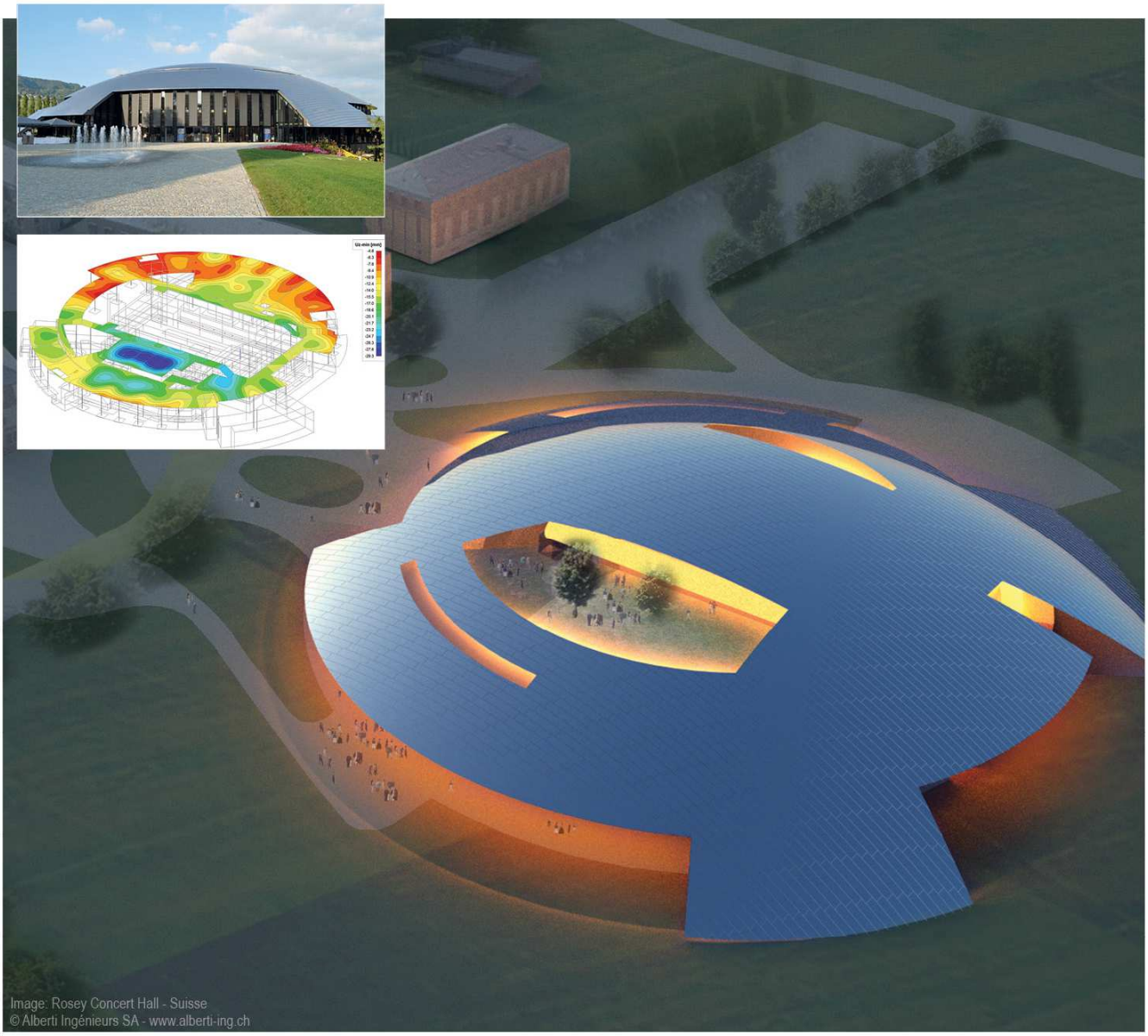
SCIAENGINEER



Topic Training

Load generators

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

© Copyright 2015 SCIA nv. All rights reserved.

Table of contents

Introduction **5**

Plane generator **6**

Load panels **9**

Load to panel nodes 9
Load panel to edges 12
Panel to edges and beams 15
Panel with parallel beams 17

Free loads **21**

2D Wind- and snow generators **27**

Wind generator 28
Snow generator 34
Wind & Snow generator 37

3D Wind Generator **39**

Mobile loads **43**

Principle 43
Load system 46
Train Loads 61

References and literature **74**

Introduction

This course 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 functionality an extra module (or edition) is required, but this will always be indicated in those paragraphs.

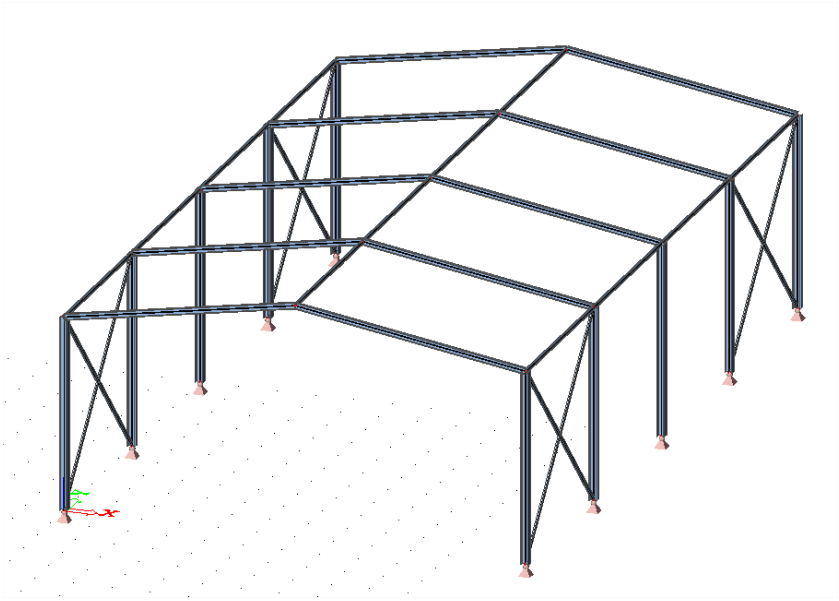
Plane generator

The plane generator is a tool which automatically transforms a defined area load into line loads acting on beams located in the specified plane.

This generator can be used for flat frames which are subjected to a continuous area load. With this generator, it is no longer needed to manually determine the load widths and calculate the line loads.

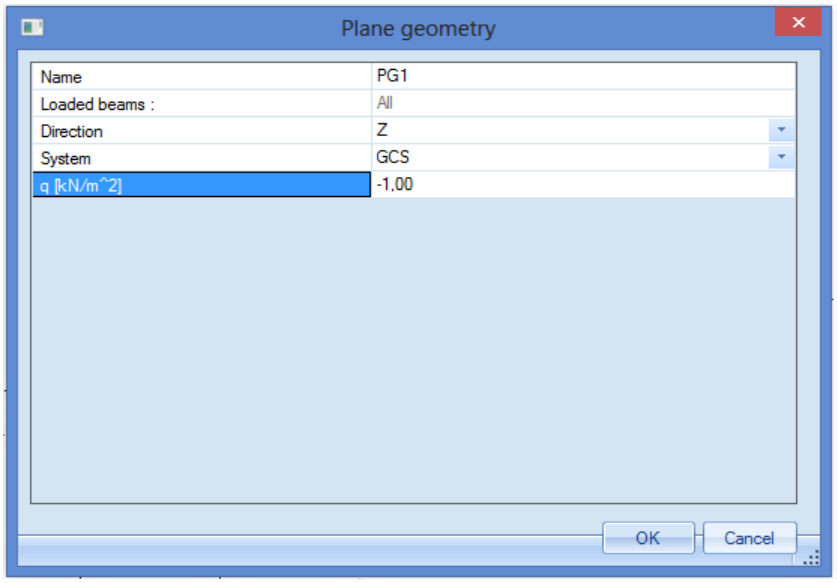
Example: Load Generators.esa

This example consists of a simple steel hall:

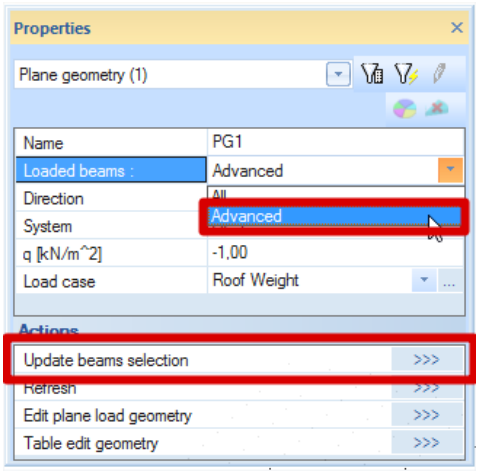


The self weight of the structure is automatically calculated by SCIA Engineer. The weight of the roof will be inserted manually using the **Plane generator** in the Load menu.

Load cases:
Self Weight (permanent)
Roof Weight (permanent), -1 kN/m²

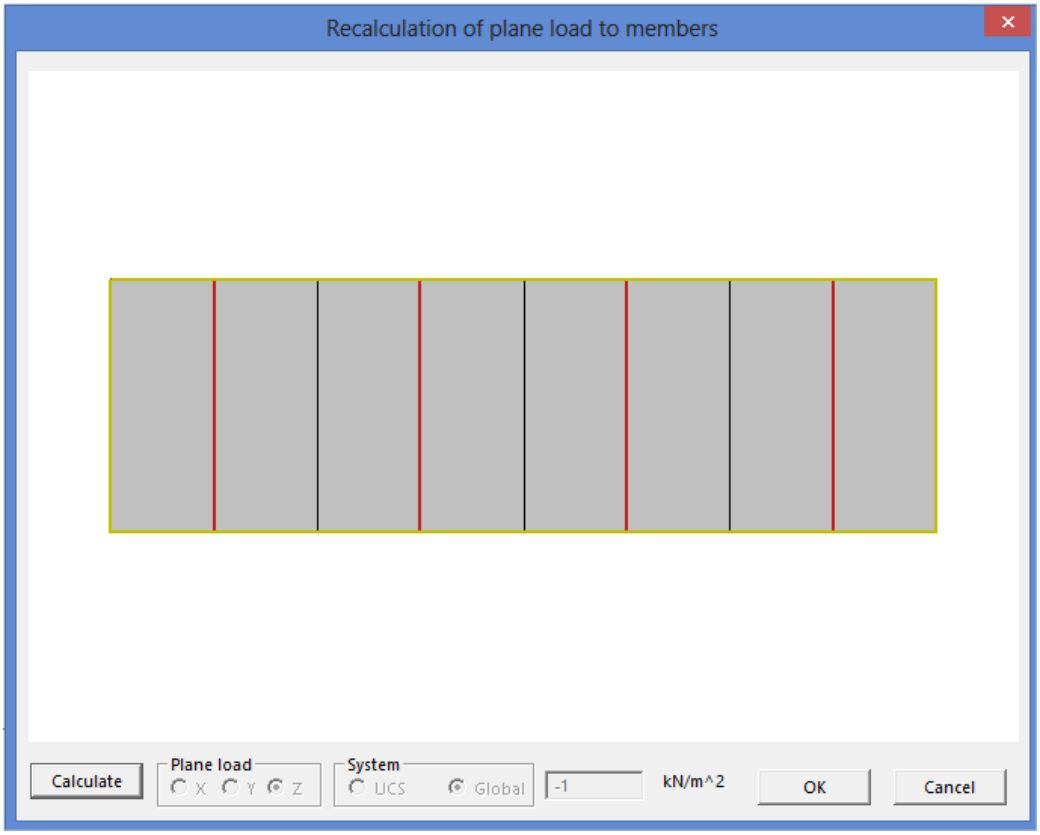


By inserting an area load using the plane generator, all beams in the specified plane are loaded. This can be changed afterwards.



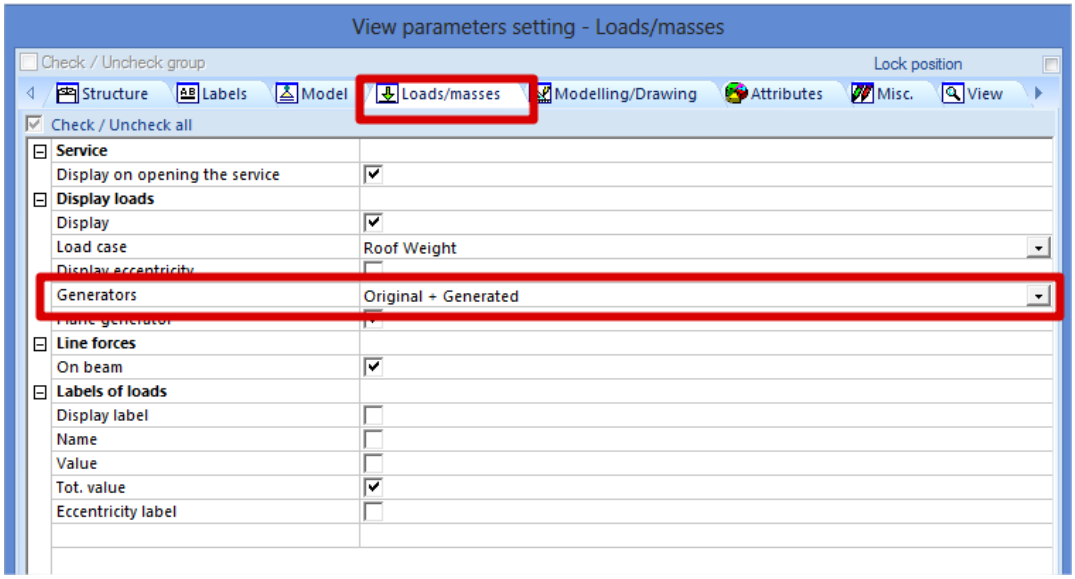
With the option **Update beams selection**, all beams that have to be loaded can be selected.

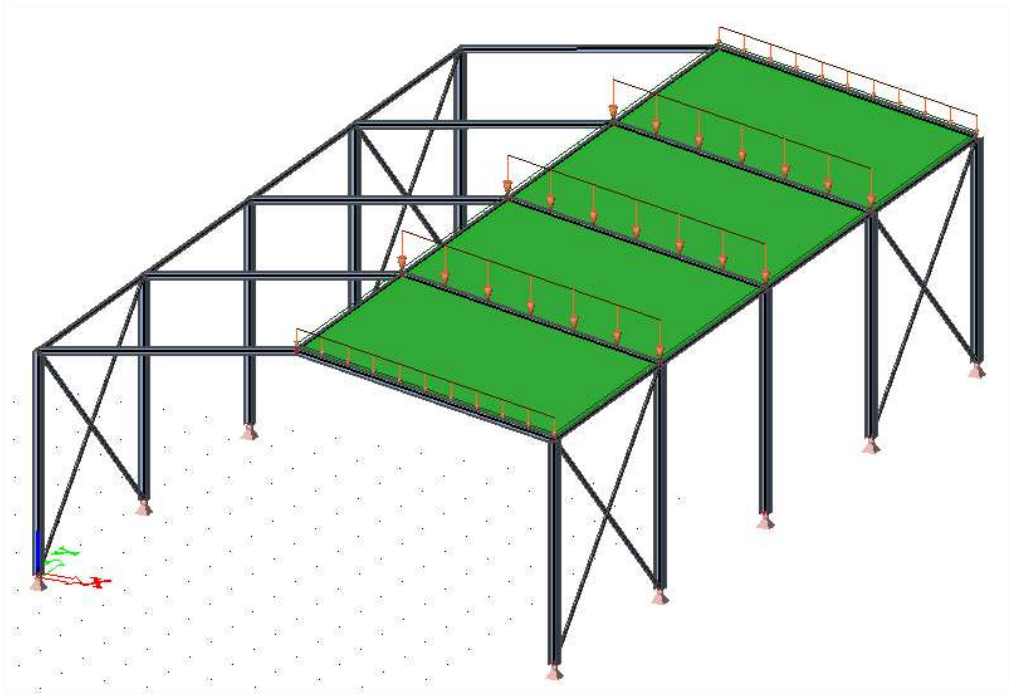
By using the action **Refresh**, the area load can be recalculated into line loads on the selected beams. After clicking on **Calculate**, the areas for the distribution of the load are calculated. The boundaries are indicated by the red lines.



Notes:

- The last step, to recalculate the area load into line loads on the beams, will be automatically done during the calculation of the project.
- Both loads, the original area load and the generated line loads, can be displayed graphically. This can be done by changing the View parameters for the loads:


View parameters setting - Loads/masses
Check / Uncheck group
Lock position
Structure
Labels
Model
Loads/masses
Modelling/Drawing
Attributes
Misc.
View
Check / Uncheck all
Service
Display on opening the service
Display loads
Display
Load case
Roof Weight
Generators
Original + Generated
Line forces
On beam
Labels of loads
Display label
Name
Value
Tot. value
Eccentricity label



Load panels

Load panels are entities which are not taken into account in the FEM calculation (Finite Element Method Calculation).

This means that load panels:

- Have no self weight
- Do have a certain stiffness to distribute the loads to the correct members (nodes, beams, plate edges, ...).
- This stiffness is not taken into account for the stiffness of the structure.

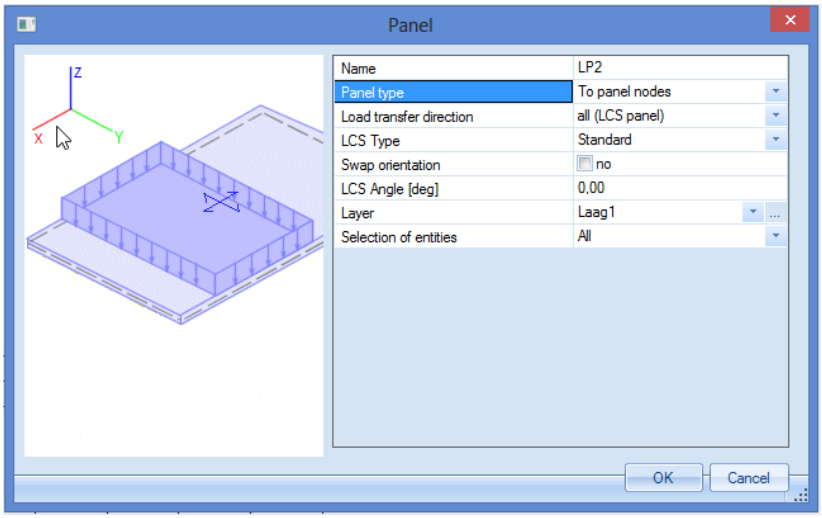
All type of loads can be inserted on load panels: point loads, line loads, surface loads, ... The load panels can be found in the **Structure menu**.

There are 4 types of load panels available in SCIA Engineer:

- Load to panel nodes
- Load to panel edges
- Load to panel edges and beams
- Panel with parallel beams

Load to panel nodes

With this tool, the loads introduced on the panel, will be distributed to the nodes of the load panel.

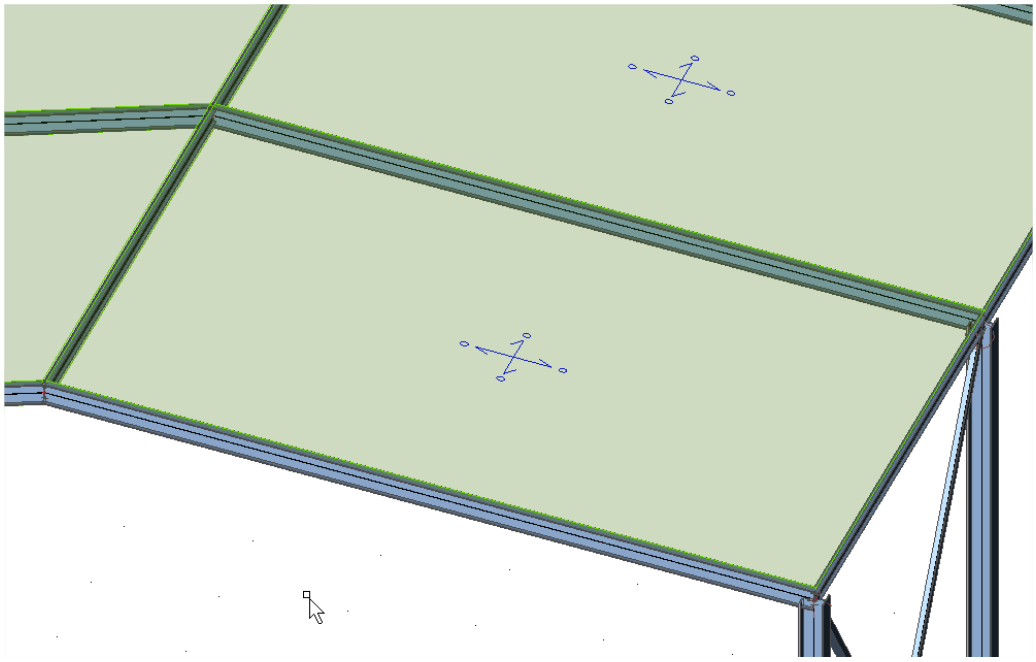


The parameters to be chosen:

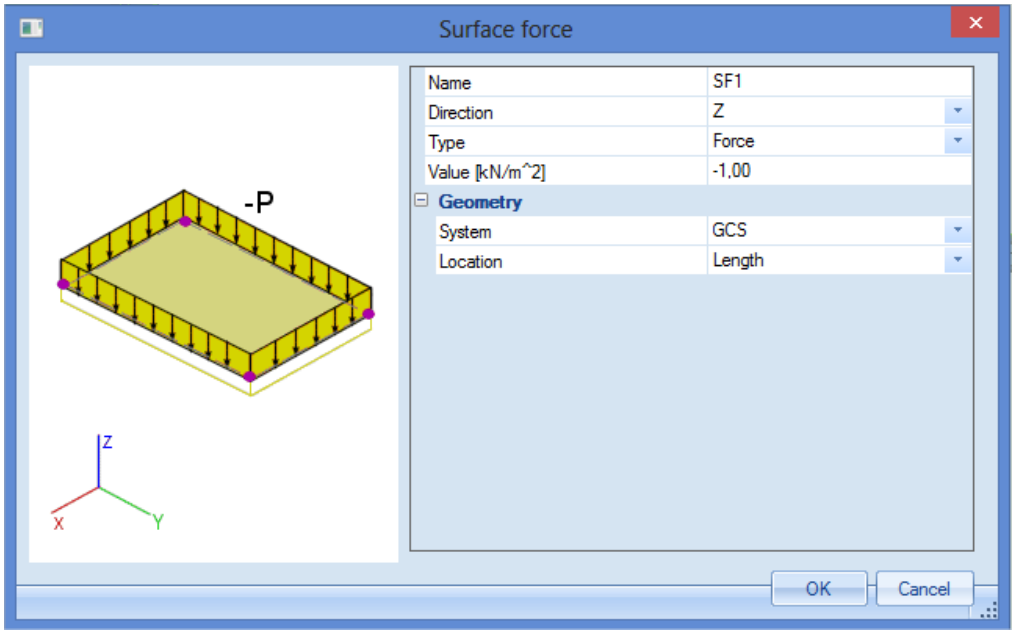
- Name: each element has a unique name.
- Panel type: the types are listed above.
- Load transfer direction: it can be chosen whether the loads should be distributed in one or all directions.
- LCS type: a load panel has its own local coordinate system. This system can be changed.
- Swap orientation: the direction of the local Z axis can be inverted.
- LCS Angle [deg]: the local axes can be rotated.

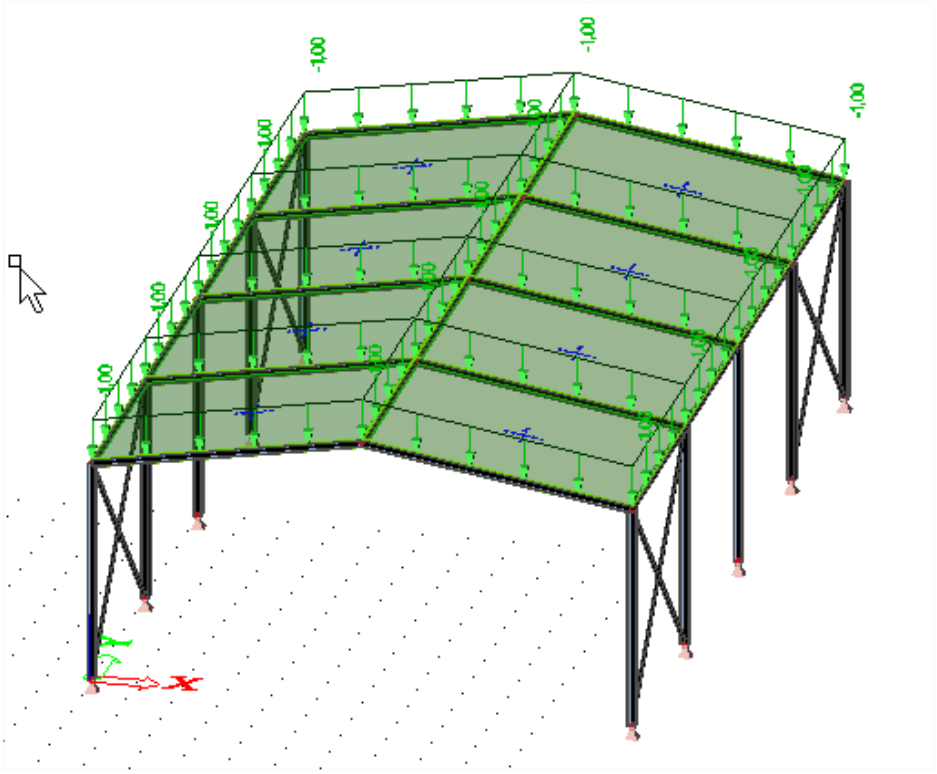
- Layer: a layer can be chosen for each element/panel.
- Selection of entities:
  - All: all nodes of the load panel will be loaded
  - User selection: the user can select the nodes of the load panel that should be loaded.

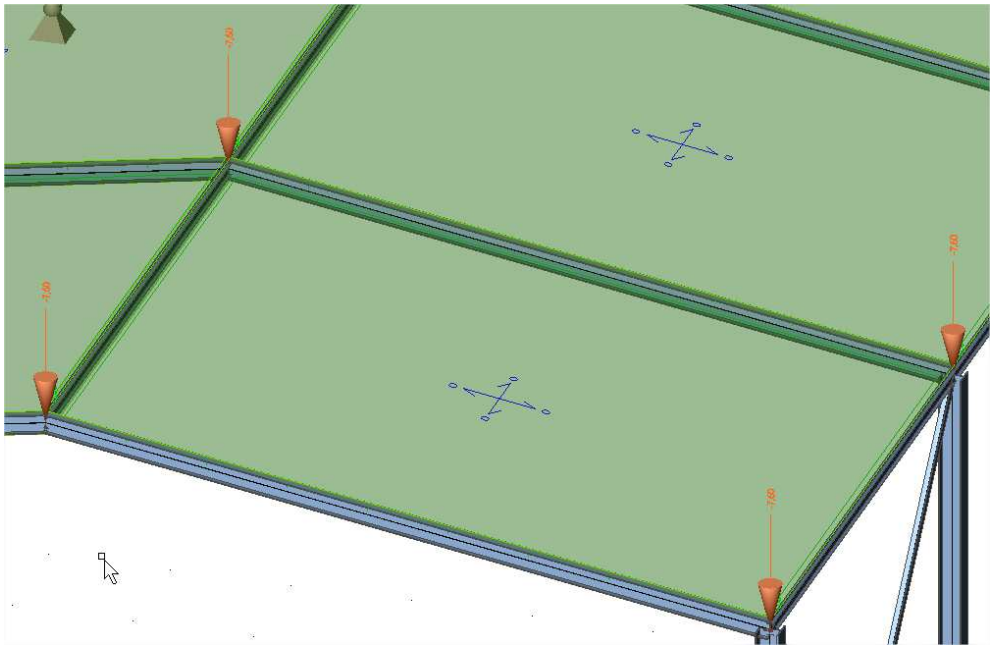
In this example, small load panels will be inserted on each part of the roof.



An area load of -1 kN/m² will be placed on each load panel. This can be done in the Load menu, using the option **Surface load, on 2D member**.



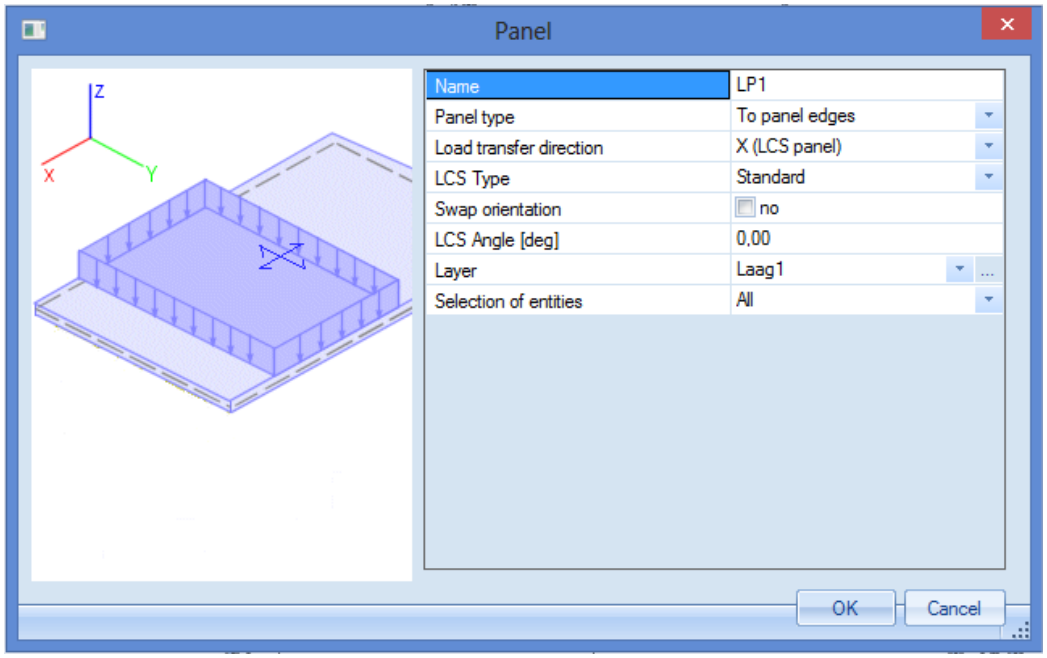

-1,00
-1,00
-1,00



Note
It is not necessary to select all the load panels and use the option Generate loads. The loads are automatically generated during the calculation of the project.

Load panel to edges

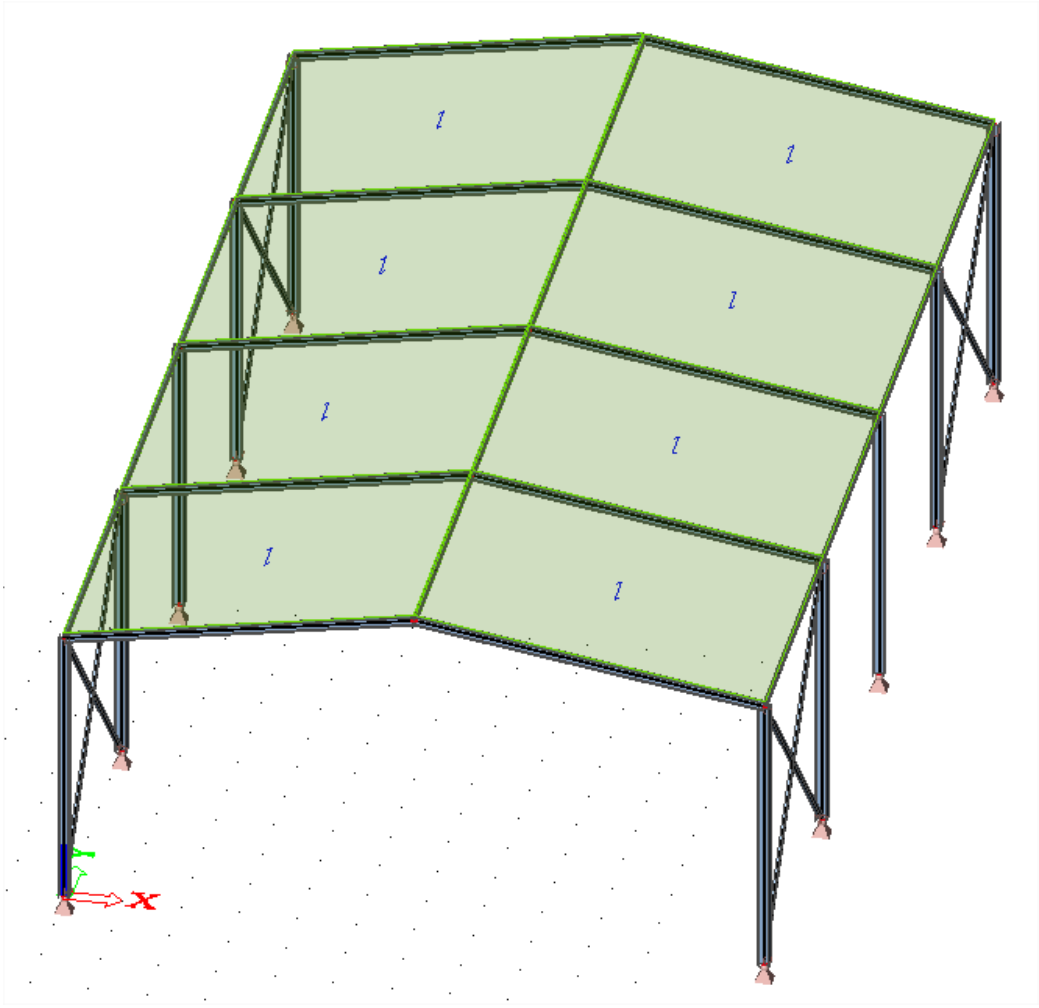
With this tool, the loads introduced on the panel, will be distributed to the edges of the load panel.



The parameters to be chosen are explained in the previous chapter (Load panel to nodes).

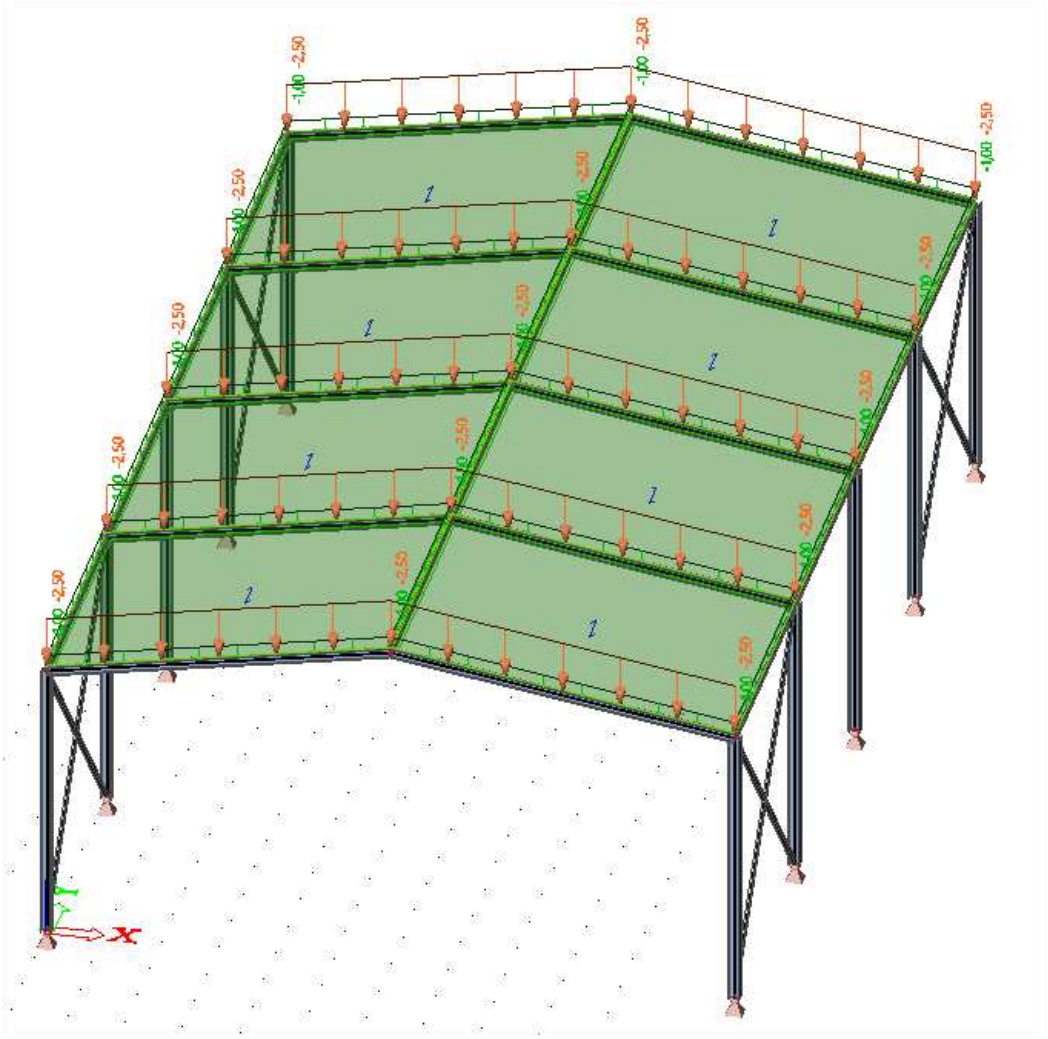
In this example, the **Load transfer direction** is set to **X (LCS panel)**. That way, the correct beams are loaded.

Analogously to the previous example, small load panels are inserted.

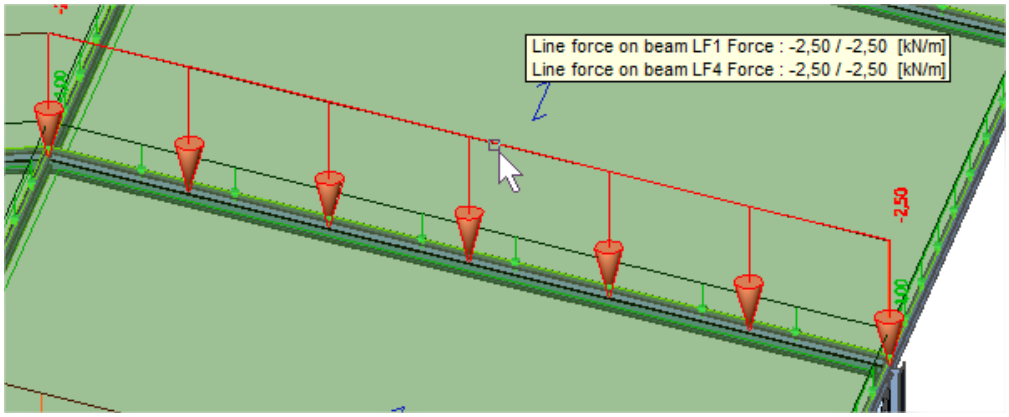


These load panels will be loaded with area loads of **-1 kN/m².** This can be done in the Load menu using the option **Surface load, on 2D member.**

Next, the area loads can be transformed into line loads on the beams by selecting the load panels and using the action **Generate loads**.



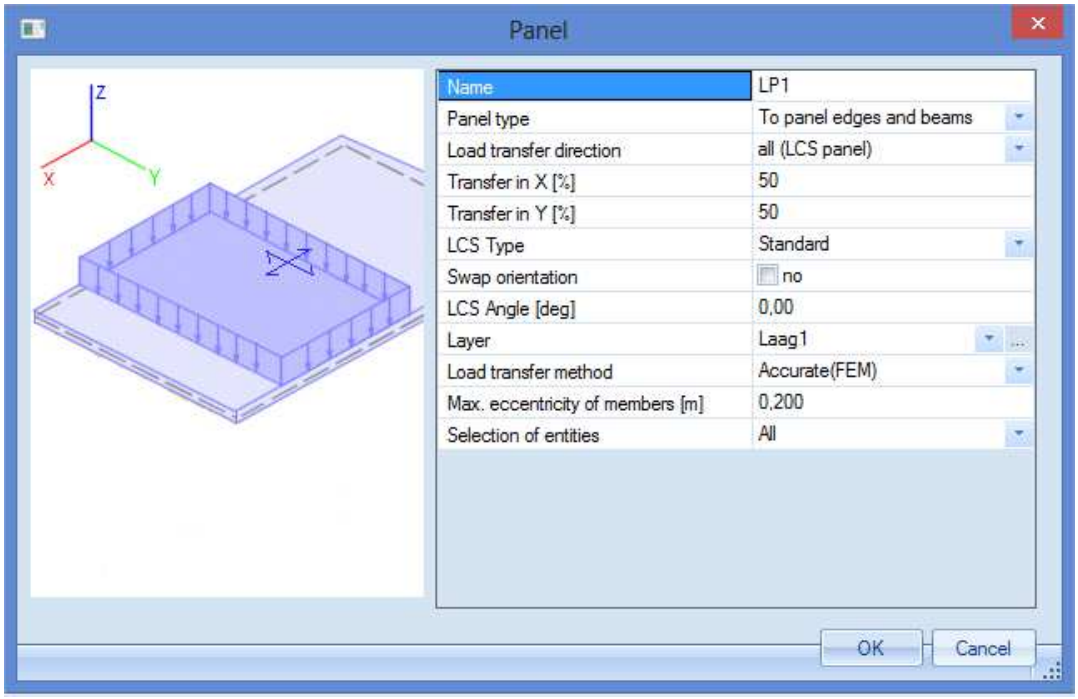
Since the beams in the middle of the roof have a load panel on each side, these beams are loaded twice with the same line load.



Note
It is not necessary to select all the load panels and use the option Generate loads. The loads are automatically generated during the calculation of the project.

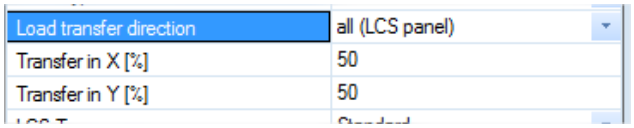
Panel to edges and beams

With this tool, the loads introduced on the panel, will be distributed to the edges of the load panel and to the supporting beams of the panel.

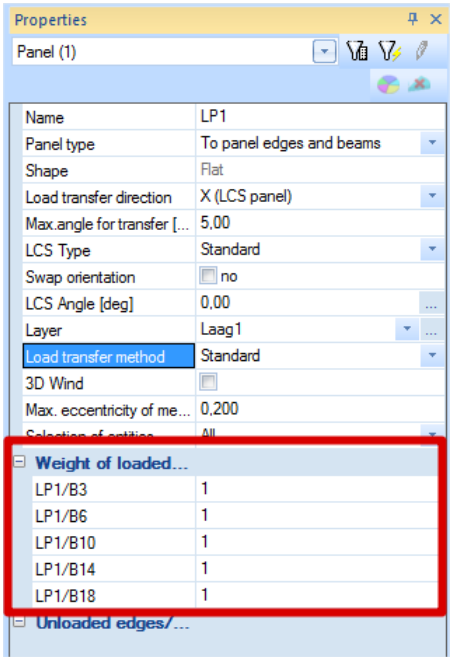


Most of the parameters to be chosen are already explained in previous chapters. The parameters specific for this type of load panel will be discussed:

- Load transfer direction:
  - X direction of the panel LCS
  - Y direction of the panel LCS
  - All directions of the panel LCS. When this option is chosen, a percentage has to be inserted for both directions.



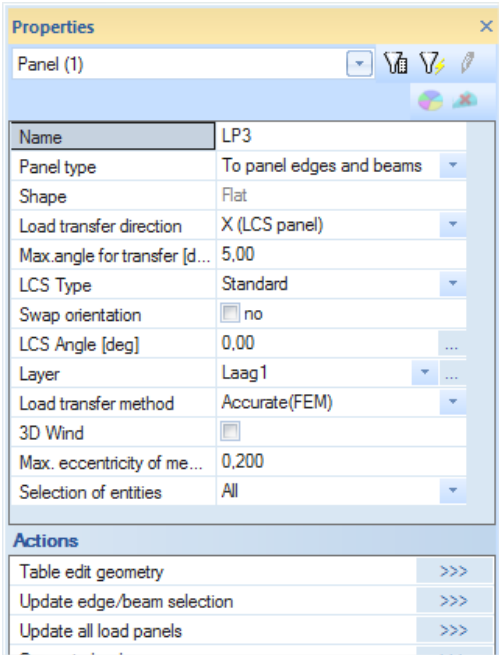
  - If one direction is chosen (X or Y of the panel LCS), there is an extra option, namely **Max. Angle for transfer [deg]**. If this angle is set to zero, only beams that are perpendicular to the chosen transfer direction can be loaded.
- Load transfer method:
  - Standard: all beams are loaded equally loaded. The user have to give in the load factors.

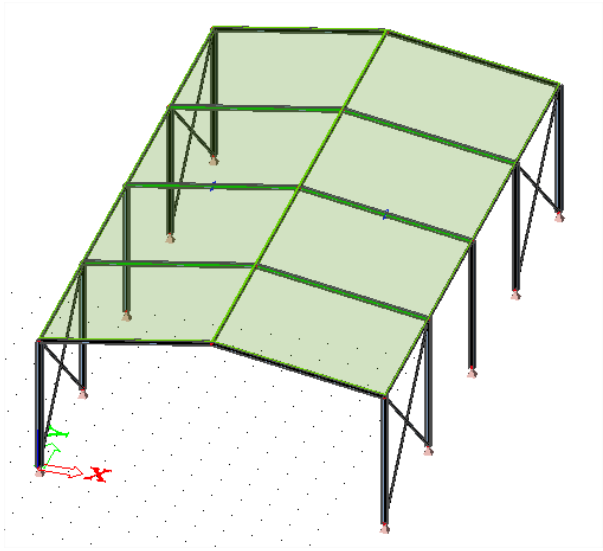


- o Accurate (FEM): the finite element calculation is used for the distribution of the area load to line loads on the edges/beams.

- Max. Eccentricity of members [m]: also the members which have a position between this eccentricity above or beneath the plane of the load panel, will be loaded.

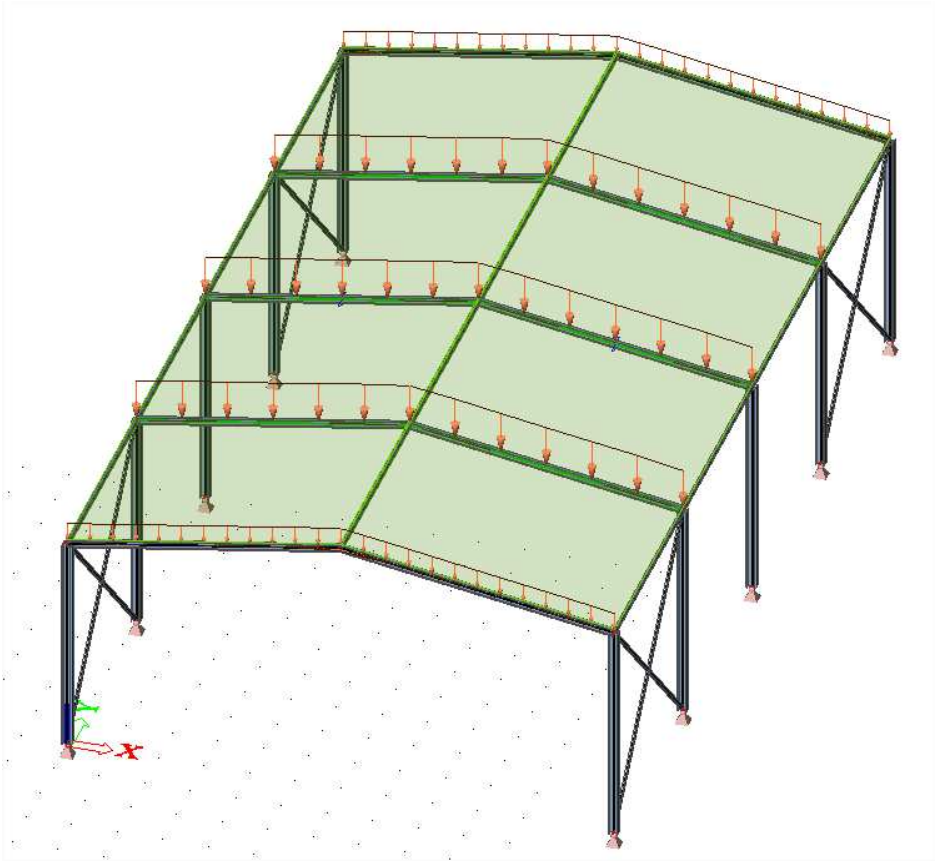
In this example 2 load panels will be inserted with the following properties:





These load panels will be loaded with an area load of **-1 kN/m²**. The loads can be inserted in the Load menu with the option **Surface load, on 2D member.**

Afterwards, the load panels can be selected. With the action **Generate loads**, the area loads will be recalculated into line loads on the beams.

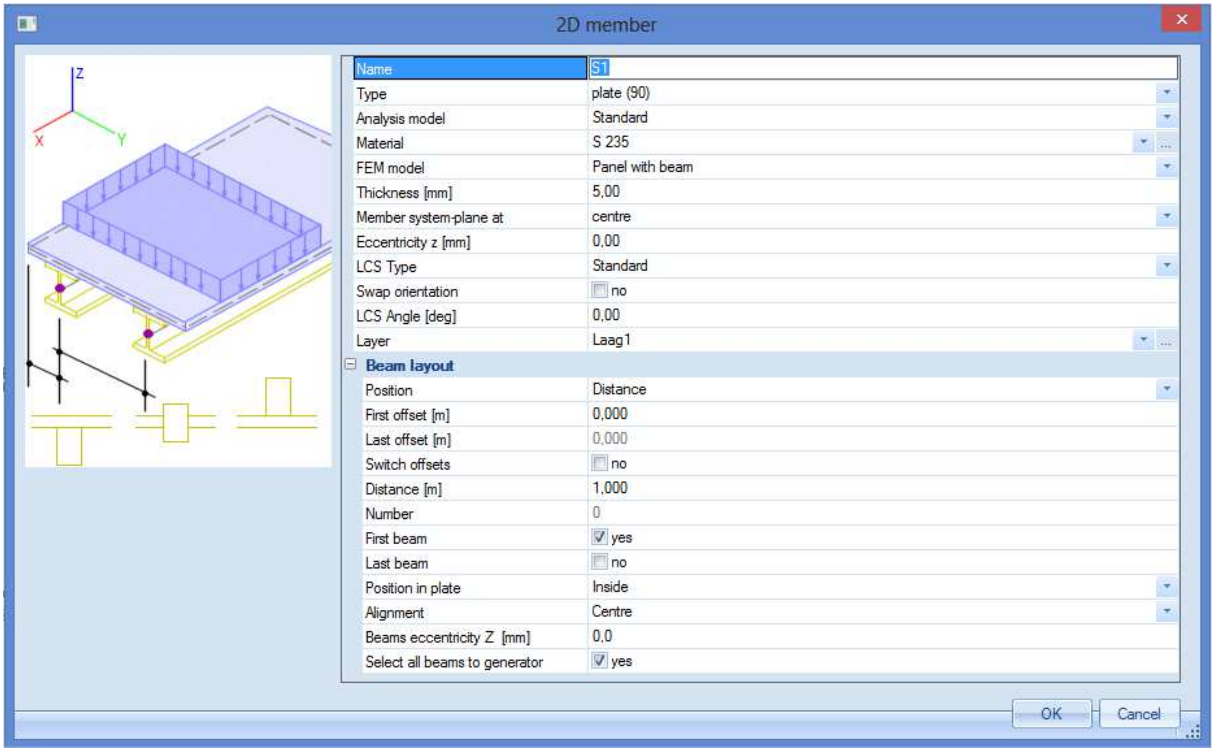


Since the **Accurate (FEM)** load transfer method is used, the loads are generated correctly without the need for the user to insert the load factors.

Panel with parallel beams

With this option, a load panel can be inserted directly with supporting beams. The **beams** are placed in the **X direction** of the panel LCS, and the **load transfer** is done in the **Y direction** of the panel LCS.

A load panel with parallel beams will be inserted with the following parameters:



Most of the parameters to be chosen are already explained in previous chapters. The parameters specific for this type of load panel will be discussed:

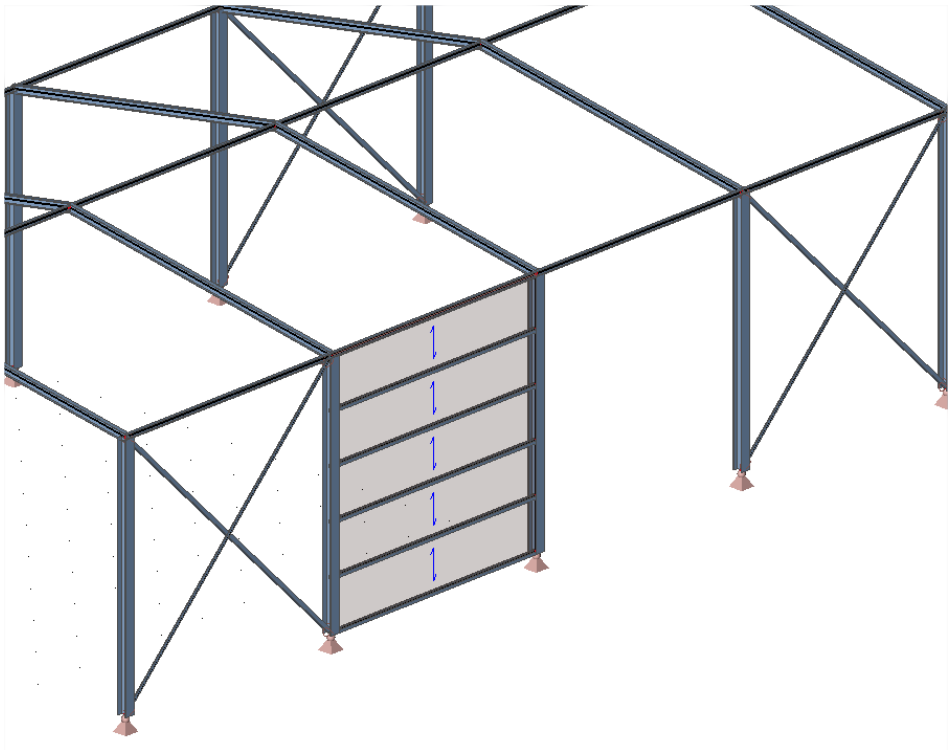
- Type: specifies the type of the plate (plate, wall or shell). The chosen type has for instance an influence on the check according to the code (future development).
- Analysis model: only Standard is available.
- Material: a material can be chosen for the calculation of the self weight of the panel/plate (future development).
- FEM model: only Panel with beams is available
- Thickness [mm]: the thickness of the plate has to be given. This thickness has an influence on:
    - The eccentricity of the beams
    - The self weight of the panel (future development)
- Member system-plane at: center, top or bottom.
- Eccentricity z [mm]: if needed, an extra eccentricity can be inserted.

And for the Beam layout:

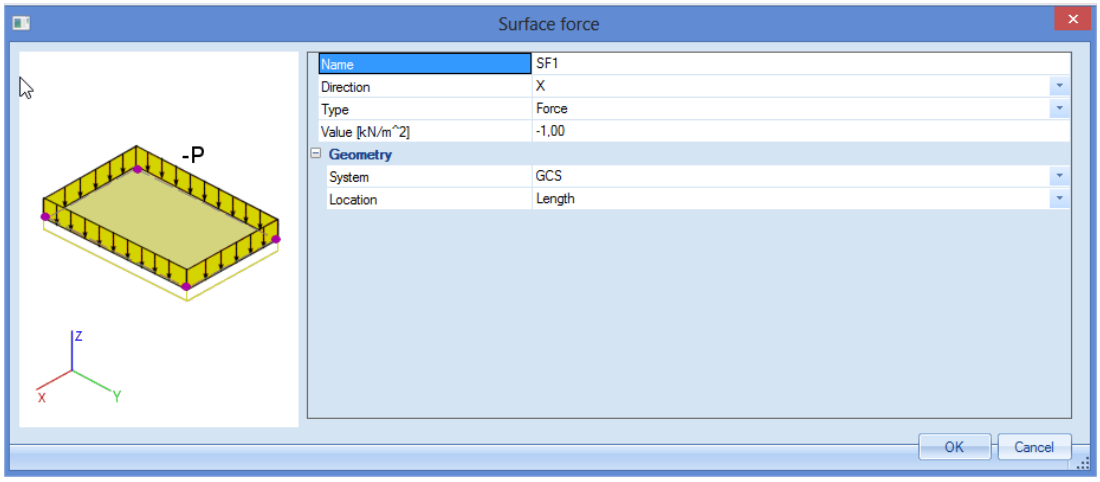
- Position: the position of the beams can be defined in several ways.
    - Distance: distance between the beams
    - Number: the amount of beams
    - Width: the part of the panel that has to be carried by the beams
    - Generic: the distance/width does not have to be same for all beams
- First/Last offset [m]: the offset of the first/last beam.
- Switch offsets: to switch between the possibility to give in the first or last offset.
- Distance/Width/Number: the distance between the beams, the part of the panel that has to be carried by the beams or the amount of beams.

- First/Last beam: the option to use/not use the first/last beam.
- Position in plate: specifies the position of the beam in the panel.
- Alignment:
    - Bottom: the beams are attached to the lower surface of the panel.
    - Centre: the axis of the beam is located at the same level as the center plane of the panel.
    - Top: the beams are attached to the upper surface of the panel.
- Beams eccentricity Z [mm]: defines the eccentricity of the beams.
- Select all beams to generator: all the beams will be loaded by the generated loads.

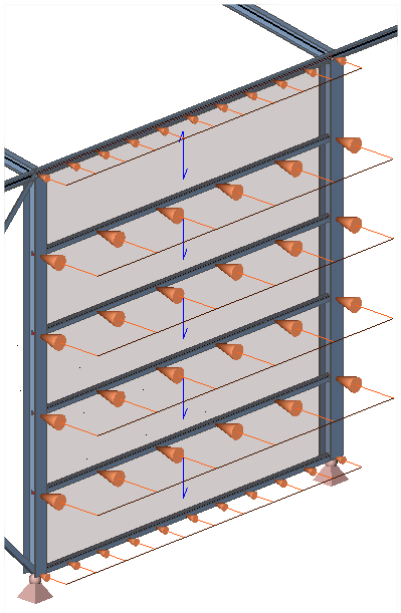
In this example, one load panel with parallel beams will be inserted to show principle. The parameters can are viewed on the previous page. After modeling this panel, the parameters for the beams are asked for. In this example the IPE 100 profile is chosen.



This load panel will be loaded with a surface load of -0,30 kN/m² in the global X direction (load case – X):



Afterwards, the panel can be selected and by clicking on the action button **Load generation**, the surface load will be distributed to the beams:



Free loads

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

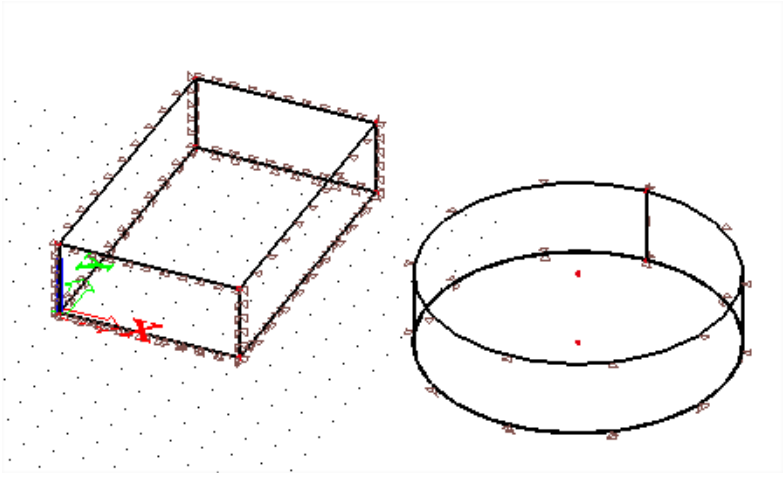
Definition of free loads is composed of their geometry, which is independent on geometry of structural members, direction of load effect and a list of 2D members which are influenced by the free loads. Free loads are in fact easy load generators. A free load differs from a 'regular load' by the fact that it is NOT attributed as an additional data to a specific 2D member.

Since free loads do not have to be inserted on the entities itself, multiple loads can be generated using only one free load.

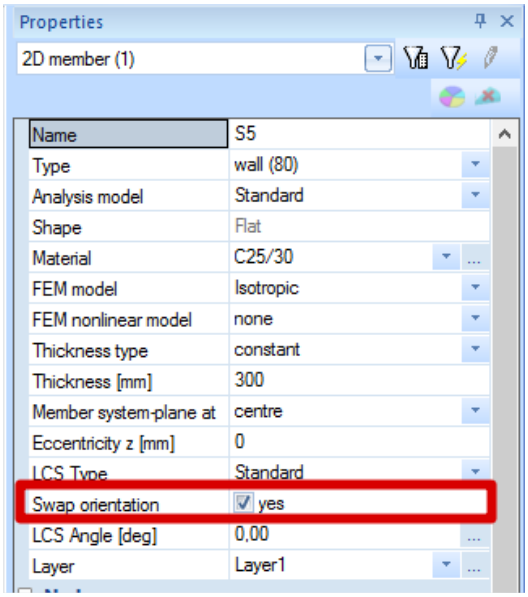
This is illustrated with an example of a swimming pool.

Example: Free Loads.esa

In this example, two swimming pools are modeled (rectangular and circular).
Both structures are supported by surface supports.



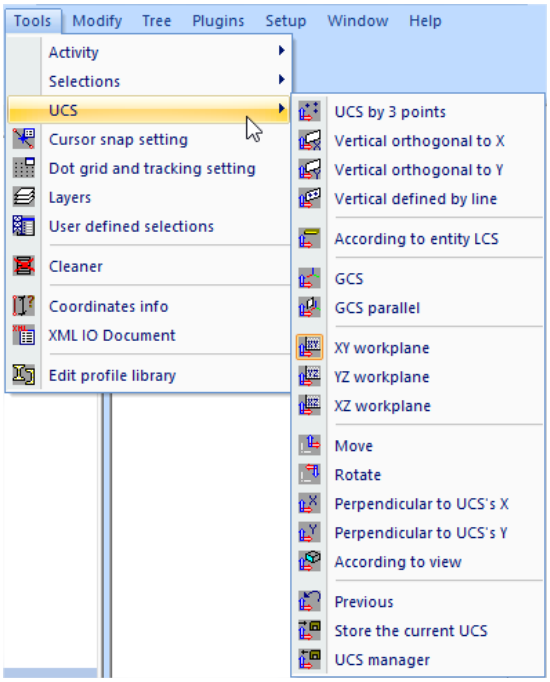
Surface supports are always placed on the negative site of the local Z axis of the 2D element.
For that reason, the positive direction of the local Z axis for each wall element should be pointing towards the inside of the pool. If this is not the case, the option **Swap Orientation** can be used in the property window of the 2D element.



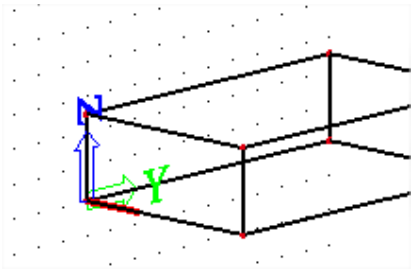
There are two load cases:
Self Weigth (Permanent)
Water pressure (Variable)

After the load cases are defined, the water pressure can be inserted in the Load menu, using a **Surface load, free**.

A free surface load can only be inserted in the working plane of the UCS (User Coordinate System). For that reason, it can be necessary to change the UCS. This can be done through **Tools > UCS**.

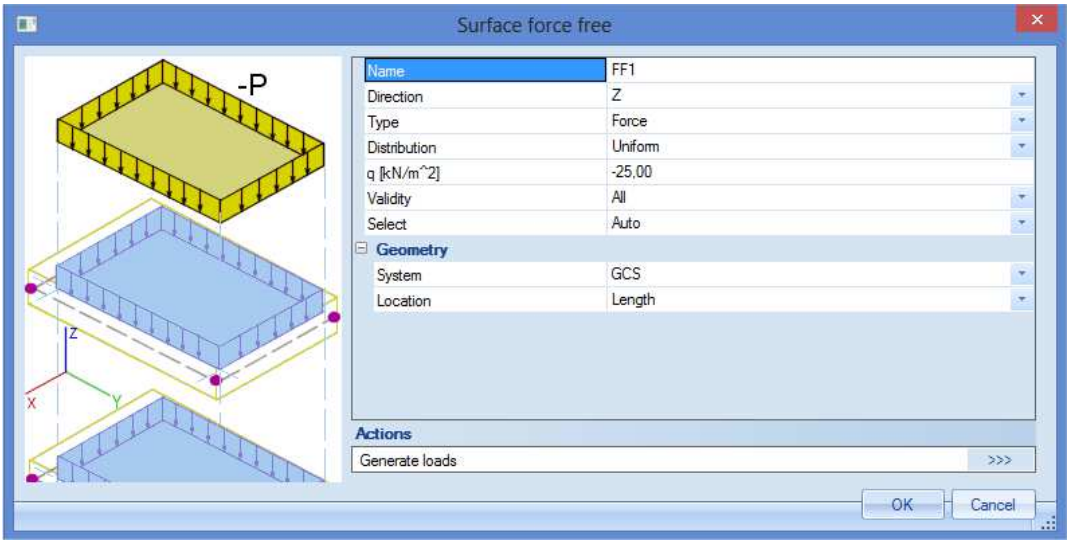


In this example, the **YZ workplane** is chosen, to be able to insert the water pressure on the walls of the swimming pools.



It is the intention to load all walls (4 walls of the rectangular pool and 1 wall of the circular pool), by only introducing one free surface load. To be able to do this, the Z axis of the LCS of all walls, needs to be pointing to the inside of the pool. In this project, this is already done by changing the LCS of 2 walls by inserting the surface supports.

Next the **Free surface load** can be inserted.

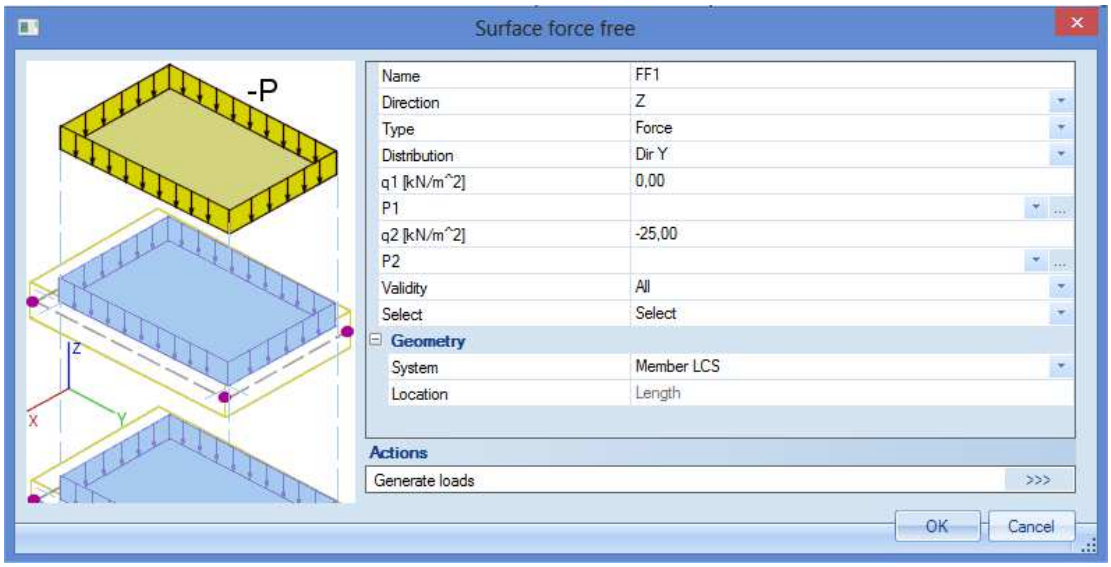


Explanation of all parameters:

- Name: each element in SCIA Engineer has a unique name, also the loads.
- Direction: the direction in which the loads have to be generated.
- Type: this load will be introduced as o force. In some cases also self weight can be chosen.
- Distribution:
    - 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.
- q [kN/m²]: the value of the load in kN/m² (more values if there is a variable course).
- Validity:
  Every free surface load has its own member LCS. Which elements should be loaded by the generated loads, depends on the validity. This validity applies to the Z axis of the free surface load LCS.
    - All: all 2D members on both sides of the free surface load will be loaded with the generated loads.

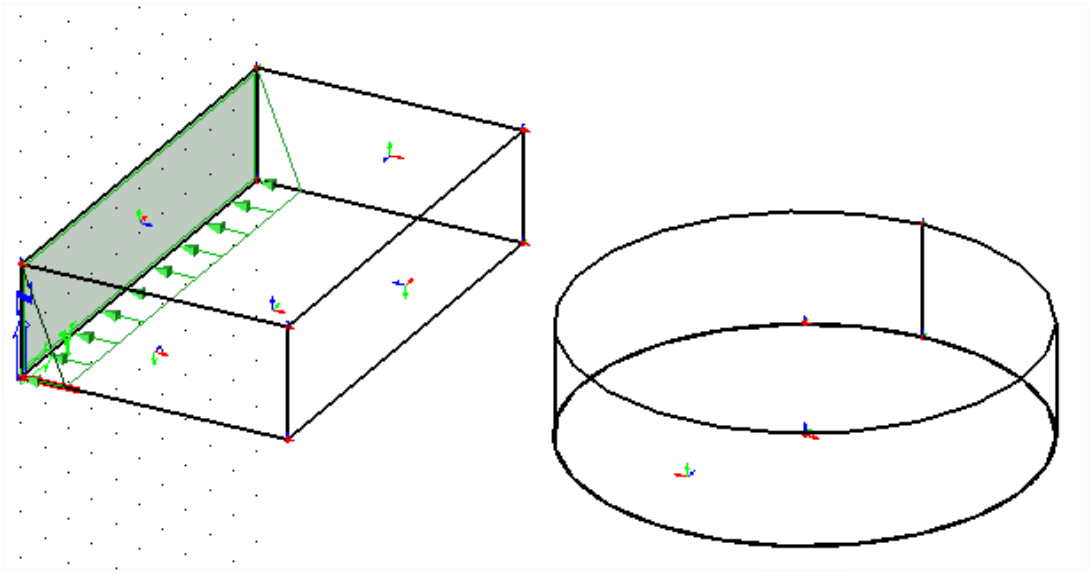
  - –Z: only the elements situated *under* the free load (situated in the half-space defined by the negative Z direction of the free load LCS), can be loaded.
  - –Z (incl. 0): only the elements situated *under and in the plane* of the free load (situated in the half-space defined by the negative Z direction of the free load LCS), can be loaded.
  - Z = 0: only the elements situated in the plane of the free surface load can be loaded.
  - +Z: only the elements situated *above* the free load (situated in the half-space defined by the positive Z direction of the free load LCS), can be loaded.
  - +Z (incl. 0): only the elements situated *above and in the plane* of the free load (situated in the half-space defined by the positive Z direction of the free load LCS), can be loaded.
  - From-to: only the elements within a given interval can be loaded.
- Select:
  - Auto: all the elements, which correspondent with the validity, will be loaded.
  - Select: the user can select the elements, which correspondent with the validity, to be loaded.
- System:
  - 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.
- Location: length or projection (= only possible when System GCS is used).

In this example, the following parameters are used:



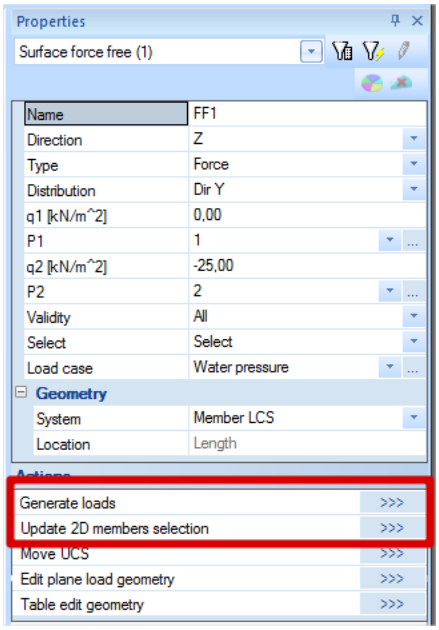
Since the value of the loads has to be 0 at the top and -25 at the bottom, it is necessary to use this order of nodes by drawing the free surface load:

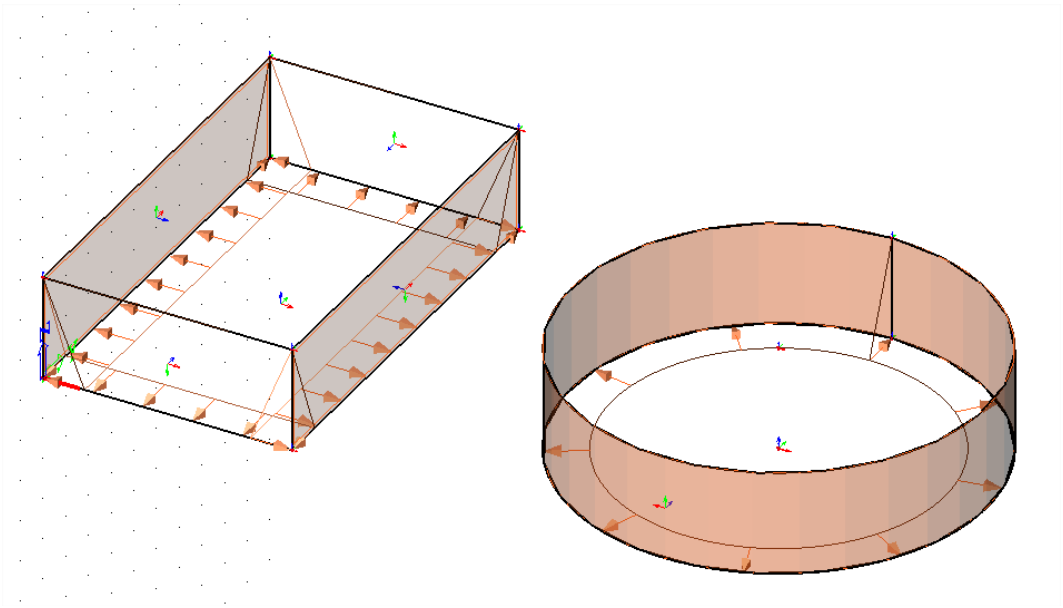
First point, a node at the top of the swimming pool.
Second point, a node at the bottom of the swimming pool.



Since all walls need to be loaded by this water pressure, the walls can be selected using the action **Update 2D member selection** in the property window of the free surface load. To end the selection functionality, click **Esc**.

Next, the action button **Generate loads** can be used to generated the loads on all selected 2D elements.

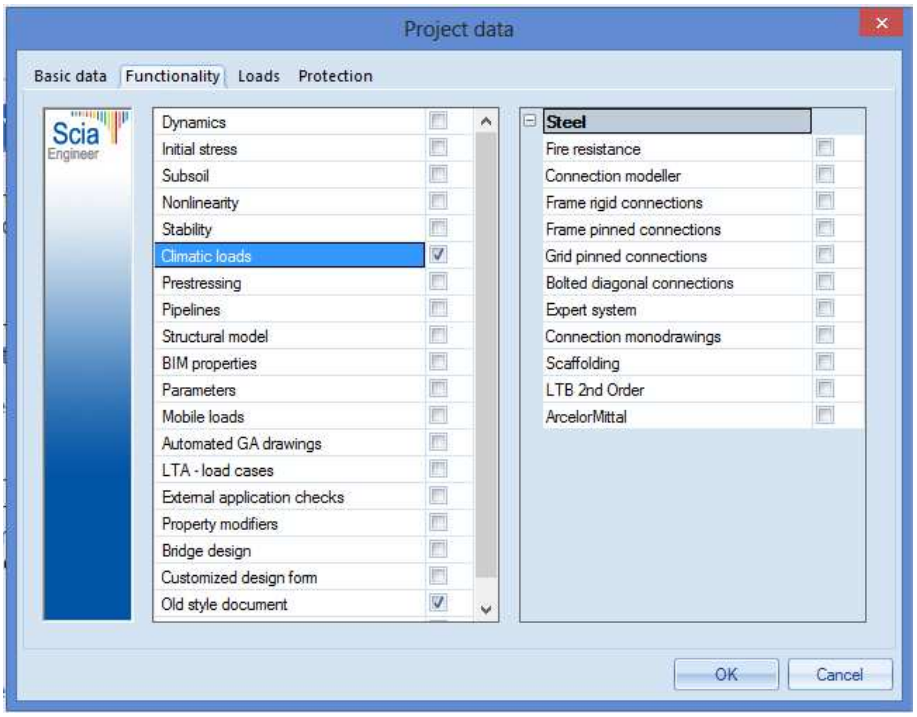




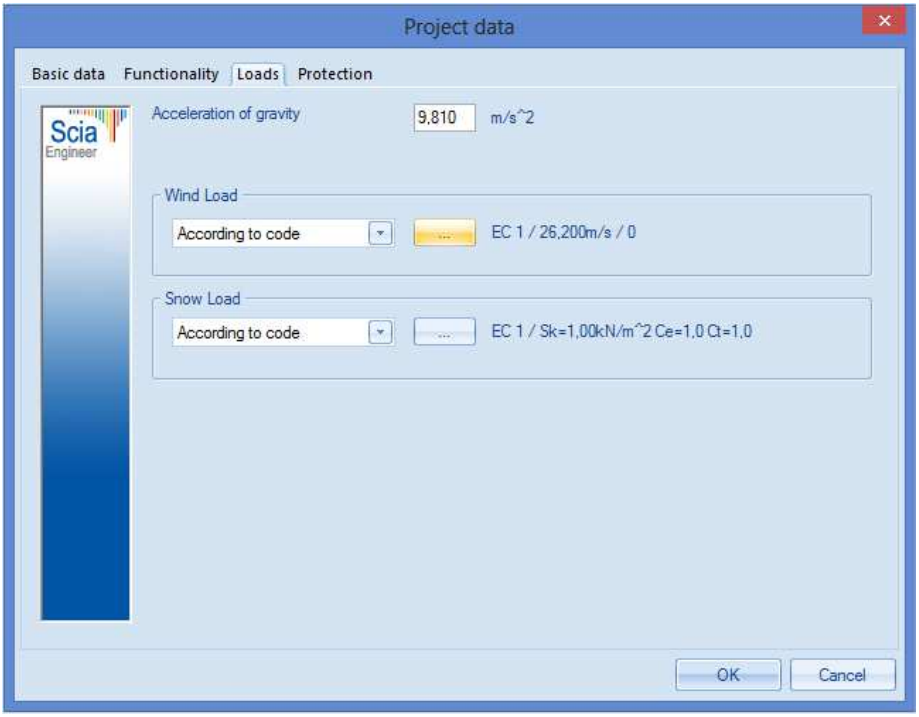
Now all walls are loaded by the water pressure, by inserting only one free surface load.

2D Wind- and snow generators

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

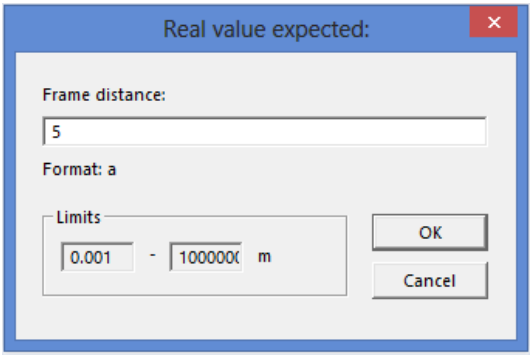
Wind generator

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

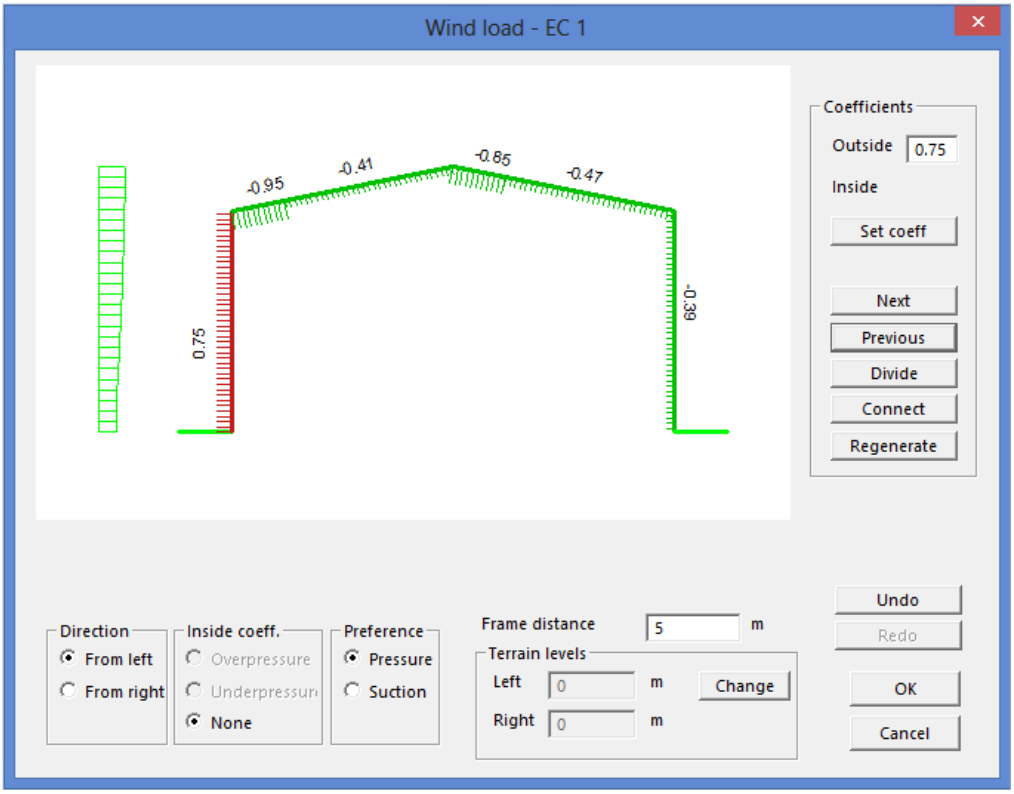
Load cases:
Self Weight (Permanent)
Wind Left (Variable, load group exclusive)
Wind Right (Variable, load group exclusive)

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

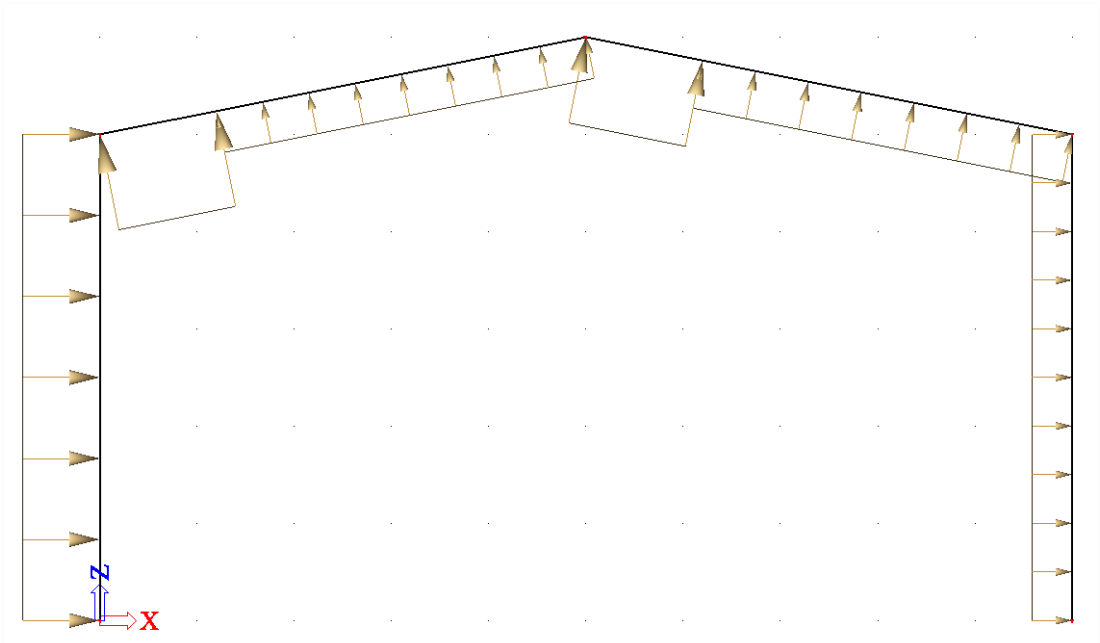
By using this wind generator, a frame distance has to be inserted. This has to be done to simulate the wind on a 2D frame as if the wind would be on a complete 3D structure.



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:

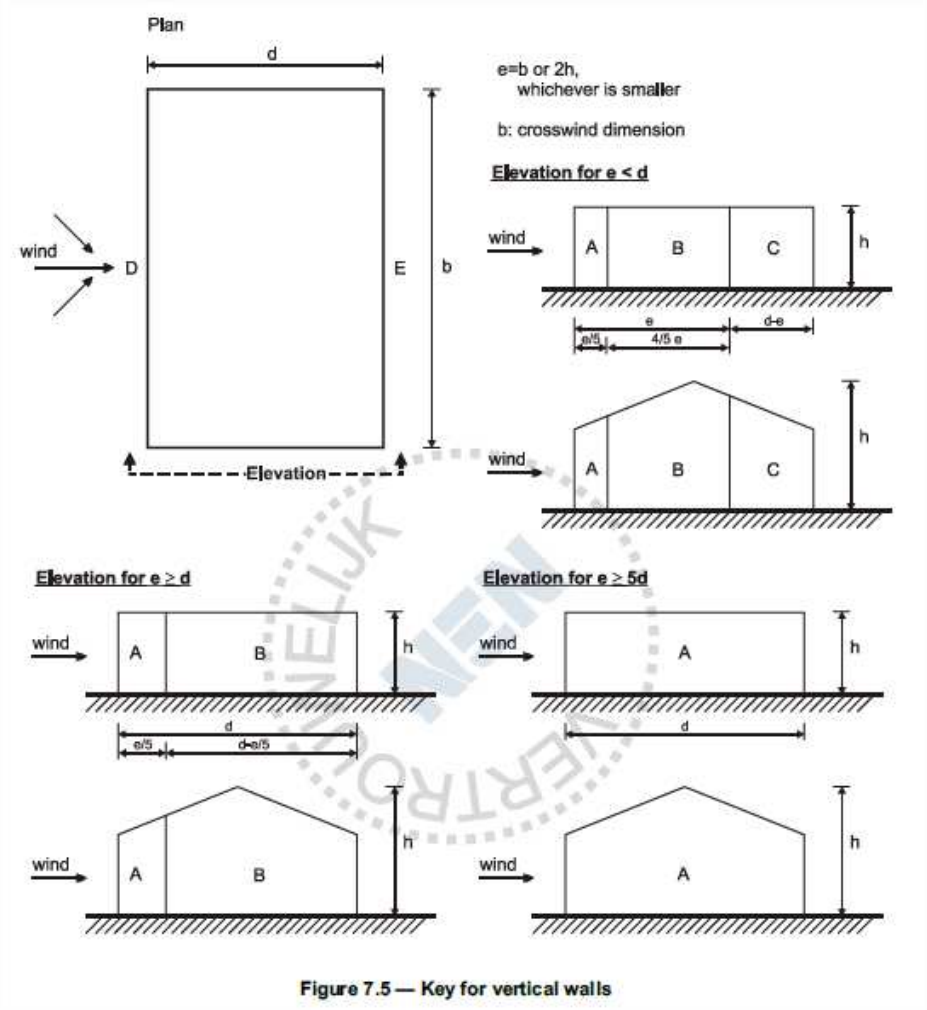


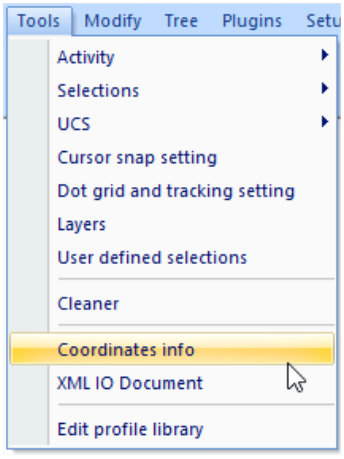
Figure 7.5 — Key for vertical walls

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:

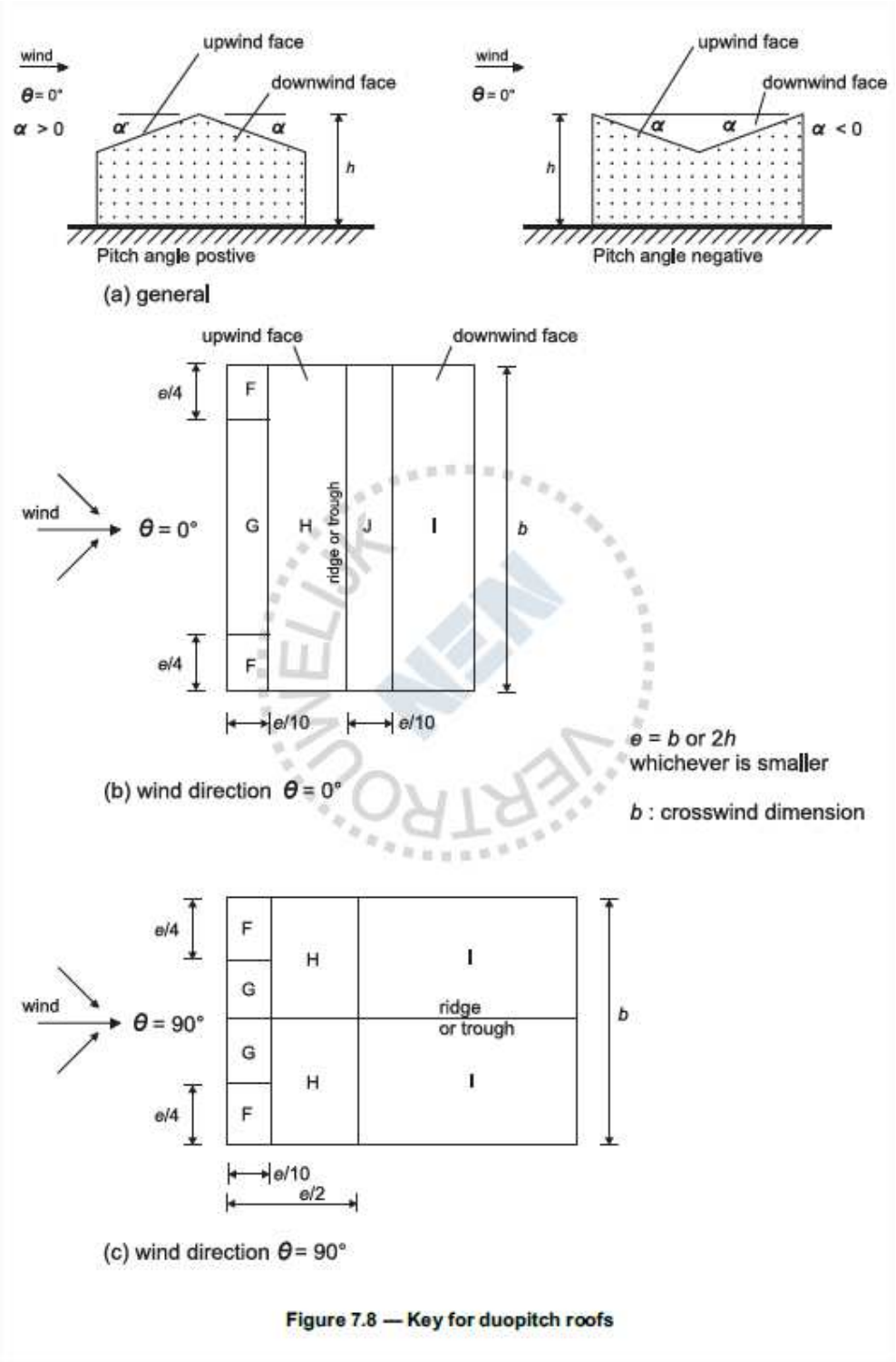
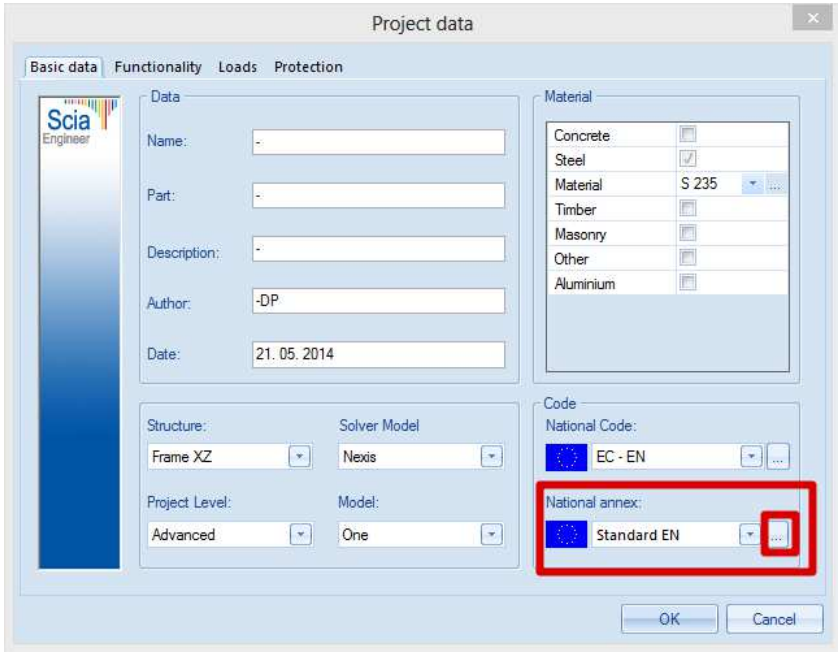
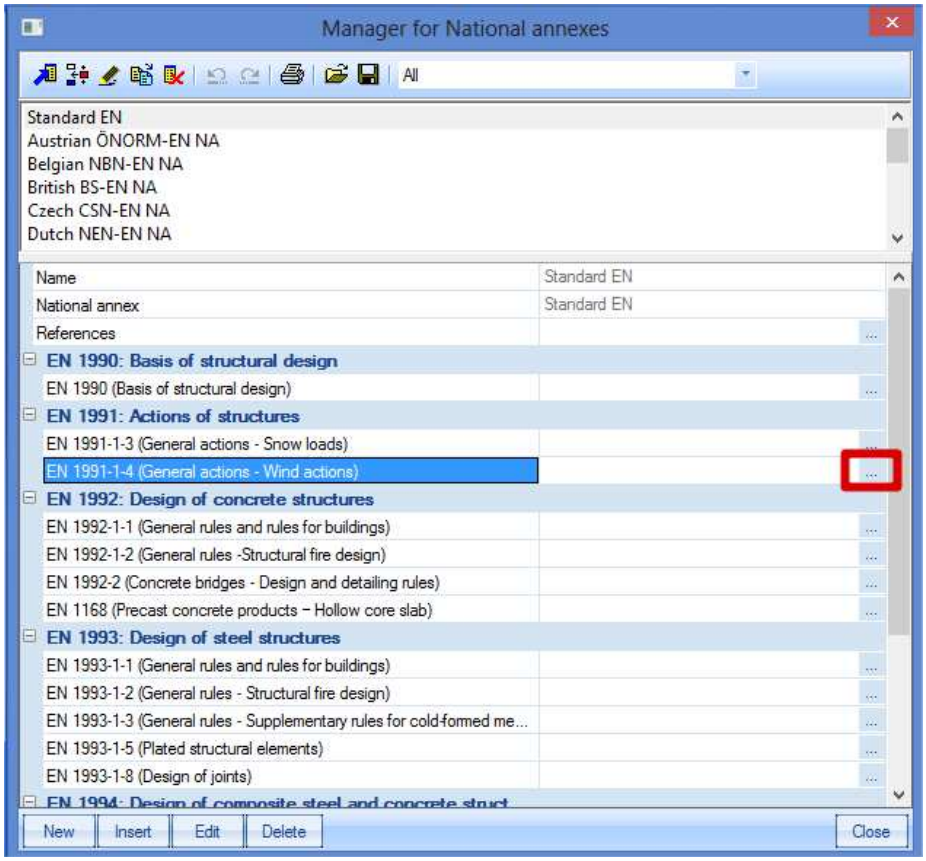


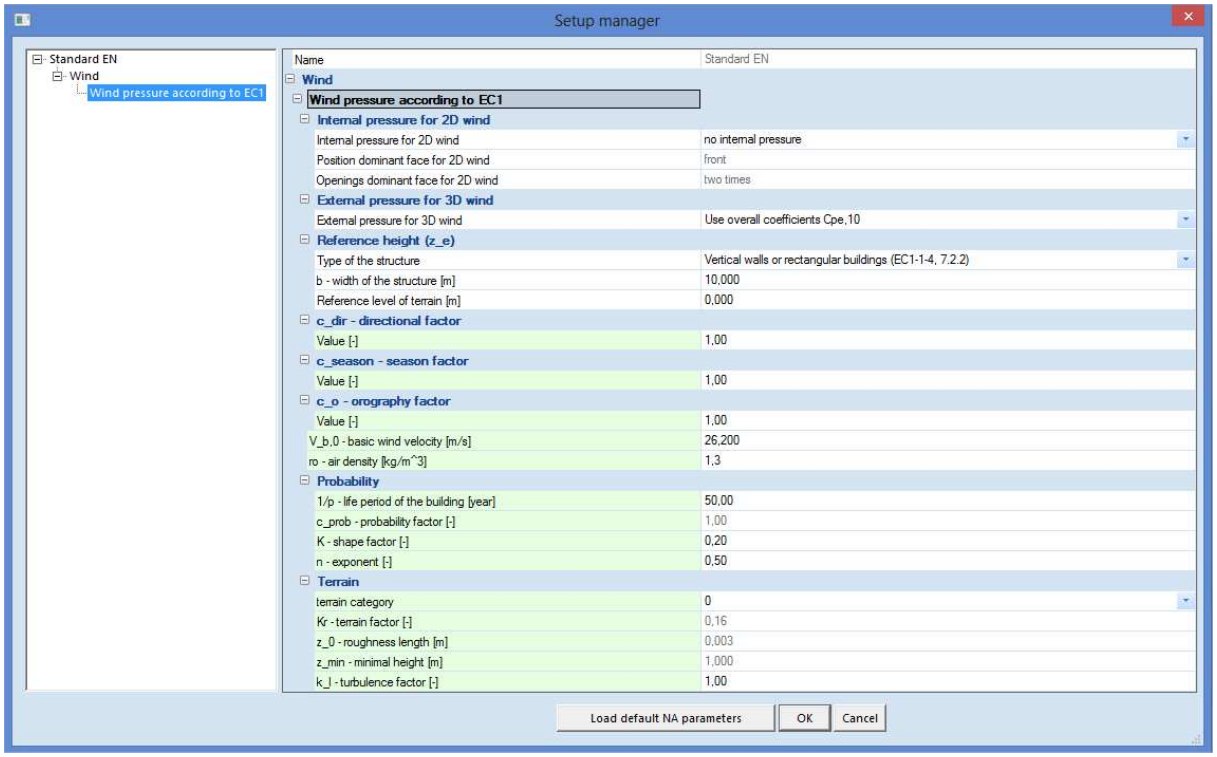
Figure 7.8 — Key for duopitch roofs

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:


Project data
Basic data
Functionality
Loads
Protection
Scia Engineer
Data
Name:
-
Part:
-
Description:
-
Author:
-DP
Date:
21. 05. 2014
Material
Concrete
Steel
Material
S 235
Timber
Masonry
Other
Aluminium
Structure:
Frame XZ
Solver Model
Nexis
Project Level:
Advanced
Model:
One
Code
National Code:
EC - EN
National annex:
Standard EN
OK
Cancel


Manager for National annexes
All
Standard EN
Austrian ÖNORM-EN NA
Belgian NBN-EN NA
British BS-EN NA
Czech CSN-EN NA
Dutch NEN-EN NA
Name
Standard EN
National annex
Standard EN
References
EN 1990: Basis of structural design
EN 1990 (Basis of structural design)
EN 1991: Actions of structures
EN 1991-1-3 (General actions - Snow loads)
EN 1991-1-4 (General actions - Wind actions)
EN 1992: Design of concrete structures
EN 1992-1-1 (General rules and rules for buildings)
EN 1992-1-2 (General rules -Structural fire design)
EN 1992-2 (Concrete bridges - Design and detailing rules)
EN 1168 (Precast concrete products – Hollow core slab)
EN 1993: Design of steel structures
EN 1993-1-1 (General rules and rules for buildings)
EN 1993-1-2 (General rules - Structural fire design)
EN 1993-1-3 (General rules - Supplementary rules for cold-formed me...
EN 1993-1-5 (Plated structural elements)
EN 1993-1-8 (Design of joints)
New
Insert
Edit
Delete
Close



Snow generator

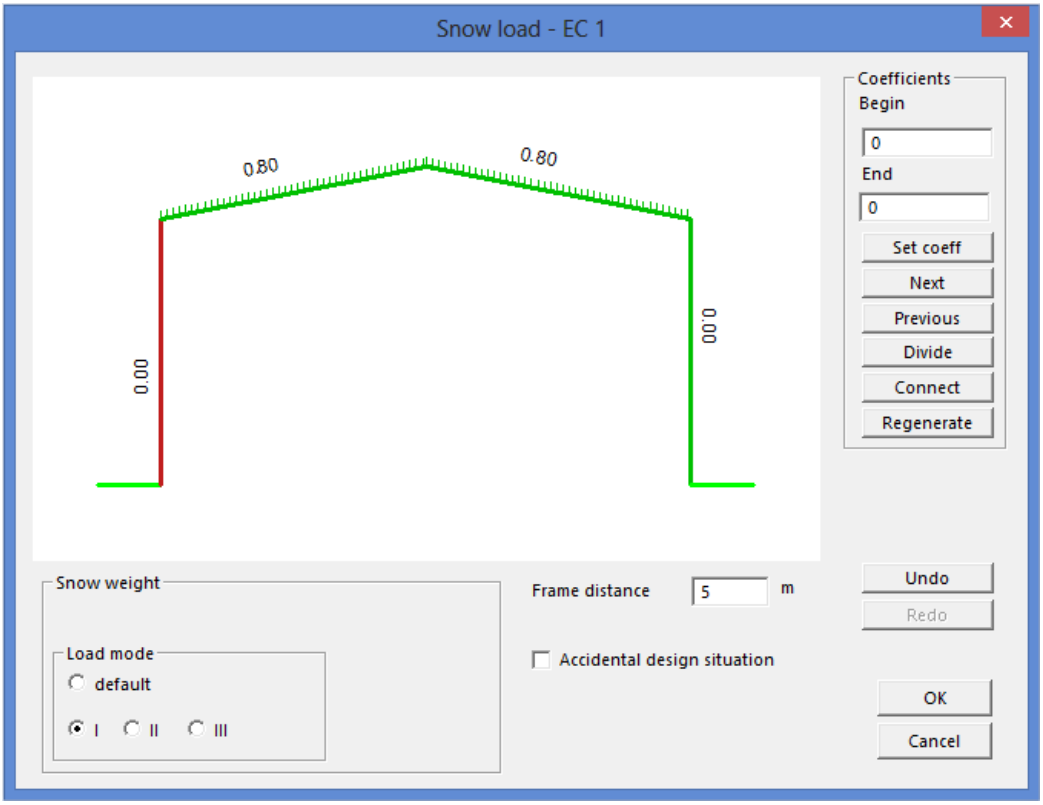
After generating the wind load in both directions, also the snow loads will be generated in this example. For the snow loads, three load cases have to be created.

Extra load cases:

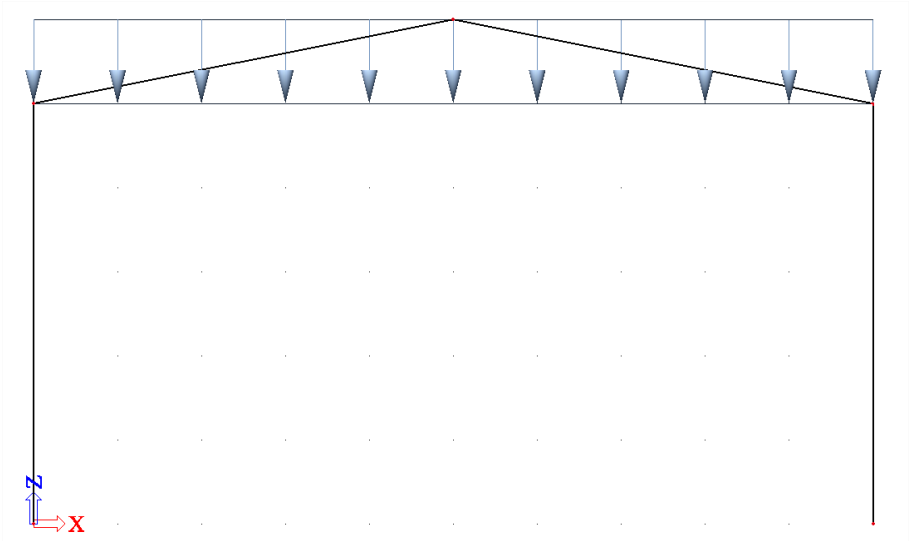
- Snow load 1 (Variable, exclusive)
- Snow load 2 (Variable, exclusive)
- Snow load 3 (Variable, exclusive)

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	$0{,}8(60 - \alpha)/30$	0,0
μ_2	$0{,}8 + 0{,}8\ \alpha/30$	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].

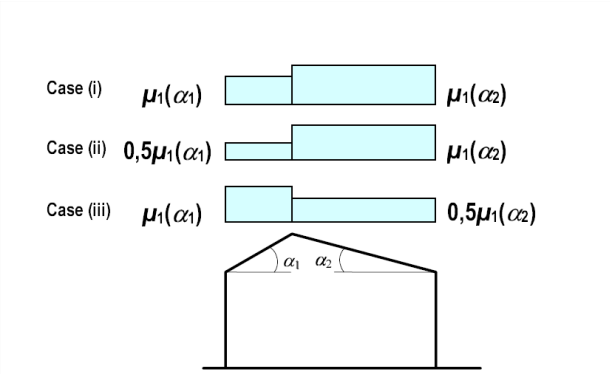
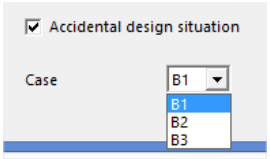


Figure 5.3: Snow load shape coefficients - pitched roofs

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 $\mu_i C_e C_t s_k$ [2] drifted $\mu_i C_e C_t s_k$	*Persistent/transient design situation* [1] undrifted $\mu_i C_e C_t s_k$ [2] drifted $\mu_i C_e C_t s_k$ *Accidental design situation (where snow is the accidental action)* [3] undrifted $\mu_i C_e C_t C_{esl} s_k$ [4] drifted $\mu_i C_e C_t C_{esl} s_k$	*Persistent/transient design situation* [1] undrifted $\mu_i C_e C_t s_k$ [2] drifted $\mu_i C_e C_t s_k$ (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 $\mu_i C_e C_t s_k$ [2] drifted $\mu_i C_e C_t s_k$ (except for roof shapes in AnnexB) *Accidental design situation (where snow is the accidental action)* [3] undrifted $\mu_i C_e C_t C_{esl} s_k$ [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.

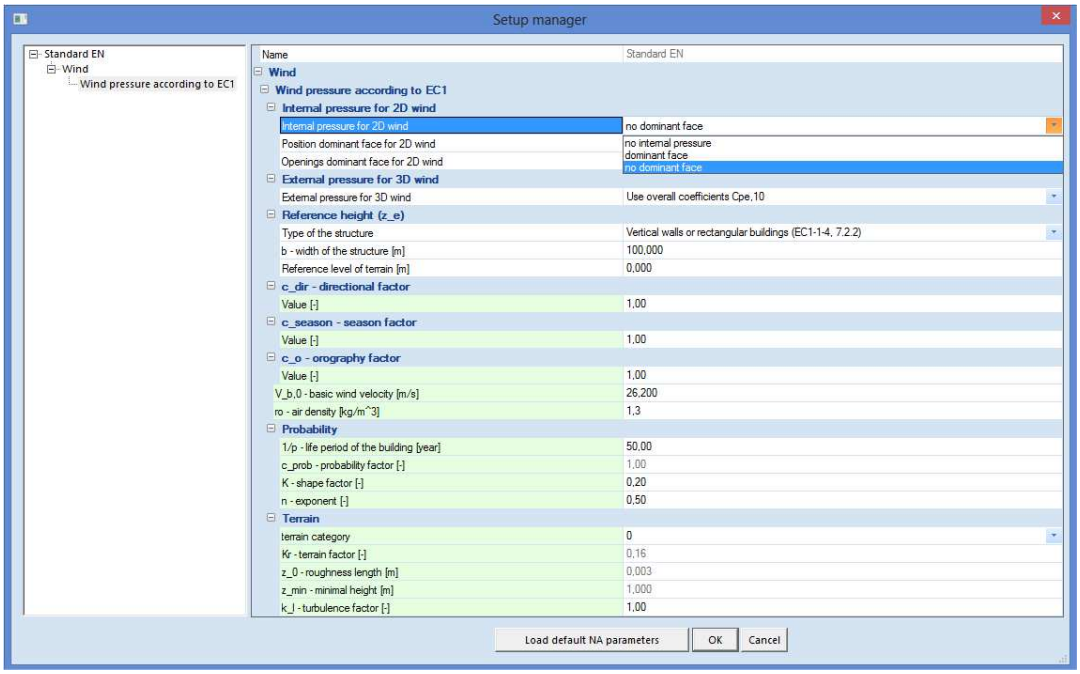
Wind & Snow generator

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**.


Setup manager
Standard EN
Wind
Wind pressure according to EC1
Name
Standard EN
Wind
Wind pressure according to EC1
Internal pressure for 2D wind
Internal pressure for 2D wind
no dominant face
Position dominant face for 2D wind
Openings dominant face for 2D wind
no internal pressure
dominant face
no dominant face
External pressure for 3D wind
External pressure for 3D wind
Use overall coefficients Cpe,10
Reference height (z_e)
Type of the structure
Vertical walls or rectangular buildings (EC1-1-4, 7.2.2)
b - width of the structure [m]
100,000
Reference level of terrain [m]
0,000
c_dir - directional factor
Value [-]
1,00
c_season - season factor
Value [-]
1,00
c_o - orography factor
Value [-]
1,00
V_b,0 - basic wind velocity [m/s]
26,200
ro - air density [kg/m^3]
1,3
Probability
1/p - life period of the building [year]
50,00
c_prob - probability factor [-]
1,00
K - shape factor [-]
0,20
n - exponent [-]
0,50
Terrain
terrain category
0
Kr - terrain factor [-]
0,16
z_0 - roughness length [m]
0,003
z_min - minimal height [m]
1,000
k_I - turbulence factor [-]
1,00
Load default NA parameters
OK
Cancel

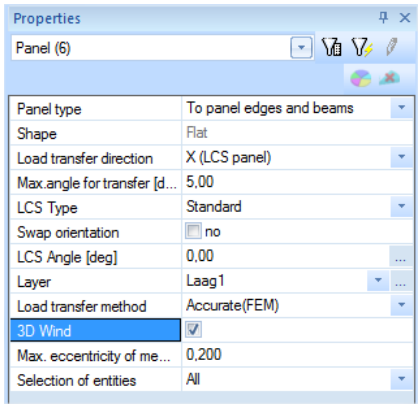
3D Wind Generator

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 **closed structures**.

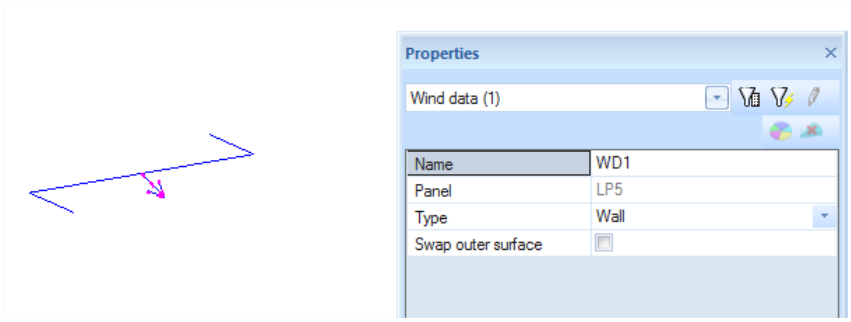
Open the example 3D Wind generator.

The structure consists of a steel hall and is closed using load panels. To be able to use the 3D wind generator, the functionality **Climatic loads** is used with the wind load according to the code (EN1991-1-4).

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.

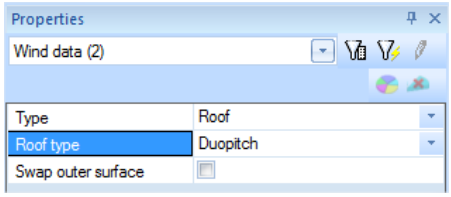


This means that **Wind data** will be created on the load panels. This wind data is shown as a small blue arrow. By selecting this arrow, the properties can be viewed and changed.

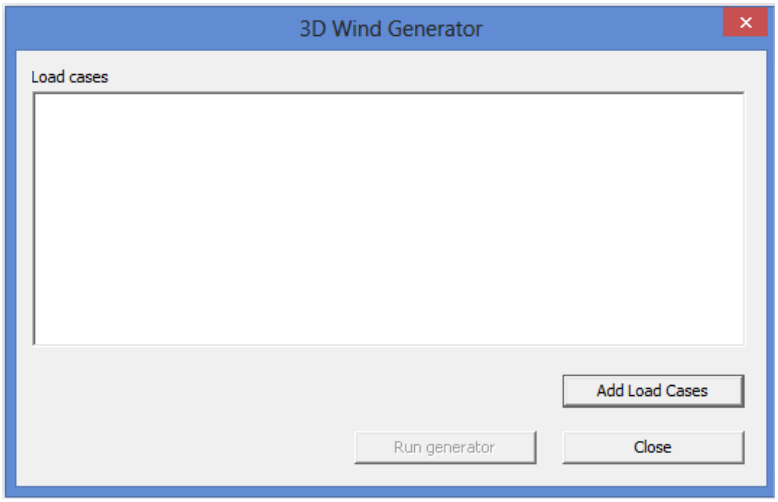


It is necessary that the arrows are pointing in the direction outside the structure. For that reason, there are two Wind Data for which the option **Swap outer surface** has to be marked.

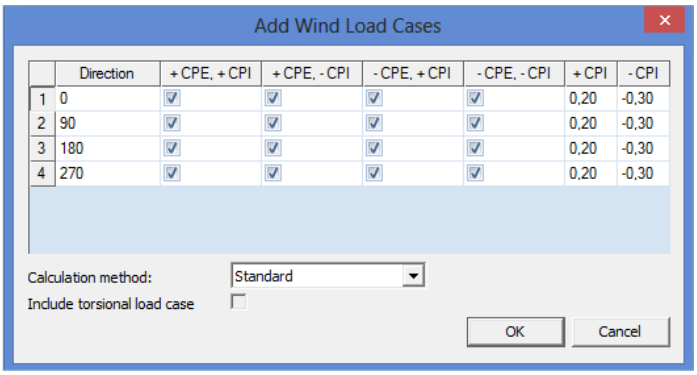
Also the Wind Data for the load panels on the roof should be adapted. By default the **Roof type** is set to Monopitch. This needs to be changed to **Duopitch**.



Afterwards, the **3D Wind Generator** can be used in the load menu. Since there is no load case yet defined, the self weight will be created automatically by opening the Load menu.
This window can be closed, to start using the 3D Wind Generator.

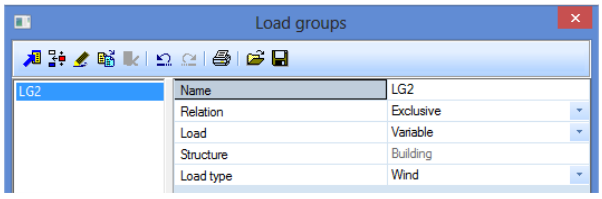


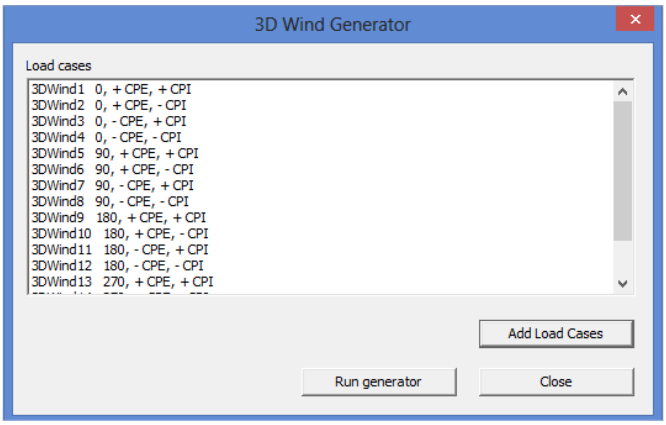
Through the button **Add Load Cases**, it can be chosen which load cases have to be generated.



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, see Chapter 2D Wind generator [1]), the Cpi coefficients needs to be inserted manually.

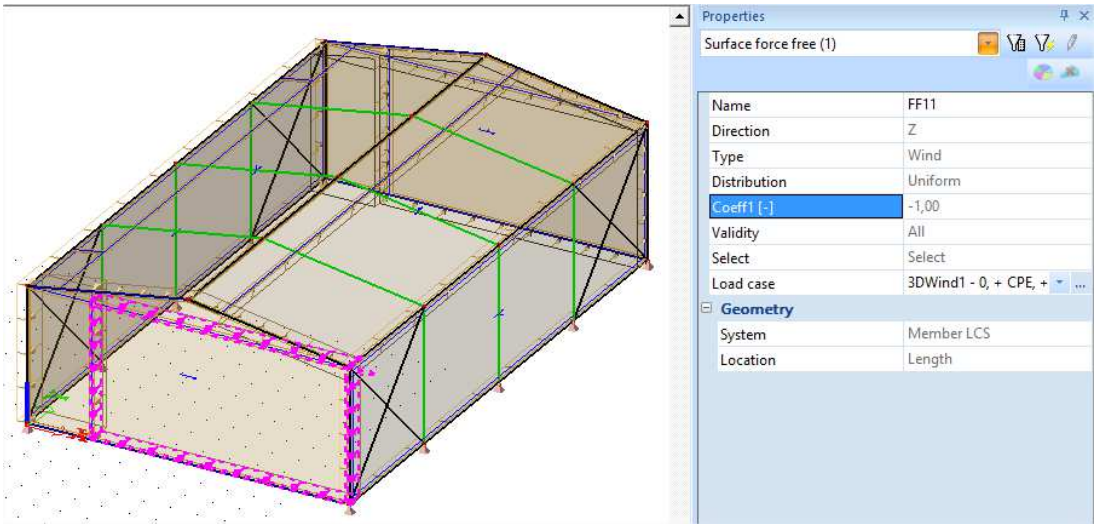
After clinking on OK, all the load cases that are marked will be created. These load cases will be gathered in a new automatic created variable load group.



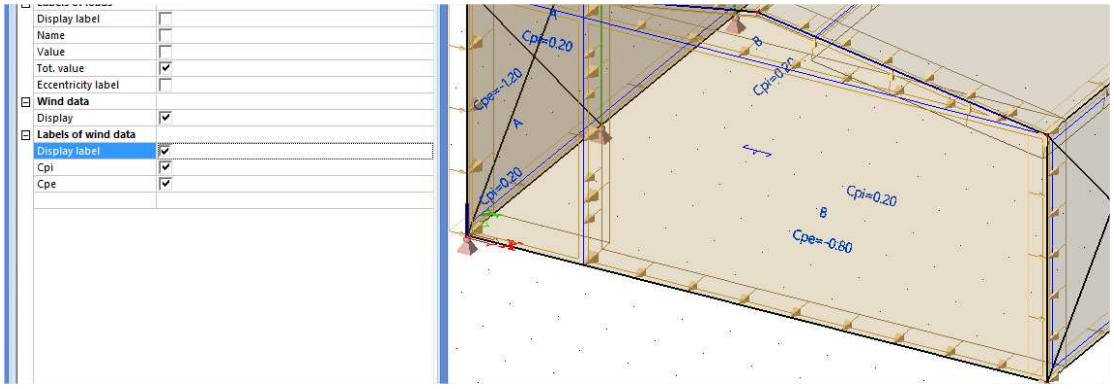


Afterwards, the generating of the loads in these load cases can be started by using the option **Run generator.**

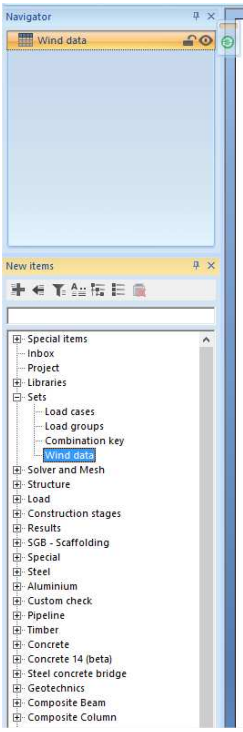
The loads can be viewed in the Load menu. The loads are generated as surface loads for each zone. By selecting such a surface load, the calculated coefficient can be viewed in the property window:



This coefficient is a combination of Cpe and Cpi (see previous chapters). The Cpe and Cpi coefficients can showed graphically by marking this option in the **View parameters**.



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



1. Wind data

Name	Type	Roof type	Swap outer surface	Load direction	Region	Zones	+Cpe	-Cpe
WD1	Wall		X	0	1	A	-1.2000	-1.2000
					2	B	-0.8000	-0.8000
					3	A	-1.2000	-1.2000
					4	B	-0.8000	-0.8000
				90	1	E	-0.3133	-0.3133
					2	E	-0.3133	-0.3133
				180	1	A	-1.2000	-1.2000
					2	B	-0.8000	-0.8000
					3	A	-1.2000	-1.2000
					4	B	-0.8000	-0.8000
				270	1	D	0.7067	0.7067
					2	D	0.7067	0.7067
WD2	Roof	Duopitch	X	0	1	F1	0.0900	-1.3400
					2	F2	0.0900	-1.3400
					3	G	0.0900	-1.0200
					4	H	0.0900	-0.4650
				90	1	F	-1.4650	-1.4650
					2	G	-1.3000	-1.3000
					3	H	-0.6550	-0.6550
					4	I	-0.5550	-0.5550
				180	1	J	0.1100	-0.7800
					2	I	-0.5100	-0.5100
				270	1	F	-1.4650	-1.4650
					2	G	-1.3000	-1.3000
					3	H	-0.6550	-0.6550
					4	I	-0.5550	-0.5550
WD3	Wall		X	0	1	E	-0.3444	-0.3444
				90	1	A	-1.2000	-1.2000
					2	B	-0.8000	-0.8000
					3	C	-0.5000	-0.5000

Mobile loads

The modules needed for the use of mobile loads, are not included in the concept edition.

The following modules are needed:

- **Mobile loads**	
esas.02 (1D members)	**Professional edition**
esas.35 (2D members)	
- **Advanced mobile loads**	
esas.03 (1D members)	**Expert edition**
esas.36 (2D members)	

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:

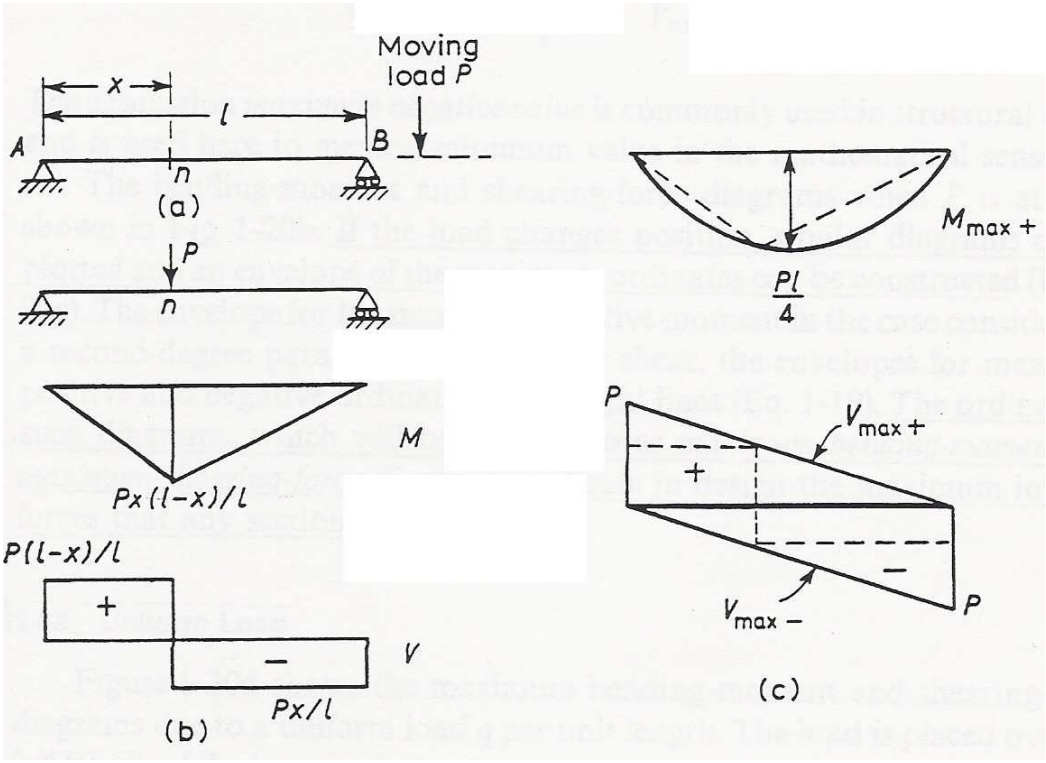


Figure (a) represents a simple beam on 2 supports, across which a concentrated load P can move.

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.

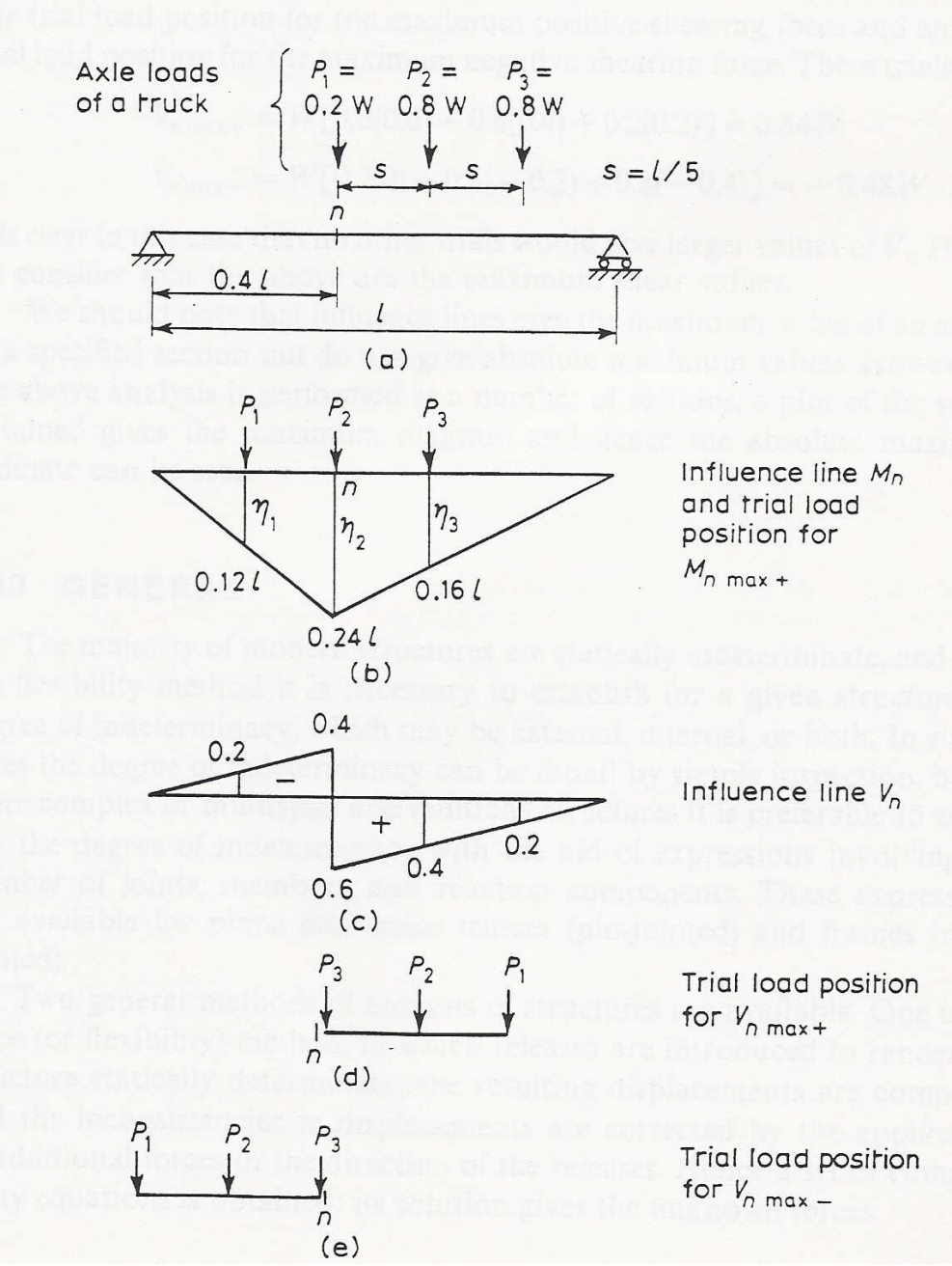


Figure (a) represents a simple beam on two supports again. Across the beam, a system of three point loads can move which represent e.g. the axis loads of a lorry. We look for the position of the load system for which the moment and the shear force are maximal in the section “n”.

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 η_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:

$$M_n = Wl[0,2(0,12) + 0,8(0,24) + 0,8(0,16)] = 0,344Wl$$

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:

- Input Track across which a Unit load can move
- Input Unit load
- Representation Influence lines
- Input Load system
- Exploitation in a point at which the Load system is linked to the Unit load
- Generation load case for exploitation in a point
- Generation enveloping load cases to gain insight in the global behaviour of the structure.

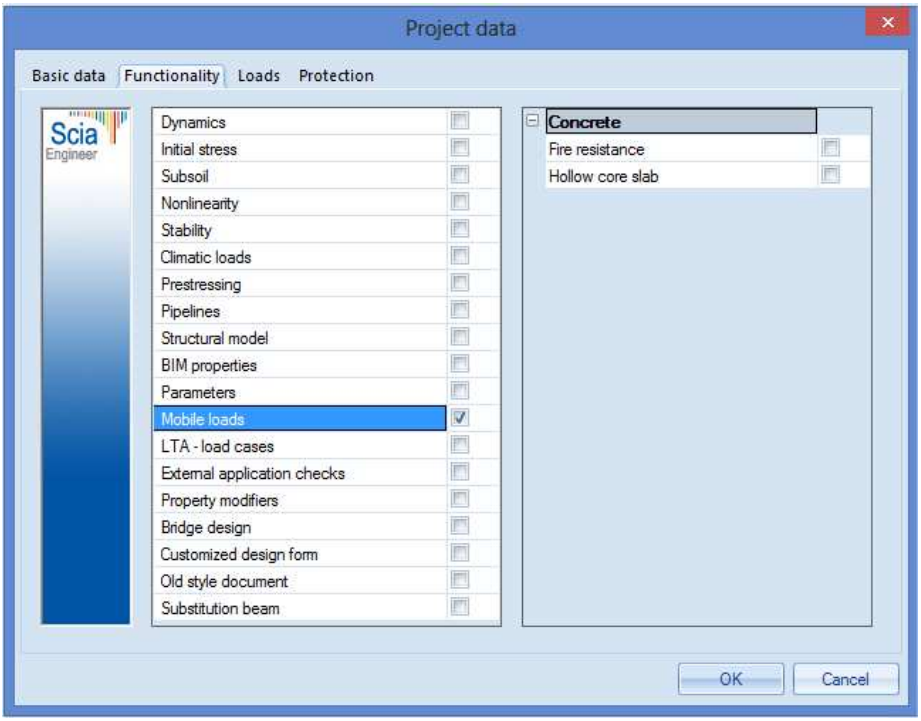
Load system

Project data

For the explanation about the influence lines and the load systems, a new project is created with the following parameters:

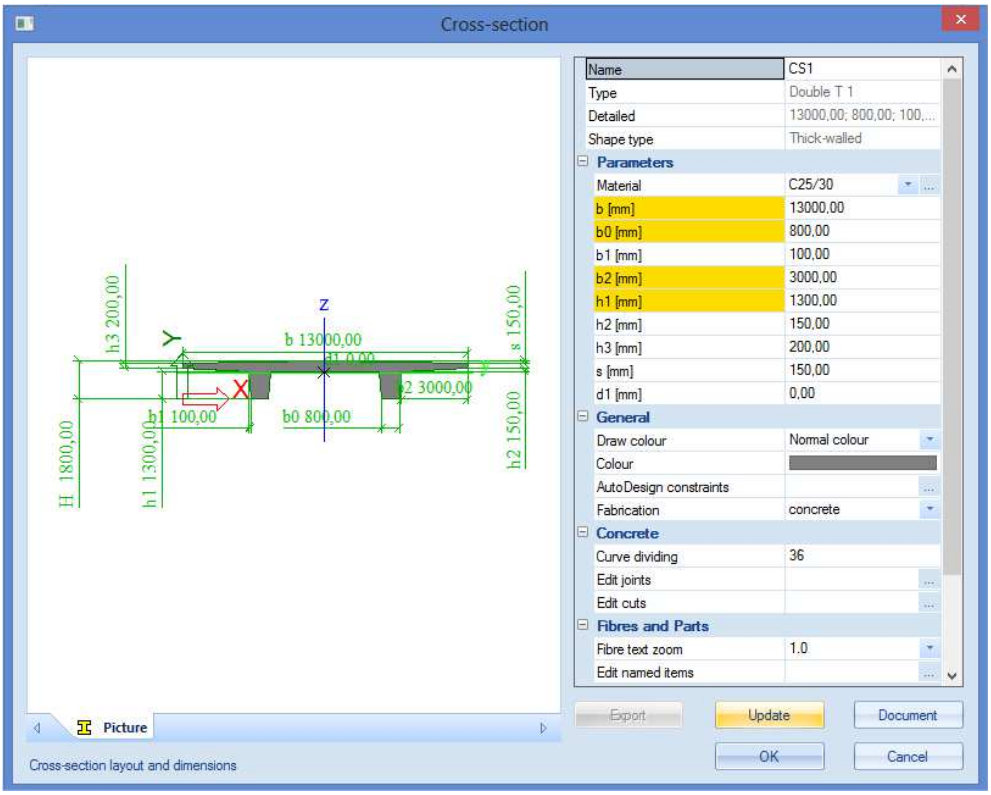
- Code: Eurocode
- Material: Concrete C25/30
- Structure: Frame XZ
- Project level: advanced

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

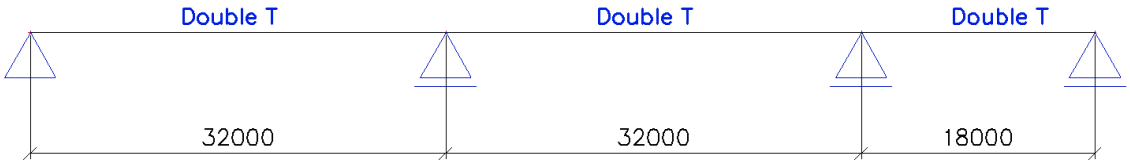


Construction

The construction is built from a **Double T** bridge girder with the default dimensions given by SCIA Engineer.



The construction can be inserted as 3 horizontal beams, at which the begin node is imposed hinged and the other nodes are rolled.

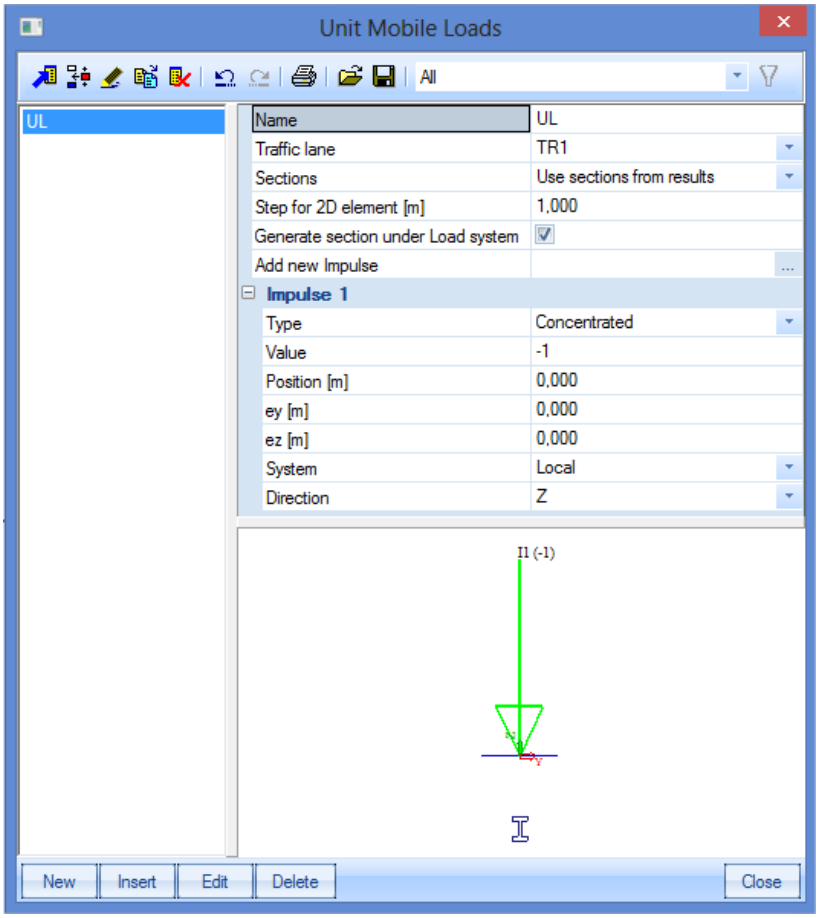


To be able to calculate the project, one load case is created, the Self Weight.

Input track and unit load

In the menu **Mobile Loads**, a track can be defined from node N1 to N4 with the option **Traffic Lane**.

Afterwards, a unit load can be inserted through the option **Unit loads**.



The option **Sections** determines the density of the used sections.

Use sections from results
The unit load is positioned in every section of the beam that lies in the area of the track. The number of sections on a member is indicated in the **Solver Setup**.

Use step according 2D element
The unit load is positioned with the step entered of "*step for 2D element [m]*". If an element has a length that is shorter than the adjusted step, it is not loaded by the unit load.

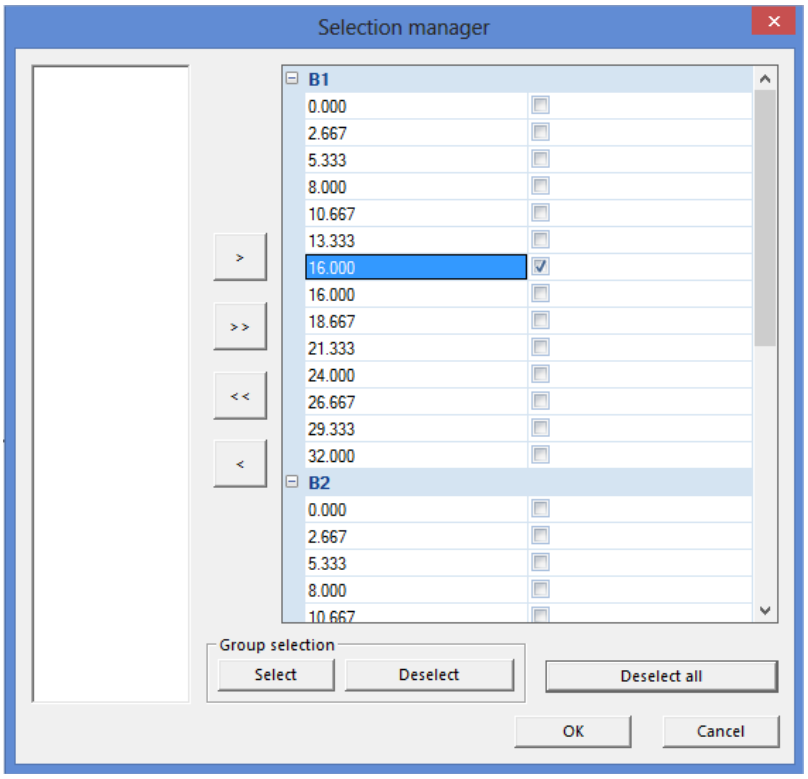
Generate at least one section on member
Analogously to the previous option; here the unit load is also positioned on elements with a shorter length than the adjusted step.

Through the option **Generate section under Load system**, a section is made under every concentrated load of a load system when showing the results. This way the result can be exactly reviewed under the concentrated load.

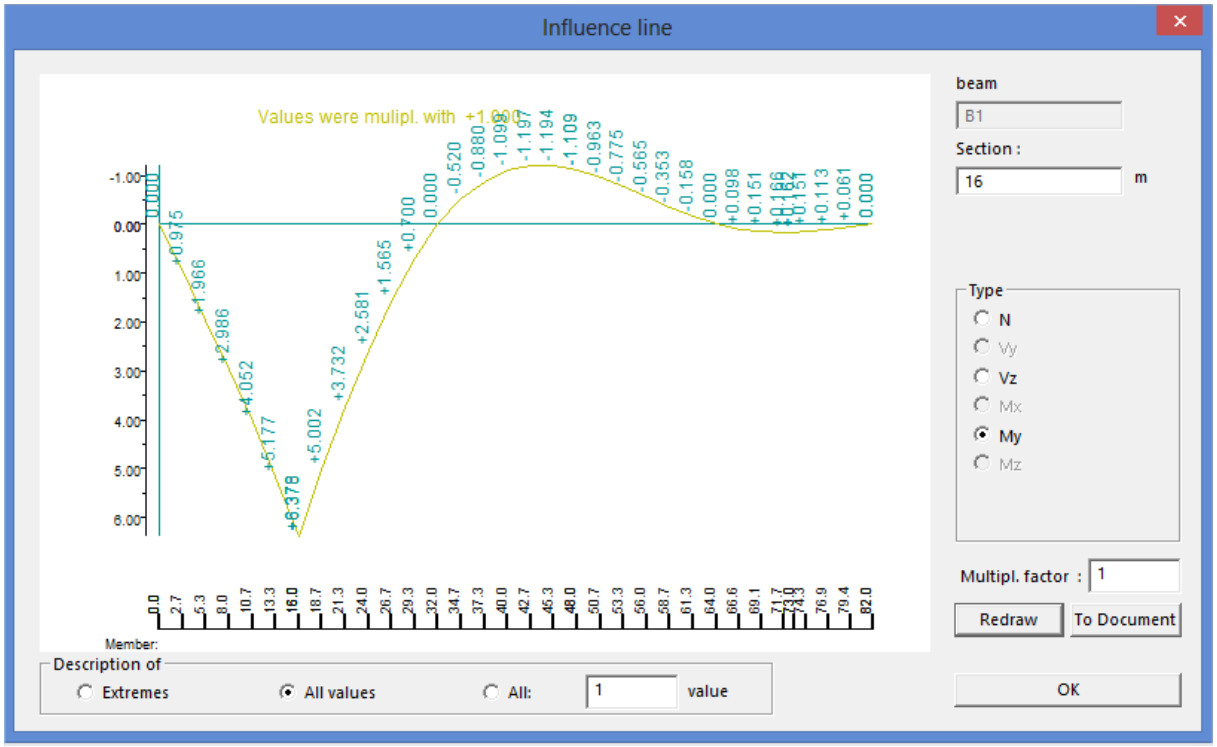
Influence lines

After defining the track and the unit load, the linear calculation can be started. When the calculation is finished, a new group **Influence lines** appears in the menu Mobile Loads.

By looking at the results, there has to be indicated on which member(s) and in which section(s) the results have to be shown by using the **Selection Tool**.



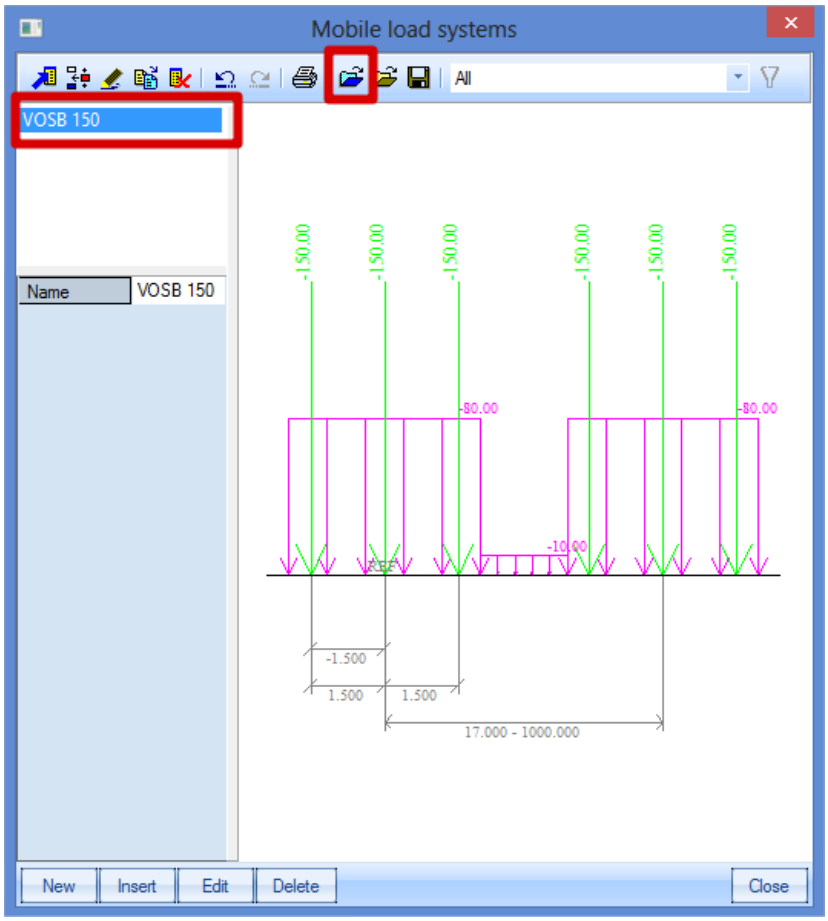
The **Preview** shows the results numerically, and with the **Single Check**, the influence line can be shown graphically.



Input of load system

Through the use of a unit load, the influence lines have been generated. Now this unit load can be linked to a load system. The load systems can be inserted through **Load System Database** in the Mobile Loads menu.

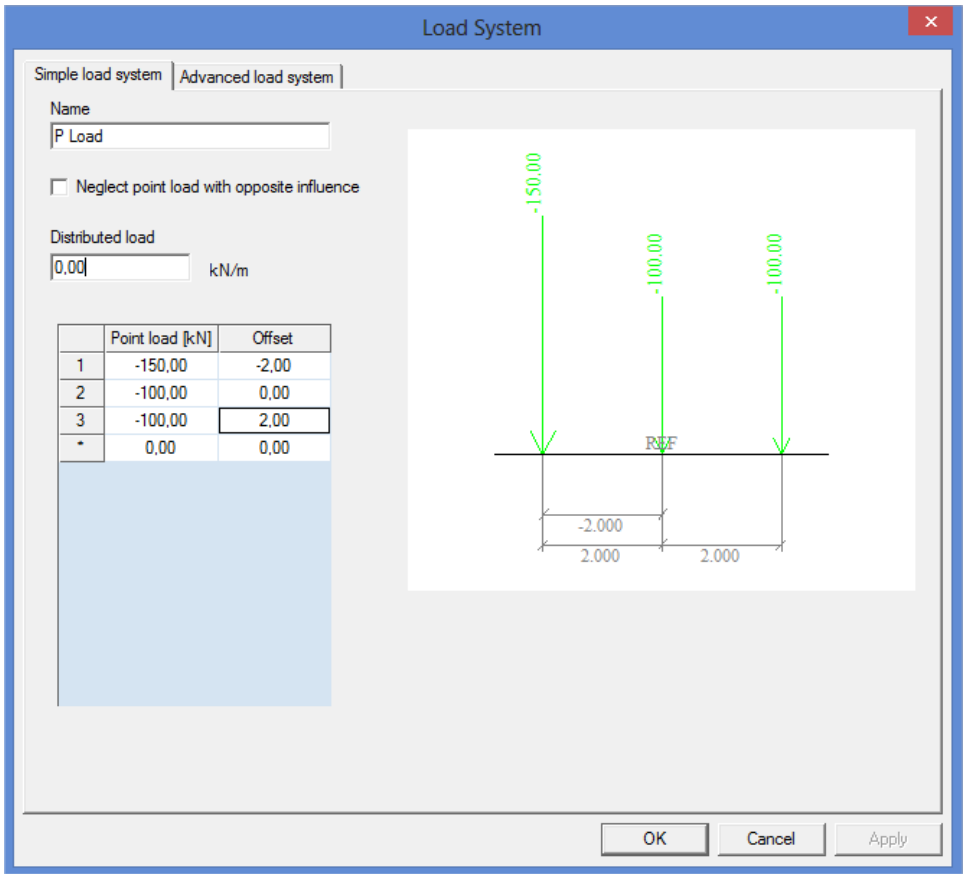
The Load Systems can be inserted manually, or a Load System from the **System Database** can be chosen:



In this example, the load systems will be added manually.
Two load systems will be inserted.

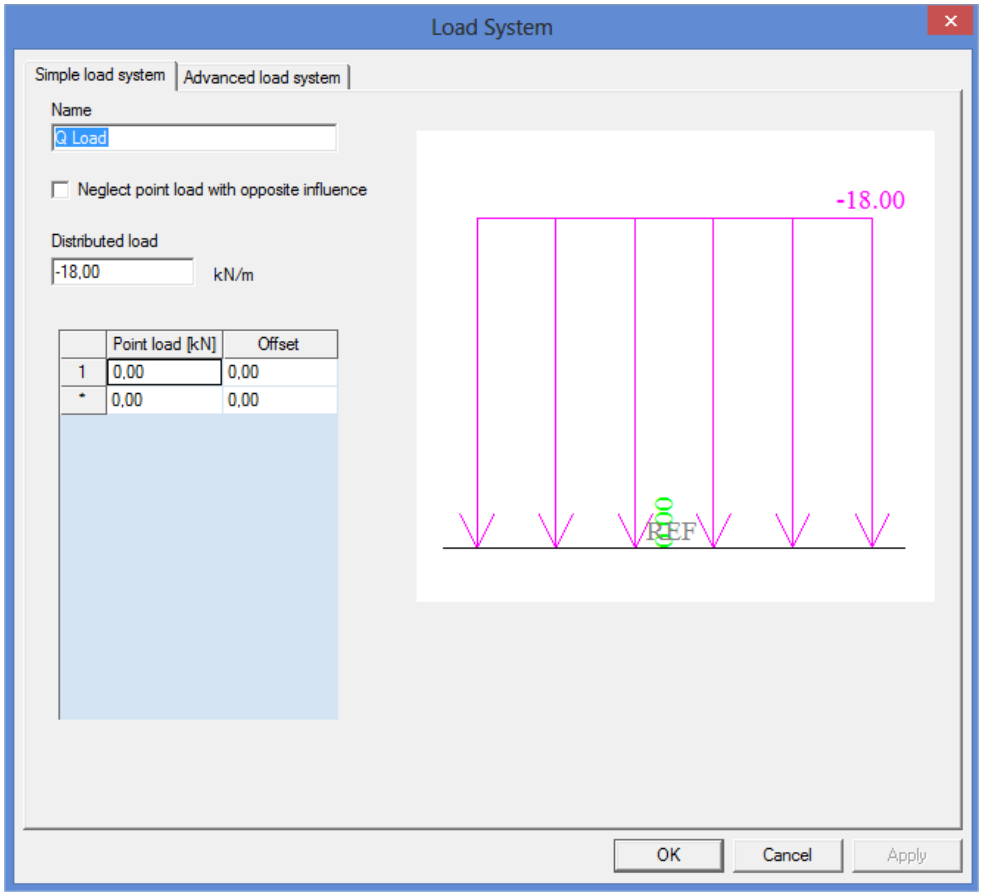
1) *Single Load System* ***P Loads***

This load consists of a point load of 150 kN and 2 point loads of 100 kN with a mutual distance of 2m.



2) *Single Load System* ***Q Loads***

This load system consists of a line load of 18 kN/m with an indefinite length.



When entering a Single Load system, you have the possibility to mark the option **Neglect point load with opposite influence**. If this option is activated, the complete concentrated load, which lies in the negative area of the influence line, will be taken in account in the calculation. By activating this option, the found maximum will be reduced.

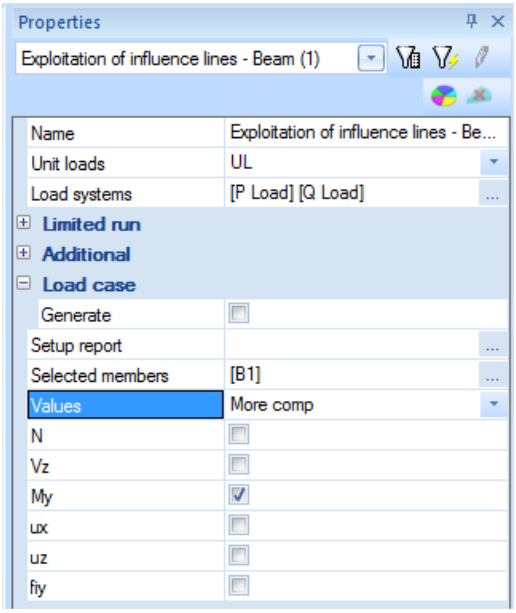
Exploitation of load systems

After defining the load systems, the linear calculation can be started. Afterwards, the new group **Detail analysis** appears in the Mobile Loads menu.

With the **Detailed Analysis**, the load systems can be linked to the unit load. For every desired position on the structure, between all the selected tracks, SCIA Engineer determines the system that is most adverse for the chosen design parameter.

An exploitation is performed for the moment **My** on a position **24m** on the first beam **B1**. The exploitation is performed for the load both the load systems **P Load** and **Q Load**.

The following option can be adjusted in the Property window for **Member force, deformation**.



Through the action **Preview**, the results of the required exploitation can be called up:

Exploitation of influence lines - Beam

1. Description of the influence line +
The selected load systems for which the exploitation is done:
Influence line:
Member B1, Position : 24.00[m], Type : My
Considered load systems:
P Load
Q Load
Unit Load : UL

2. Co-ordinates of the nodes of the the loadtrack and their ordinates:

Node	X [m]	Y [m]	Z [m]
1	0.000	0.000	0.000
2	32.000	0.000	0.000
3	64.000	0.000	0.000
4	82.000	0.000	0.000

3. Areas of the fields of the influence line:

Area Nr	Area
1	43.676
2	-34.575
3	2.695

4. Co-ordinates at the points where the sign of the influence line changes:

Sign Nr	X [m]	Y [m]	Z [m]
0	0.000	0.000	0.000
1	32.000	0.000	0.000
2	64.000	0.000	0.000

5. Additional factors:
Mult. factor results except deformations : 1.000
Mobile factor: 1.000
6. The data of load system which gives the maximum / minimum values:
Negative maximum position : Q Load

Sum P [kNm]	Sum Q [kNm]	X1 [m]	X2 [m]
0.000	-622.355	0.000	0.000

Positive maximum position : P Load

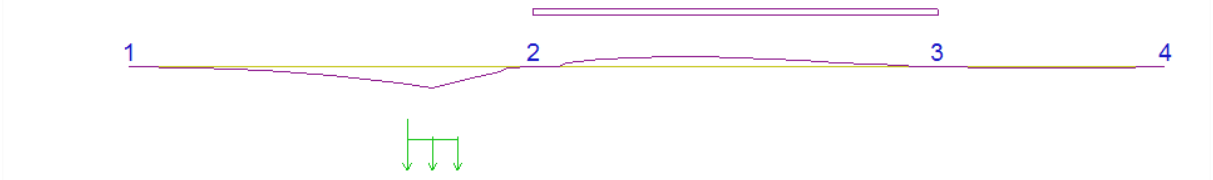
Sum P [kNm]	Sum Q [kNm]	X1 [m]	X2 [m]
1134.903	0.000	24.000	24.000

7. Results:
Negative maximum position : Q Load

Description	Due to P	Due to Q	P + Q	Units
My negative	0.000	-622.355	-622.355	[kNm]

Positive maximum position : P Load

Description	Due to P	Due to Q	P + Q	Units
My positive	1134.903	0.000	1134.903	[kNm]



The parts that should be displayed in the report can be indicated through the options **Setup report**.

Under *Title 1.* It can be seen that the position for which the design parameter My is extreme on a position **24m** on member **B1**.

Under *Title 6. and 7.*, is indicated that two extremes have been found.

My is minimal **(-622,355 kNm**) on **24m** if the reference point of the load system **Q Load** is located at **0,000m** from the begin point of the track.
My is maximal (**1134,903 kNm**) on **24m** if the reference point of the load system **P Load** is located at **24m** from the begin point of the track.
The values X1 and X2 are the same since a single load system is used.

Through the action **Single Check** the results are shown in a window, at which the position for the exploitation can be simply changed.

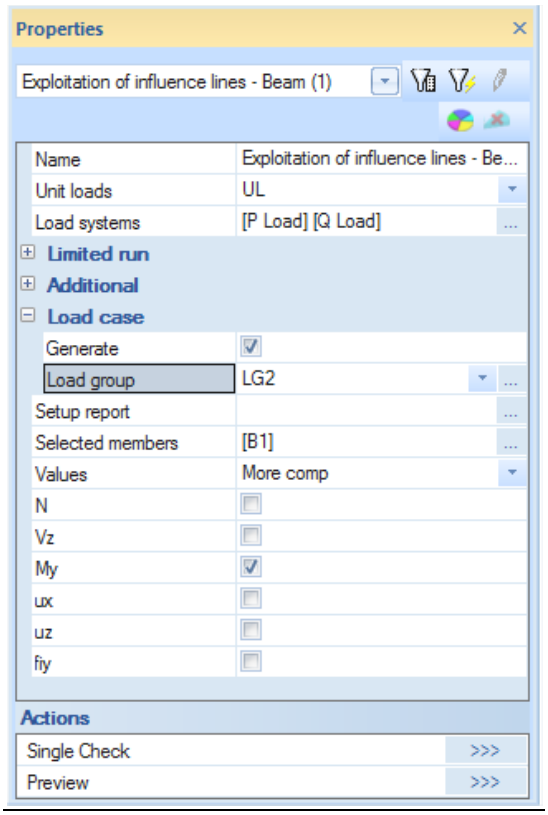
Generating Load cases – Enveloping Load cases

Generating Load cases

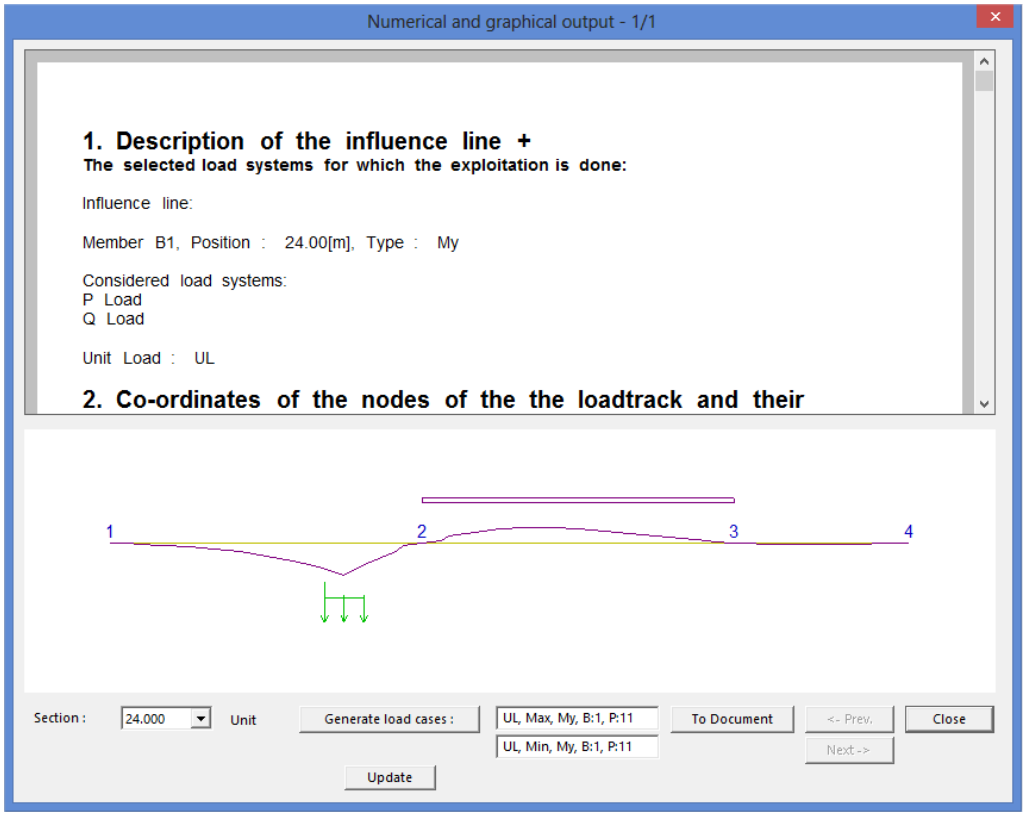
With the exploitation of a design parameter in a section you have the possibility to generate several exclusive variable load cases.

First of all the option Load case - generate has to be marked at the Detailed Analysis.

If no variable load group is found, the program asks whether a new group has to be made.

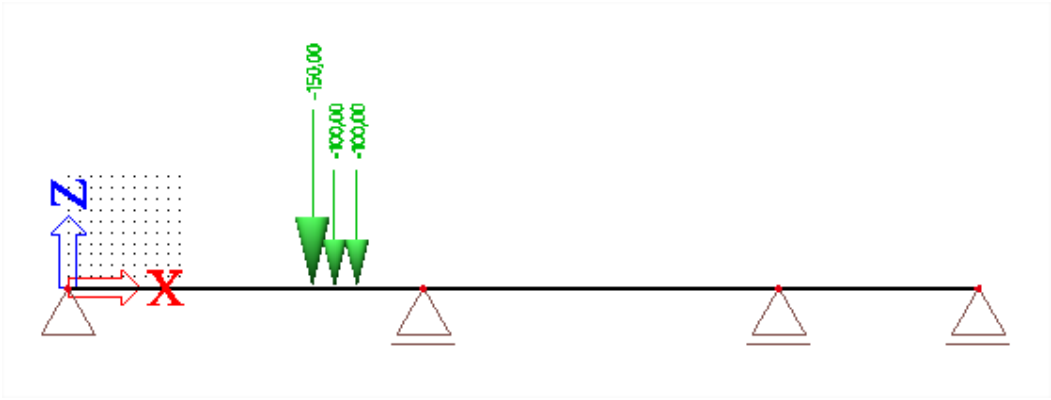


After activating this option, a Single check is performed on the member B1 through the action Single check.

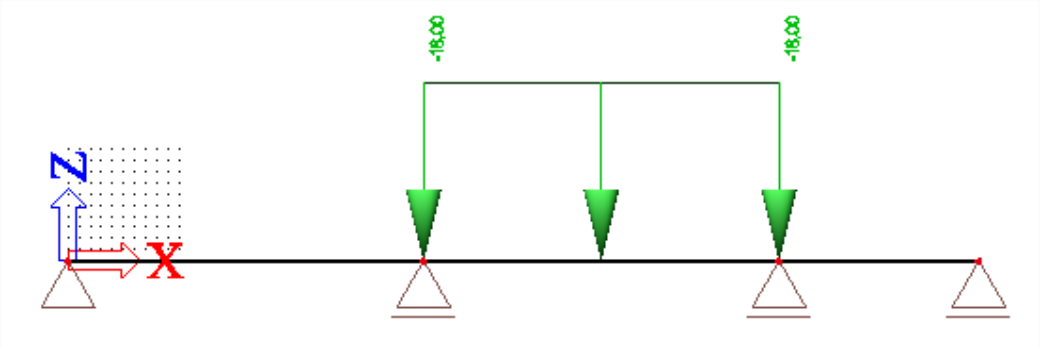


Through the button **Generate Load Cases** two load cases are generated, one for the minimal My on 24m and one for the maximal My on 24m. The parameter B indicates the member, parameter P the position on the member.
Since this option is used to make real load cases, the content of these load cases can be seen.

Max My:



Min My:



After a linear calculation these load cases can be combined with other load cases.

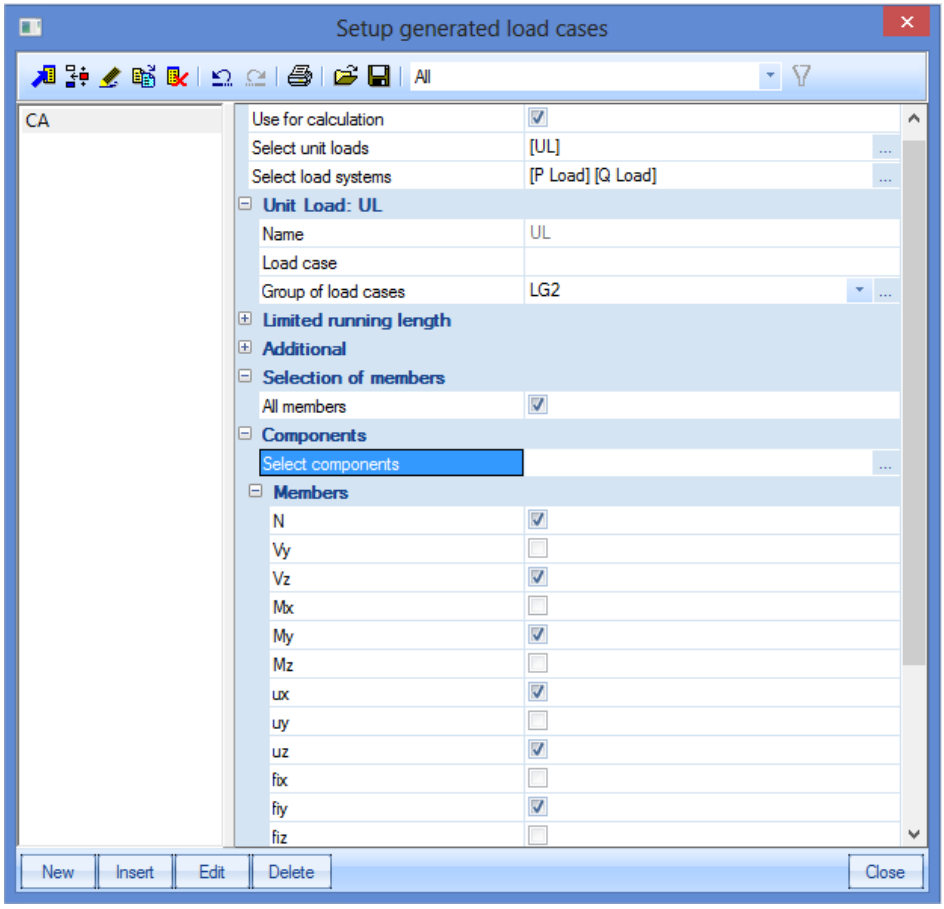
Generating Envelope Load cases

During the exploitation of the influence line, the individual sections of the track are evaluated for the design components (e.g. My). During this exploitation the critical position of the load system is determined. This position causes a maximal value of the design component in the appropriate section. This value is saved together with the corresponding values of this design component in other sections and the procedure is repeated for the following section.

As soon as the calculation is performed for every section, the envelope can be created. Subsequently the system can create envelopes for other design components (e.g. Vy, Vz, etc.). It is important to see that the envelope doesn't represent a realistic load case, so it is not possible to show the content.

The envelope represents a fictive load case that shows the found extremes (envelopes). For this reason it is not useful to use this envelope e.g. for a steel check. This envelope can be combined with other load cases to obtain insight in the global behaviour of the structure.

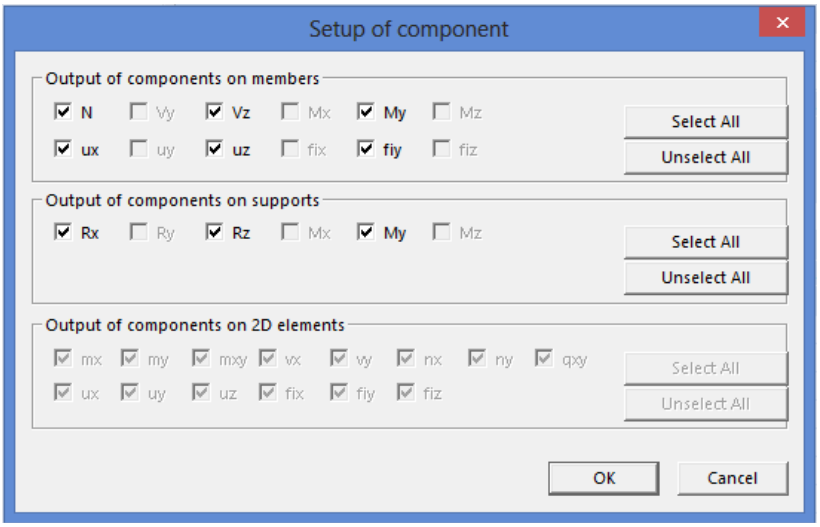
To generate the envelope load cases, the option **Setup generated load cases** in the Mobile Loads menu has to be used.



In the window **Load case** you can enter a name for the load cases you have to make. In this example the names of the load cases are automatically generated by the program by leaving the window blank.

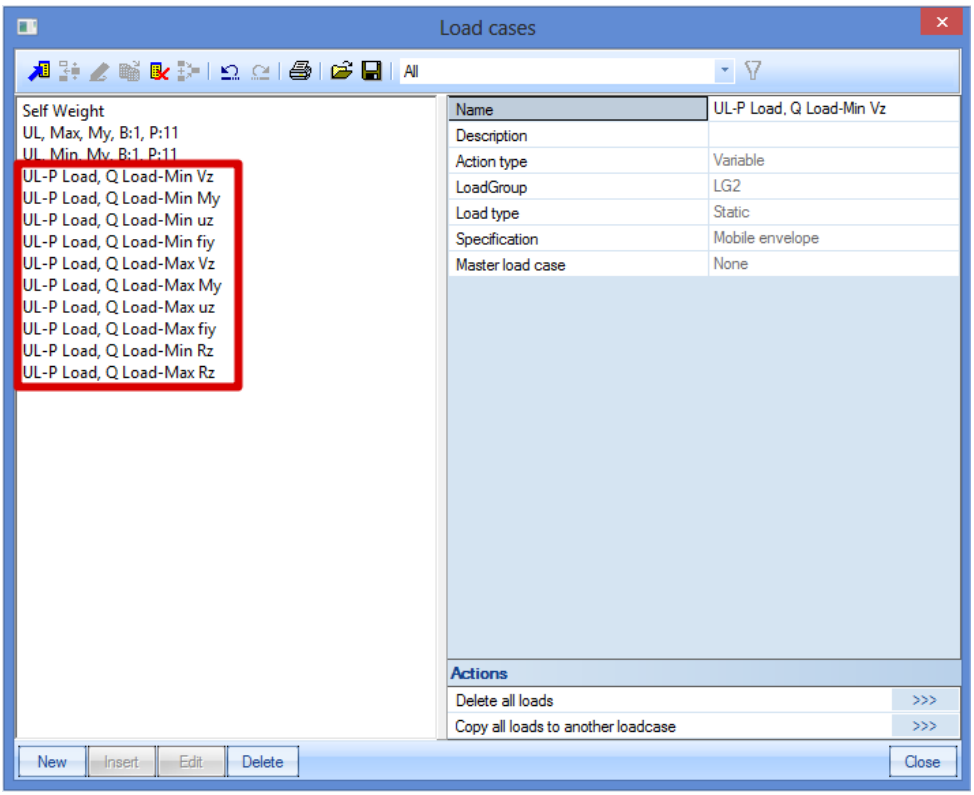
With **Selection of member** the option **All members** is marked, so all the members are taken into account in the calculation.

Through **Select Components** you can indicate for which components a envelope has to be generated. In this example all components are considered.



After importing these data a linear calculation can be performed, so the envelope load cases are made.

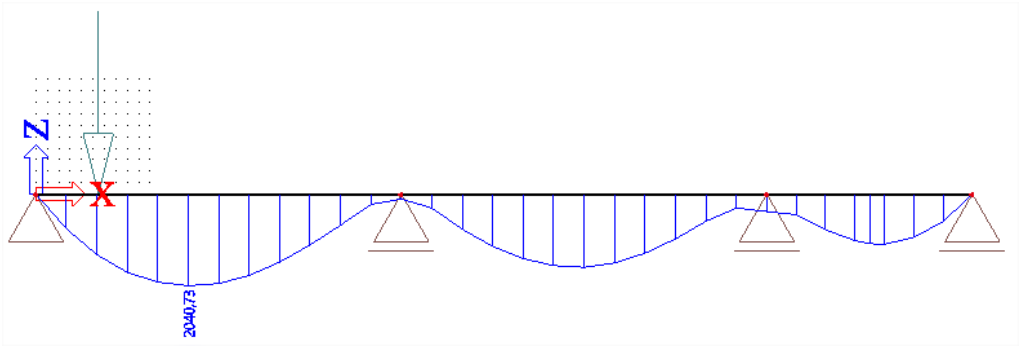
After the calculation the Load cases manager shows the following:



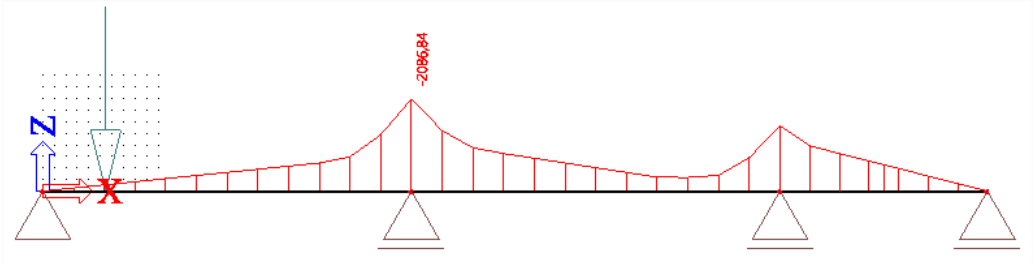
The load cases have ‘Mobile envelope’ as a description and are in an exclusive load group.

In the results menu, the results of these envelope load cases can be reviewed.

Max My:



Min My:



Remarks:

When performing a Detailed analysis or generating the enveloping load cases, a number of advanced options is available:

Limited run:

During the exploitation the critical position of the load system is determined. However, it may happen that the extreme is reached if the mobile load is partially outside the structure. With this option you can indicate whether the mobile load can only appear on a restricted interval of the track so you can avoid that a part of the load system falls partially outside the structure.

The restriction of the track will be executed in such a way that the values of the influence lines will be zero outside the given interval.

Additional multiplication factor results except deformations:

The VOSB code (NEN code) shows that every internal force and reaction for the position of a mobile load has to be multiplied by this coefficient. The results of influence lines for deformations are not multiplied with this factor.

It is possible that a deformation of a load case, associated with internal forces such as Max My, has a larger deformation than e.g. the load case Min uz.

Additional Mobile factor:

The mobile factor is used e.g. to consider a single or double traffic lane. All results are multiplied with this factor, also the deformations.

Train Loads

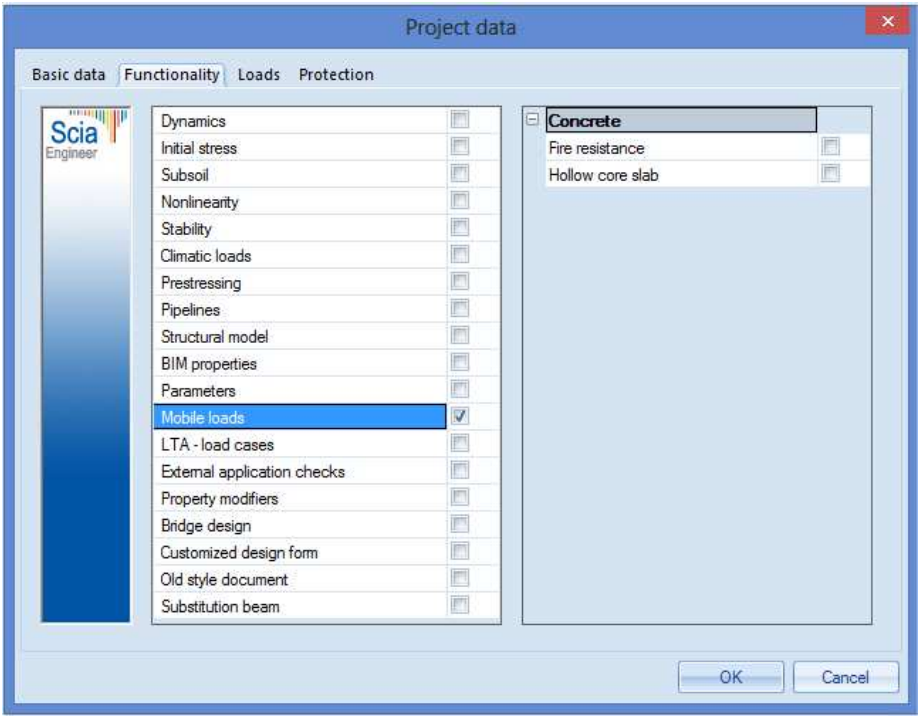
Approximately the same as in the previous example, can be done with 2D members. In this example, a bridge deck will be modelled as a concrete plate on three line supports.

Project Data

A new project is created with the following parameters:

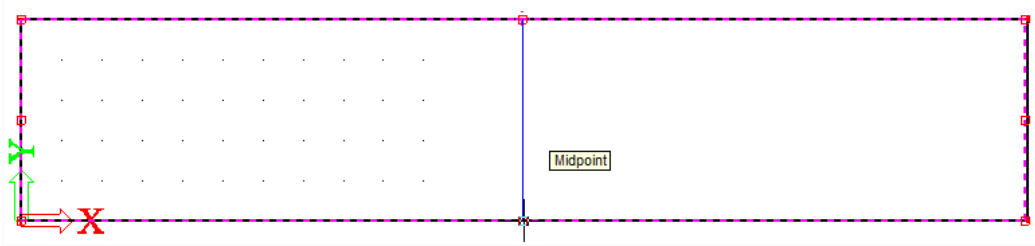
- Code: Eurocode
- Material: Concrete C25/30
- Structure: Plate XZ
- Project level: advanced

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

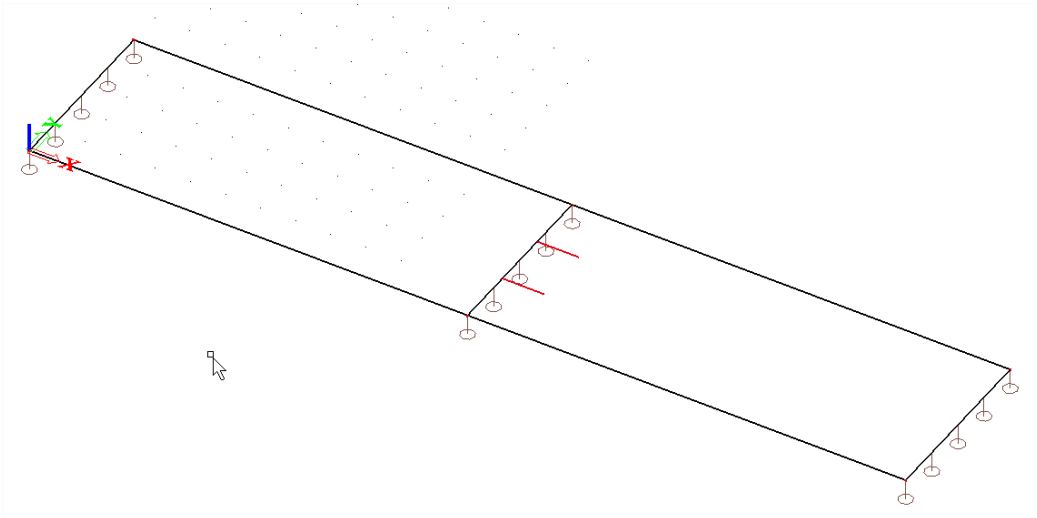


Construction

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**.



Next the line supports are inserted on the three short edges, with only the translation is the Z-direction prevented.



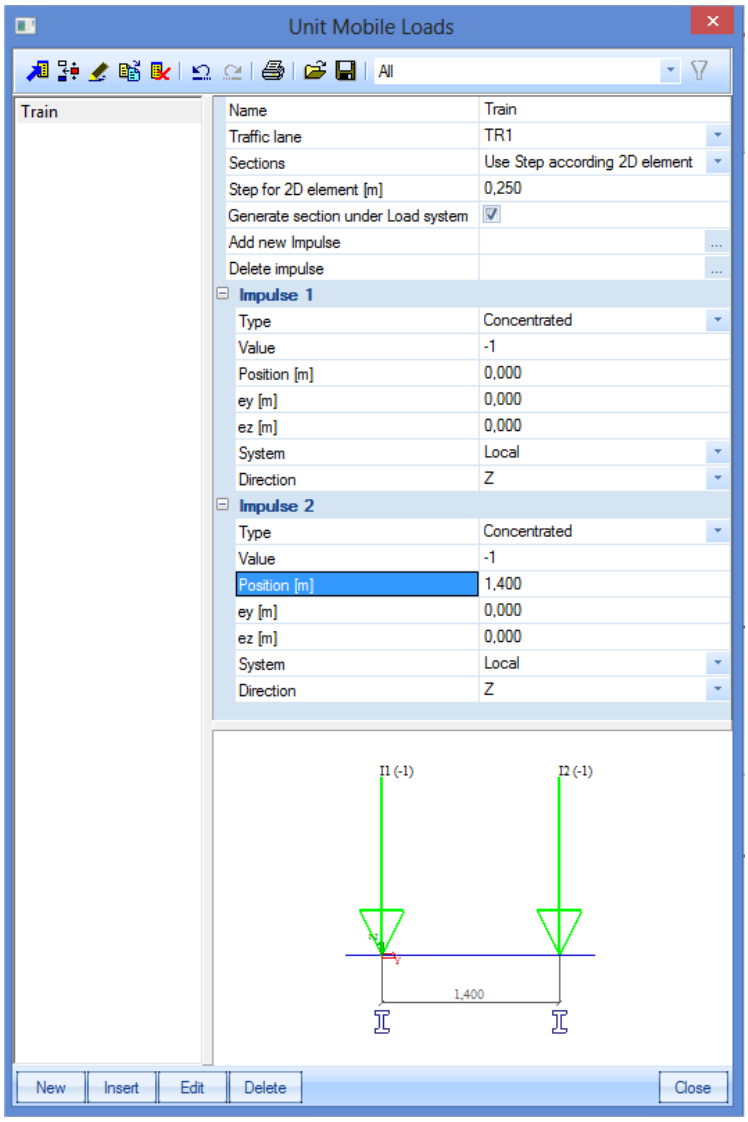
To be able to calculate the project, one load case is created, the Self Weight.

Input track and unit load

In the Mobile Loads menu, the train track can be inserted. This track consists 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 mobile load track is entered with a unit and two impulses on it.

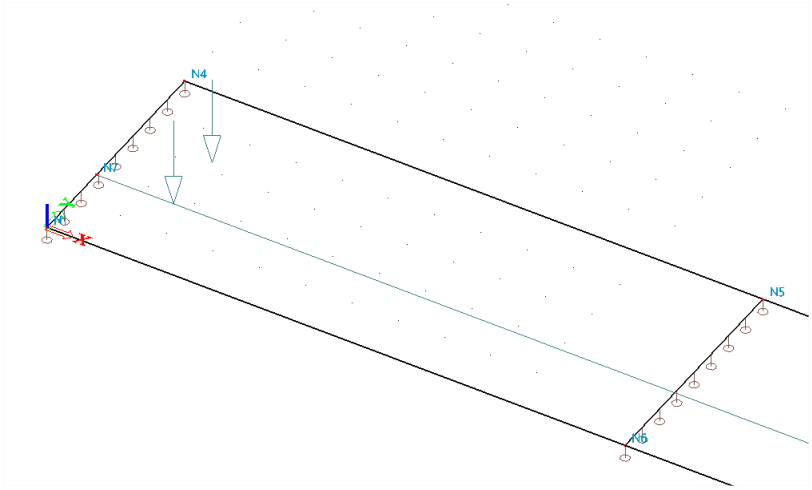
The track has to be entered on **1.8m** from the edge to be able to place the train track in the middle of the bridge. The coordinates can be entered in the Command line (0;1,8 and 25;1,8)

Afterwards, a unit load can be inserted through the option **Unit loads**.



With **Sections**, the option **Use step according 2D element** is chosen with **0,25m** as step.
To have both rails of the train track, a second impulse is added trough the option **Add new impulse**.
The position of the second impulse is **1,4m**.

Both impulses are displayed on the bridge deck:



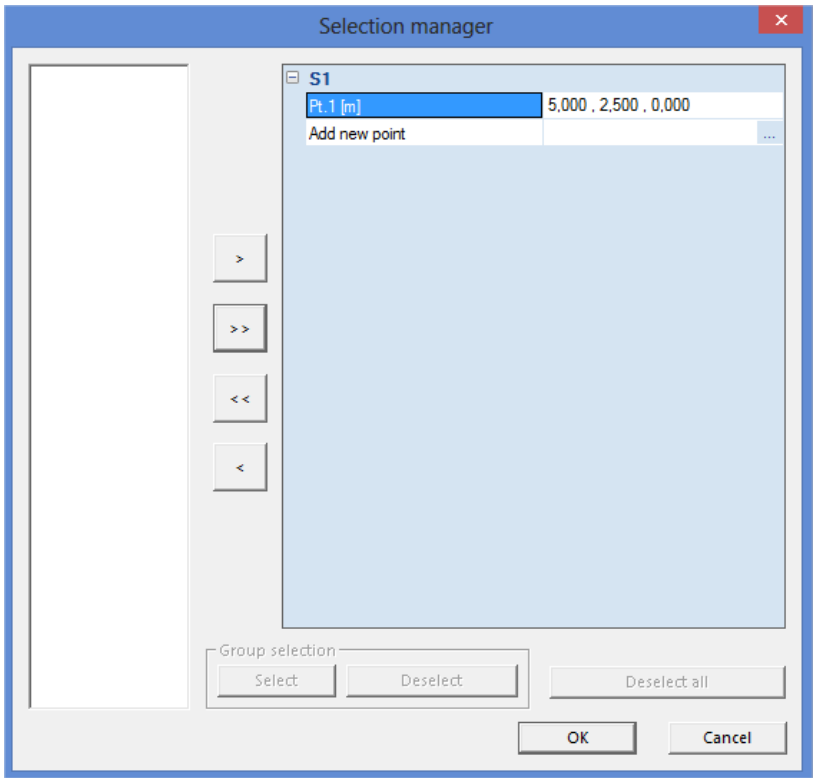
Influence lines

After defining the train track and the unit load that represent both rails, the calculation can be done. To have more accurate results, the mesh is refined in the **Mesh Setup**. For **Average size of 2D element/curved element**, a value of **0,5m** in inserted.

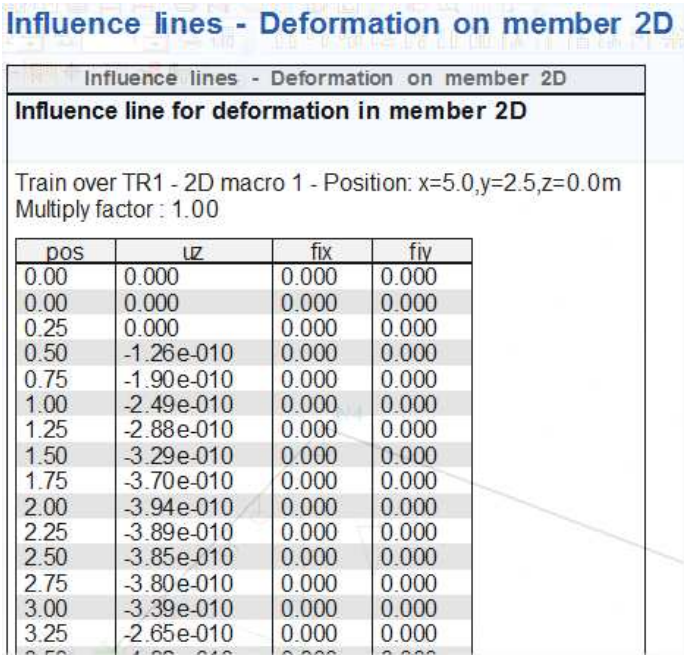
When the calculation is done, a new group **Influence lines** appears in the Mobile Loads menu.

By looking at the results, there has to be indicated on which 2D element and in which point the results have to be shown by using the **Selection Tool**.

The results are asked for the **Deformation on slab** in the point **(5 ; 2,5 ; 0)**.



The **Preview** shows the results numerically. The step of 0,25 is clearly shown in this table.



Influence lines - Deformation on member 2D

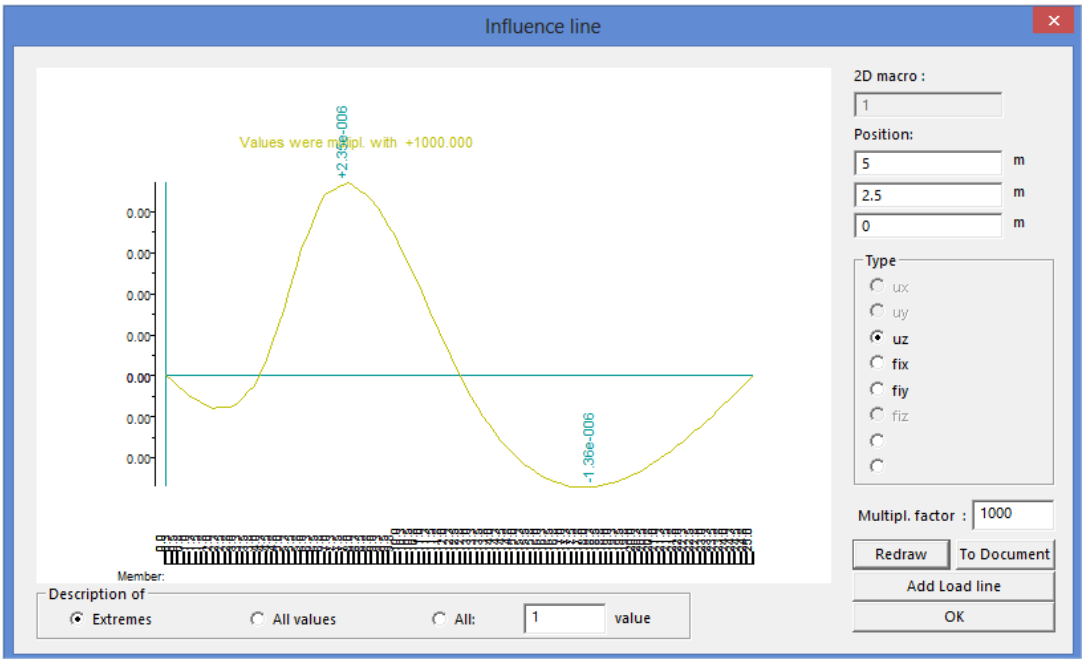
Influence lines - Deformation on member 2D

Influence line for deformation in member 2D

Train over TR1 - 2D macro 1 - Position: x=5.0,y=2.5,z=0.0m
Multiply factor : 1.00

pos	uz	fix	fiy
0.00	0.000	0.000	0.000
0.00	0.000	0.000	0.000
0.25	0.000	0.000	0.000
0.50	-1.26e-010	0.000	0.000
0.75	-1.90e-010	0.000	0.000
1.00	-2.49e-010	0.000	0.000
1.25	-2.88e-010	0.000	0.000
1.50	-3.29e-010	0.000	0.000
1.75	-3.70e-010	0.000	0.000
2.00	-3.94e-010	0.000	0.000
2.25	-3.89e-010	0.000	0.000
2.50	-3.85e-010	0.000	0.000
2.75	-3.80e-010	0.000	0.000
3.00	-3.39e-010	0.000	0.000
3.25	-2.65e-010	0.000	0.000

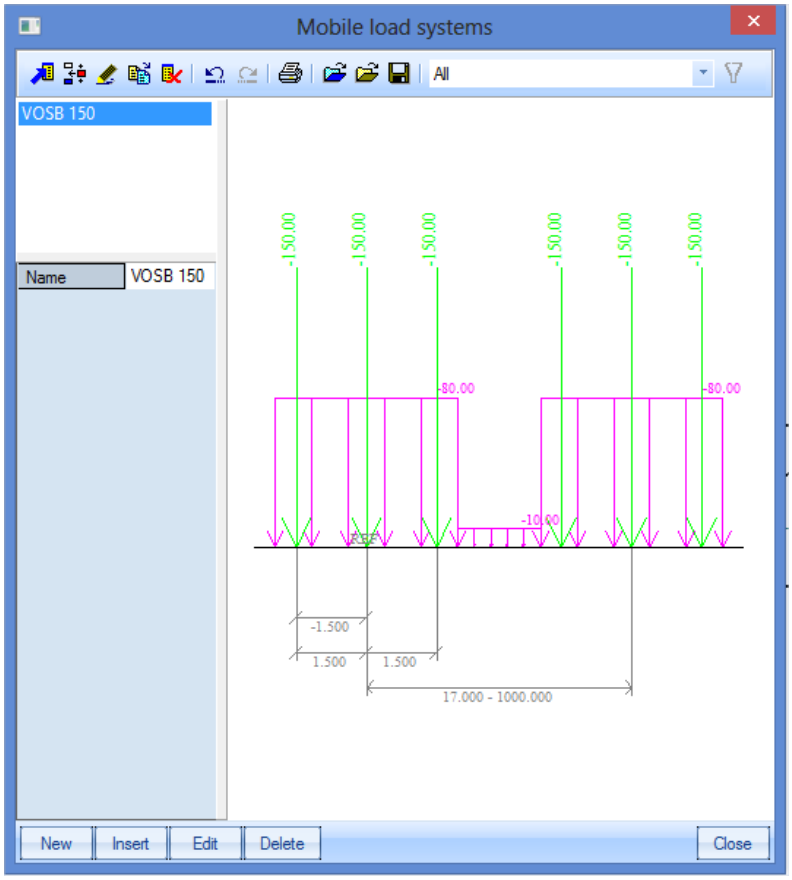
Trough the **Single Check**, the results can be viewed graphically.



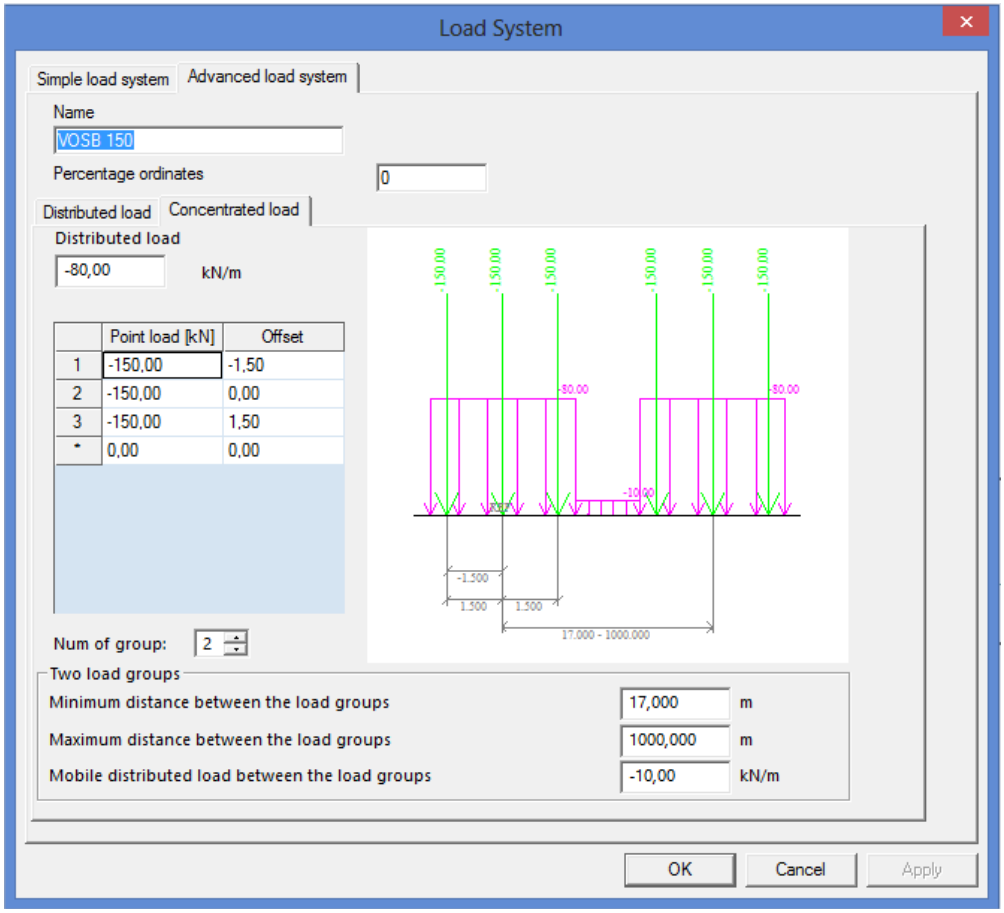
Input Load systems

Through the option **Load System Database**, a load system can be entered.

In this example, a predefined load system is used, namely VOSB 150.



Through the option **Edit**, the properties can be viewed and changed.



The load system consists of 2 groups of three point loads and a divided load. The point loads have a value of 150 kN and a mutual distance 1.5m. The divided load has a value of 80 kN/m.

The **Minimum distance between the load groups** is 17m, the **Maximum distance** is 1000m. SCIA Engineer will let the distances of the load groups between these two boundaries vary to obtain the maximal effect on the bridge deck.

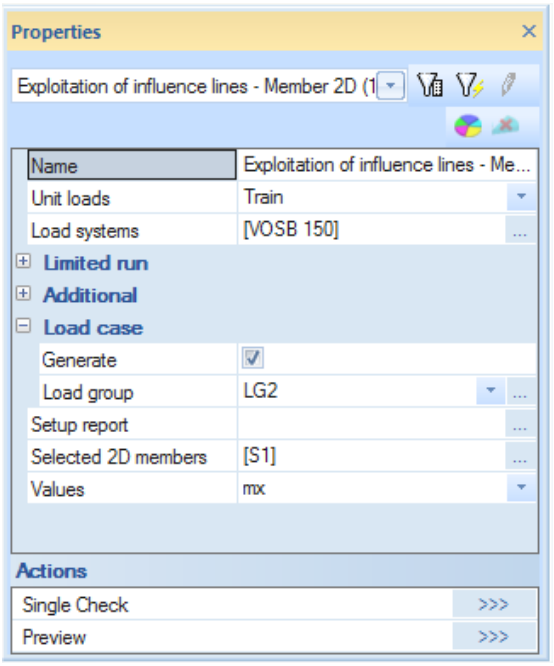
The **Mobile distributed load between the load groups** is 10 kN/m. This value will reduce the found maximum.

Exploitation of the load systems

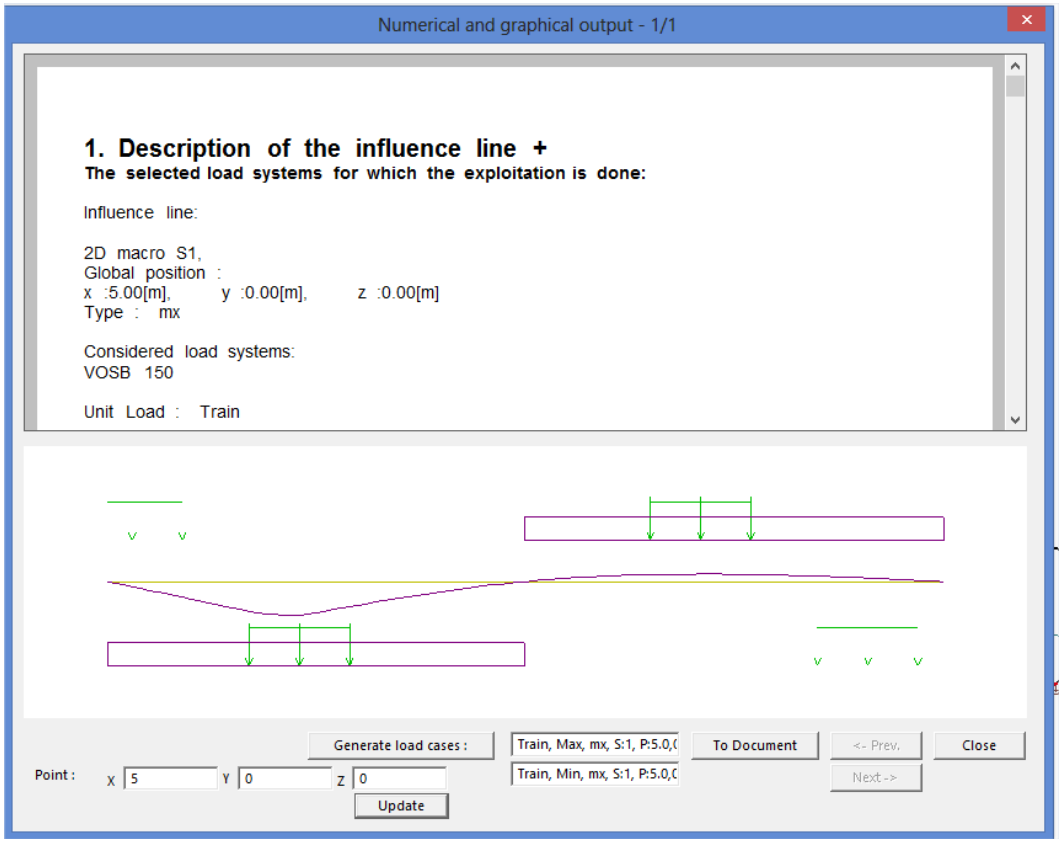
After defining the mobile Unit load and the load systems, the linear calculation can be started. After the calculation, a new group appears in the Mobile Loads menu, namely **Detail analysis – Member 2D force, deformation**.

With the Detailed Analysis the load system can be linked to the Unit load. For every desired position on the structure, between all the selected tracks, SCIA Engineer determines the system that is most adverse for the chosen design parameter.
E.g. an exploitation is performed for the moment **mx**. The parameters can be set in the Property window and through **Selected 2D members** is indicated that results are called up for 2D element S1.

The option **Load case - generate** has to marked at the Detailed Analysis. If no variable load group was found, the program asks if a new group had to be made.



Next through **Single Check**, the bridge deck can be indicated. The exploitation is performed e.g. in the point **(5 ; 0 ; 0)**.



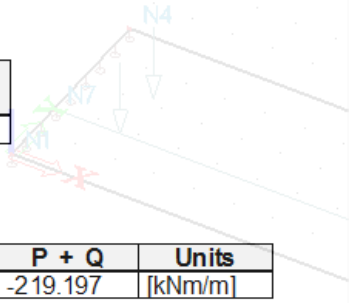
Under *Title 6. and 7.* is indicated that two extremes have been found.

6. The data of load system which gives the maximum / minimum values:
Negative maximum position : VOSB 150

Sum P [kNm/m]	Sum Q [kNm/m]	X1 [m]	X2 [m]
-87.504	-131.693	0.750	17.750

Positive maximum position : VOSB 150

Sum P [kNm/m]	Sum Q [kNm/m]	X1 [m]	X2 [m]
363.775	473.744	5.750	22.750



7. Results:
Negative maximum position : VOSB 150

Description	Due to P	Due to Q	P + Q	Units
mx negative	-87.504	-131.693	-219.197	[kNm/m]

Positive maximum position : VOSB 150

Description	Due to P	Due to Q	P + Q	Units
mx positive	363.775	473.744	837.519	[kNm/m]

mx is minimal **(-219,197 kNm/m**) in point **(5 ; 0 ; 0)** if the reference point of the first group of point loads is on **0,75m** from the begin point of the track and the reference point of the second group of point loads that is on **17,75m**.
mx is maximal (**837,519 kNm/m**) in point **(5 ; 0 ; 0)** if the reference point of the first group of point loads is on **5,75m** from the begin point of the track and the reference point of the second group of point load is on **22,75m**.

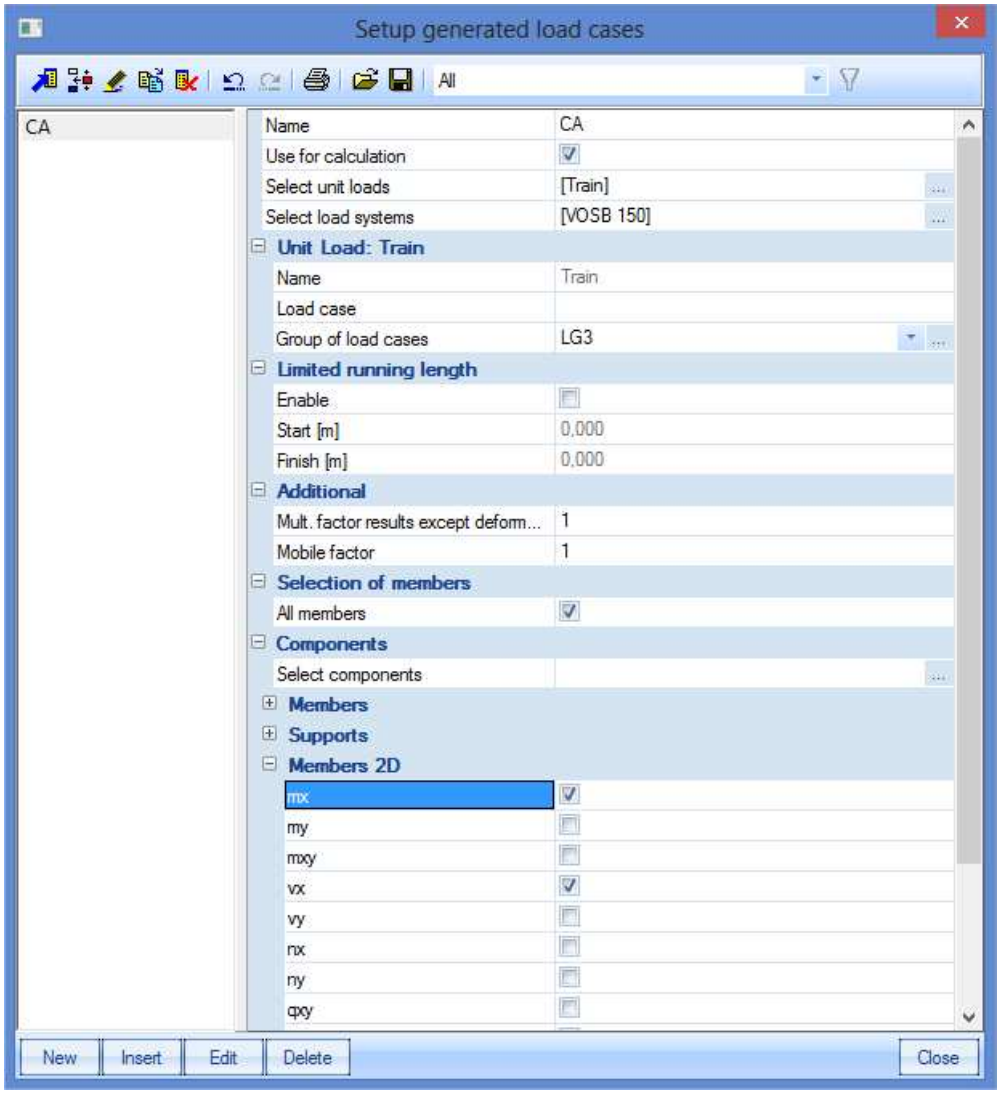
In this example it is clear that the distance between both load groups is always 17m, as set at the VOSB 150 load system.

Generate load cases – Envelope load cases

In this example, the envelope load cases will be generated for the moment mx and the shear force vx. After drawing up the envelopes, a selective exploitation is performed in a point from the bridge deck.

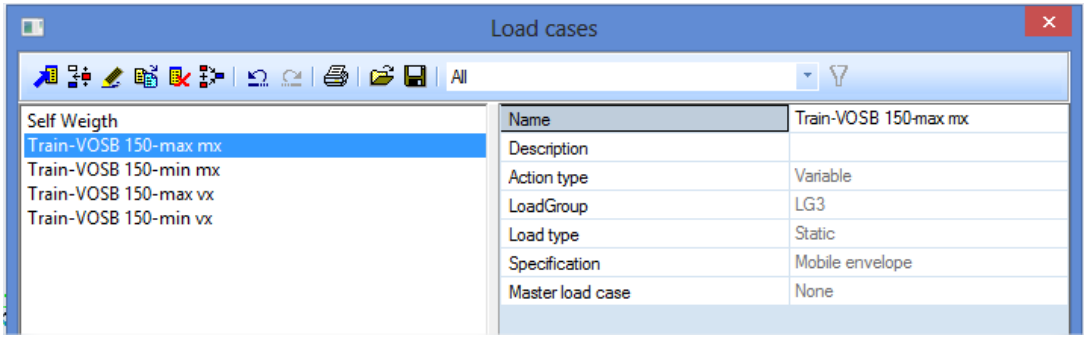
Generation Envelope Load Cases

The option **Setup generated load cases** is used.



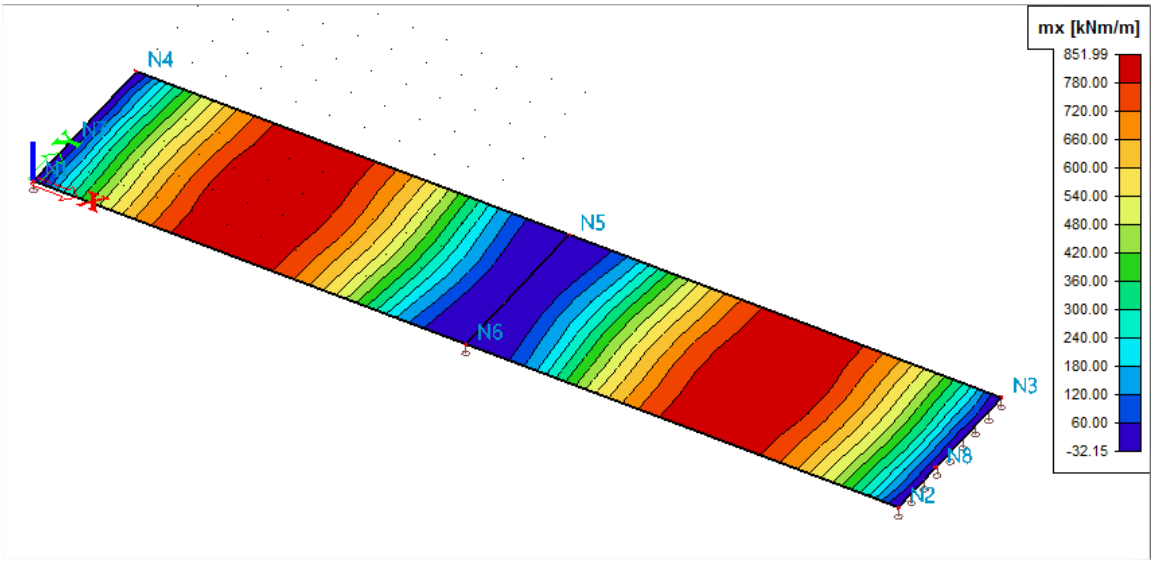
After entering these data, a linear calculation can be performed so the enveloping load cases are created.

After the linear calculation, the following load cases are generated:

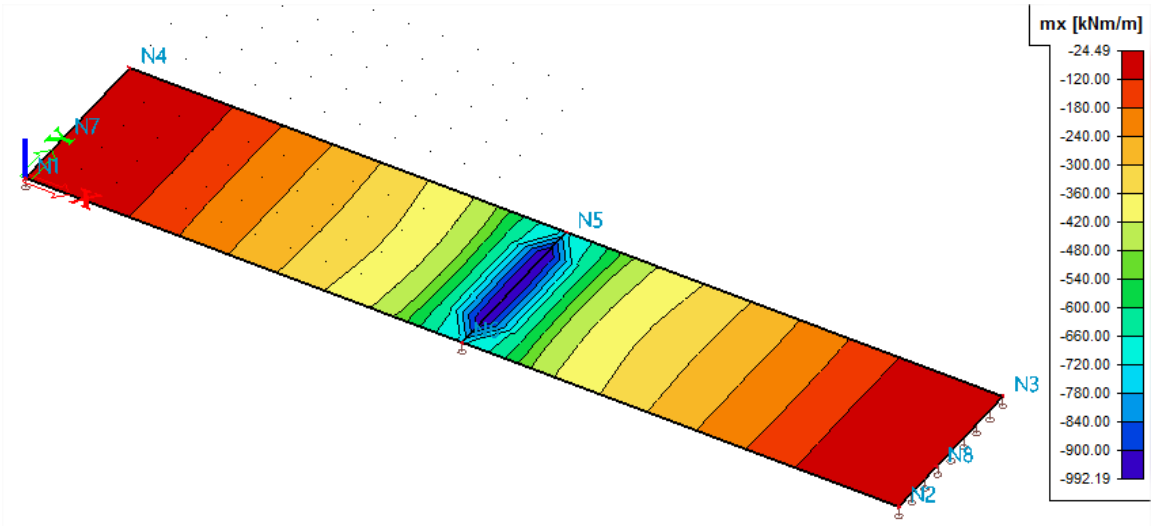


Subsequently the results of these envelopes can be viewed for e.g. the moment mx:

Max mx:



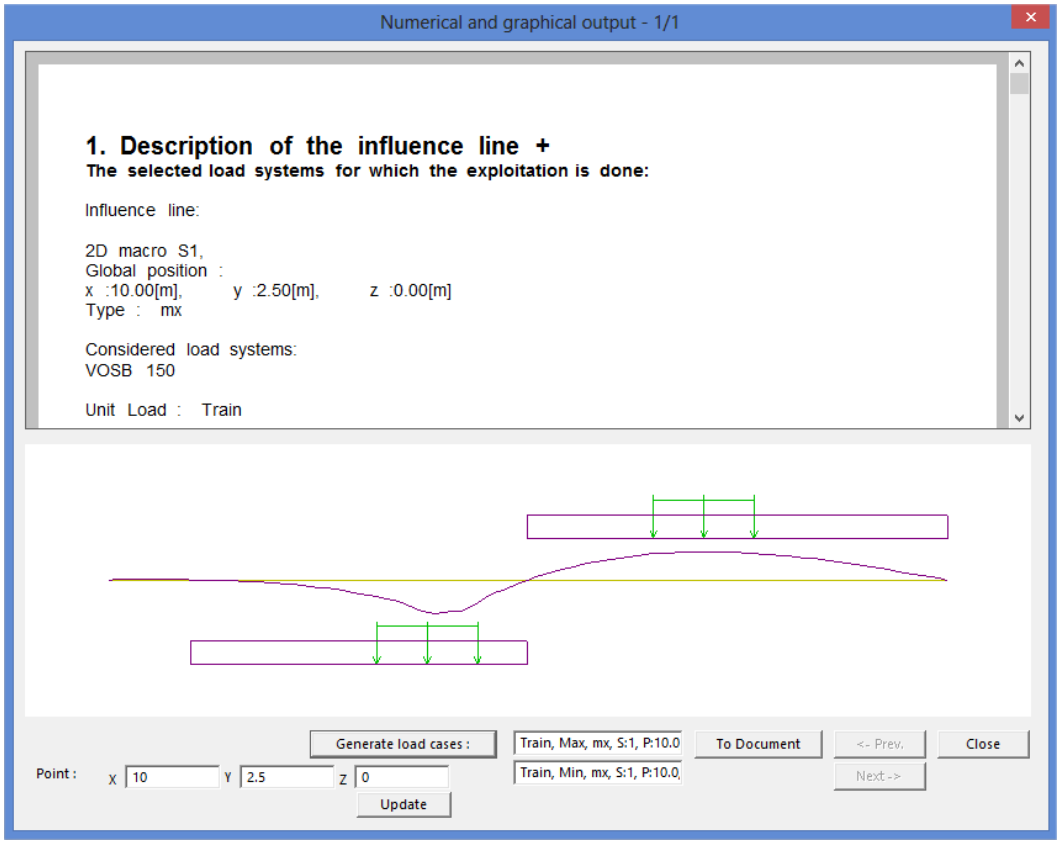
Min mx:



Generation of load cases

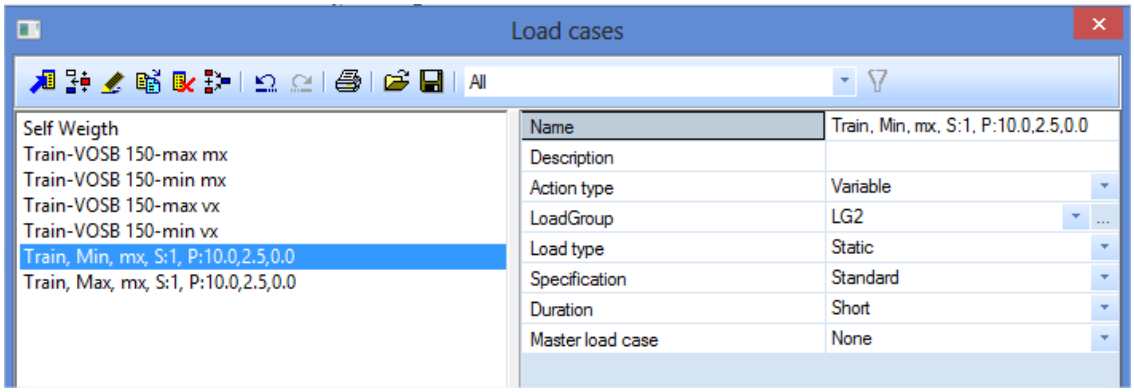
After setting the envelopes, a selective exploitation is performed for the moment, indicated on position **(10 ; 2,5 ; 0)**.

The option **Load case – generate** had to be marked in the Detail Analysis.
After activating this option, a **Single Check** is performed on the bridge deck and the desired position is set.



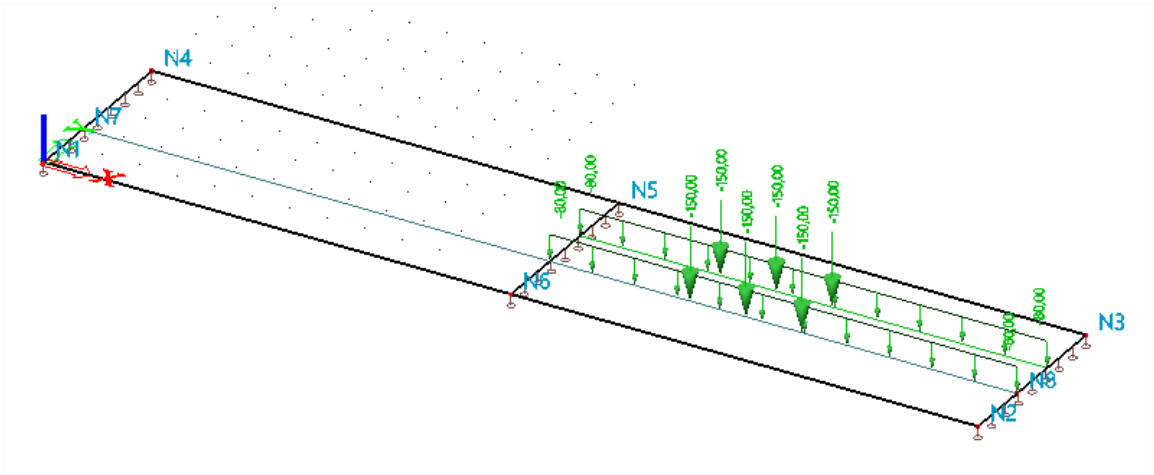
Through **Generate load cases**, the load cases are generated.

In the Load cases manager, a description can be added to these load cases:

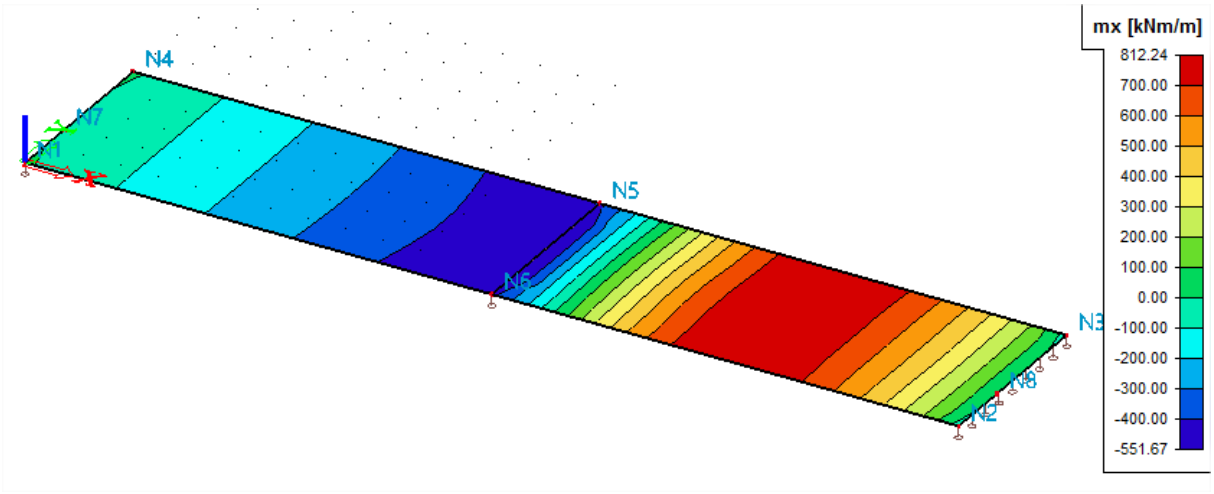


After rerunning the linear calculation, the results for these generated load cases can be viewed.

Load case Max, mx:



Results max mx:



References and literature

[1] Eurocode 1
Actions on structures
Part 1-4: General actions - Wind actions

[2] Eurocode 1:
Actions on structures
Part 1-3: General actions- Snow loads