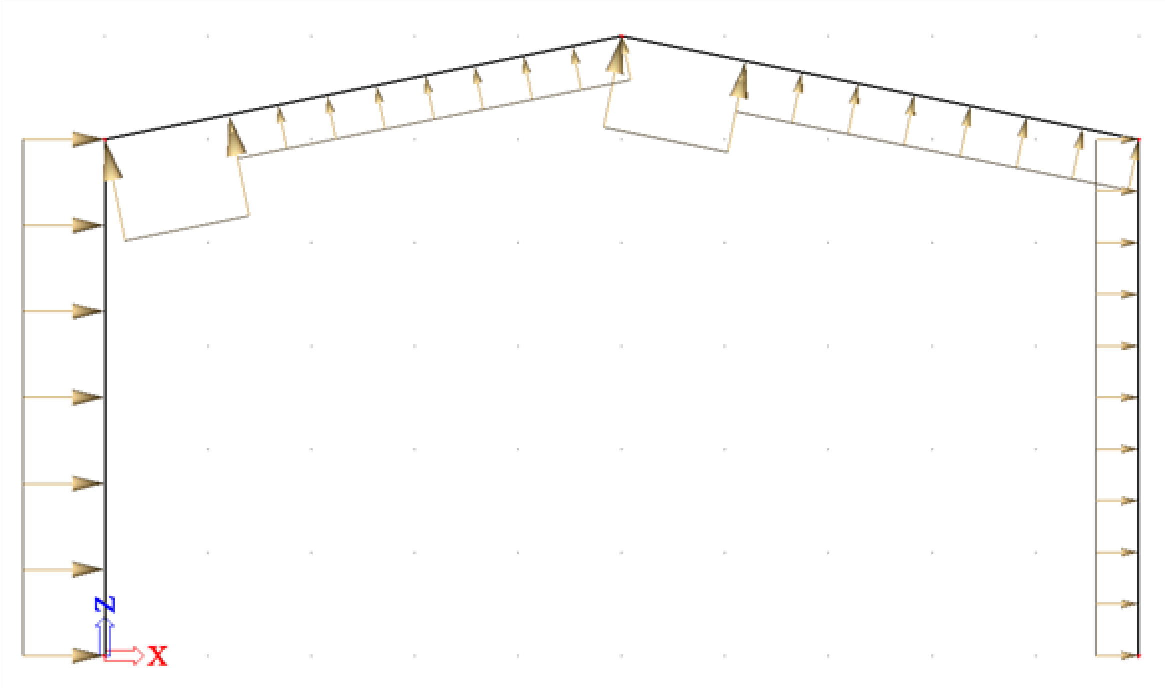
By using this wind generator, a frame distance has to be inserted. This has to be done to simulate the wind on a 2D frame as if the wind would be on a complete 3D structure.



In the next window, the settings for the wind load calculation have to be inserted:



For the load case Wind Left, the direction is set to 'From left'.
After clicking on OK in this window, the wind load from the left is generated and placed on the frame.
The same can be done for the wind load from the right.



The **Load Coefficients** are calculated according to the code, EN 1991-1-4.
For the vertical walls, table 7.1 of EN 1991-1-4 is used [1]:

Table 7.1 — Recommended values of external pressure coefficients for vertical walls of rectangular plan buildings

Zone	A		B		C		D		E	
h/d	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$
5	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,7	
1	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,5	
≤ 0,25	-1,2	-1,4	-0,8	-1,1	-0,5		+0,7	+1,0	-0,3	

In this example, h = 6m and d = 10m, so h/d = 0,6.
This value is between 0,25 and 1, so an interpolation has to be done between these two rows.
In the following picture [1], it can be seen that the zones D and E should be calculated:

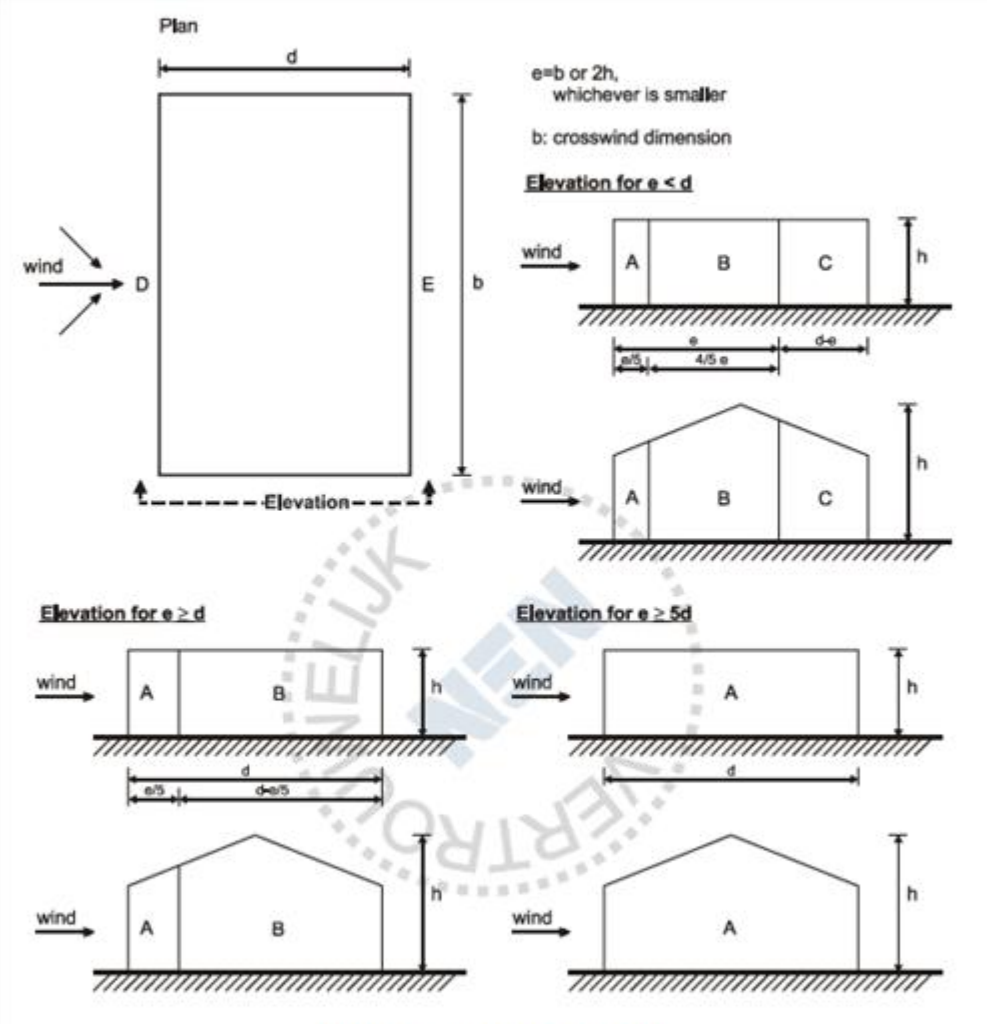


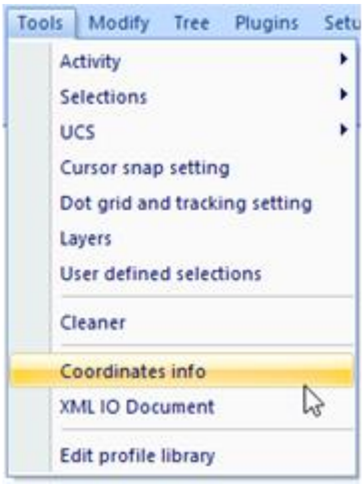
Figure 7.5 — Key for vertical walls

For the roof, table 7.4a of EN 1991-1-4 is used [1]:

Table 7.4a — External pressure coefficients for duopitch roofs

Pitch Angle α	Zone for wind direction $\theta = 0°$									
	F		G		H		I		J	
	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$	$c_{pe,10}$	$c_{pe,1}$
-45°	-0,6		-0,6		-0,8		-0,7		-1,0	-1,5
-30°	-1,1	-2,0	-0,8	-1,5	-0,8		-0,6		-0,8	-1,4
-15°	-2,5	-2,8	-1,3	-2,0	-0,9	-1,2	-0,5		-0,7	-1,2
-5°	-2,3	-2,5	-1,2	-2,0	-0,8	-1,2	+0,2		+0,2	
							-0,6		-0,6	
5°	-1,7	-2,5	-1,2	-2,0	-0,6	-1,2	-0,6		+0,2	
	+0,0		+0,0		+0,0				-0,6	
15°	-0,9	-2,0	-0,8	-1,5	-0,3		-0,4		-1,0	-1,5
	+0,2		+0,2		+0,2		+0,0		+0,0	+0,0
30°	-0,5	-1,5	-0,5	-1,5	-0,2		-0,4		-0,5	
	+0,7		+0,7		+0,4		+0,0		+0,0	
45°	-0,0		-0,0		-0,0		-0,2		-0,3	
	+0,7		+0,7		+0,6		+0,0		+0,0	
60°	+0,7		+0,7		+0,7		-0,2		-0,3	
75°	+0,8		+0,8		+0,8		-0,2		-0,3	

The angle α of the roof is 11,31°. The angle can be checked with the option **Coordinates info** in the Tools menu:



This means that an interpolation has to be done between the values for angle α 5° and 15°.

In the following picture [1], it can be seen that the zones G,H,I and J should be calculated:

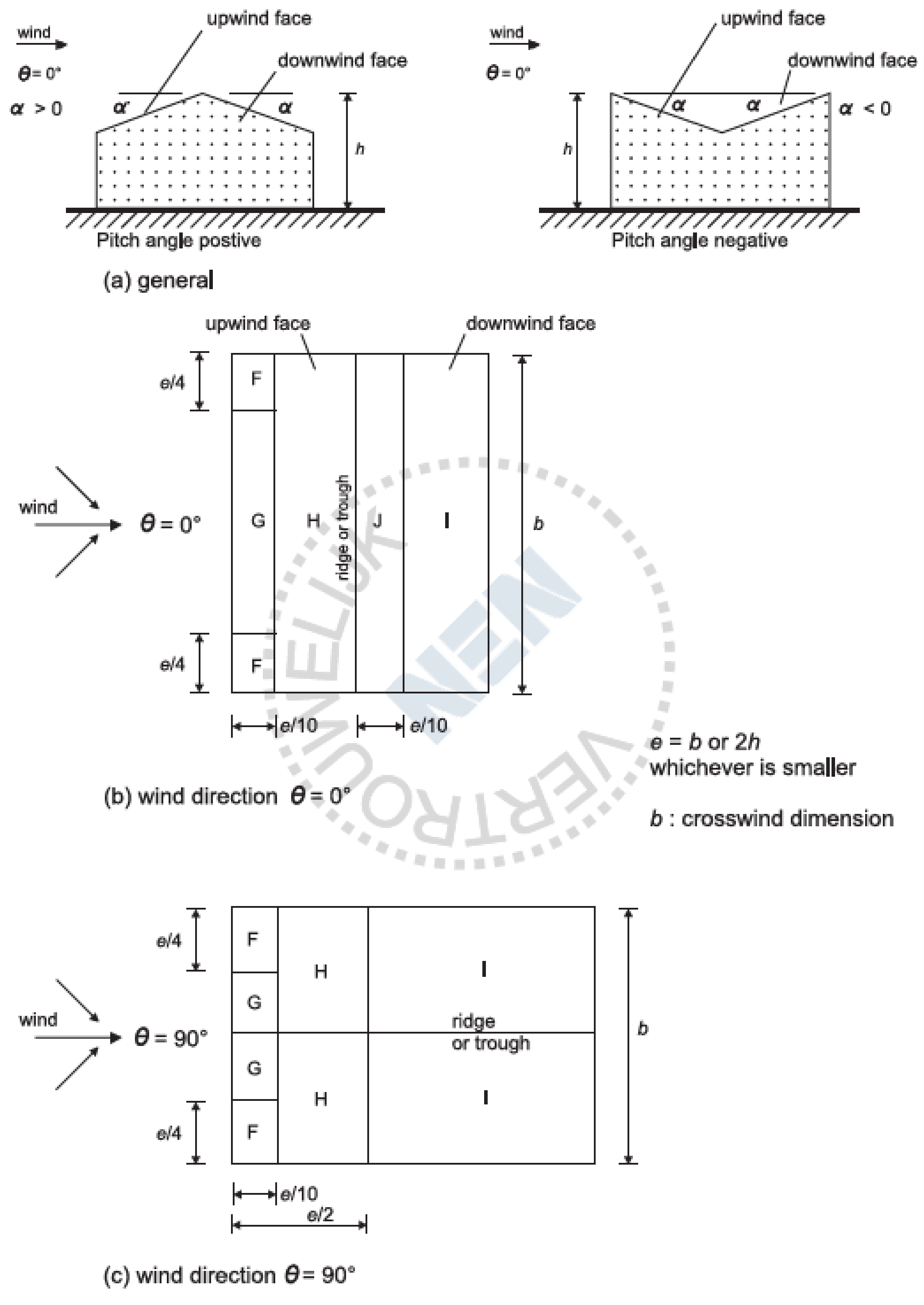
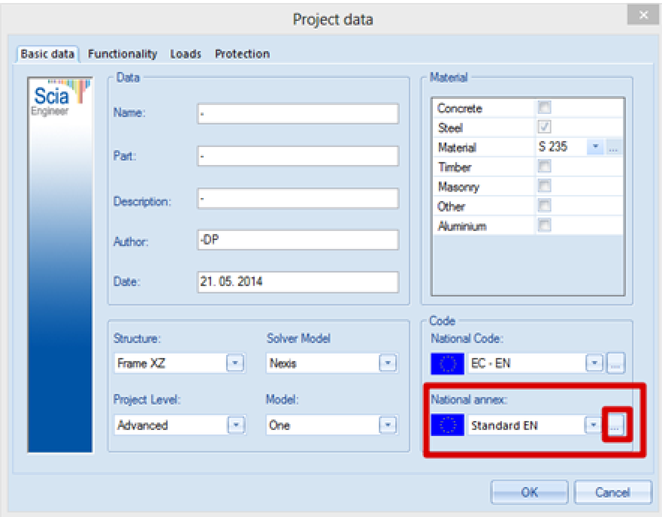
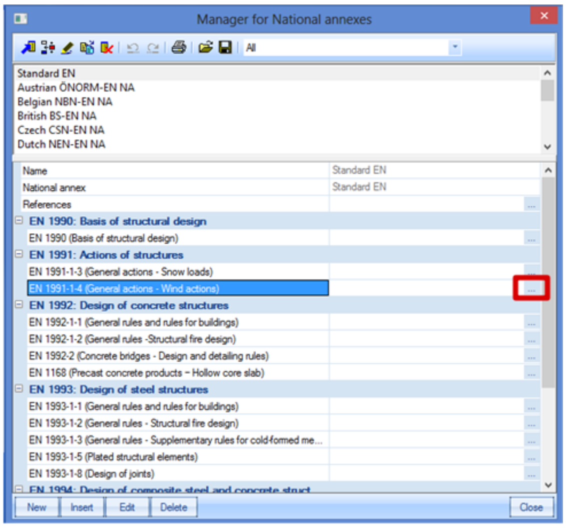


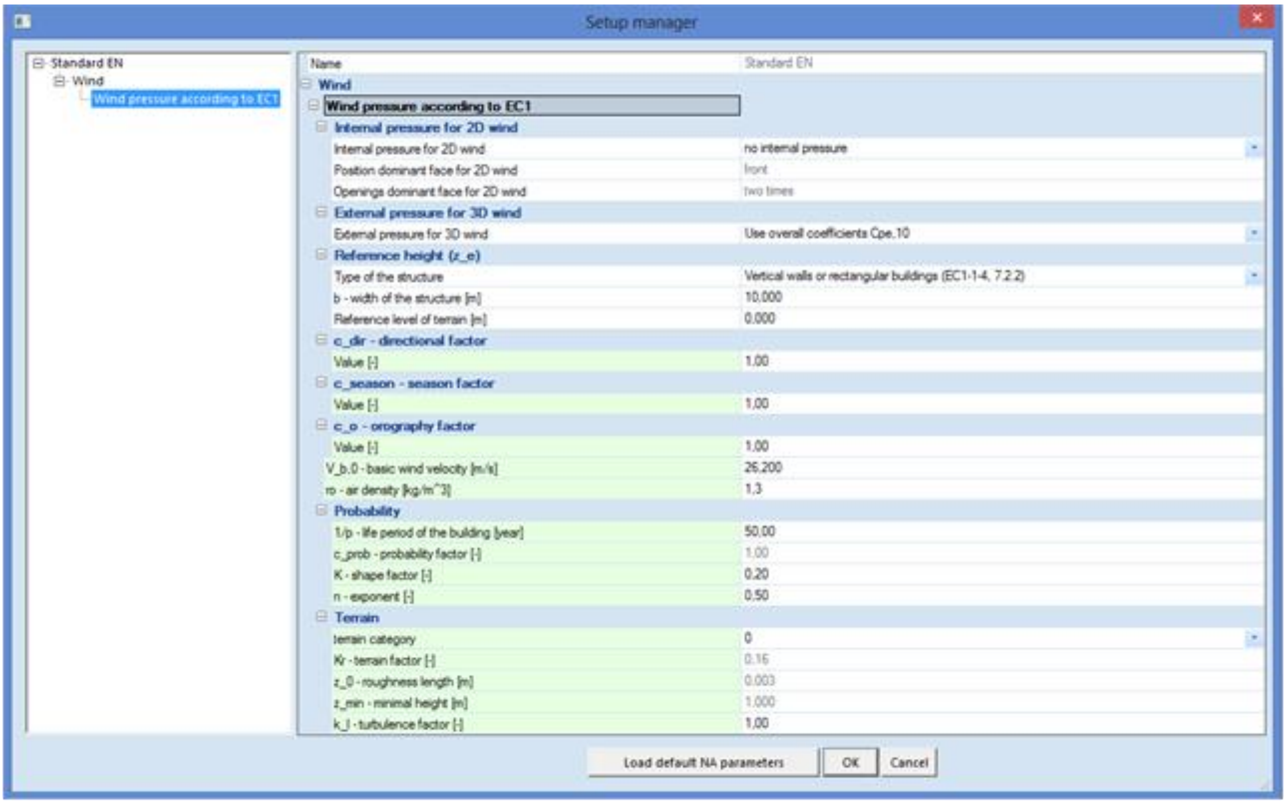
Figure 7.8 — Key for duopitch roofs

In this example, the Cpe,10 values are used. The choice to use the Cpe,10 or the Cpe,1 values, can be made in the **National annex** parameters. Also the option to take into account the internal pressure coefficients has to be set in these parameters.

The national annex parameters can be opened in the project data:







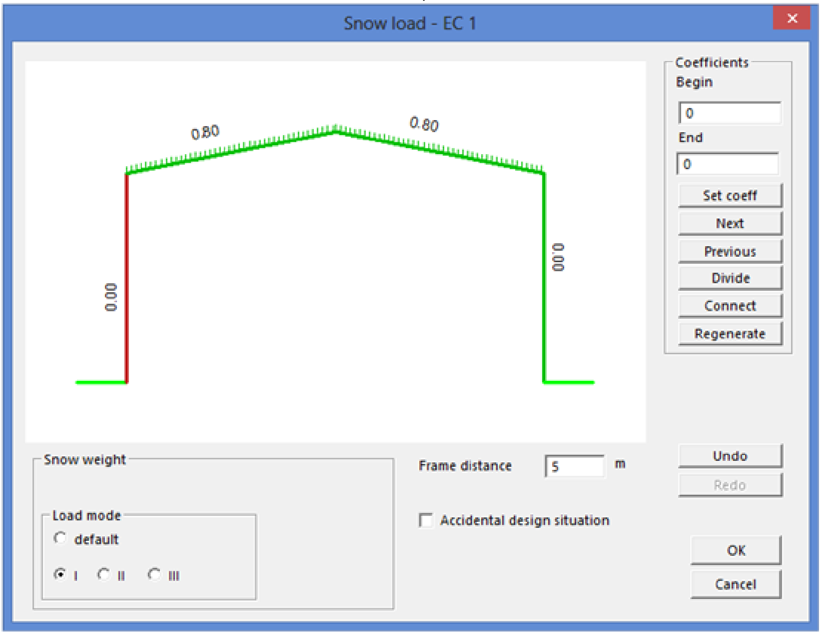
8.2. Snow generator

After generating the wind load in both directions, also the snow loads will be generated in this example.
For the snow loads, three load cases have to be created.
Extra load cases:

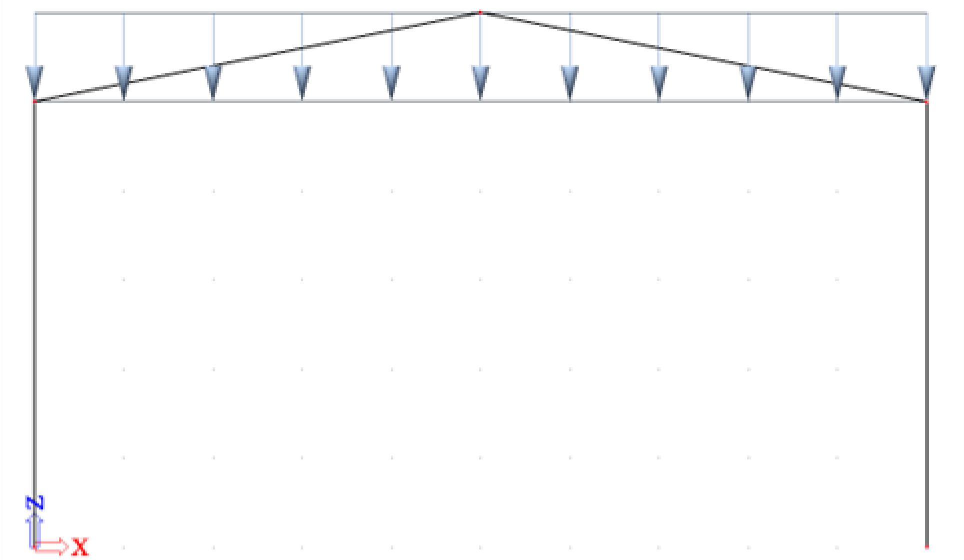
- Snow load 1 (Variable, exclusive)
- Snow load 2 (Variable, exclusive)
- Snow load 3 (Variable, exclusive)

After creating these load cases, the snow loads can be added in the Load menu with the option **Snow generator**.

By using this snow generator, a frame distance has to be inserted. This has to be done to simulate the wind on a 2D frame as if the wind would be on a complete 3D structure.



After clicking on OK in this window, the snow load 1 will be generated on the roof elements.
The same can be done for Snow load 2 and 3.



The **Load Coefficients** are calculated according to the code, EN 1991-1-3.
Table 5.2 [2] is used for the load coefficients.

Table 5.2: Snow load shape coefficients

Angle of pitch of roof α	$0° \le \alpha \le 30°$	$30° < \alpha < 60°$	$\alpha \ge 60°$
μ_1	0,8	0,8(60 - α)/30	0,0
μ_2	0,8 + 0,8 α/30	1,6	--

Since the angle α is 11,31° for this example, a load coefficient of 0,80 is used.
This is the case for **Load mode 1**.

The different modes (cases) are explained in article 5.3.3 of EN1991-1-3, figure 5.3 [2].

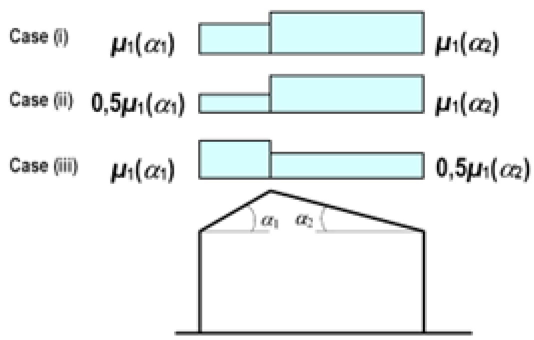
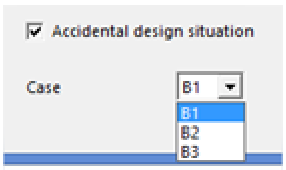


Figure 5.3: Snow load shape coefficients - pitched roofs

So for Snow load 2 and 3, there is a reduction of 50% of the snow weight.
Additionally, an accidental design situation can be taken into account. The need to take this into account is National Annex dependent.



The different cases are shown in table A.1 of EN1991-1-3 [2].

Table A.1 Design Situations and load arrangements to be used for different locations

Normal	Exceptional conditions		
Case A	Case B1	Case B2	Case B3
No exceptional falls No exceptional drift	Exceptional falls No exceptional drift	No exceptional falls Exceptional drift	Exceptional falls Exceptional drift
3.2(1)	3.3(1)	3.3(2)	3.3(3)
Persistent/transient design situation [1] undrifted $\mu_i\ C_eC_t\ s_k$ [2] drifted $\mu_i\ C_eC_t\ s_k$	*Persistent/transient design situation* [1] undrifted $\mu_i\ C_eC_t\ s_k$ [2] drifted $\mu_i\ C_eC_t\ s_k$ *Accidental design situation (where snow is the accidental action)* [3] undrifted $\mu_i\ C_eC_t\ C_{esl}\ s_k$ [4] drifted $\mu_i\ C_eC_t\ C_{esl}\ s_k$	*Persistent/transient design situation* [1] undrifted $\mu_i\ C_eC_t\ s_k$ [2] drifted $\mu_i\ C_eC_t\ s_k$ (except for roof shapes in AnnexB) *Accidental design situation (where snow is the accidental action)* [3] drifted $\mu_i\ s_k$ (for roof shapes in AnnexB)	*Persistent/transient design situation* [1] undrifted $\mu_i\ C_eC_t\ s_k$ [2] drifted $\mu_i\ C_eC_t\ s_k$ (except for roof shapes in AnnexB) *Accidental design situation (where snow is the accidental action)* [3] undrifted $\mu_i\ C_eC_t\ C_{esl}\ s_k$ [4] drifted $\mu_i\ s_k$ (for roof shapes in AnnexB)
NOTE 1: Exceptional conditions are defined according to the National Annex. **NOTE 2**: For cases B1 and B3 the National Annex may define design situations which apply for the particular local effects described in section 6.			

In Belgium and the Netherlands, the cases B1, B2 and B3 do not need to be considered.

8.3. Wind & Snow generator

This generator is a combination of the wind and snow generator, which are described in the previous chapters.
This generator automatically creates:

- 2 new load groups
  - Wind (variable, exclusive)
  - Snow (variable, exclusive)
- Without over- and underpressure taken into account
  - 3 new load cases
    - WND – L – Wind from the left
    - WND – R – Wind from the right
    - SN – Snow loads
- With over- and underpressure taken into account
  - 5 new loadcases
    - WND – LO – Wind from the left - overpressure
    - WND – LU – Wind from the left - underpressure
    - WND – RO – Wind from the right - overpressure
    - WND – RU – Wind from the right - underpressure
    - SN – Snow loads

By default, the option to use over- and underpressure is grayed out. To be able to use this additional pressure, the option **Internal pressure for 2D wind** needs to be activated in the **National Annex parameters**.