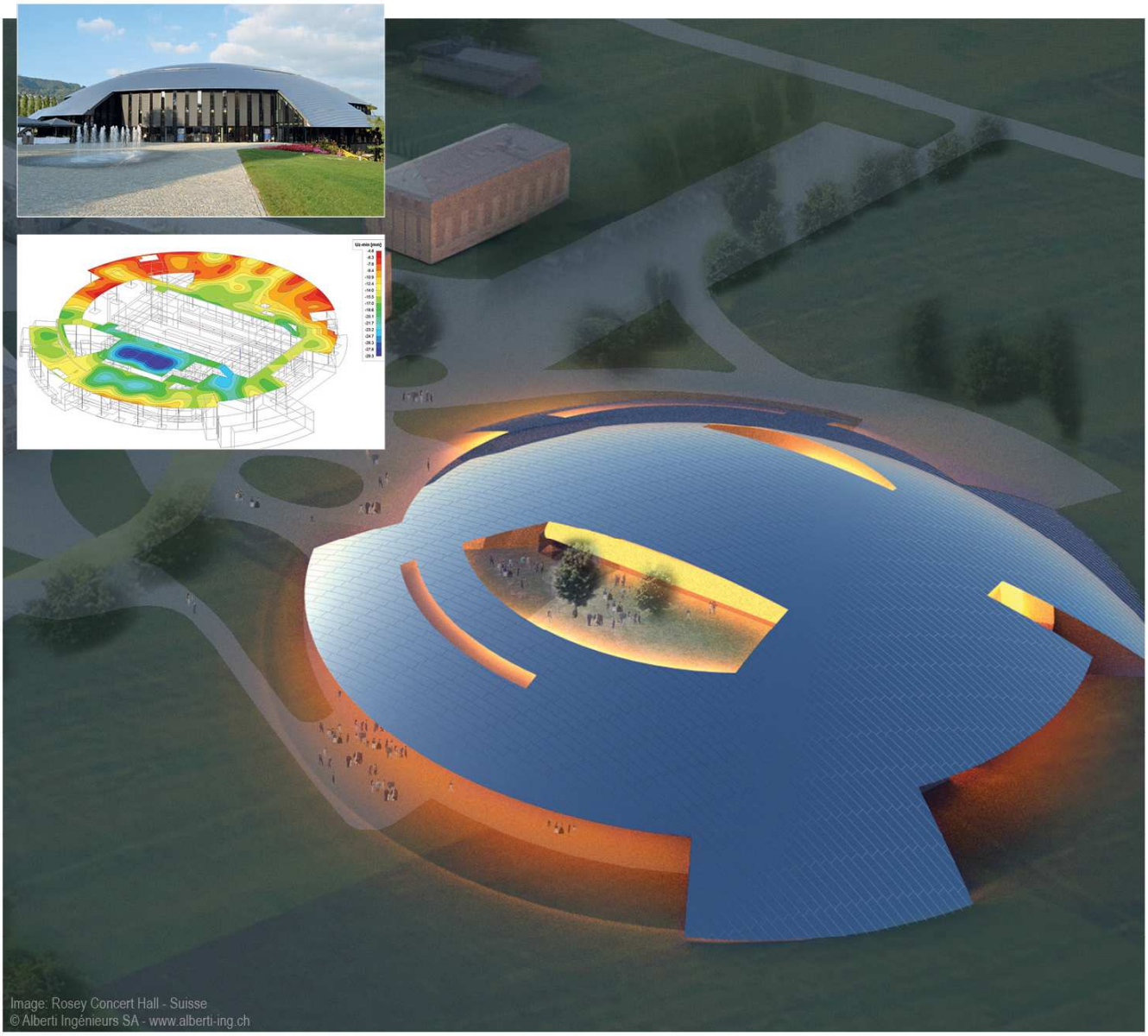
SCIAENGINEER



Advanced Professional Training

Mobile Loads

Table of contents

Introduction

The examples in this manual can be made in a full licensed as well as in a tryout or student version of SCIA Engineer.

Here follows an overview of the required SCIA Engineer modules / editions, per subject:

- **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)

In this course the 'Mobile loads' functionality will be examined in detail. With this functionality, mobile load systems, connected to a track, 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.

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:

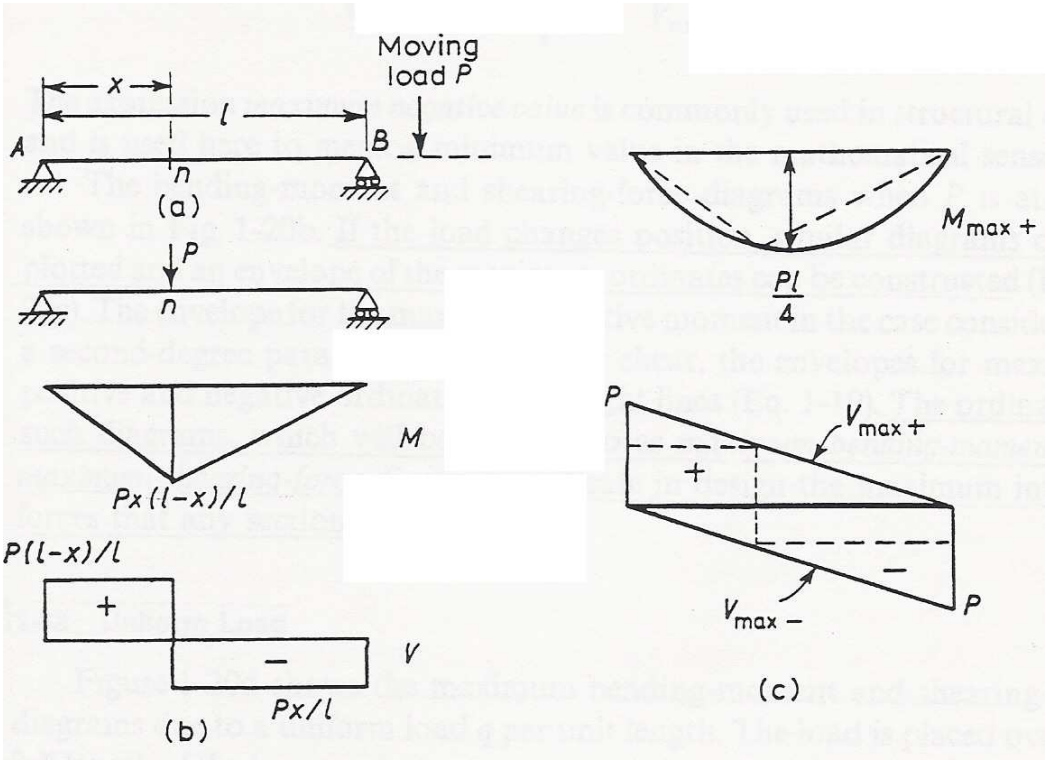


Figure (a) represents a simple beam on 2 supports, across which a concentrated load P can move.

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.

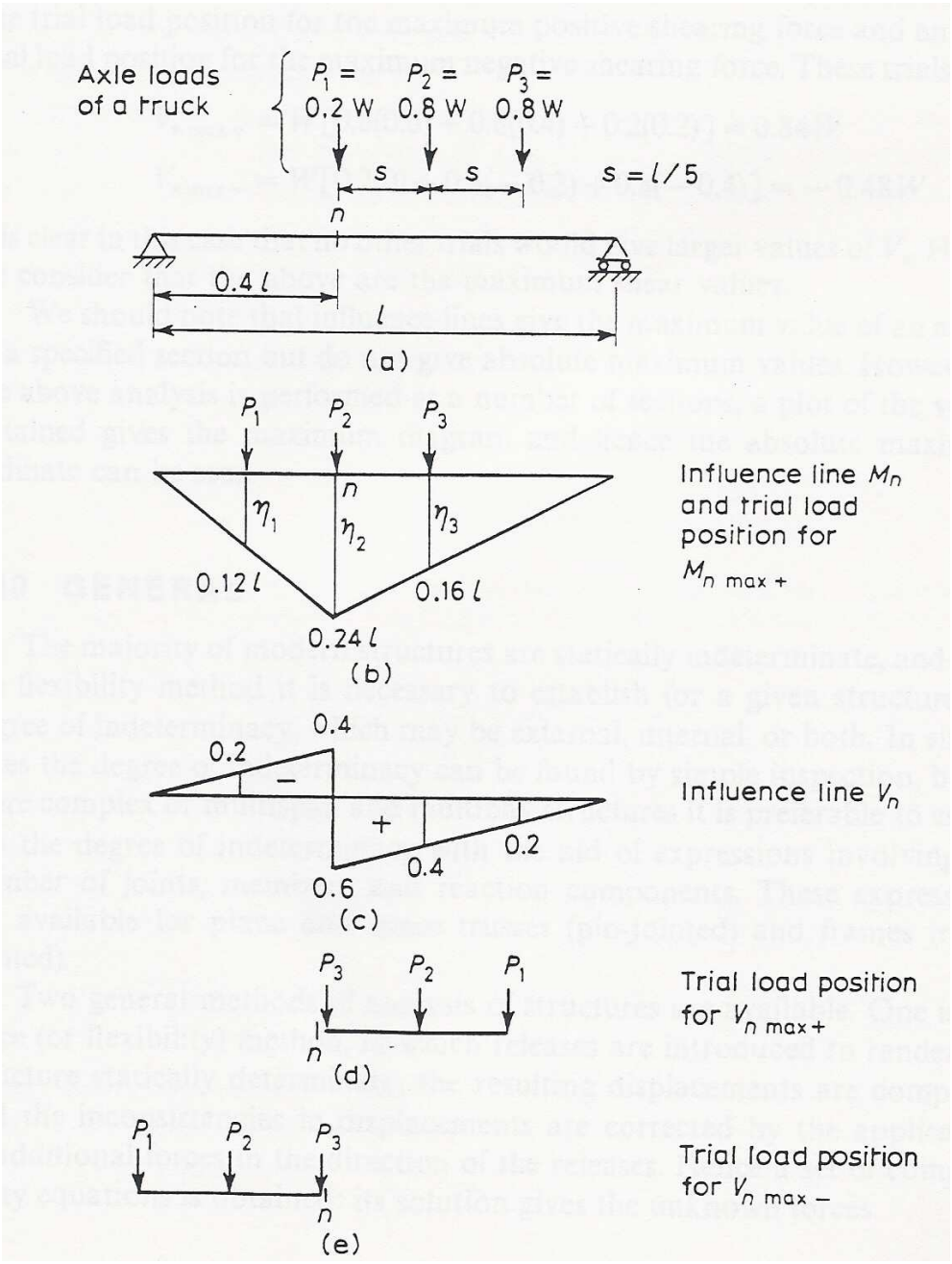


Figure (a) represents a simple beam on two supports again. Across the beam, a system of three point loads can move which represent e.g. the axis loads of a lorry. We look for the position of the load system for which the moment and the shear force are maximal in the section “n”.

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.

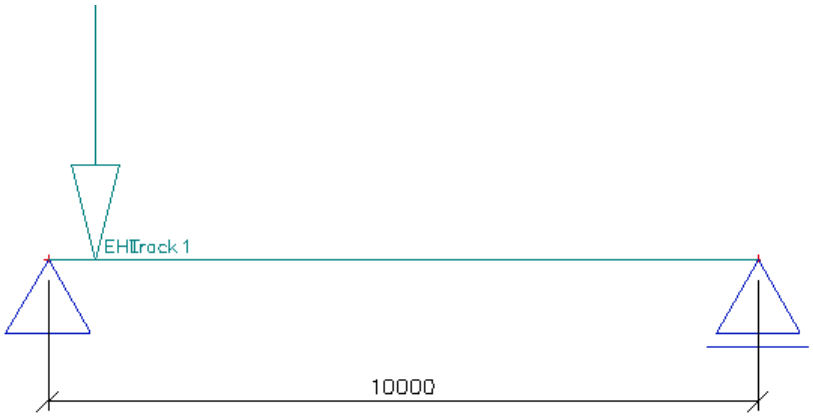
Figures (d) and (e) show the positions of the load system for the maximal positive shear force and the maximal negative shear force.

In SCIA Engineer these various steps appear as follows:

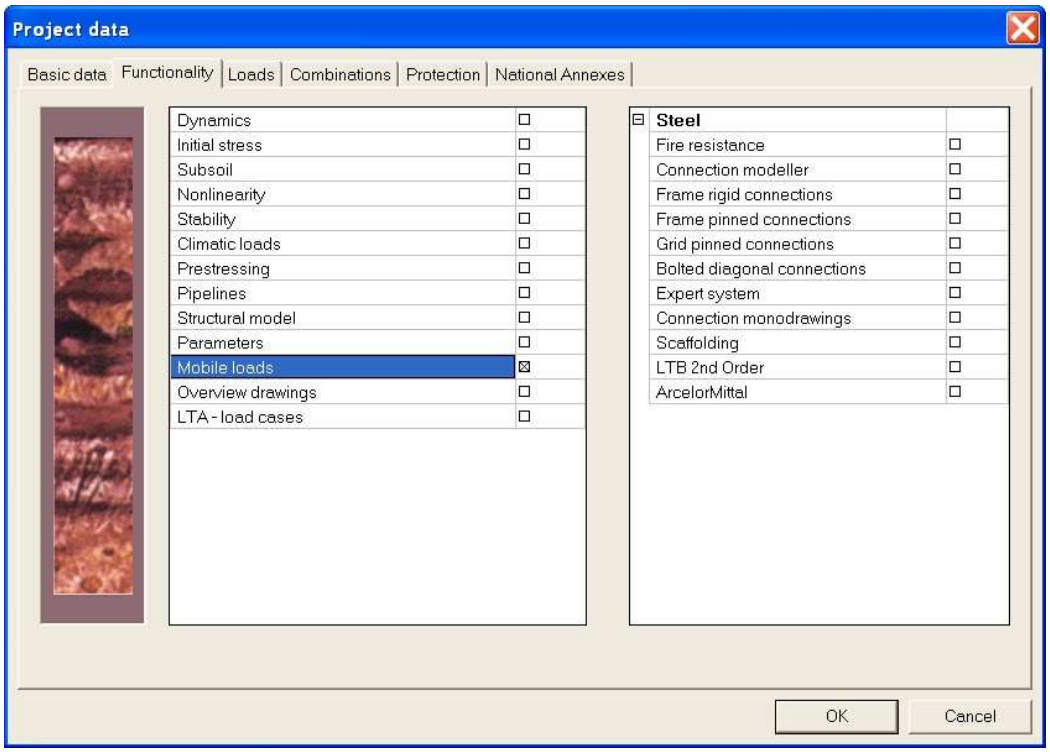
- Input Track across which a Unit load can move
- Input Unit load
- Representation Influence lines
- Input Load system
- Exploitation in a point at which the Load system is linked to the Unit load
- Generation load case for exploitation in a point
- Generation enveloping load cases to gain insight in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

Project M1: Influence lines

In this first project a simple beam is modelled on 2 supports. By means of the module Mobile Loads, a track and a unit load are defined on this beam so the influence lines of the various design components can be reviewed.

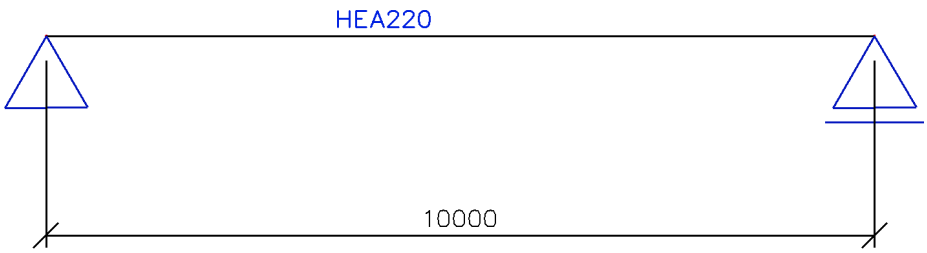


a) Project data



b) Construction

The construction can be imported as Beam at which the begin node is imposed hinged and the end node rolled.



c) Load

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

d) Input track and unit load

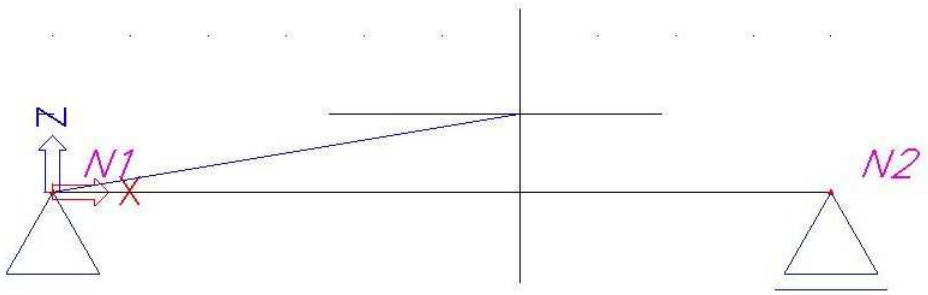
Subsequently the menu Mobile loads can be opened.

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 track has to be defined first, across which a unit load can move.

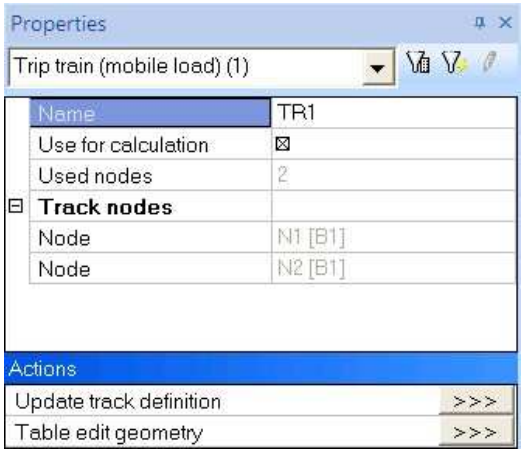
You can import this track through New mobile load track.

The program defines the track as a polyline. As a starting point of the track, the node **N1** is indicated, as an end point the node **N2**.



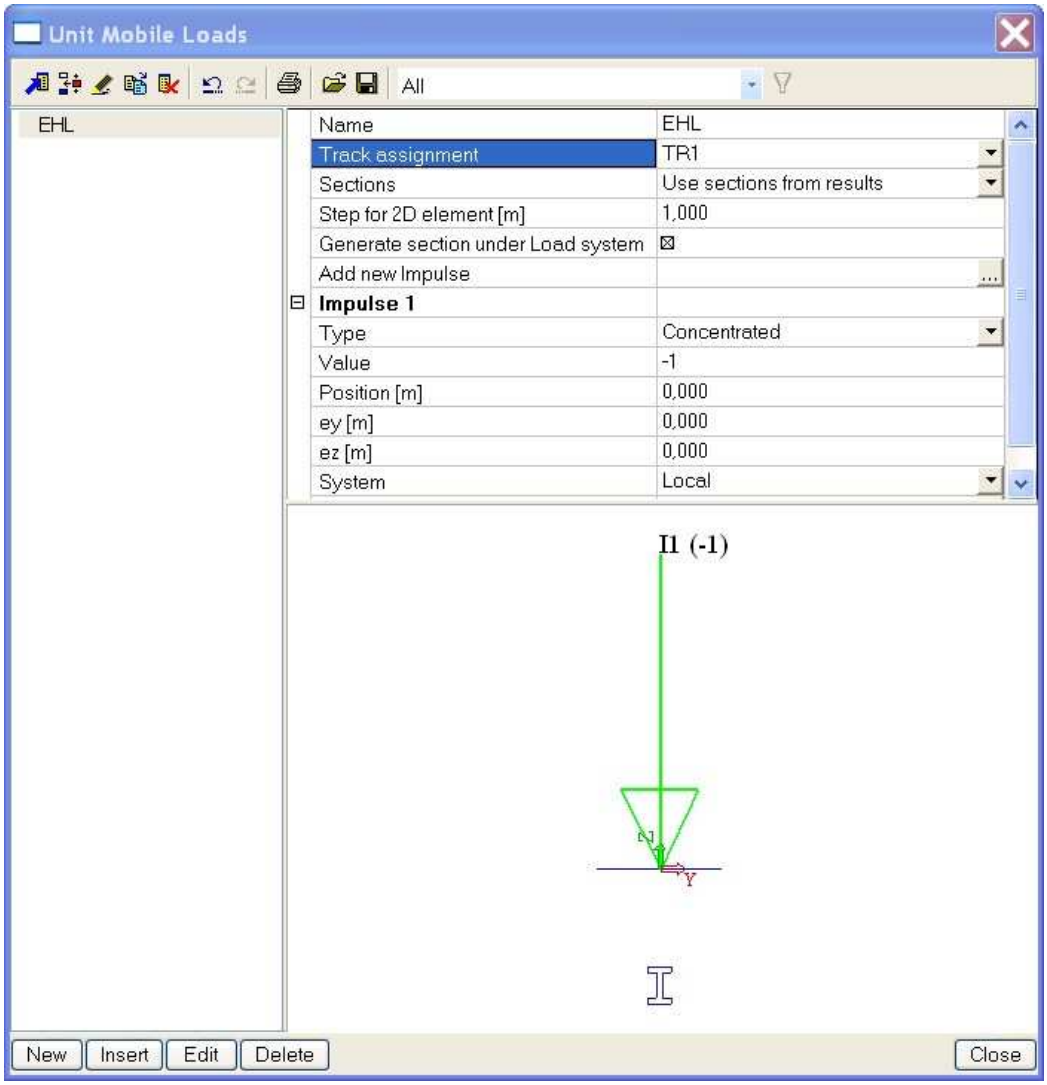
If the track moves across several members, it is important to indicate sufficient nodes.

The property window shows which nodes are used in the track. As a **Name** for the track **TR1** is entered.



The option **Use for calculation** shows that this track is taken into account in the calculation. If more tracks are defined, this is way can be used to show which tracks really have to be calculated
The action **Update track definition** allows generating the imported track again if e.g. the coordinates of a node were adapted. That way the track doesn't have to be imported again after an adjustment of the geometry.

After defining the track, the unit load can be imported through the menu Unit loads.



Through the option **Track assignment** you can indicate on which track the unit load needs to appear.

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 at Solver setup.

Use step according 2D element
The unit load is positioned with the step entered of "*step for 2D element [m]*". If a 2D 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 2D 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.
By default 1 impulse is made. In other examples also more impulses are used. The distance between two impulses is always perpendicular to the track.

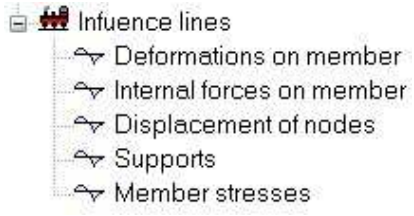
In this example the default settings are kept so the concentrated mobile unit load with value **-1** is defined.

The **Name** of the unit load is by default **EHL,** which is kept for this example.

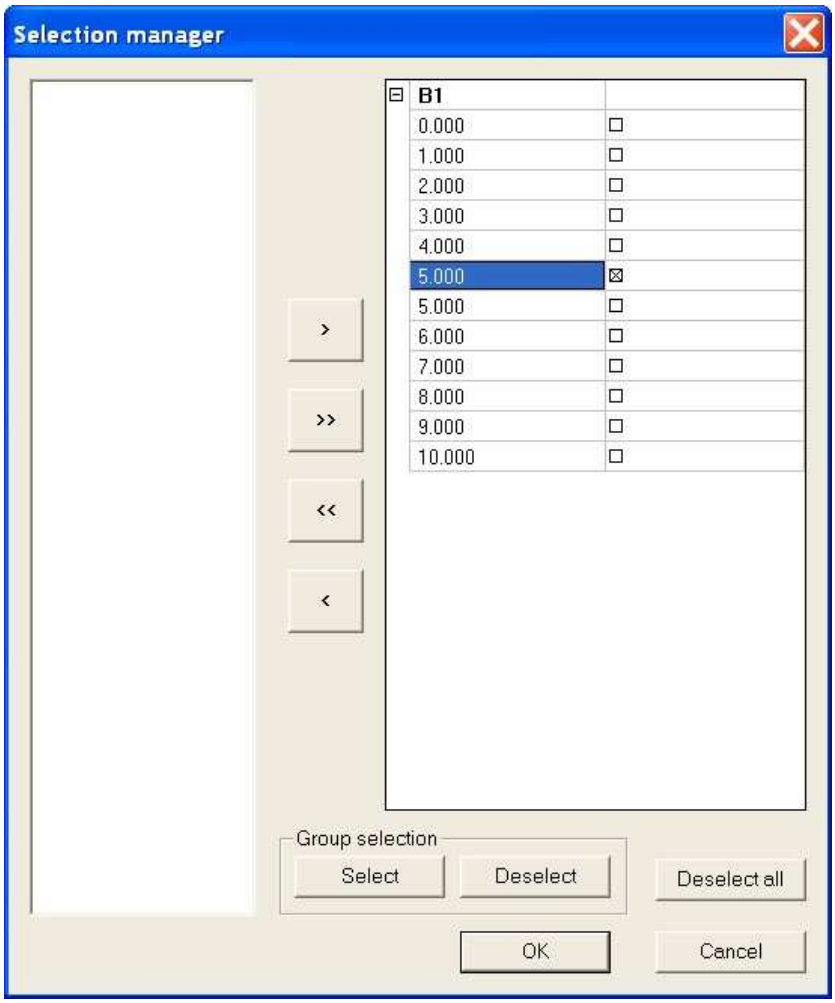
e) Influence lines

After defining the track and the unit load, the linear calculation can be started. To do this it is not necessary to exit the menu Mobile Loads, but you can use the button Calculation in the project toolbar.

After the calculation a new group appears in the menu Mobile Loads:



When choosing a result group, you have to indicate on which member and in which section the results have to be shown through the **Selection Tool**.



The **Preview** shows the results numerically.

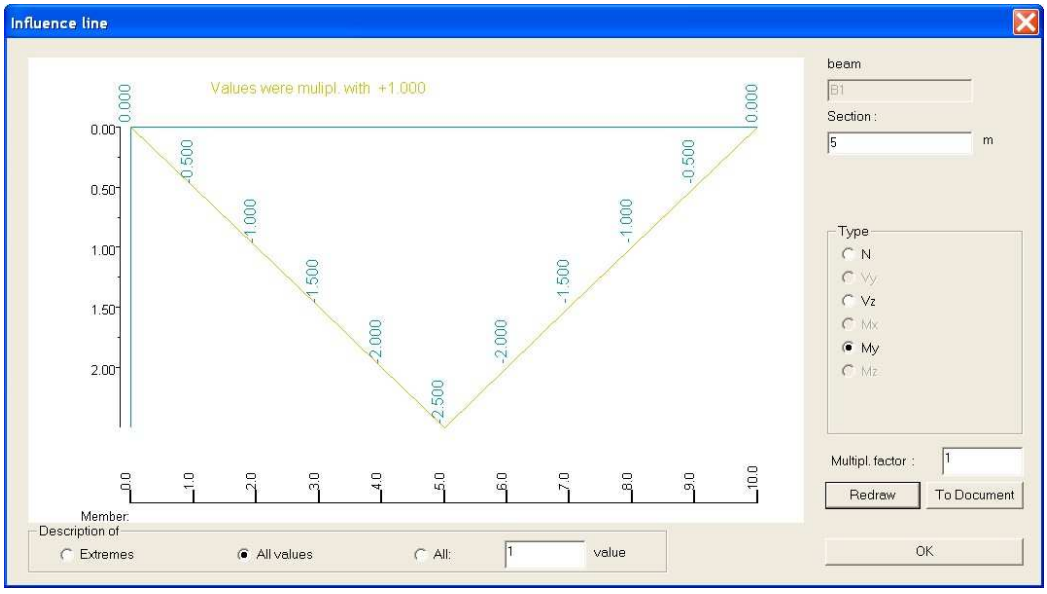
Influence lines - Internal forces on beam

Invloedslijnen - intene krachten op staaf

Influence line for internal forces on beam
EHL over TR1 - beam B1 - section x=5.0m
Multiply factor : 1.00

poz	N	Vz	My
0.00	0.000	0.000	0.000
0.00	0.000	0.000	0.000
1.00	0.000	-0.100	+0.500
2.00	0.000	-0.200	+1.000
3.00	0.000	-0.300	+1.500
4.00	0.000	-0.400	+2.000
5.00	0.000	-0.500	+2.500
5.00	0.000	+0.417	+2.500
5.00	0.000	+0.417	+2.500
5.00	0.000	+0.500	+2.500
6.00	0.000	+0.400	+2.000
7.00	0.000	+0.300	+1.500
8.00	0.000	+0.200	+1.000
9.00	0.000	+0.100	+0.500
10.00	0.000	0.000	0.000
10.00	0.000	0.000	0.000

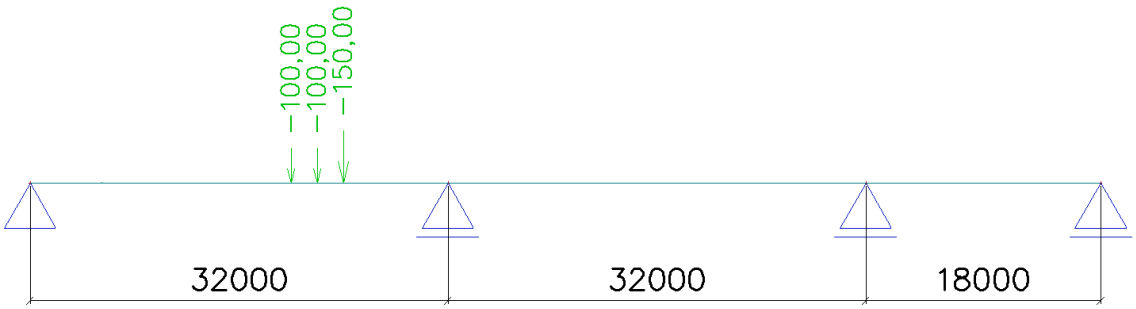
Through the action button **Single Check** the influence line can be shown graphically.



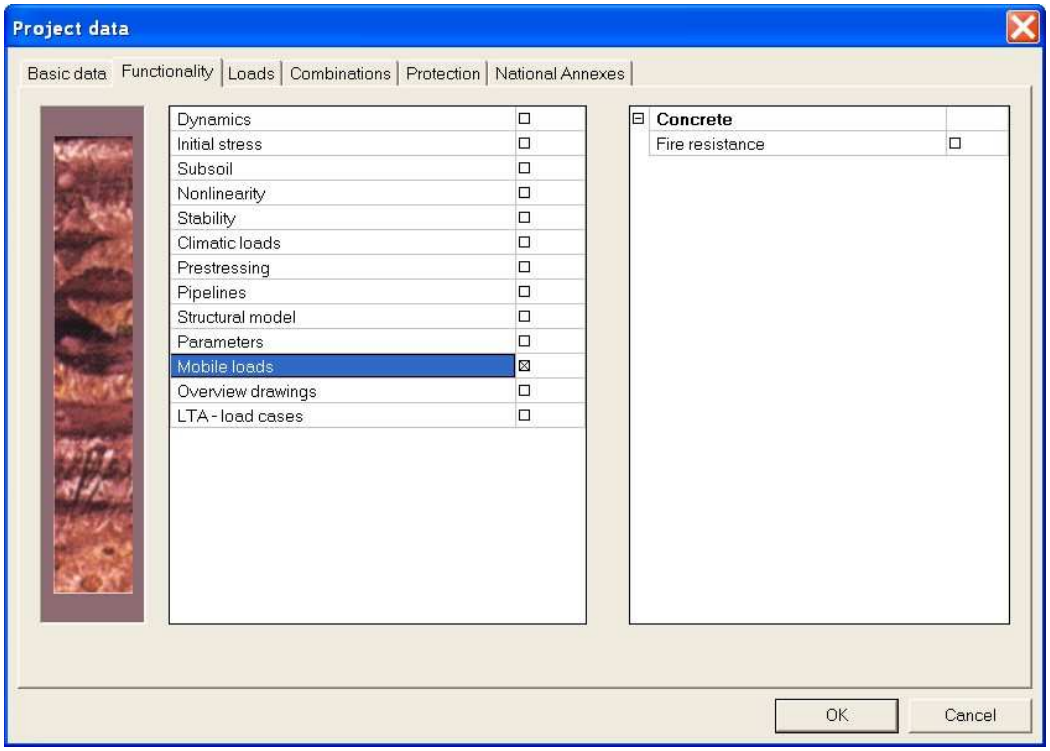
In the field **Multiplication factor**, a proportionality factor can be set. With the button **To document**, the numerical results are sent directly to the document.

Project M2: Load system

In this project a bridge deck is modelled on several supports. After defining a track and a mobile unit load, the various load systems are linked to the unit load.
Through a selective exploitation, the load cases are automatically generated 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.

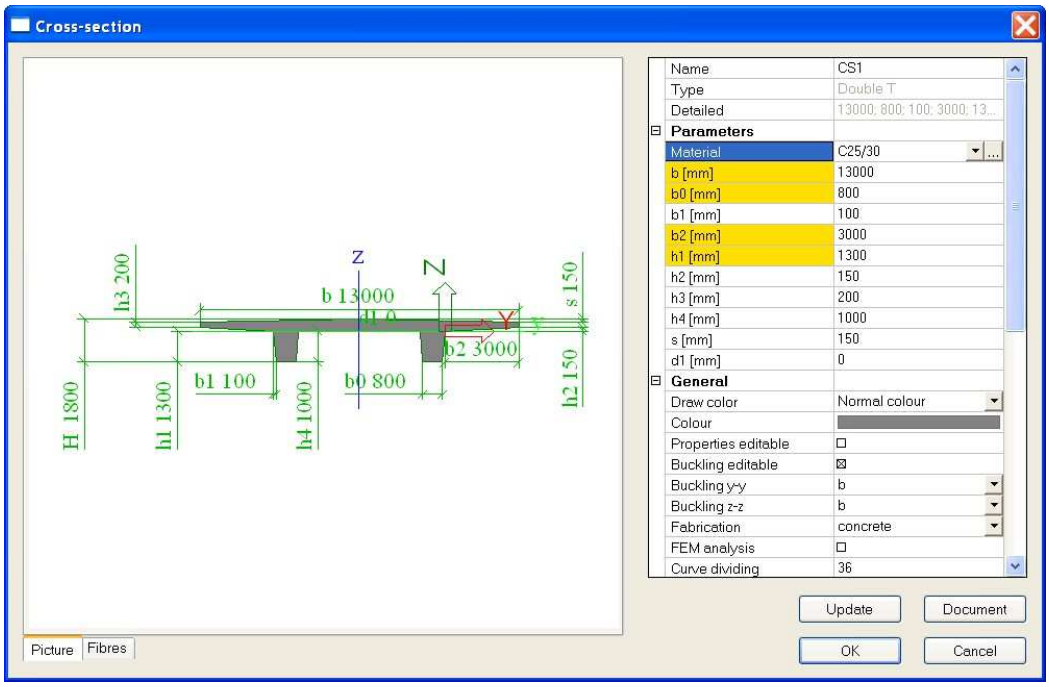


a) Project data

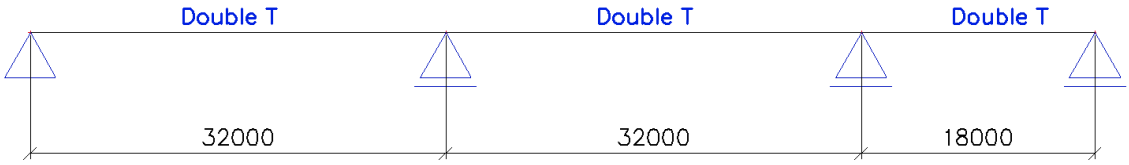


b) Construction

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



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



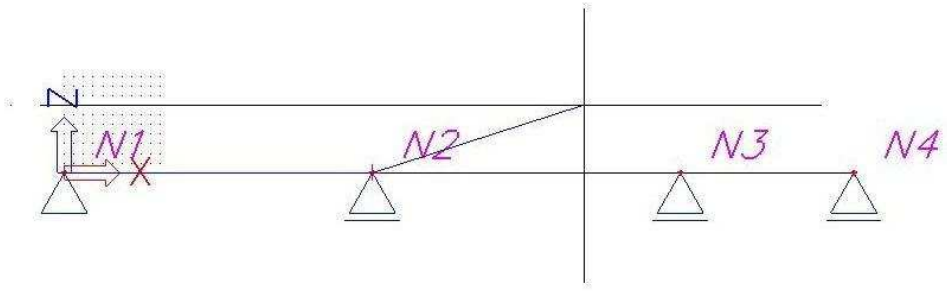
c) Load

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

d) Input track and unit load

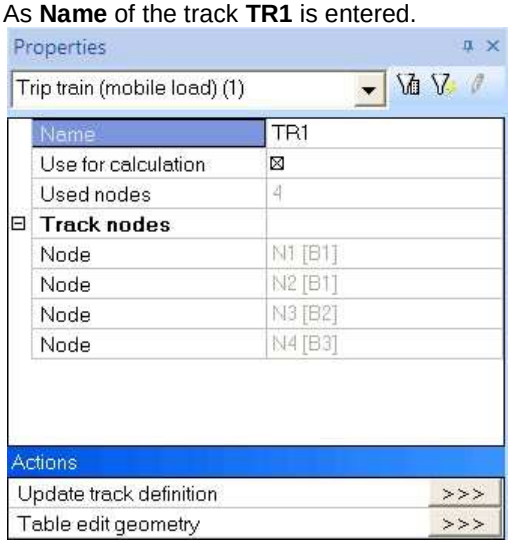
After entering the construction, the menu Mobile loads can be opened.

Through New mobile load track a track can be defined from node **N1** to node **N4**.

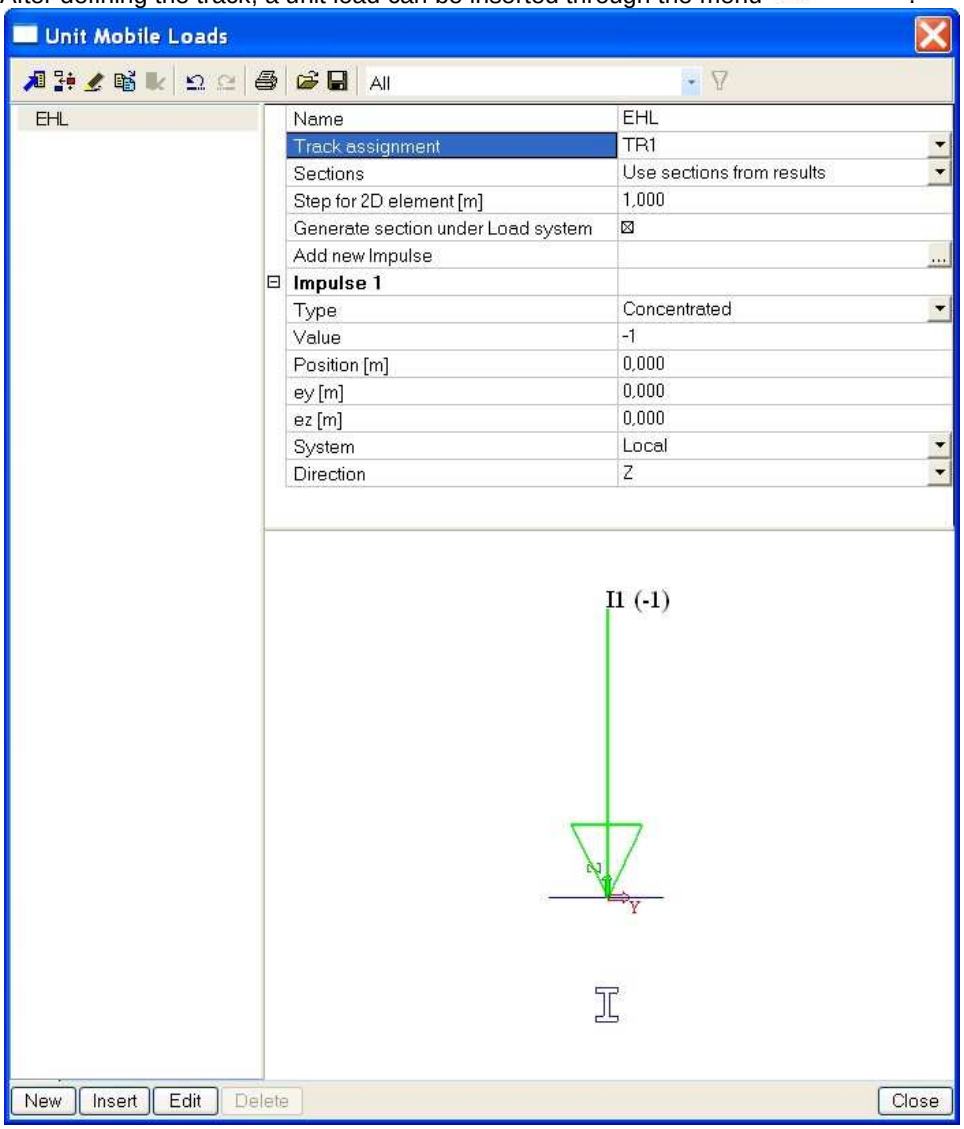


The property window shows the nodes that are recognized by the track:

As **Name** of the track **TR1** is entered.



After defining the track, a unit load can be inserted through the menu Unit loads.



e) Input load systems

By means of the unit load, the influence lines for the construction can already be generated. SCIA Engineer also allows linking this unit load to a load system.

The input of the load systems occurs through the option Load System Database.

Both Single and Multiple Load systems can be defined.

Possibilities with Single Load systems:

- A coherent combination of point loads (e.g. vehicle)
- Line loads of an indefinite length (e.g. pedestrians)
- A combination of both

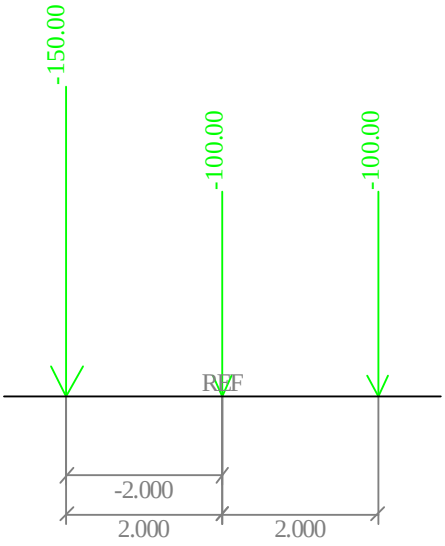
Possibilities with Multiple Load systems:

- A line load with a definite length in combination with a line load of an indefinite length.
- Two similar independent systems of point loads with variable interval in combination with a divided load of an indefinite length.
- Three or more independent systems of point loads with a fixed interval in combination with a divided load of an indefinite length.

In this project the following three load systems are considered:

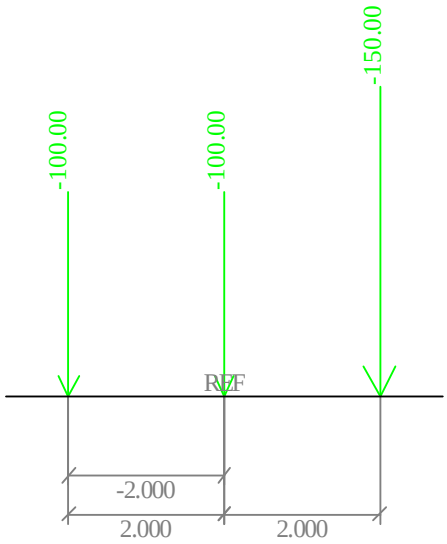
*1) Single Load system **P Loads left***

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.



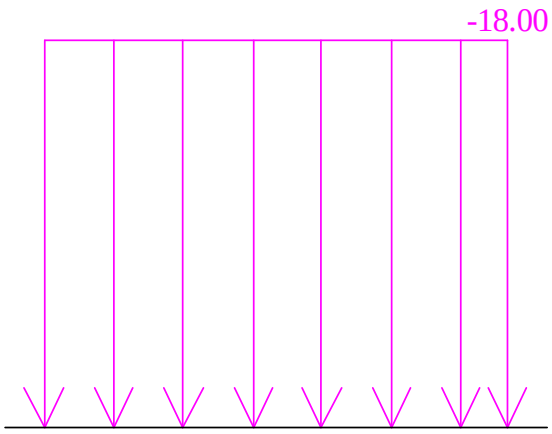
*2) Single Load system **P Loads right***

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.



*3) Single Load system **Q Load***

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

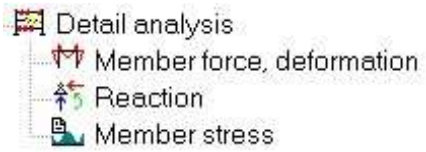


When entering a Single Load system, you have the possibility to mark the option **Neglect point load with opposite influence**. If this option is activated, the complete concentrated load, which lies in the negative area of the influence line, will be taken in account in the calculation. By activating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

In this project, the option is not activated.

f) Exploitation of load systems

After defining the mobile unit load and the load systems, the linear calculation can be started through the button Calculation in the project toolbar.
After the calculation a new group appears in the menu Mobile Loads:

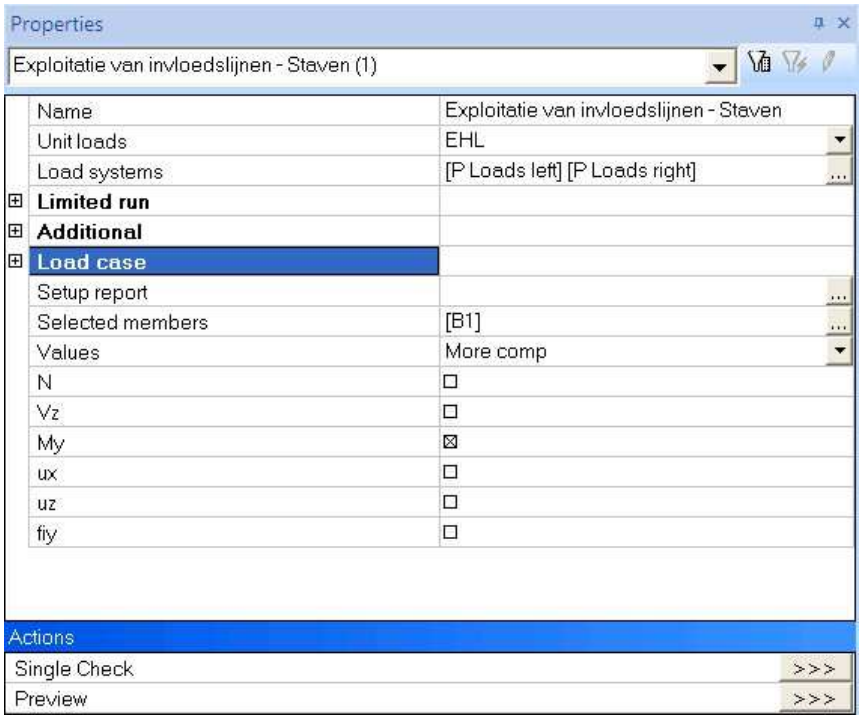


With the **Detailed Analysis**, the load systems can be linked to the unit load. For every desired position on the structure, between all the selected tracks, SCIA Engineer determines the system that is most adverse for the chosen design parameter.

This is illustrated for 2 different cases.

Case 1
An exploitation is performed for the moment **My** on a position **24m** on the first beam **B1**. The exploitation is performed for the load systems **P Loads Left** and **P Loads Right**.

In the Property window these options can be adjusted:



The advanced options **Limited run**, **Additional** and **Load case** are discussed further in this course.

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

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 Loads left
P Loads right
Unit Load : EHL

2. Co-ordinates of the nodes of the load track 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	43.527
2	-34.564
3	2.722

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 : P Loads left

Sum P [kNm]	Sum Q [kNm]	X1 [m]	X2 [m]
-621.408	0.000	44.667	44.667

Positive maximum position : P Loads right

Sum P [kNm]	Sum Q [kNm]	X1 [m]	X2 [m]
1149.982	0.000	22.000	22.000

7. Results:
Negative maximum position : P Loads left

Description	Due to P	Due to Q	P + Q	Units
My negative	-621.408	0.000	-621.408	[kNm]

Positive maximum position : P Loads right

Description	Due to P	Due to Q	P + Q	Units
My positive	1149.982	0.000	1149.982	[kNm]

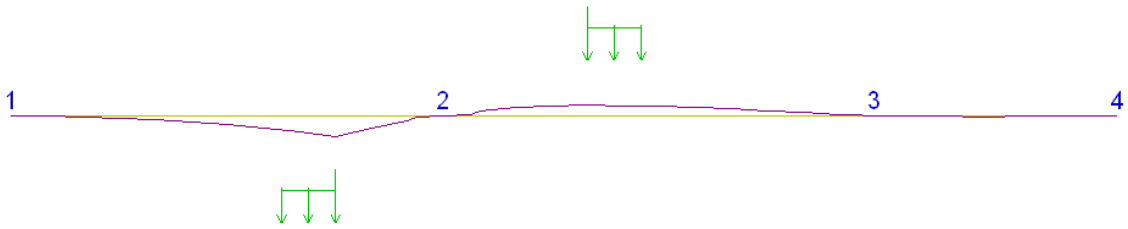
The parts that should be displayed in the report can be indicated through the options **Setup report**.

Under *Title 1.* you can see 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 **(-621,408 kNm**) on **24m** if the reference point of the load system **P Loads left** is located at **44,667m** from the begin point of the track.
My is maximal (**1149,983 kNm**) on **24m** if the reference point of the load system **P Loads right** is located at **22m** from the begin point of the track.
The values X1 and X2 are the same since a single load system is used.

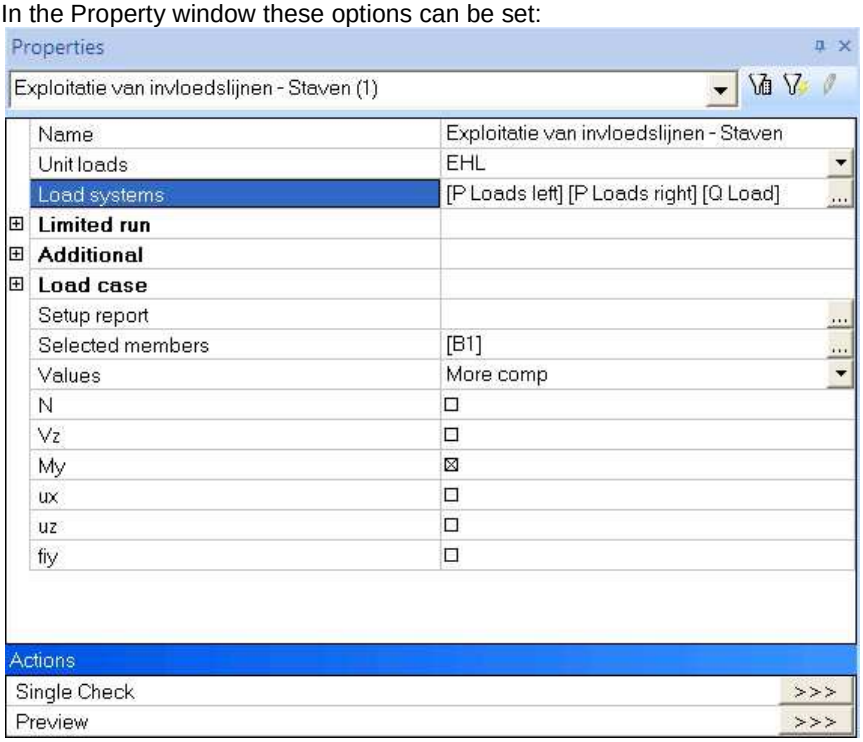
This result is also displayed graphically:



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

Case 2
An exploitation is performed for the moment **My** on a position **24m** on the first beam **B1**. The exploitation is performed for the load systems **P Loads left**, **P Loads right** and **Q Load**.

In the Property window these options can be set:



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

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 Loads left
P Loads right
Q Load
Unit Load : EHL

2. Co-ordinates of the nodes of the load track 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	43.527
2	-34.564
3	2.722

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	-622.150	0.000	0.000

Positive maximum position : P Loads right

Sum P [kNm]	Sum Q [kNm]	X1 [m]	X2 [m]
1149.982	0.000	22.000	22.000

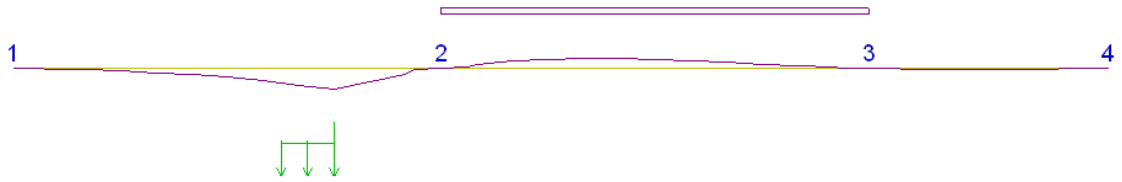
7. Results:
Negative maximum position : Q Load

Description	Due to P	Due to Q	P + Q	Units
My negative	0.000	-622.150	-622.150	[kNm]

Positive maximum position : P Loads right

Description	Due to P	Due to Q	P + Q	Units
My positive	1149.982	0.000	1149.982	[kNm]

This result is also displayed graphically:



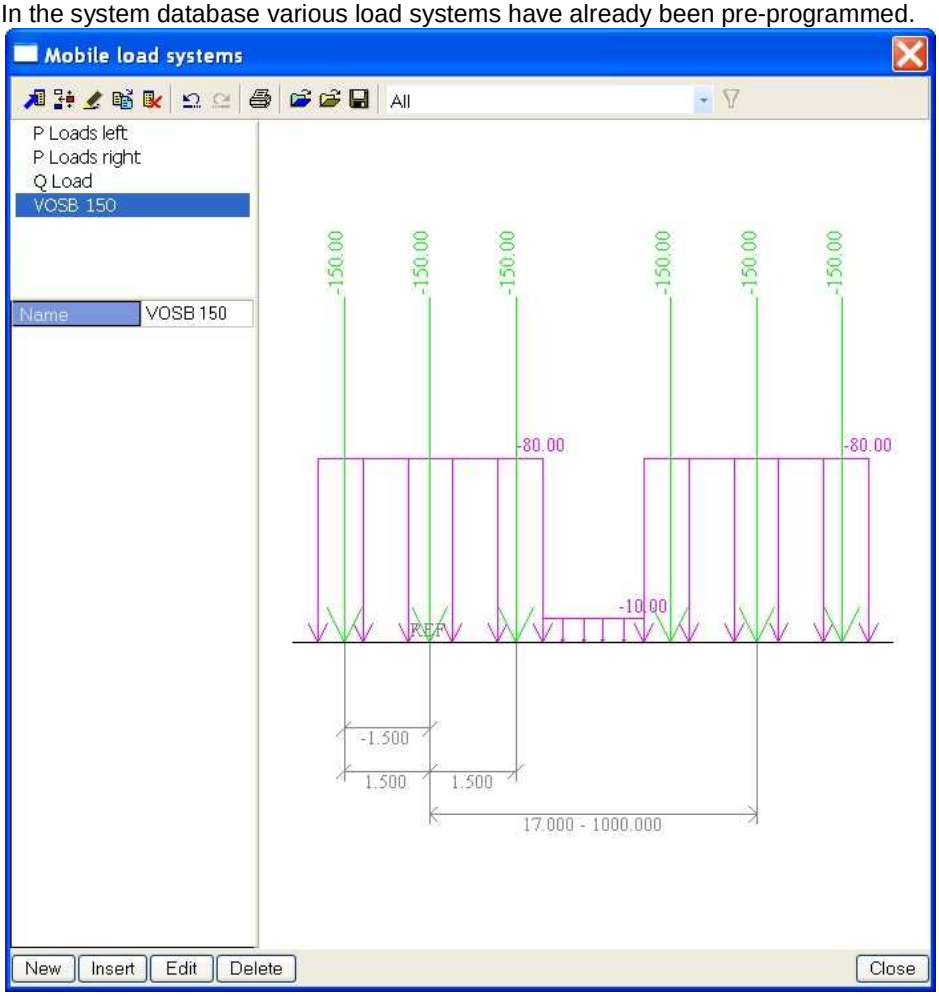
An influence line for a point of the construction is the representation of the amplitude of the design parameter in the point, if the unit load is moving across the structure. By placing the divided load on the places where the influence line has the same sign, an extreme result is obtained. In this example the moment My on 24m reaches a minimal value **-622.15 kNm** if the divided load is placed in the second field.

Remarks:
With an exploitation calculation various load systems can be selected. In the calculation, SCIA Engineer considers these load systems as individual.

To obtain an exploitation at which various systems are loading the structure at the same time, multiple systems have to be used.

In this project only one track is defined. Of course it also possible to define several tracks. With a calculation, at which various tracks and several load systems have been selected, the program considers every system on every track separately. The resulting extreme component comes from one of the systems on one of the tracks.

In the system database various load systems have already been pre-programmed.



g) Generation Load cases – Enveloping Load cases

SCIA Engineer allows making both single and enveloping load cases.

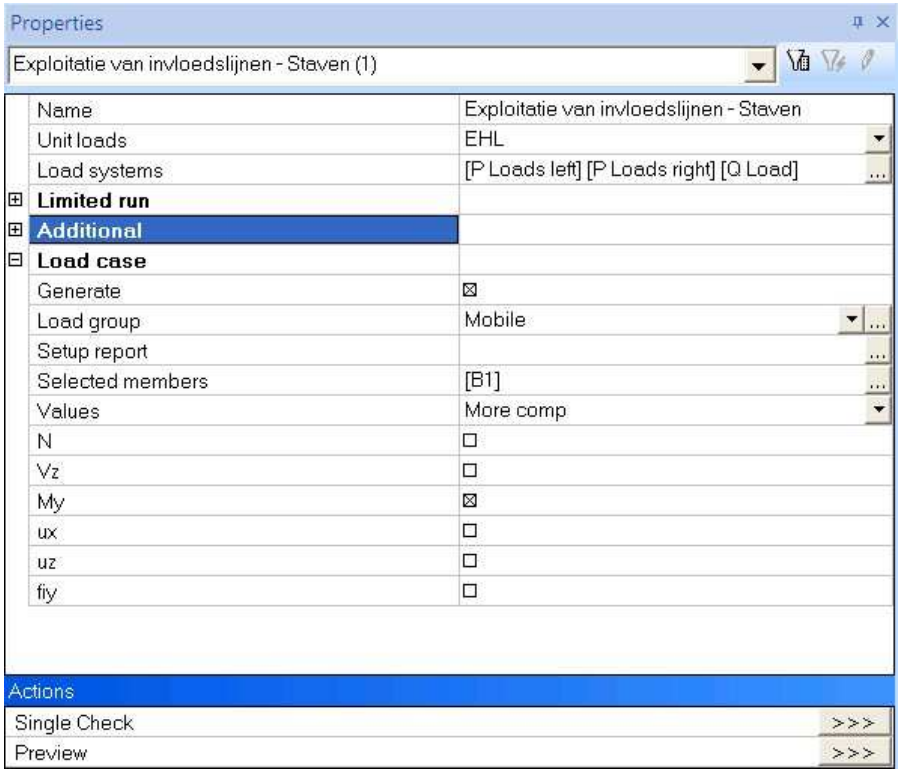
Generation Load cases

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

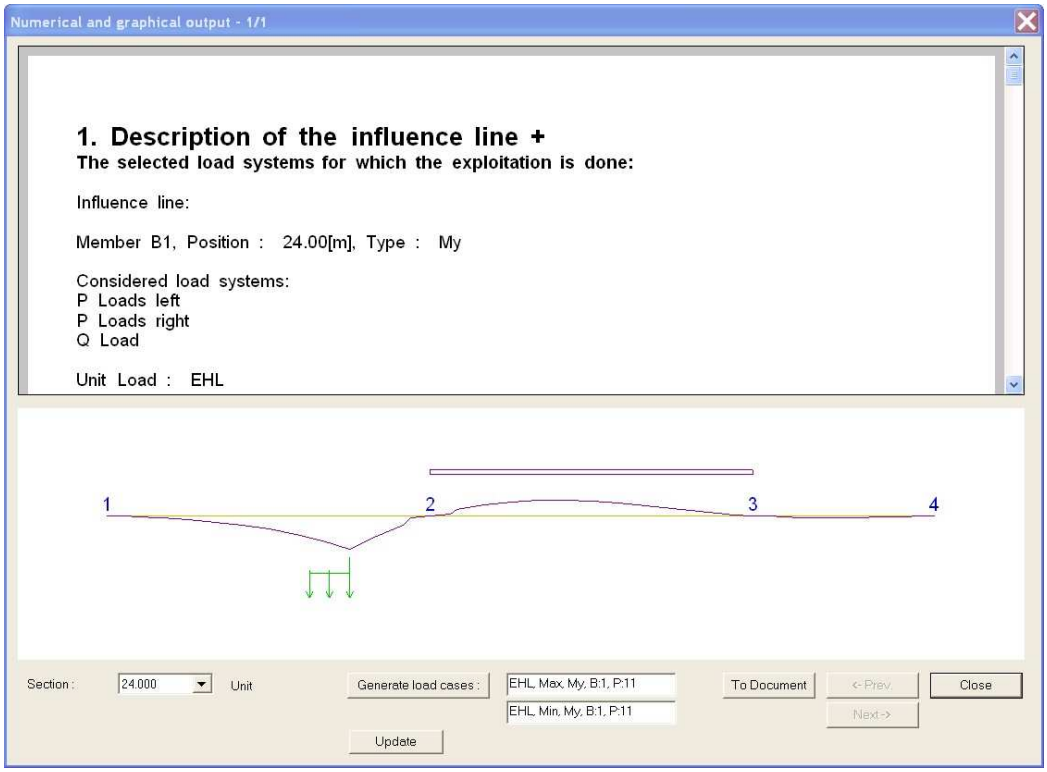
First of all the option Load case - generate has to be marked at the Detailed Analysis.

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

In this example it is applied on case 2, mentioned above:

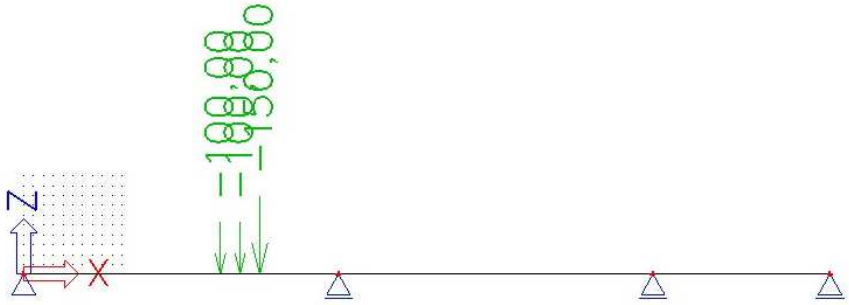


A load case Mobile is made.
After activating this option, a Single check is performed on the member B1 through the action Single check.

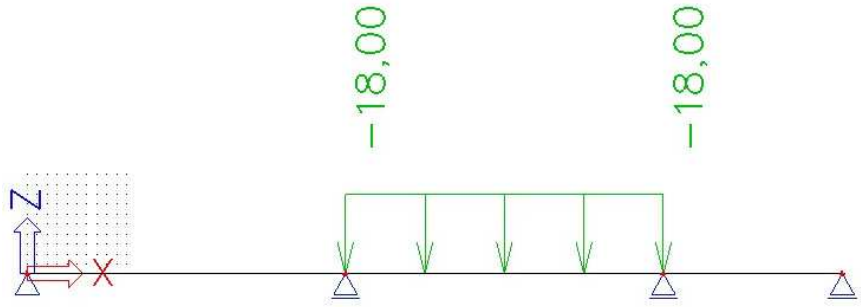


Through the button **Generate Load Cases** two load cases are generated, one for the minimal My on 24m and one for the maximal My on 24m. The parameter B indicates the member, parameter P the position on the member.
Since this option is used to make real load cases, the content of these load cases can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

Max My:



Min My:



After a linear calculation these load cases can be combined with other load cases and e.g. used for a steel check.

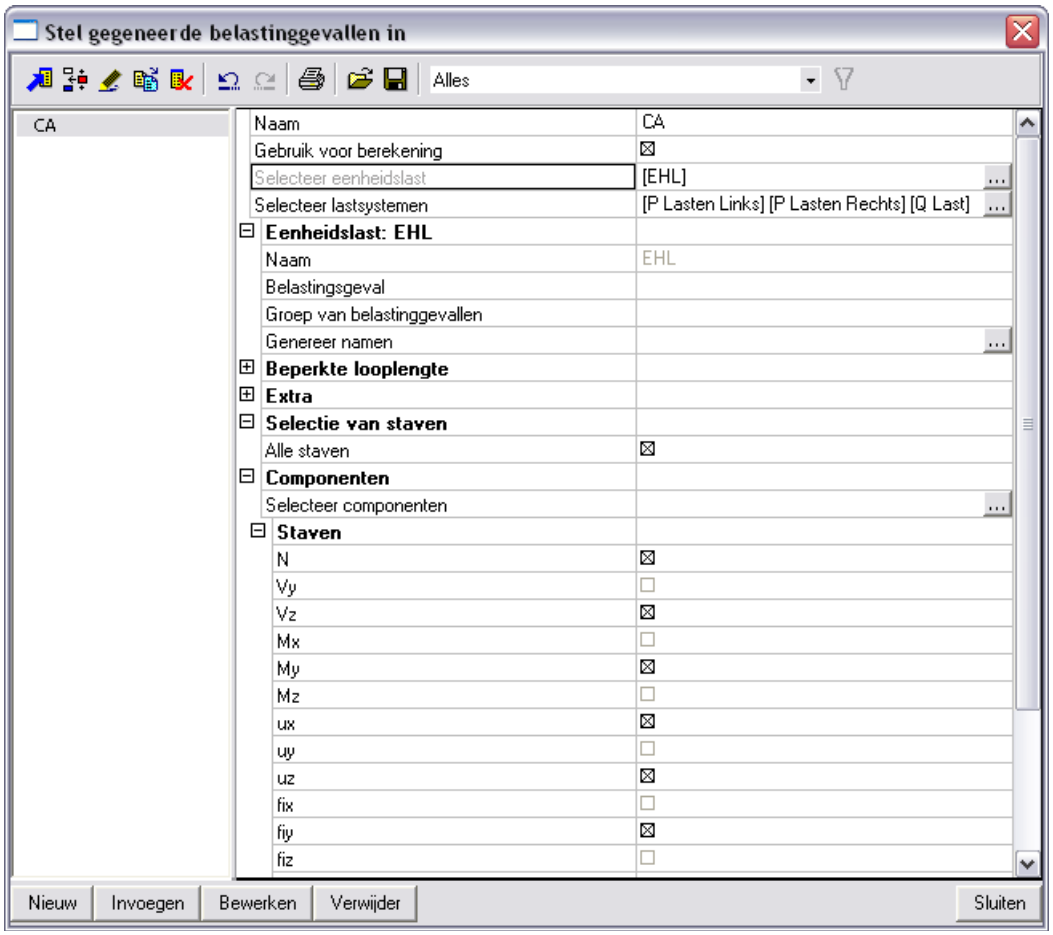
Generation Envelope Load cases

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 be able to generate such enveloping load cases, the option Setup generated load cases is used in the menu Mobile Loads.

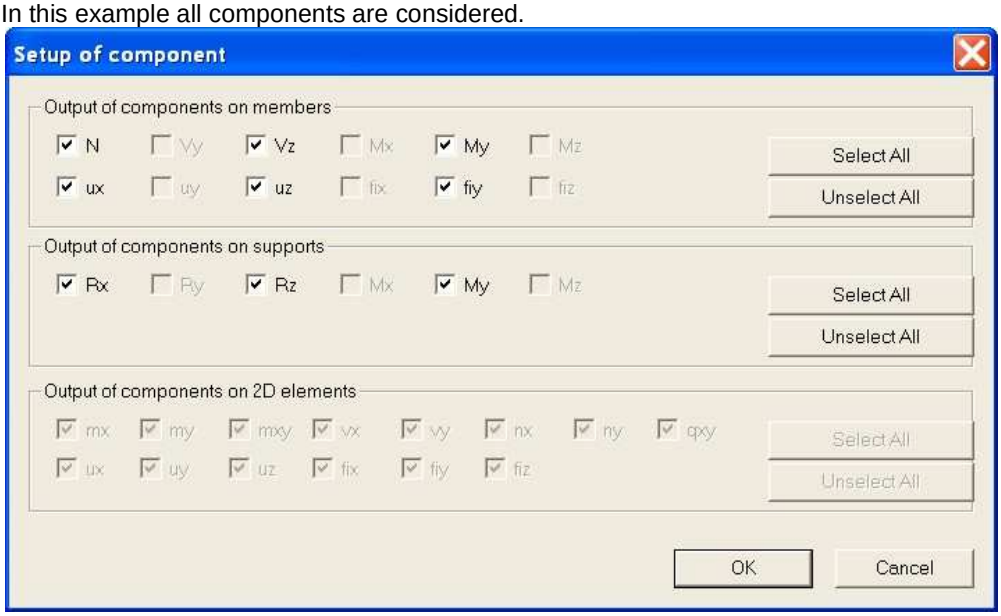


First of all you have to indicate which unit loads and which load systems have to be taken into account. In this example three imported load systems are selected.

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 and the name **Mobile** is selected for the load group.

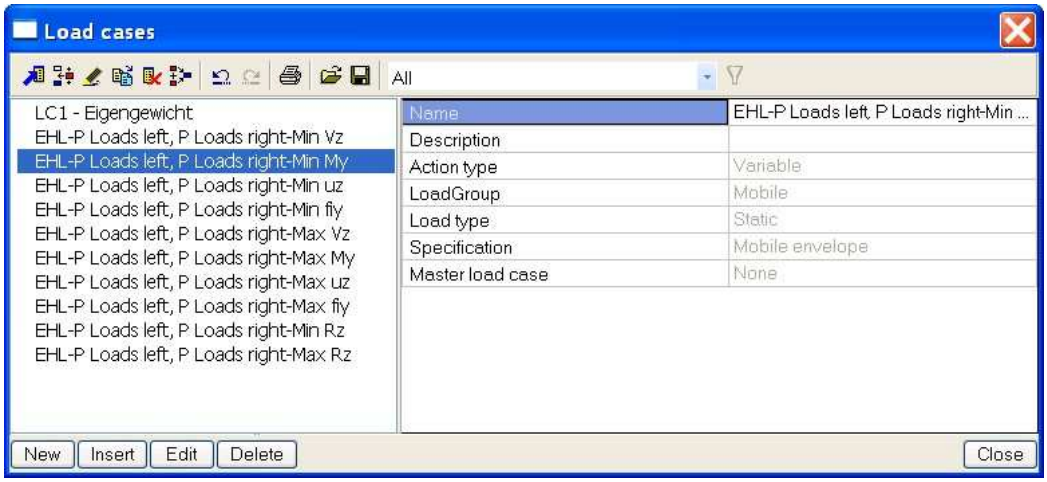
With **Selection of member** the option **All members** is marked, so all the members are taken into account in the calculation.

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.

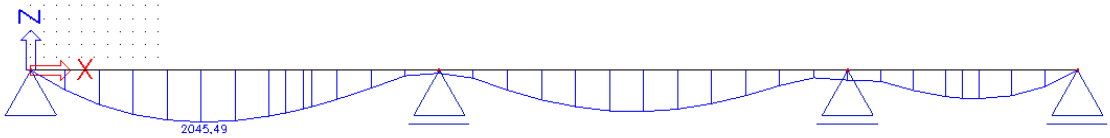
After the calculation the Load cases manager shows the following:



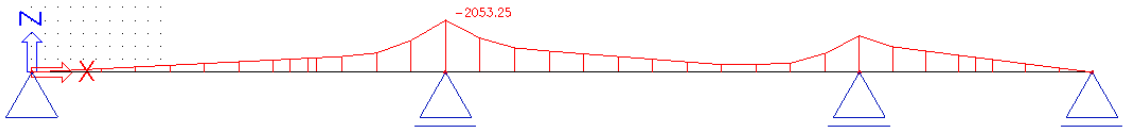
The load cases have ‘Mobile envelope’ as a description and are in an exclusive load group. If required, the load group can be adjusted to set a Load type according to EC1991.

Subsequently the results of this envelope can be reviewed, e.g. the moment My:

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.

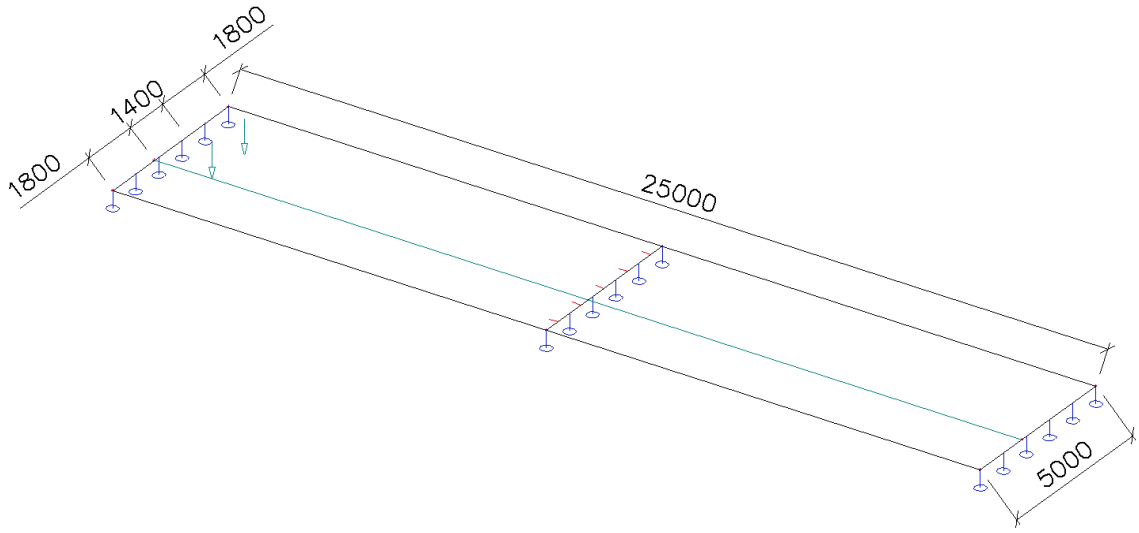
It is possible that a deformation of a load case, associated with internal forces such as Max My, has a larger deformation than e.g. the load case Min uz.

Additional Mobile factor:

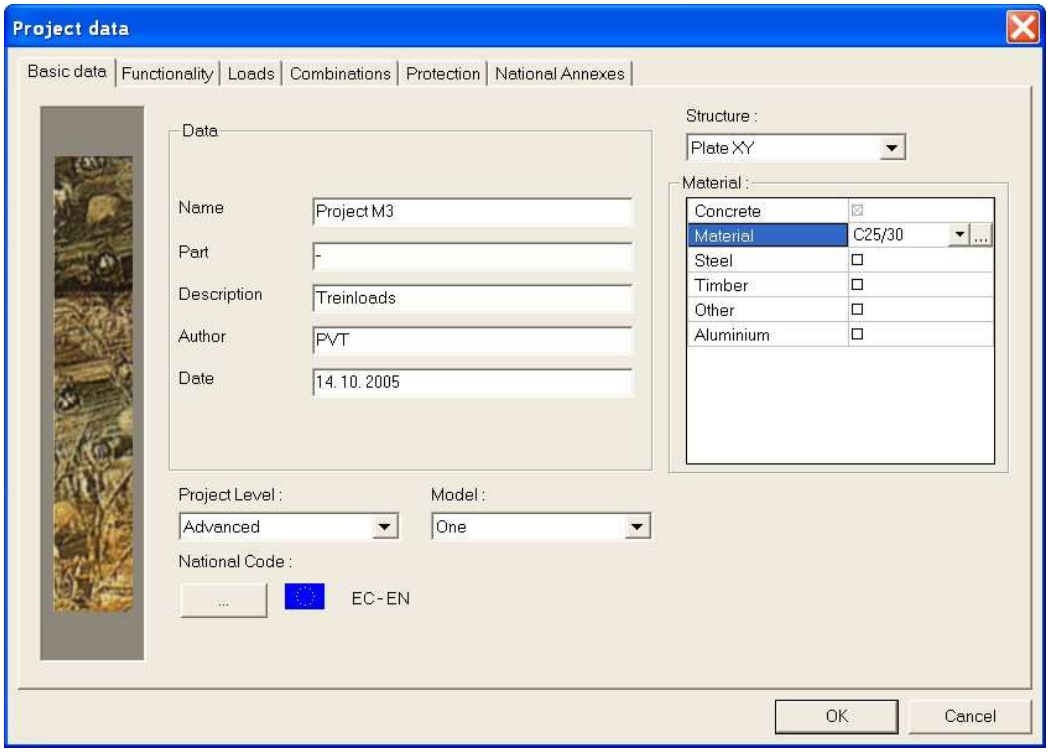
The mobile factor is used e.g. to consider a single or double traffic lane. All results are multiplied with this factor, also the deformations.

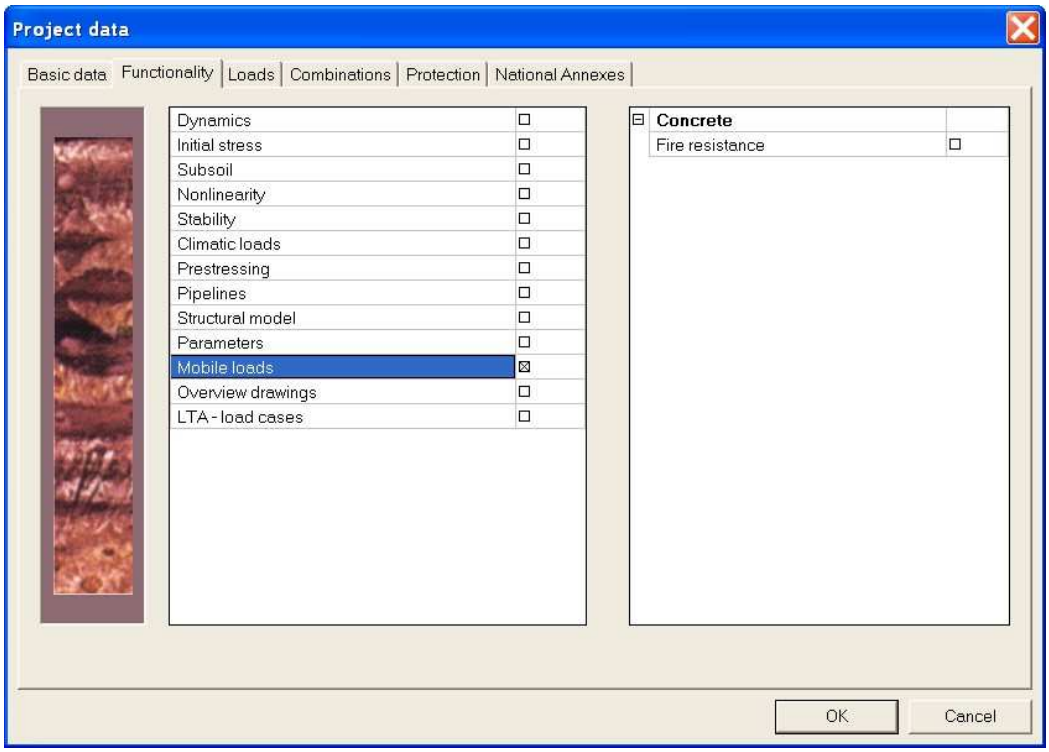
Project M3: Train loads

In this project a bridge deck is modelled as a concrete plate on three supports. Analogously to the previous projects, a track with a unit load is defined on the bridge deck so the influence lines can be determined. However, in this project a unit load with two impulses is defined to simulate both rails of a train track. In a next step, a VOSB 150 load system is linked to this unit load and the enveloping load cases are generated.



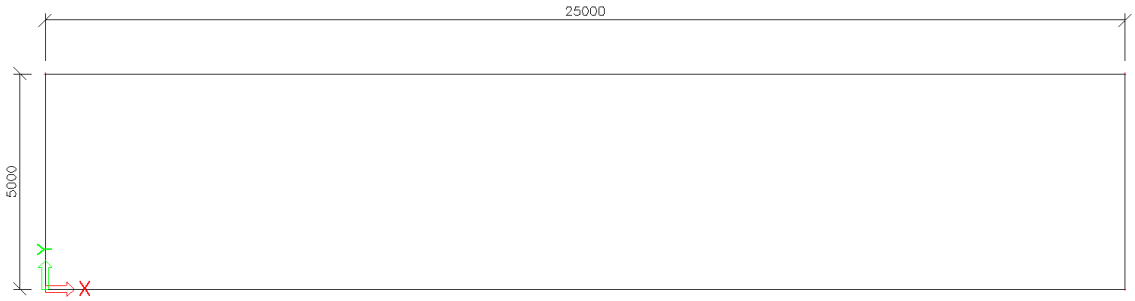
a) Project data



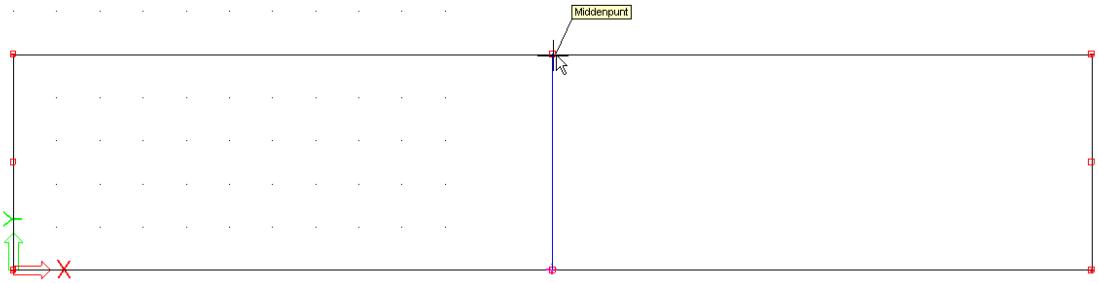


b) Construction

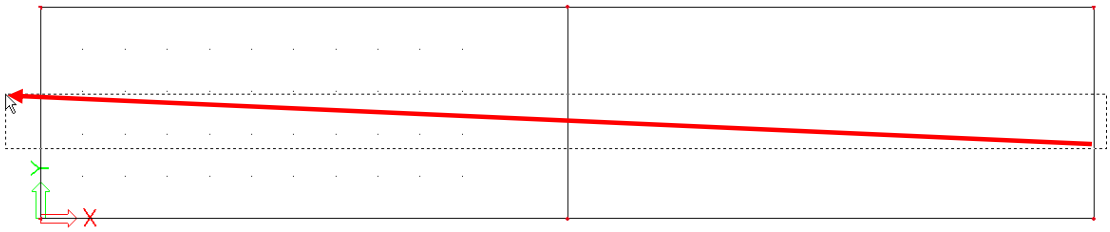
The bridge deck can be entered as Plate with thickness **500mm**.
The length of the bridge deck is **25m**, the width **5m**.



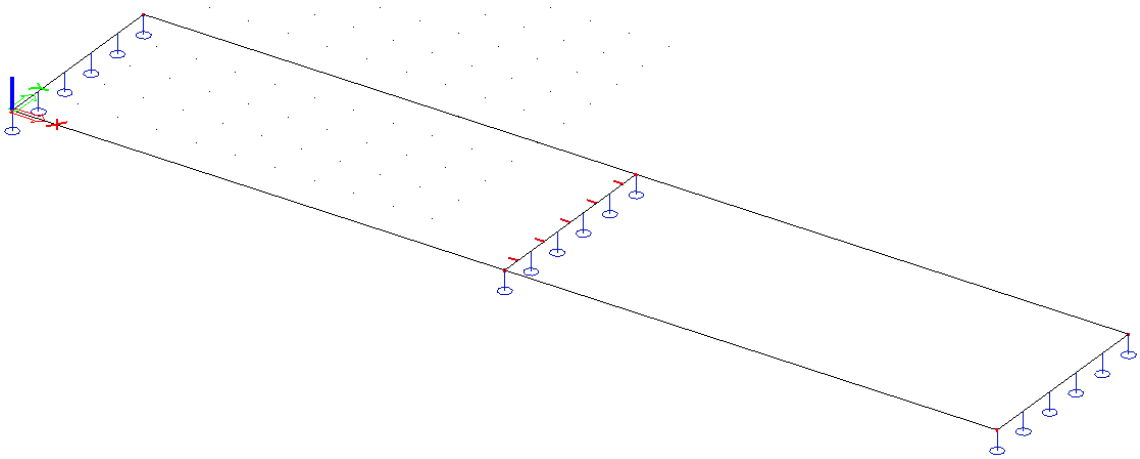
In the middle of the bridge deck an internal edge is created. Using the **Cursor Snap Settings** you can snap on midpoints of the long edges so the edge can be imported through the option Internal edge.



Using Support line on 2D member edge, the translation in the Z-direction can be prevented for the three short edges. They can be simply selected by drawing a rectangle:



Then we have the following structure:



c) Load

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

d) Input track and unit load

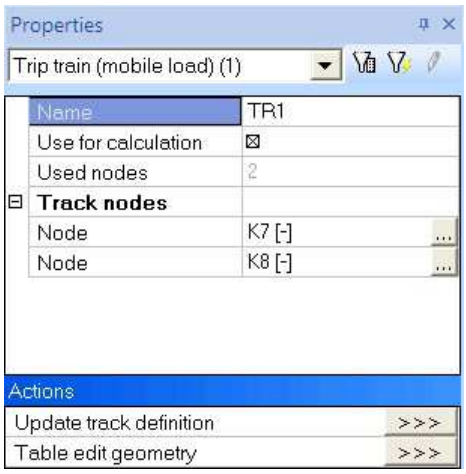
After entering the construction, the menu Mobile loads can be opened.

The train track consists in this project 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.

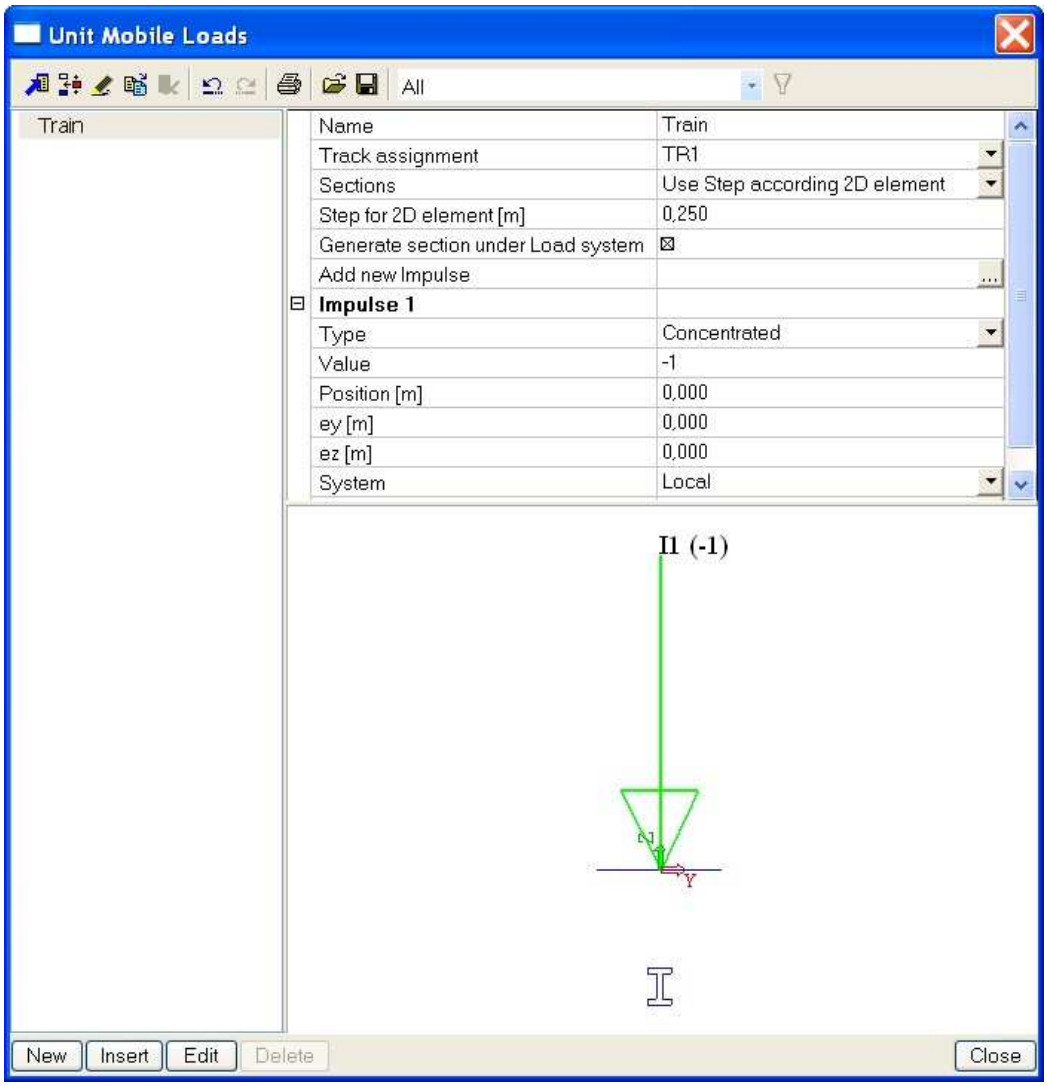
Through New mobile load track the track can be defined. The coordinates can be entered in the Command line:

As **Name** of the track, **TR1** is entered.



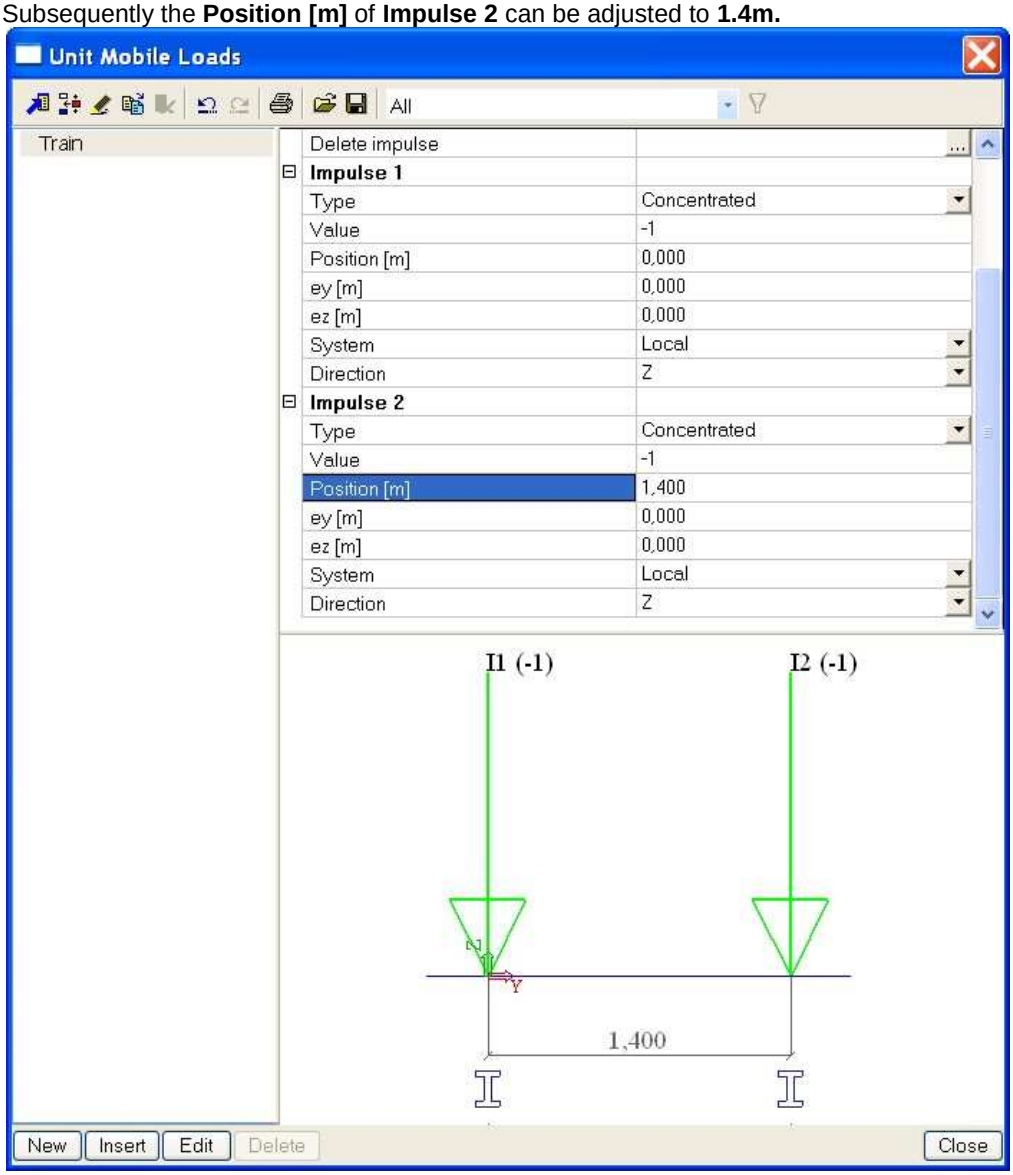
After defining the track, the unit load can be entered through the menu Unit loads.

As a **Name** of the unit load, **Train** is entered for a simple reference.
With **Sections** the option **Use step according 2D element** can be chosen and as step, **0.25m** is entered.

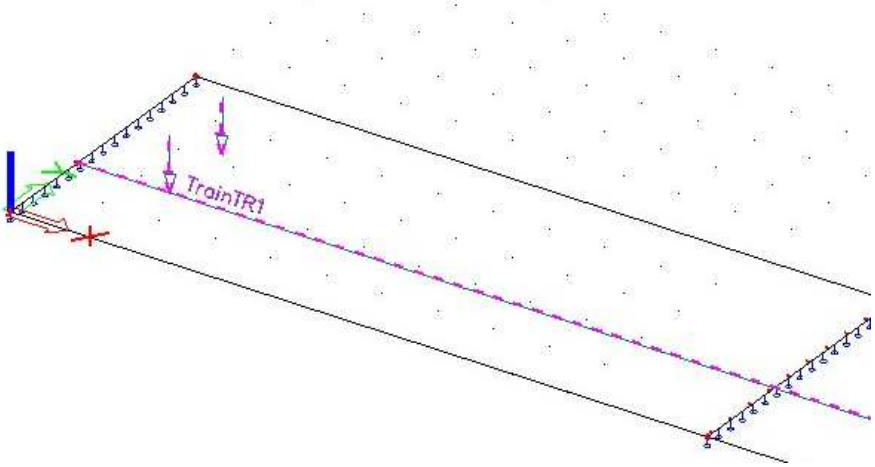


Now the unit load consists of 1 concentrated load. To show both rails of the train track, a second impulse is added through the option **Add new impulse** ….

Subsequently the **Position [m]** of **Impulse 2** can be adjusted to **1.4m.**



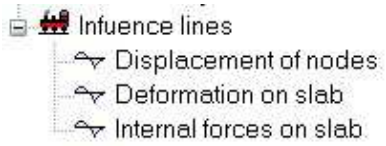
Both impulses are displayed on the bridge deck:



e) Influence lines

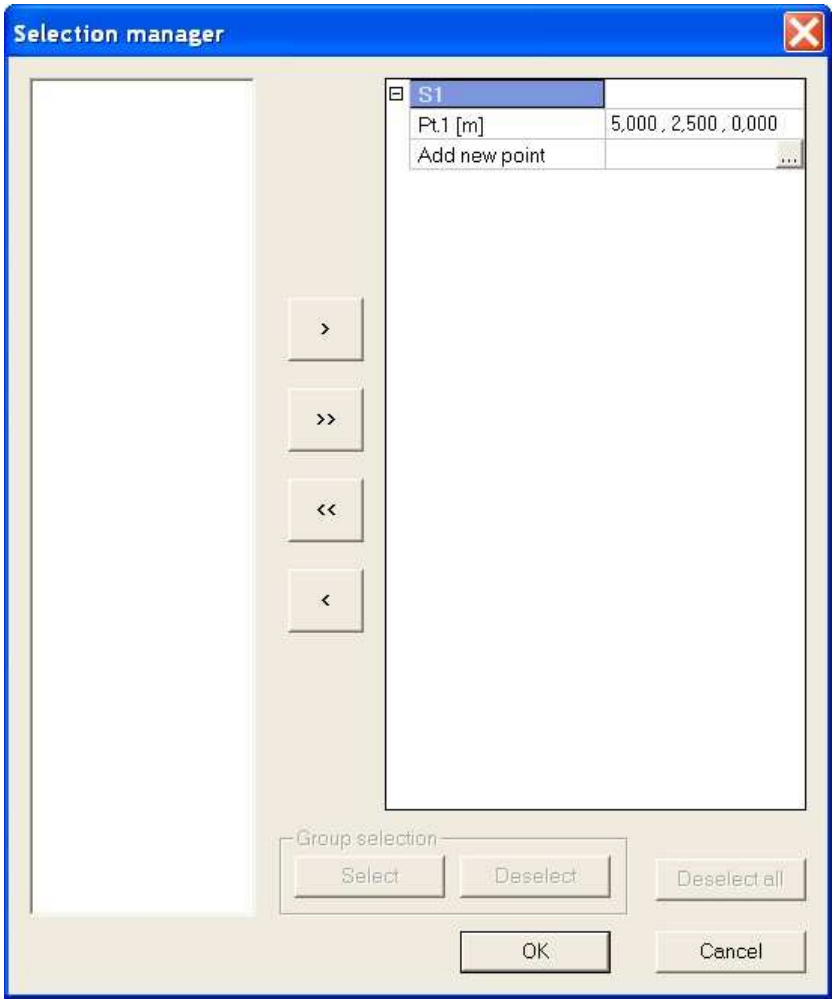
After defining the train track and the unit load that represent both rails, the linear calculation can be started. To do this it is not necessary to exit the menu Mobile Loads, but you can use the button Calculation in the project toolbar. Through Mesh setup the **Average size of 2D element/curved element** can be set to **0.5m**.

After the calculation a new group appears in the menu Mobile Loads:



When choosing a result group, you have to indicate on which 2D element in which point the results have to be displayed through the **Selection tool**.

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



The **Preview** shows the following results:

Influence lines - Deformation on member 2D

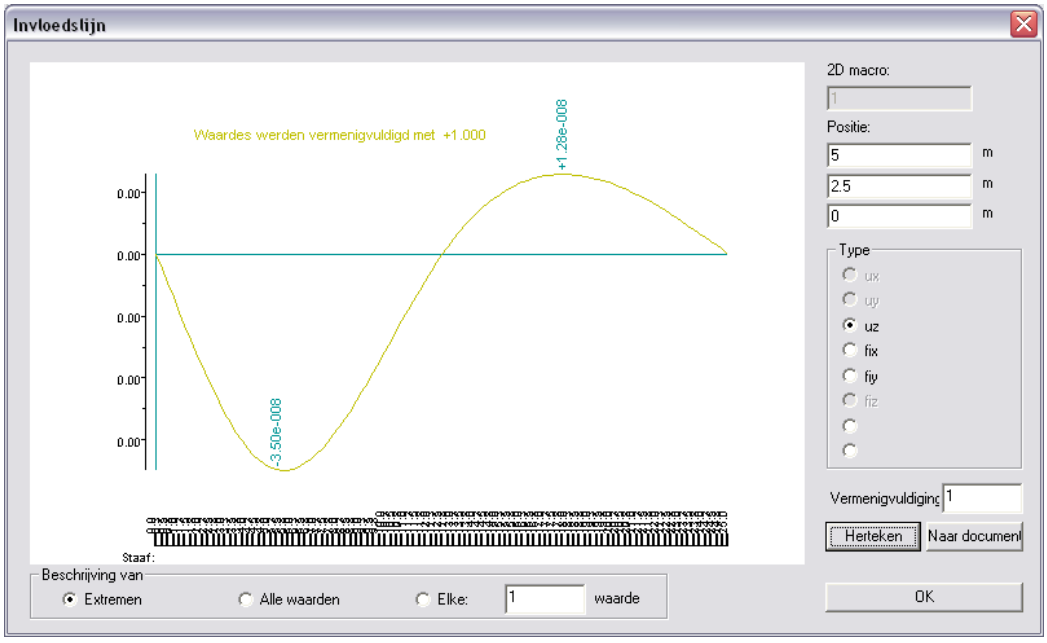
Invloedslijnen - Vervormingen op 2D elementen

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

poz	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	0.000	0.000	0.000
0.75	0.000	0.000	0.000
1.00	-1.01e-010	0.000	0.000
1.25	-1.11e-010	0.000	0.000
1.50	-1.20e-010	0.000	0.000
1.75	-1.11e-010	0.000	0.000
2.00	-1.02e-010	0.000	0.000
2.25	0.000	0.000	0.000
2.50	0.000	0.000	0.000
2.75	0.000	0.000	0.000
3.00	0.000	0.000	0.000
3.25	+2.00e-010	0.000	0.000
3.50	+3.04e-010	0.000	0.000

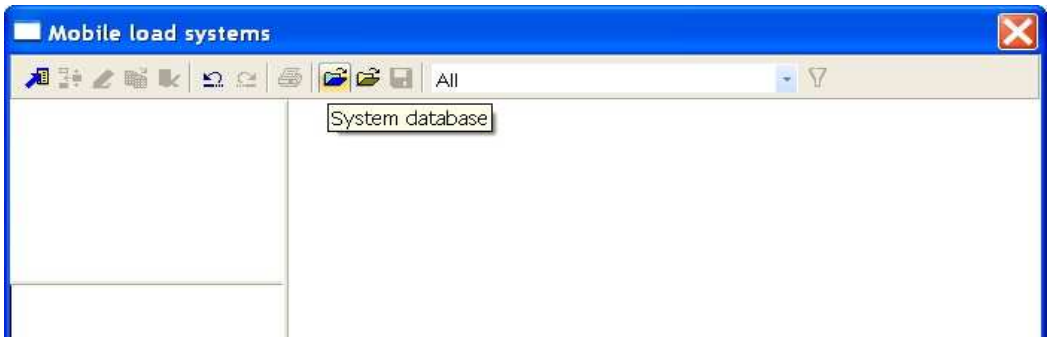
The result table clearly shows the step of 0.25m. Through **Single Check** the result can be viewed graphically.



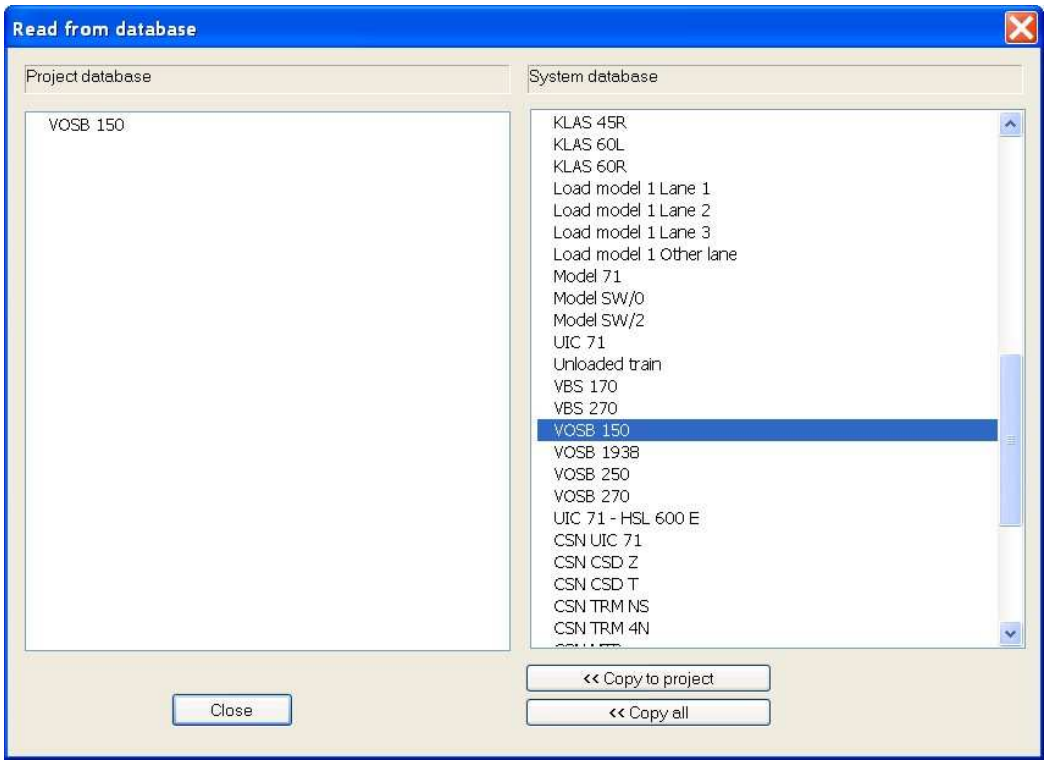
f) Input Load systems

Through the option Load System Database a load system can be entered in the project.

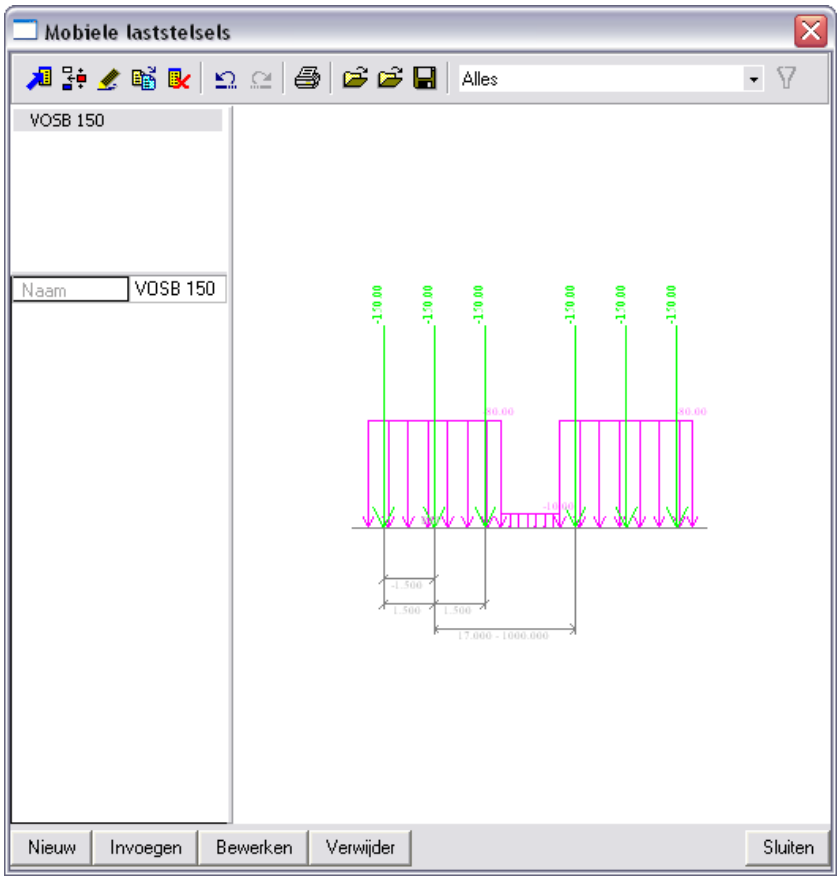
In this project a predefined load system is used; namely VOSB 150.
That is why the window Load system is cancelled so the Load system Manager is displayed.



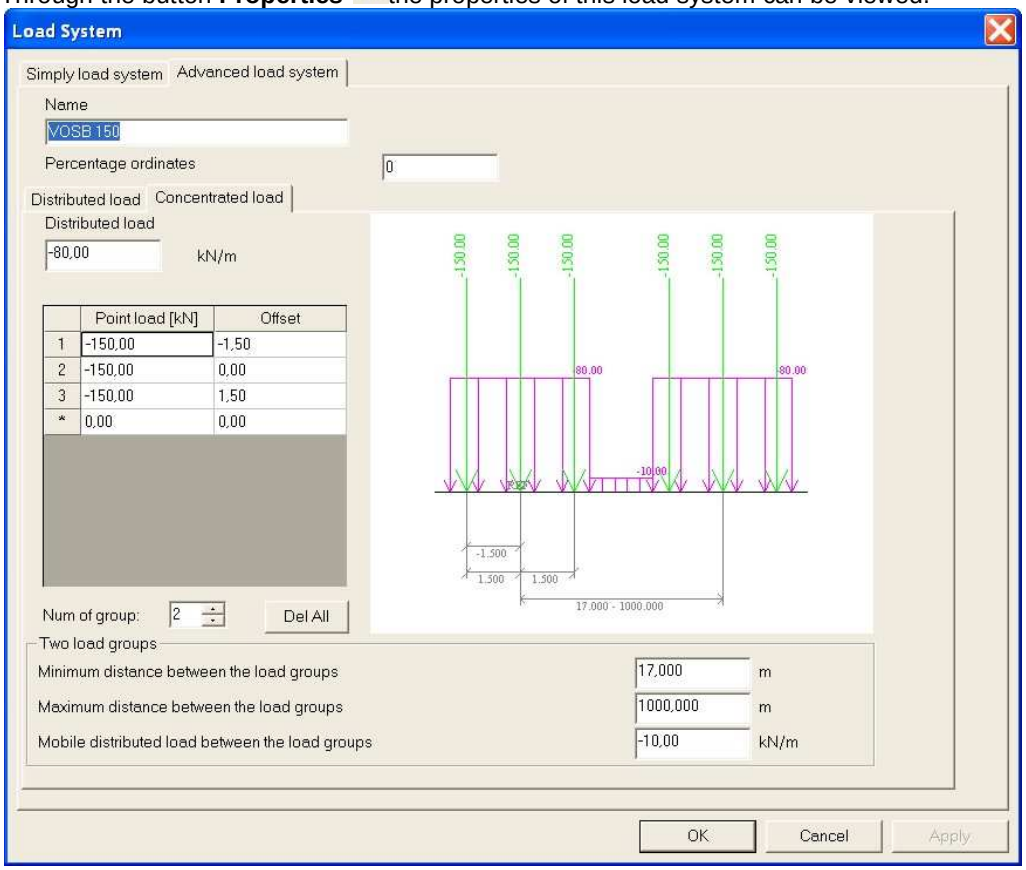
Through the button System database a predefined load system can be added to the project:



With the button << Copy to project a load system **VOSB 150** can be copied to the Project. By pressing the button Close this system is displayed in the Load system Manager.



Through the button **Properties** the properties of this load system can be viewed.



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.

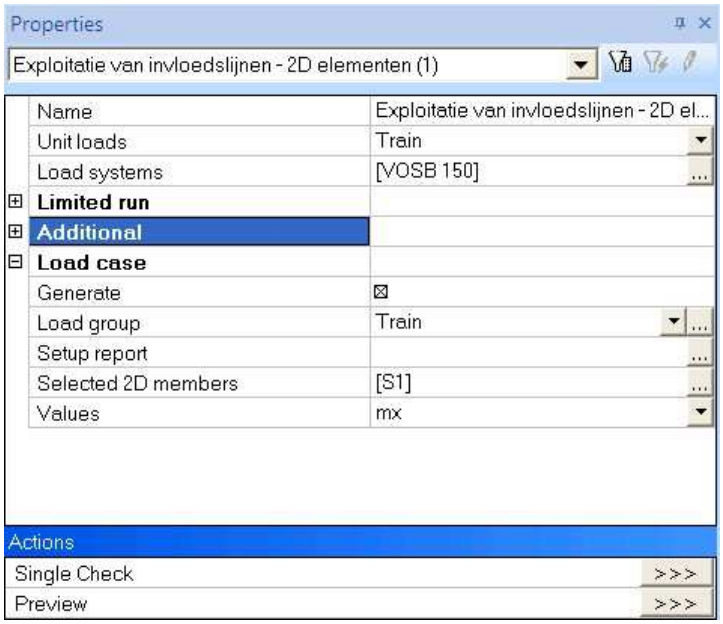
g) Exploitation of the load systems

After defining the mobile Unit load and the load systems, the linear calculation can be started by pressing the button Calculation in the project toolbar.

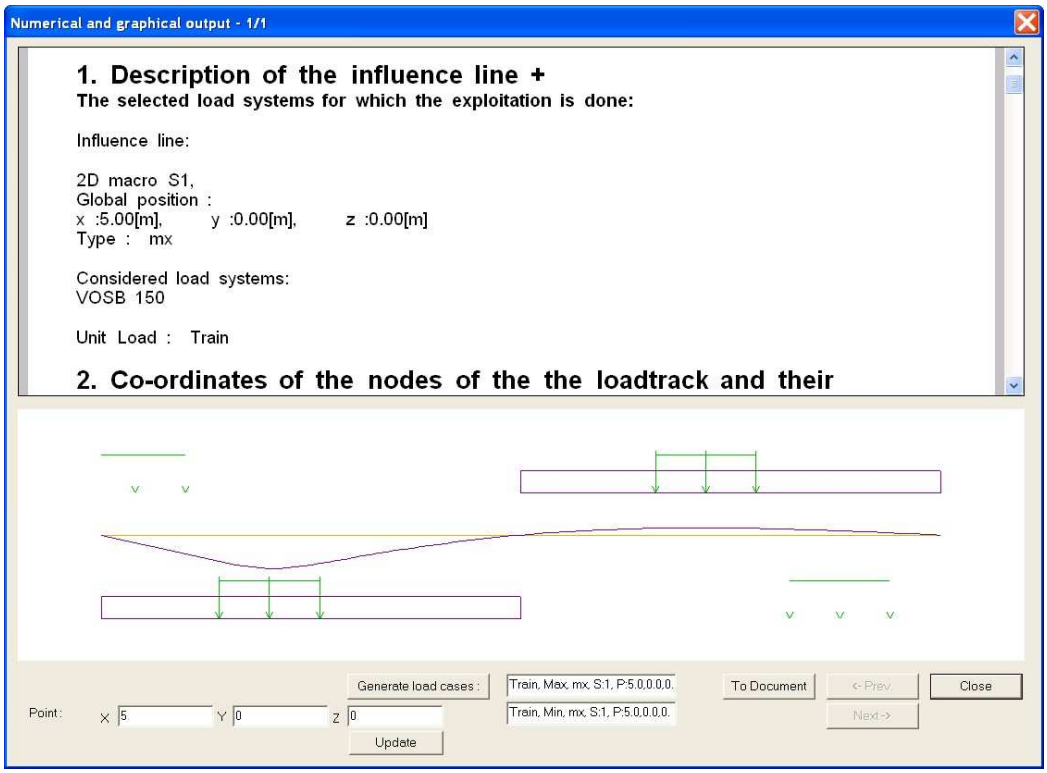
After the calculation a new group appears in the menu Mobile Loads:
Detail analysis - Member 2D force, deformation

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. The **Load group - Train** is selected/made.



Subsequently 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]
-83.093	-125.154	1.000	18.000

Positive maximum position : VOSB 150

Sum P [kNm/m]	Sum Q [kNm/m]	X1 [m]	X2 [m]
365.018	475.258	5.000	22.000

7. Results:

Negative maximum position : VOSB 150

Description	Due to P	Due to Q	P + Q	Units
mx negative	-83.093	-125.154	-208.247	[kNm/m]

Positive maximum position : VOSB 150

Description	Due to P	Due to Q	P + Q	Units
mx positive	365.018	475.258	840.277	[kNm/m]

mx is minimal **(-208,247 kNm/m**) in point **(5 ; 0 ; 0)** if the reference point of the first group of point loads is on **1m** from the begin point of the track and the reference point of the second group of point loads that is on **18m**.

mx is maximal (**840,277 kNm/m**) in point **(5 ; 0 ; 0)** if the reference point of the first group of point loads is on **5m** from the begin point of the track and the reference point of the second group of point load is on **22m**.

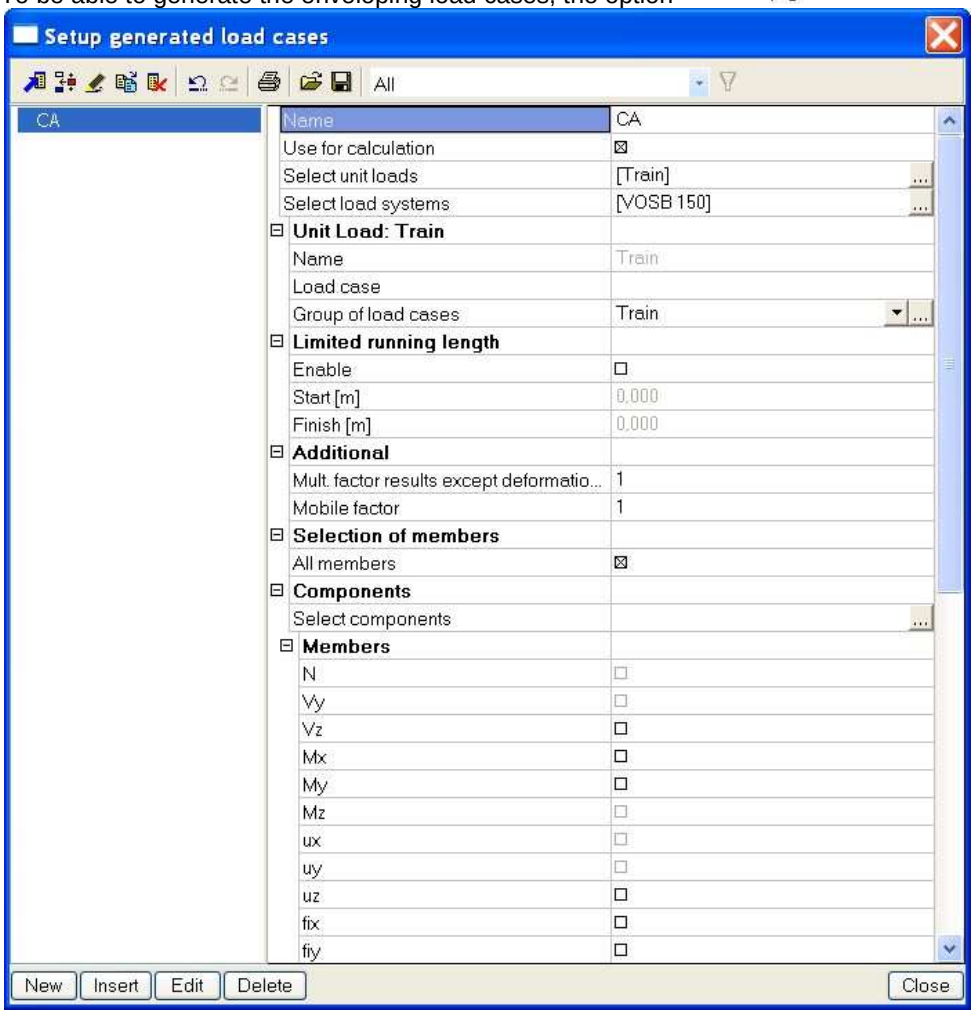
In this example it is clear that the distance between both load groups is always 17m, as set at the VOSB 150 load system.

h) Generate load cases – Envelope load cases

In this project the enveloping load cases are 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

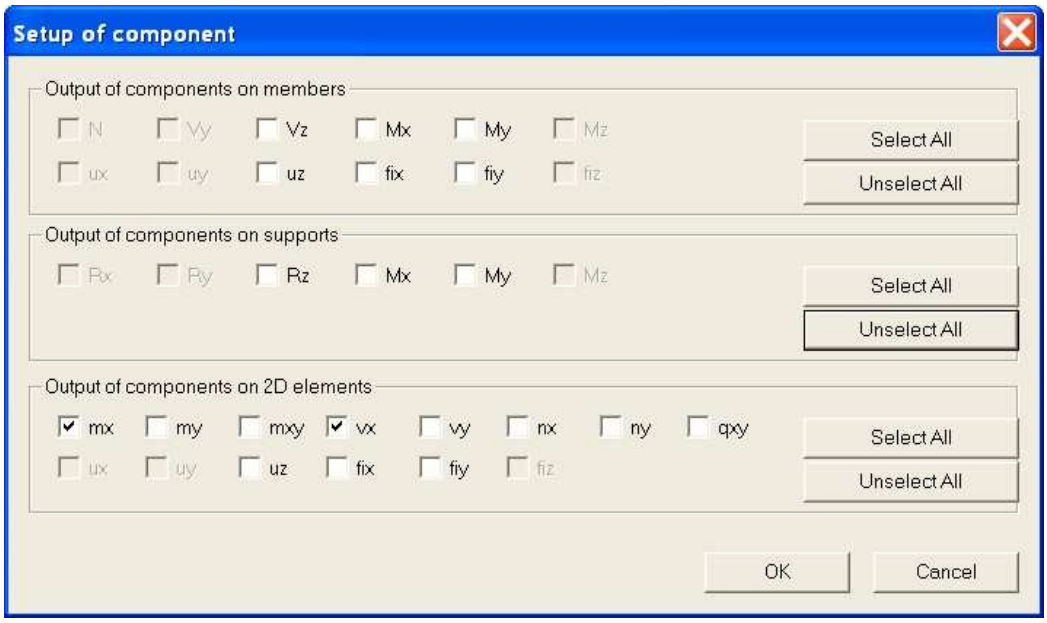
To be able to generate the enveloping load cases, the option Setup generated load cases is used.



First of all you have to indicate which Unit load and which Load system have to be taken into account.

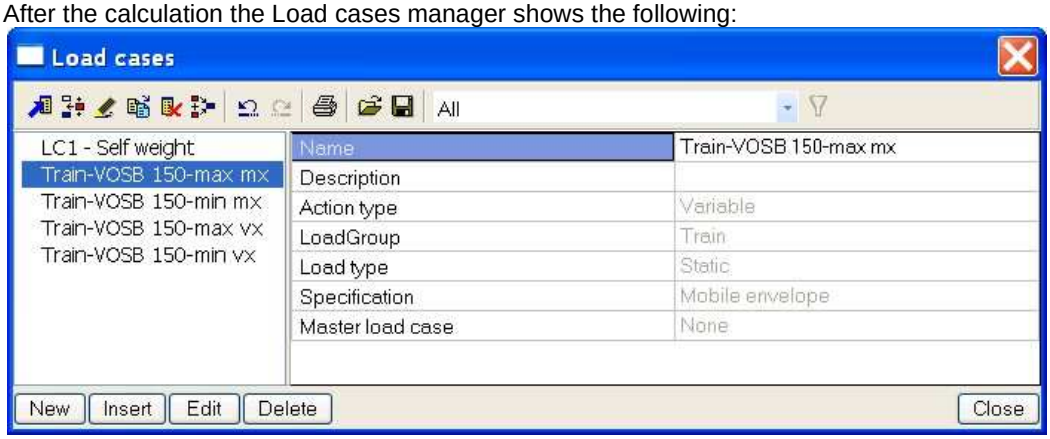
Subsequently the option **Name Load case** can be used to enter the names. This is not necessary. Nothing is filled in so the program generates the names automatically based on *Train* and *VOSB 150*.

Through **Select components** you can indicate for which components an envelope has to be generated. In this example the design parameters vx and mx are considered.



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

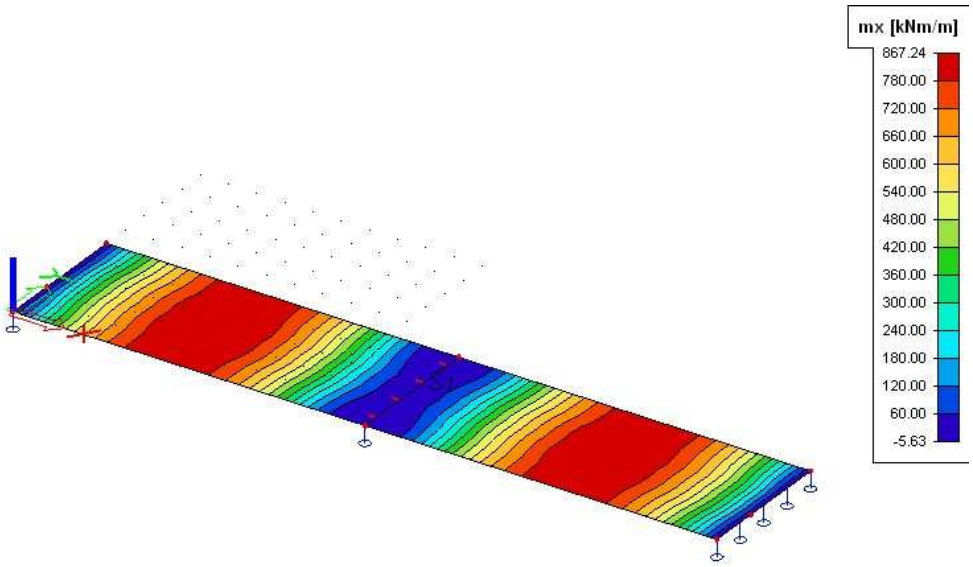
After the calculation the Load cases manager shows the following:



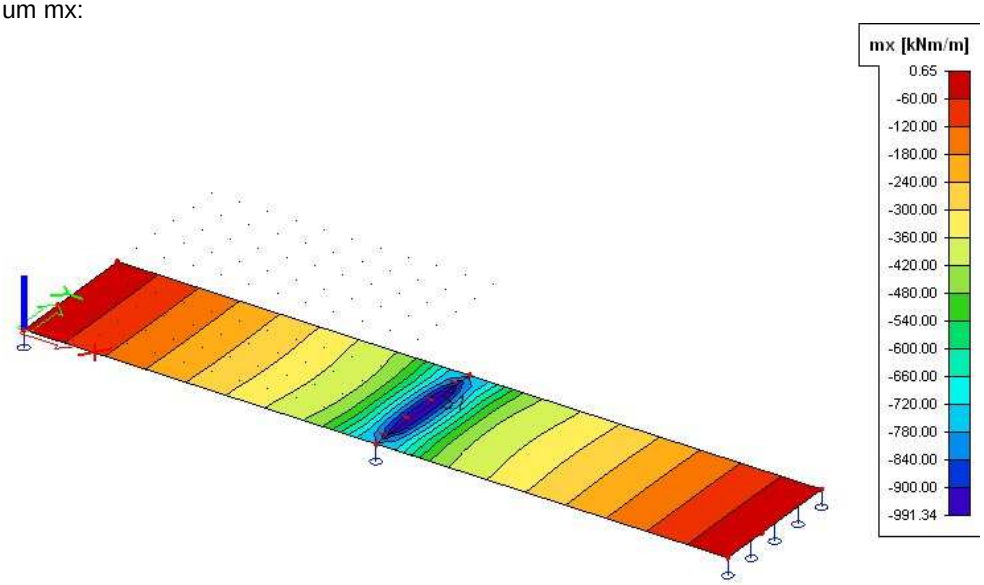
The load cases have Mobile envelope as a description and are in an exclusive load group. The load group can be adjusted if required to set a Load type according to EC1991.

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

Maximum mx:



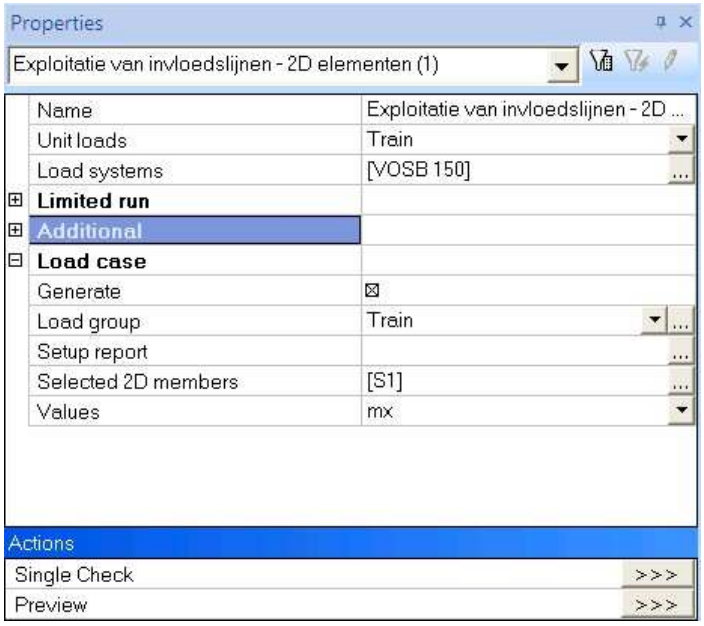
Minimum mx:



Generation of load cases

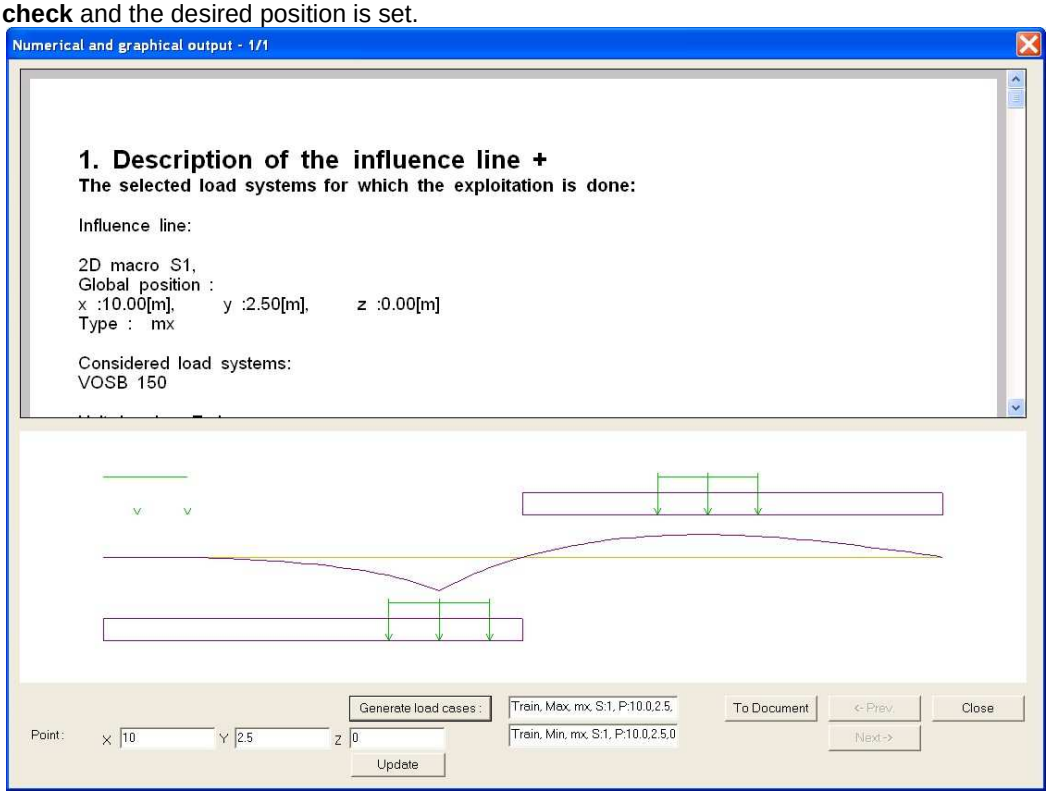
After setting the envelopes, a selective exploitation is performed for the moment, indicated on position **(10; 2,5 ; 0)**.

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



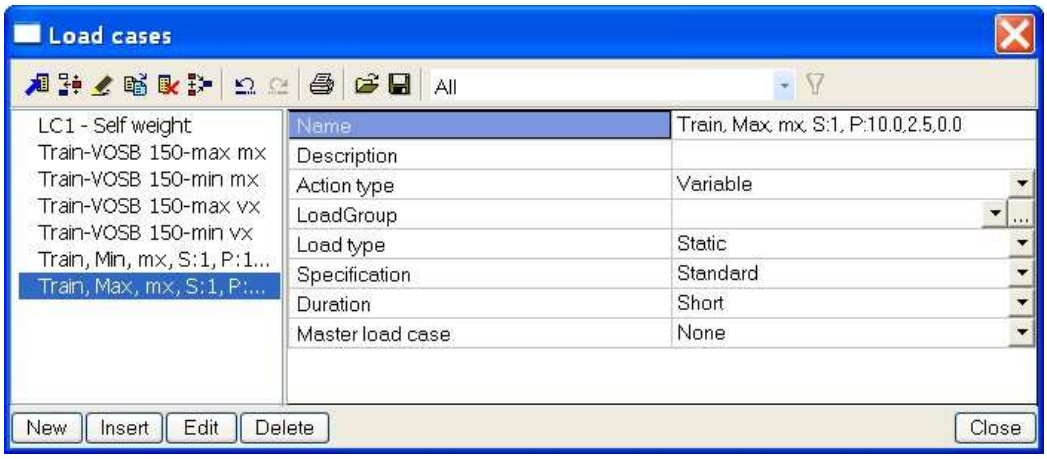
The load cases will be placed in the variable load group **Train** that has already been made.

After activating this option, a Single check is performed on the bridge deck through the option **Single check** and the desired position is set.



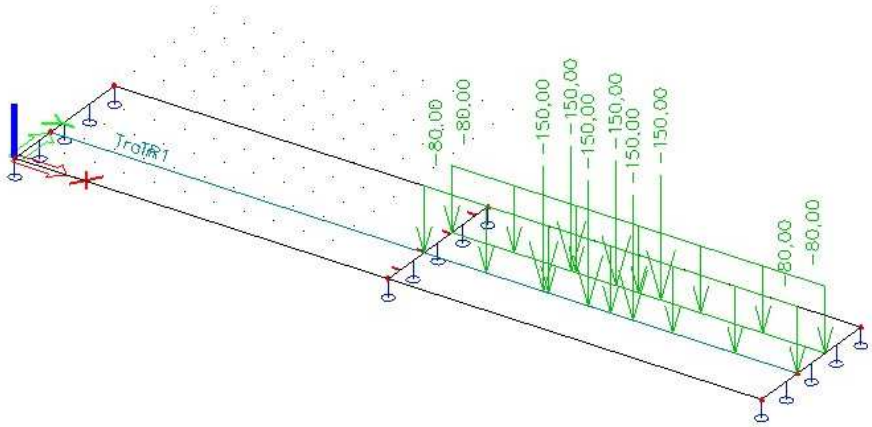
Through **Generate Load Cases** the load cases are generated.

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

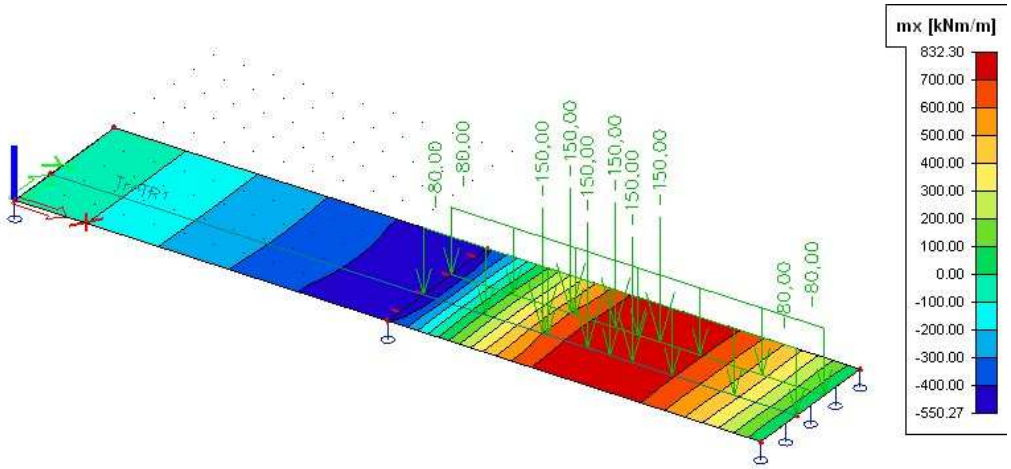


After re-running the linear calculation, the results for these generated load cases can be viewed.

Load case Max, mx:



Results:

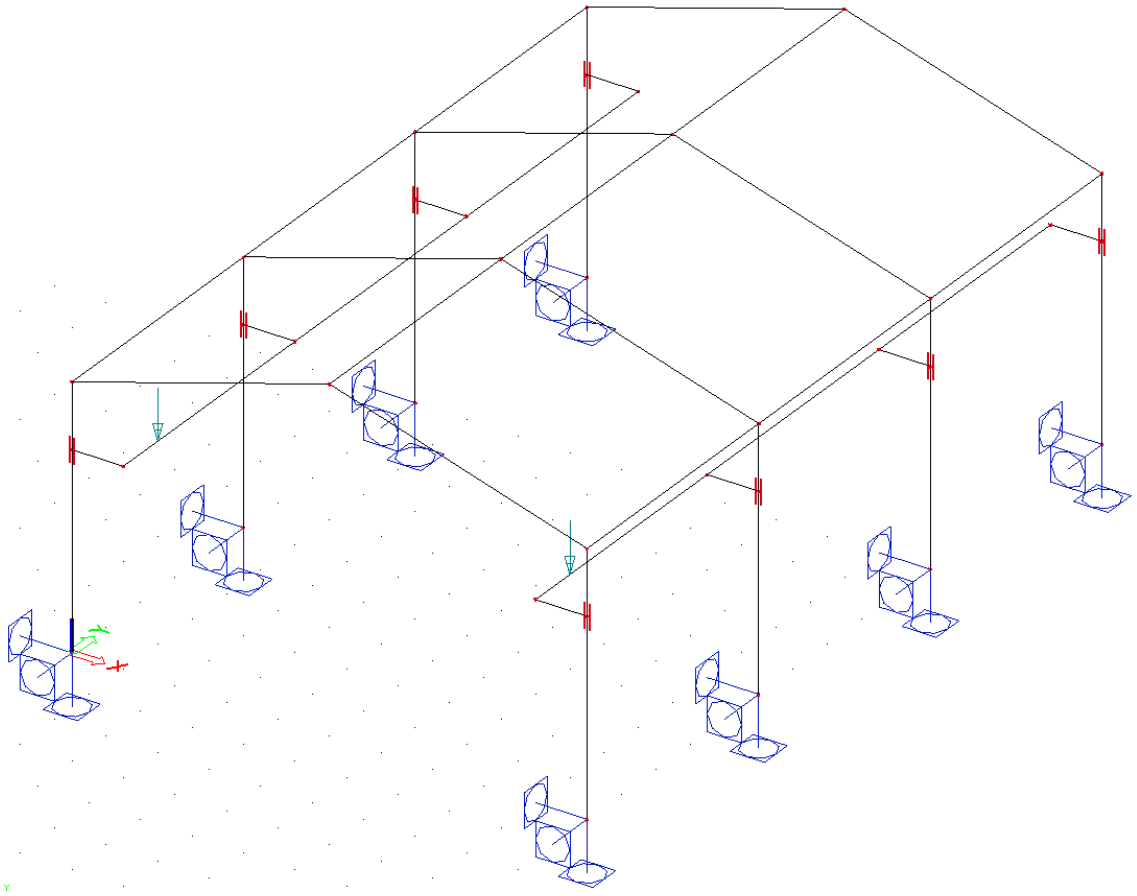


Project M4: Crane track

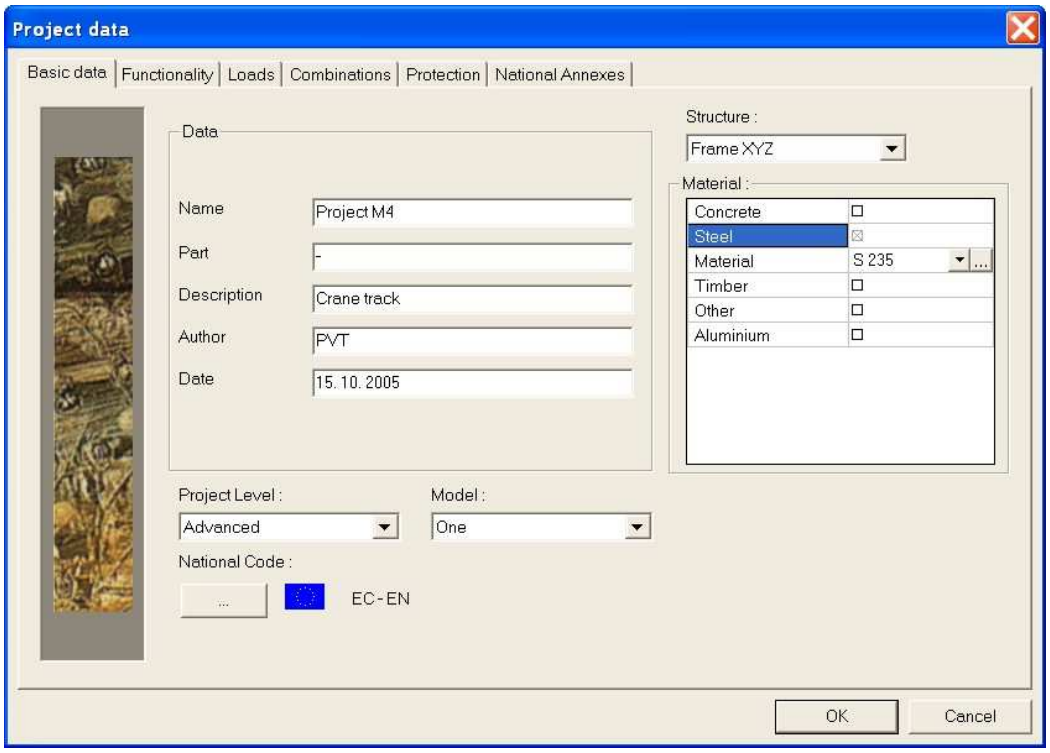
This last project shows how the position of a load system on the structure can be adapted through various unit loads. That way e.g. a crane track, which moves from left to right in a hall, can be modelled.

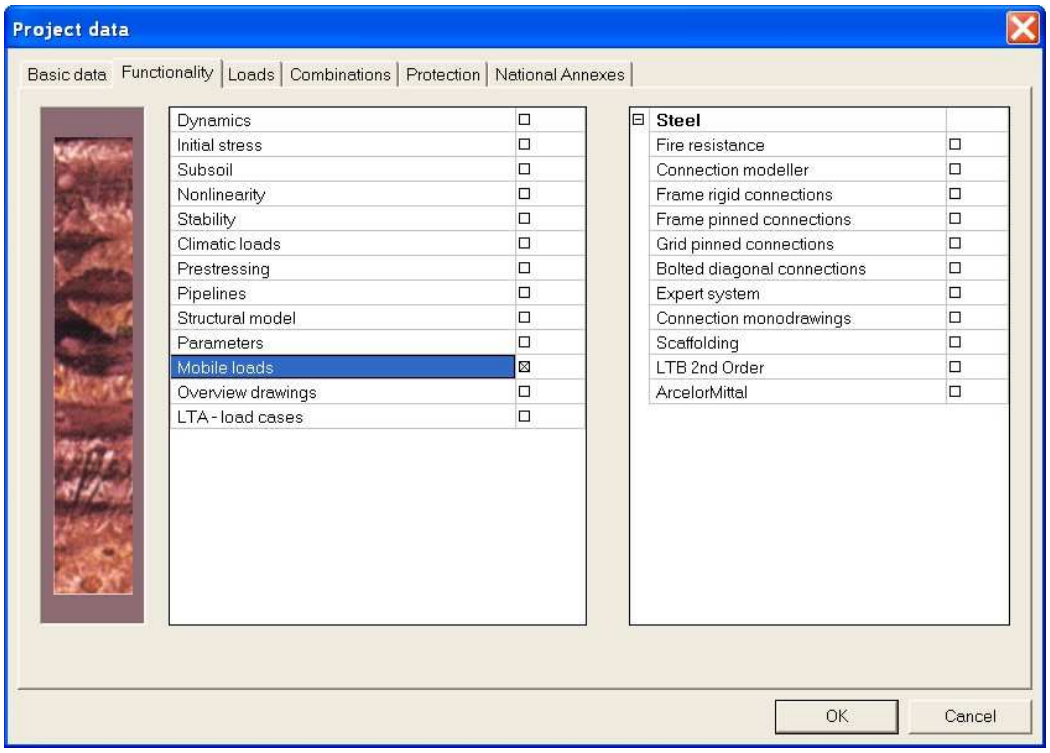
After entering a simple hall, the track of the crane track is defined. Using the Unit load with two impulses, both rails of the crane track are simulated. More Unit loads with various factors are entered to show that the crane track can also move in the transversal direction, perpendicular on the rails.

In a next step the load system is defined which represents the wheels of the crane track and this load system is linked to various unit loads so the enveloping load cases can be generated.



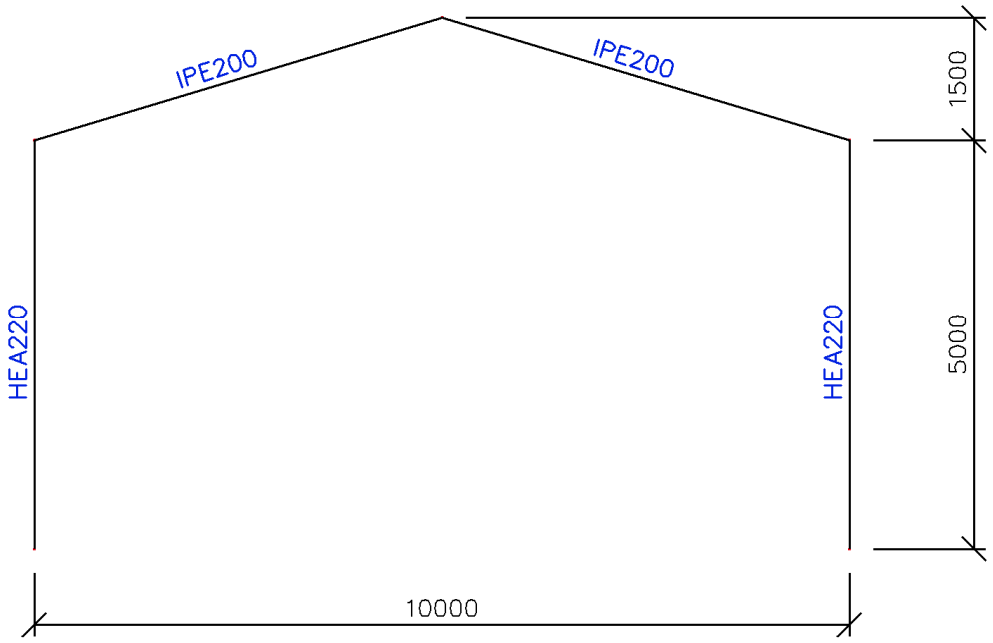
a) Project data





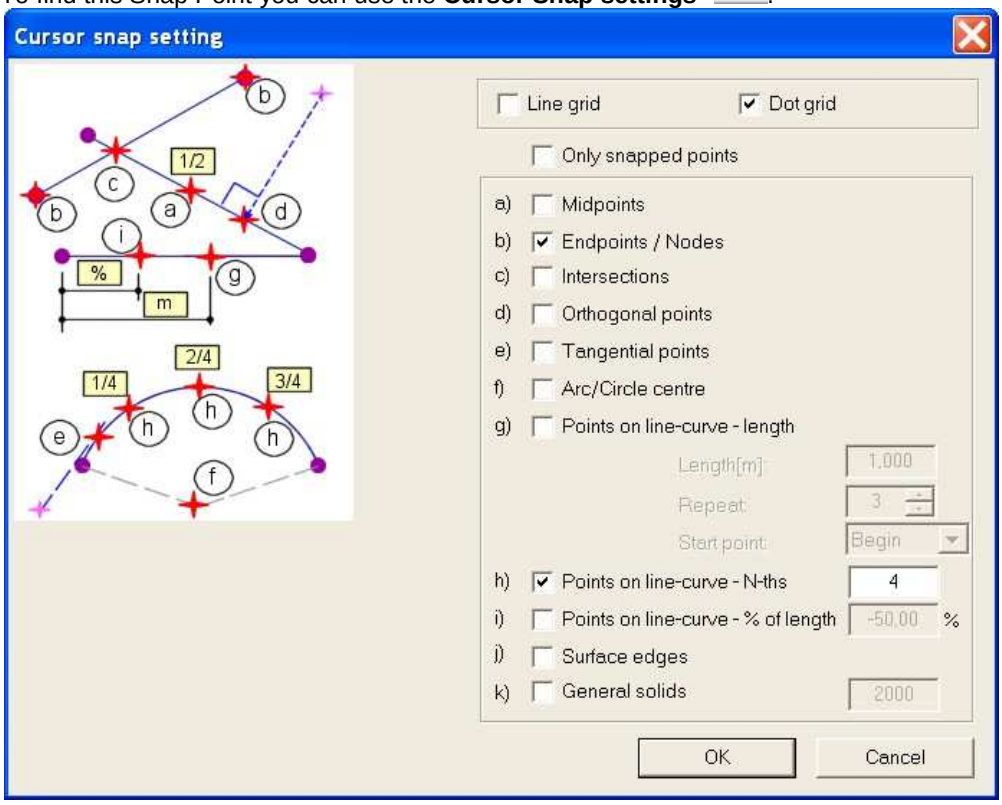
b) Construction

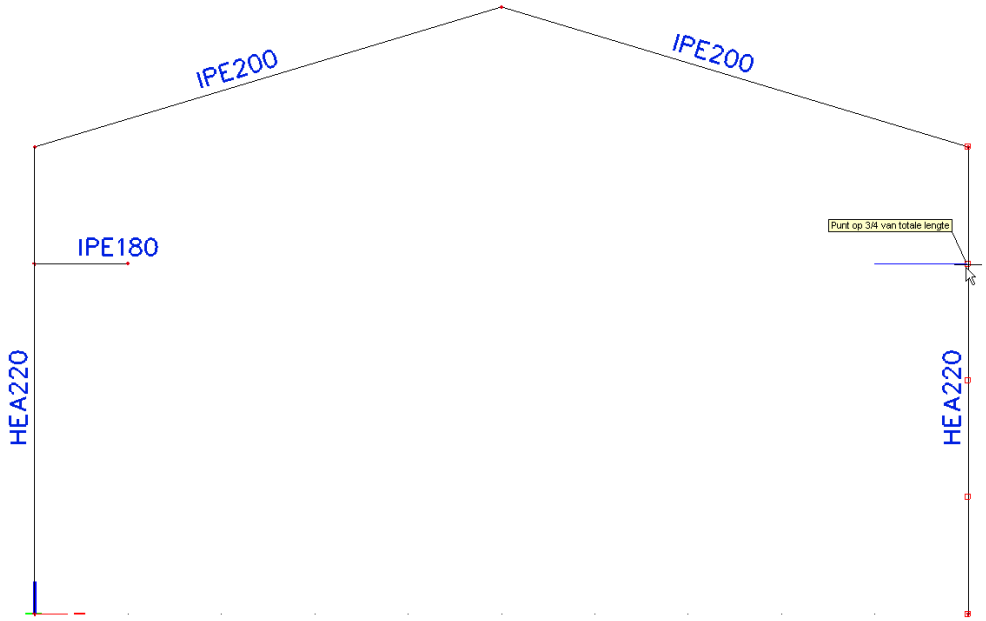
The first portal of the hall can be entered through Catalogue blocks .



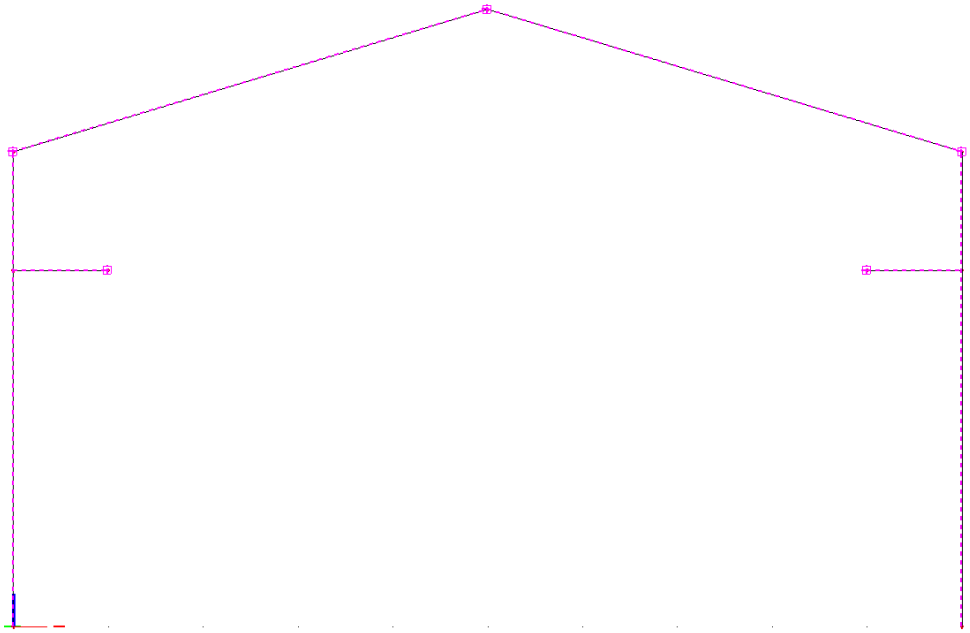
Subsequently the haunch beams on which the rail support, can be entered through Beam . The beams have a length **1m**, type **IPE 180** and move across ¾ of the length of the column.

To find this Snap Point you can use the **Cursor Snap settings** .

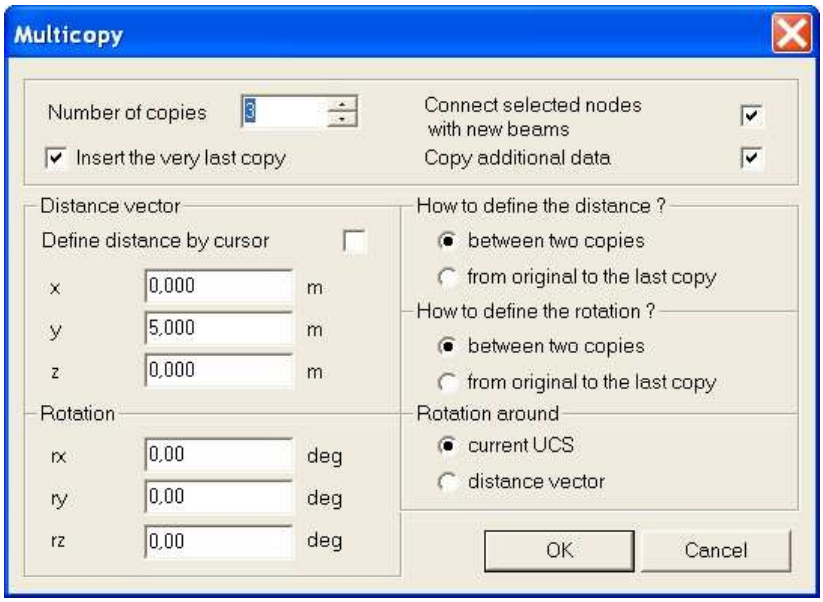




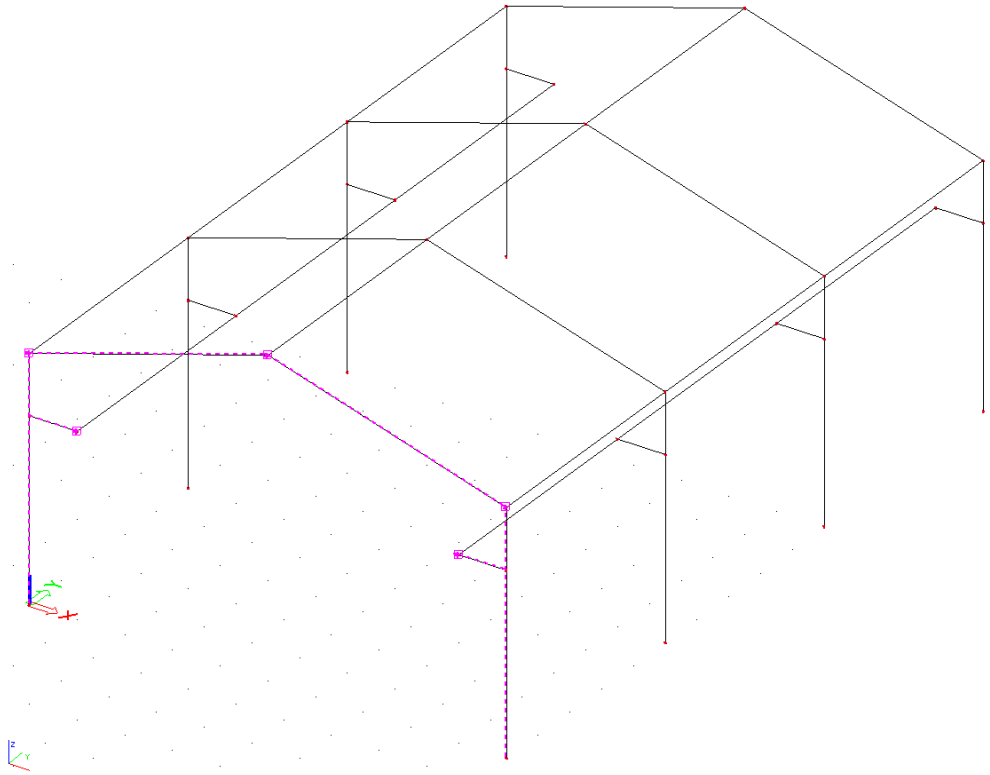
To be able to get the full hall, the option  Multiple copy is used. All members, the three nodes of the roof and the two nodes of the IPE180 beams are selected:



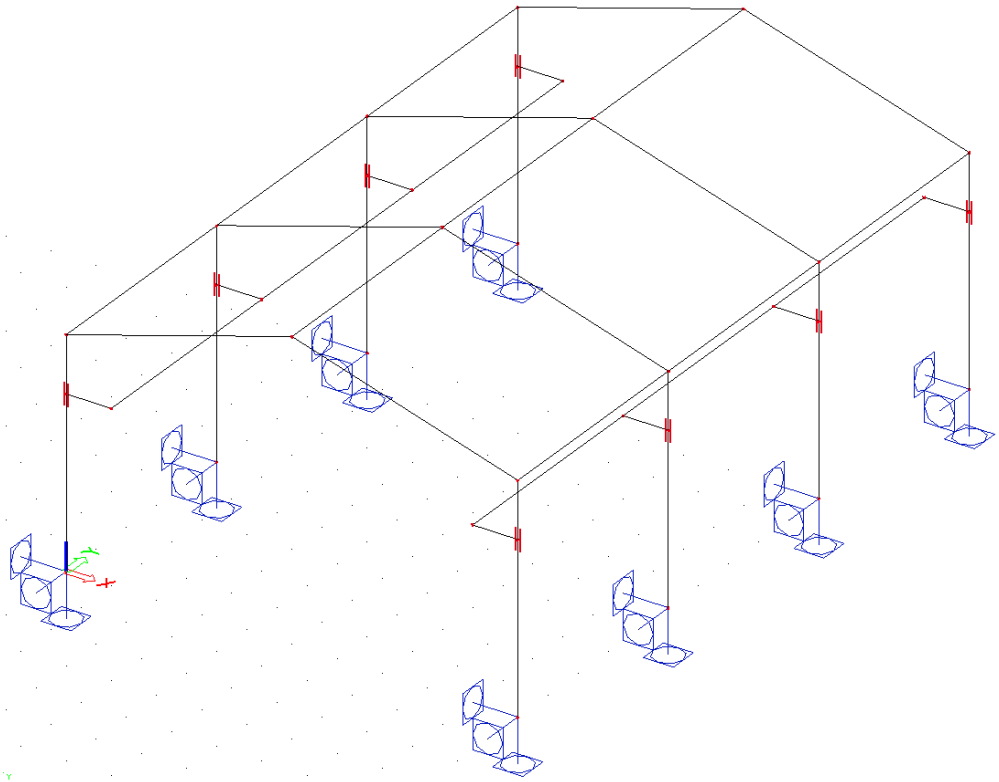
The window More copies can be set:



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 Check structure data and Connect members/nodes to connect the various members.



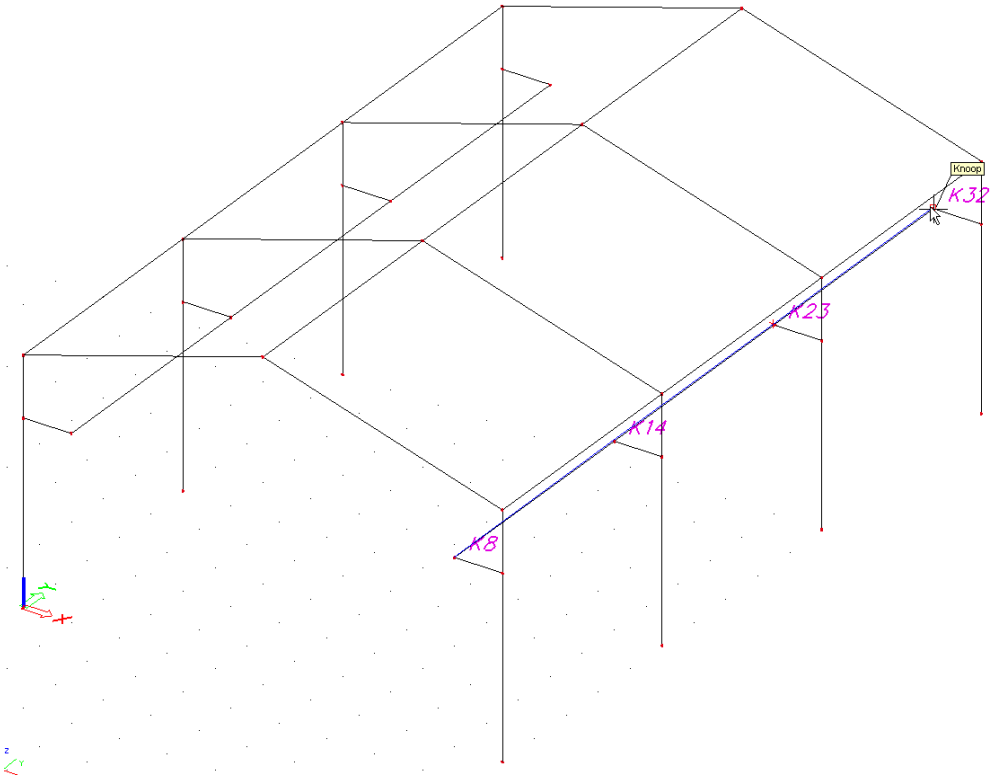
c) Load

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

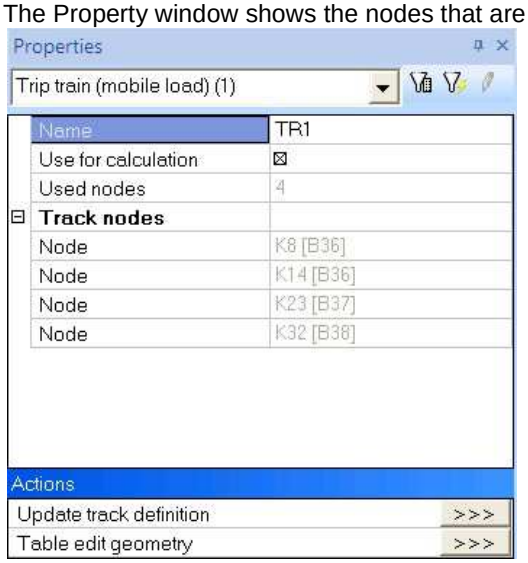
d) Input track and unit load

After entering the construction, the menu Mobile loads can be opened.

Through New mobile load track a track can be defined from node **K8** to **K14** to **K23** to **K32**.



The Property window shows the nodes that are recognized by the track:



As a **Name** for the track **TR1** is entered.

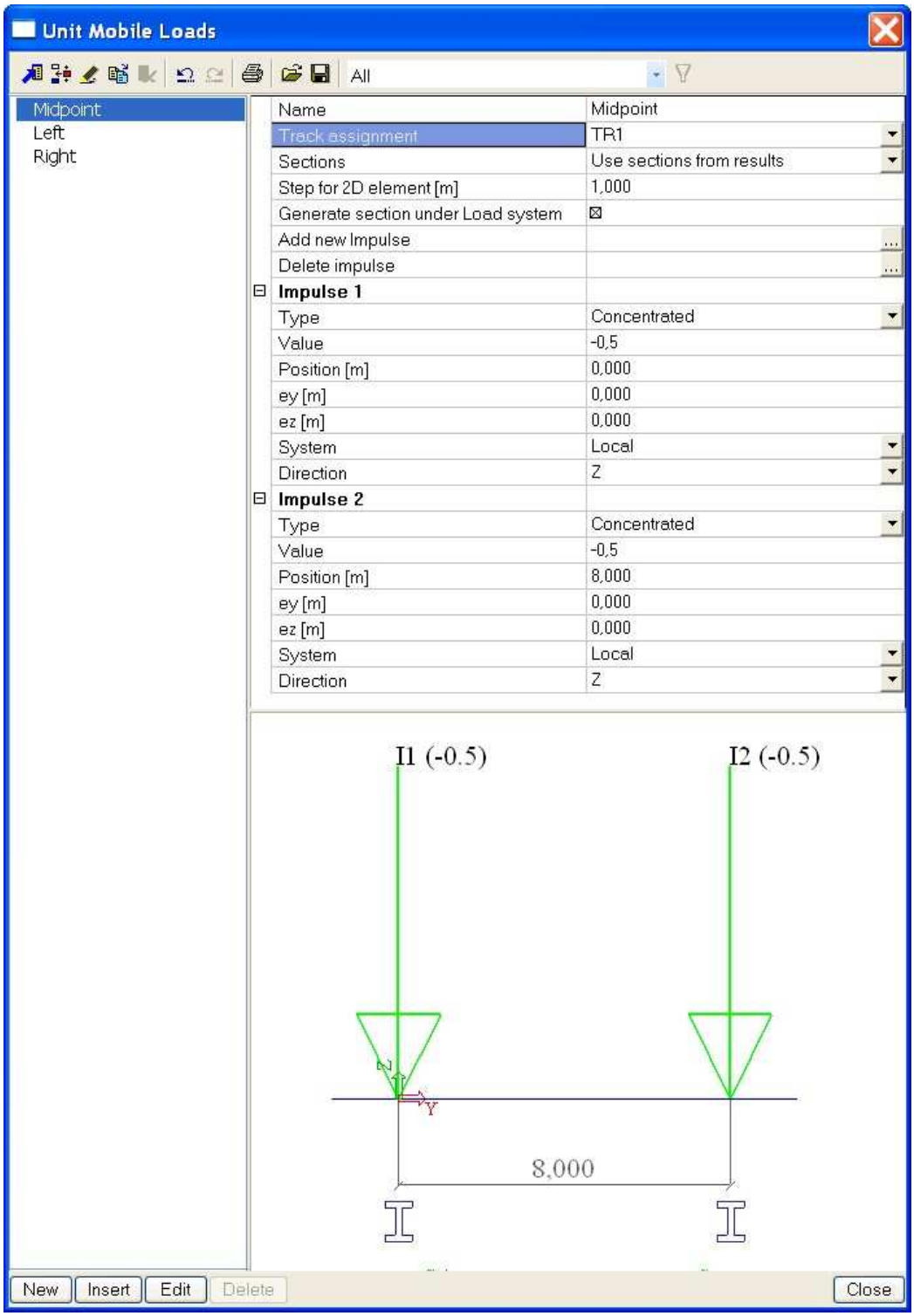
After defining the track, the Unit loads can be entered through the menu Unit loads.
In this project three unit loads are entered:

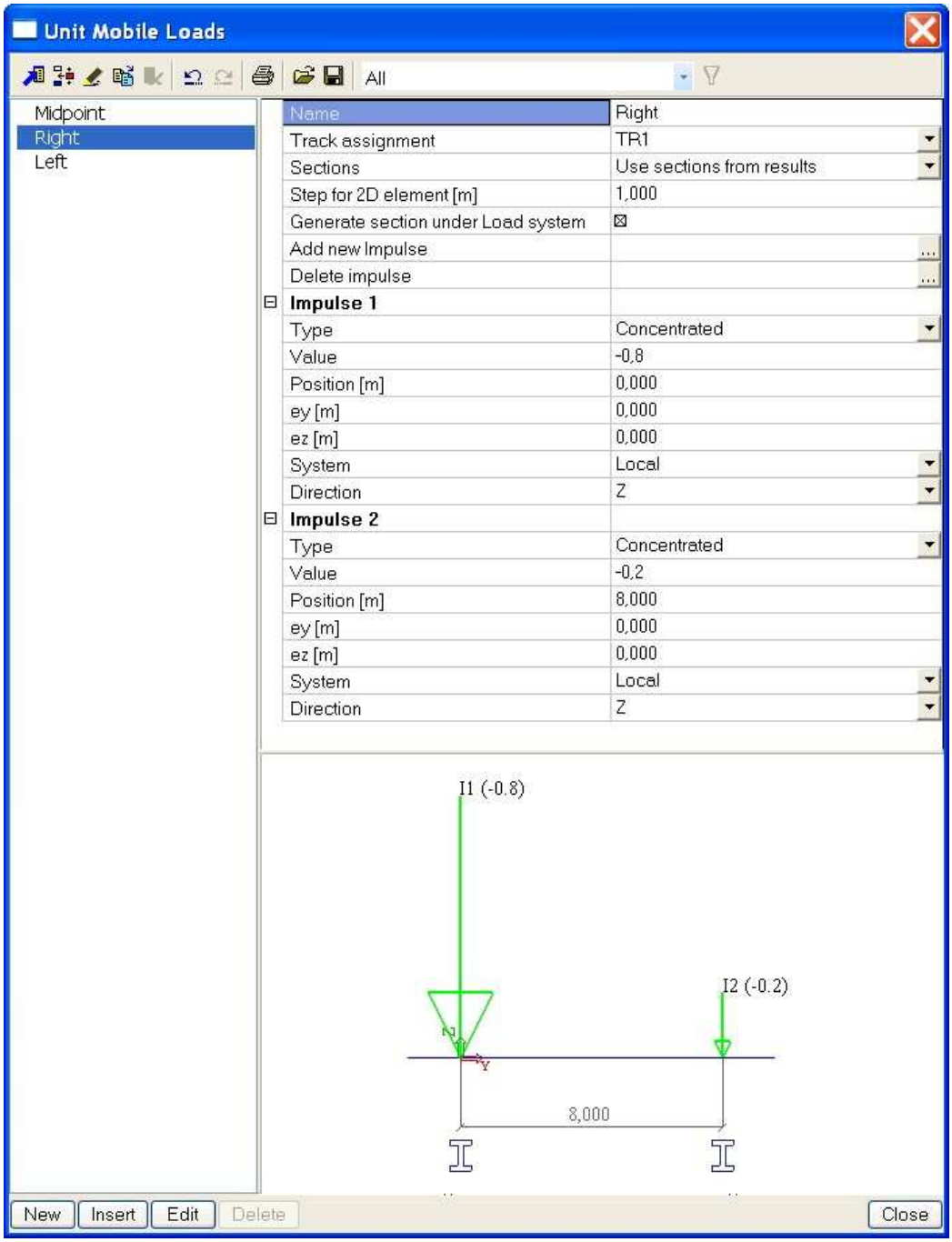
Centre: a Unit load consisting of two impulses of **0,5** simulating that the crane track is in the middle of both rails.

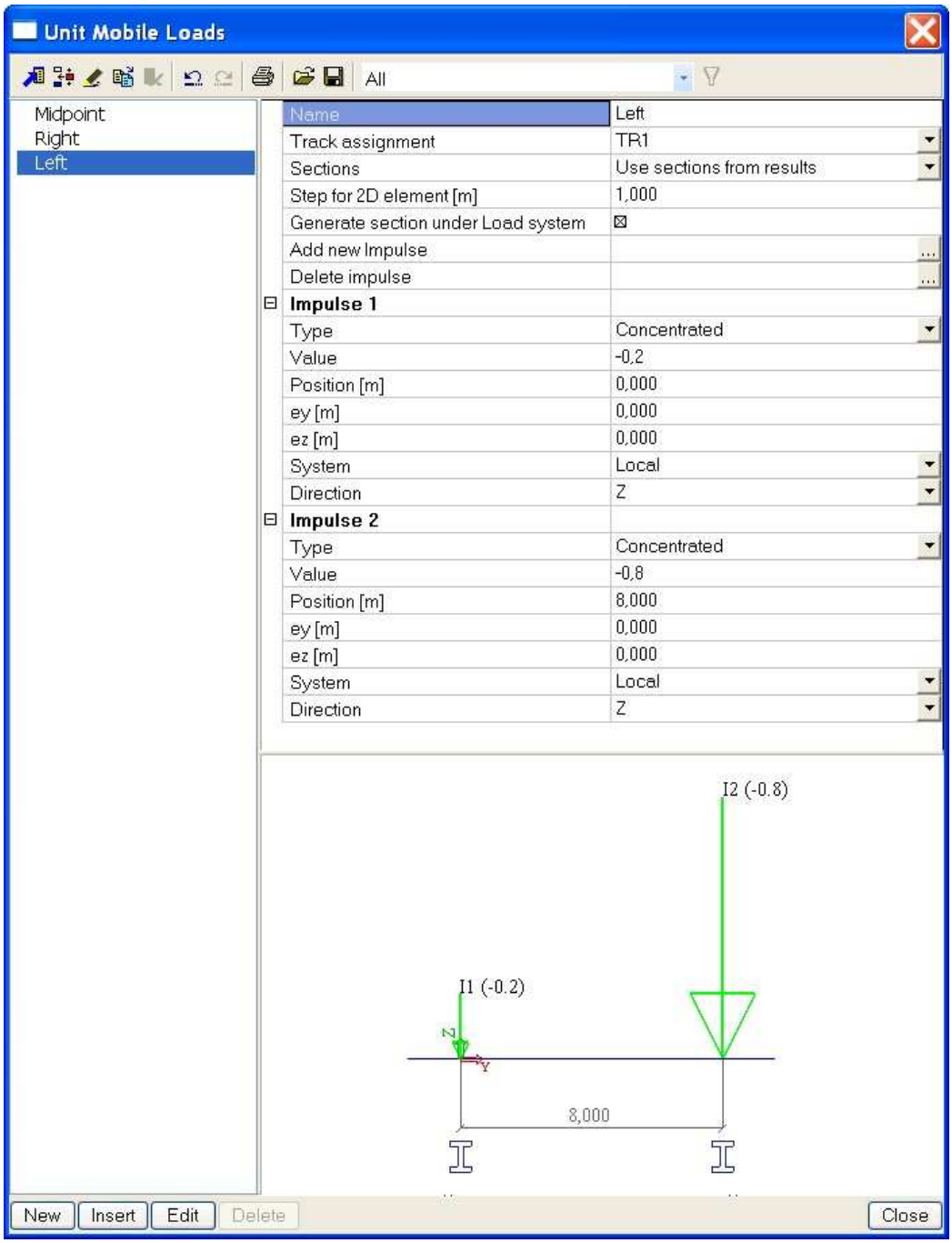
Left: a Unit load consisting of an impulse of **0,8** and an impulse of **0,2** simulating that the crane track is on the left hand side of the hall.

Right: a Unit load consisting of an impulse of **0,2** and an impulse of **0,8** simulating that the crane track is on the right hand side of the hall.

The distance between both impulses is the distance between both rails: **8m**.


Unit Mobile Loads
All
Midpoint
Left
Right
Name
Midpoint
Track assignment
TR1
Sections
Use sections from results
Step for 2D element [m]
1,000
Generate section under Load system
Add new Impulse
Delete impulse
Impulse 1
Type
Concentrated
Value
-0,5
Position [m]
0,000
ey [m]
0,000
ez [m]
0,000
System
Local
Direction
Z
Impulse 2
Type
Concentrated
Value
-0,5
Position [m]
8,000
ey [m]
0,000
ez [m]
0,000
System
Local
Direction
Z
I1 (-0.5)
I2 (-0.5)
8,000
New
Insert
Edit
Delete
Close


Unit Mobile Loads
All
Midpoint
Right
Left
Name
Right
Track assignment
TR1
Sections
Use sections from results
Step for 2D element [m]
1,000
Generate section under Load system
Add new Impulse
Delete impulse
Impulse 1
Type
Concentrated
Value
-0,8
Position [m]
0,000
ey [m]
0,000
ez [m]
0,000
System
Local
Direction
Z
Impulse 2
Type
Concentrated
Value
-0,2
Position [m]
8,000
ey [m]
0,000
ez [m]
0,000
System
Local
Direction
Z
I1 (-0.8)
I2 (-0.2)
8,000
New
Insert
Edit
Delete
Close



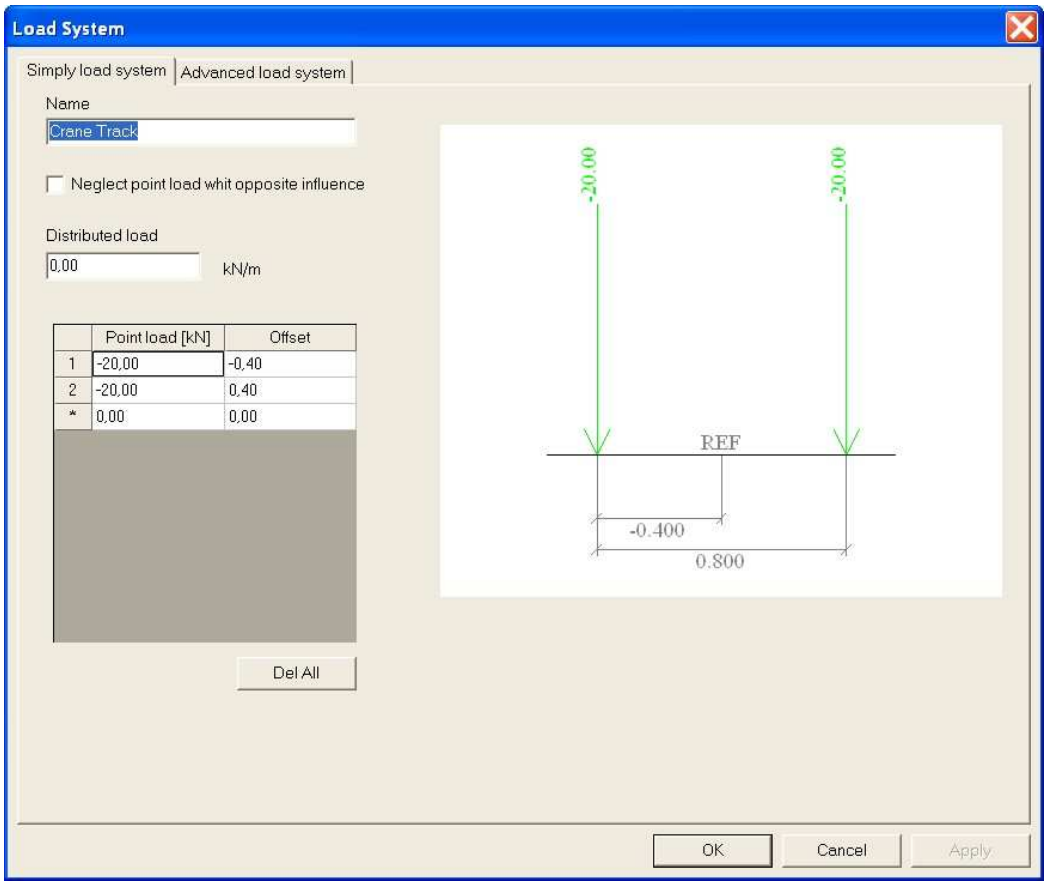
e) Input load system

The input of the load system for the crane track happens through the option Load System Database .

For the crane track a total weight of **40 kN** is taken. If the crane track is in the middle, it means **20 kN** per rail. On every rail there are two wheels so a weight of **10 kN** is calculated. The interval between the wheels is **0,8m**.

However, the defined Unit loads are entered with a factor lower than 1. For the unit load Centre a factor of **0,5** is entered per rail. Because of this the loads of the load system have to be doubled to come to the total weight of **40kN.**

The single load system can be entered as two point loads of **20kN** with a mutual distance of **0,8m.**

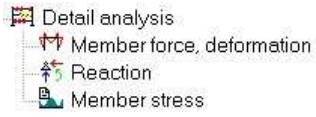


As a **Name** for the load system, **Crane Track** is entered.

f) Exploitation of the load system

After defining the mobile unit loads and the load system, the linear calculation can be started through the button Calculation in the project toolbar.

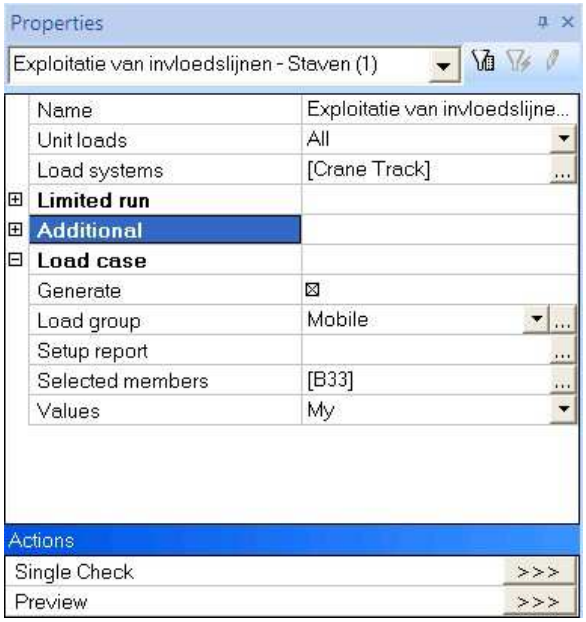
After the calculation a new group appears in the menu Mobile Loads:



With the Detailed Analysis the load system can be linked to various unit loads.

An exploitation is performed for the moment **My** on a position **2.5m** on the first beam **B33**. The exploitation is performed for the three Unit loads together.

In the Property window these loads can be set:



Through Generate a Load group Mobile is made.
Through the action **Preview** the result of the asked exploitation can be asked for.

1. Description of the influence line +
The selected load systems for which the exploitation is done:
Influence line:
Member B33, Position : 2.50[m], Type : My
Considered load systems:
Crane Track
Unit Load : Left

2. Co-ordinates of the nodes of the load track and their ordinates:

Node	X [m]	Y [m]	Z [m]
9	9.000	0.000	3.750
18	9.000	5.000	3.750
27	9.000	10.000	3.750
36	9.000	15.000	3.750

3. Areas of the fields of the influence line:

Area Nr	Area
1	-0.000
2	1.940
3	-0.471
4	0.088

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

Sign Nr	X [m]	Y [m]	Z [m]
0	9.000	0.000	3.750
1	9.000	0.049	3.750
2	9.000	5.197	3.750
3	9.000	10.418	3.750
0	9.000	0.000	3.750
1	9.000	0.049	3.750
2	9.000	5.197	3.750

3	9.000	10.418	3.750

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 : Crane Track

Sum P [kNm]	Sum Q [kNm]	X1 [m]	X2 [m]
-5.590	0.000	7.275	7.275

Positive maximum position : Crane Track

Sum P [kNm]	Sum Q [kNm]	X1 [m]	X2 [m]
27.074	0.000	2.100	2.100

7. Results:
Negative maximum position : Crane Track

Description	Due to P	Due to Q	P + Q	Units
My negative	-5.590	0.000	-5.590	[kNm]

Positive maximum position : Crane Track

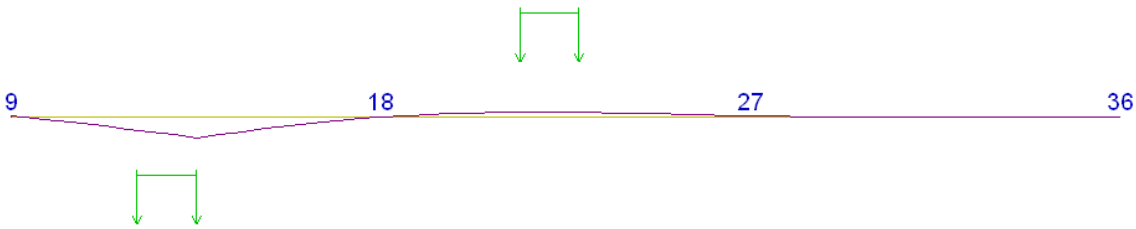
Description	Due to P	Due to Q	P + Q	Units
My positive	27.074	0.000	27.074	[kNm]

As expected the maximal moment **My** on the position **2.5m** arises when the crane track is on the left hand side of the hall:

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

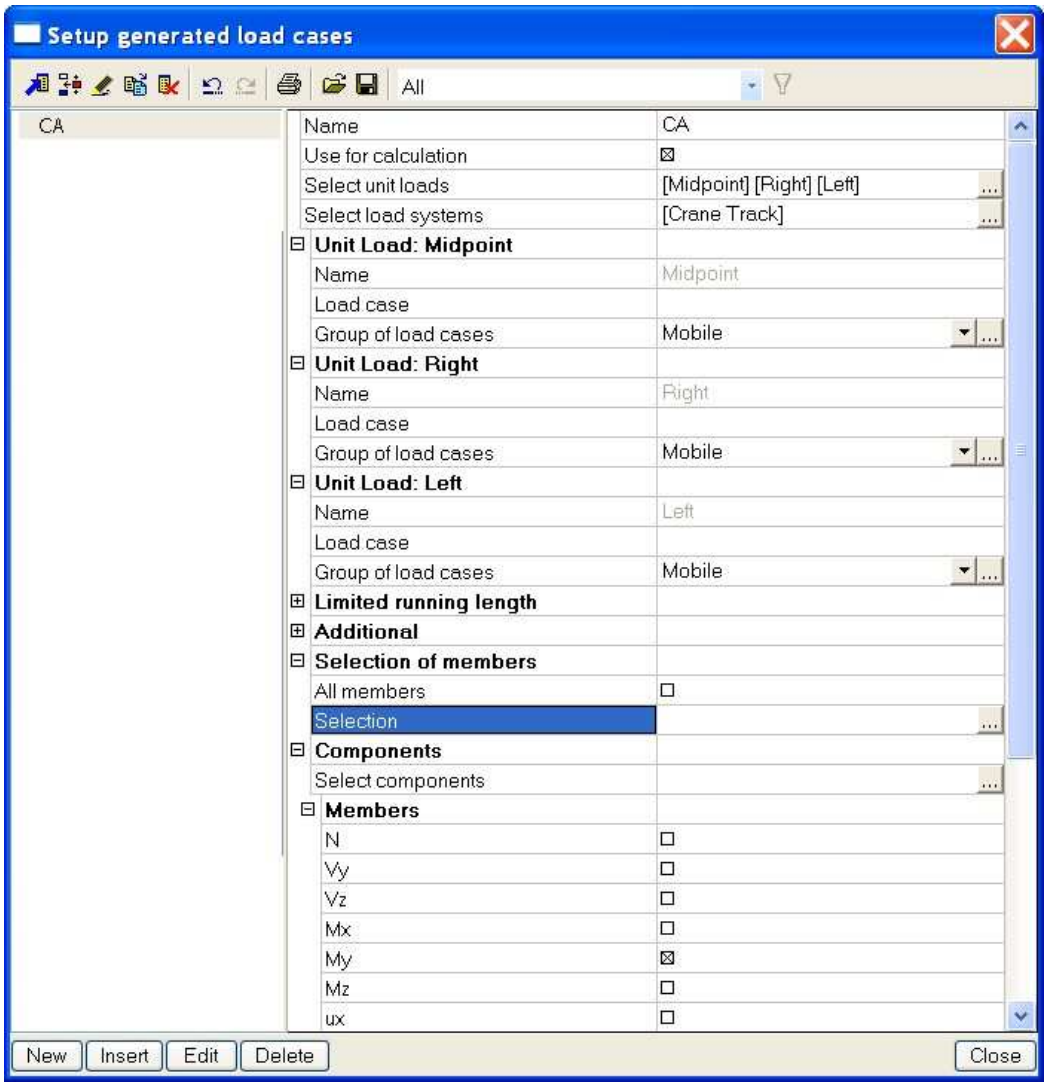
My is minimal **(-5.590 kNm**) on **2.5m** if the reference point of the crane track is on **7.275m** from the begin point of the track.
My is maximal (**27.074 kNm**) on **2.5m** if the reference point of the crane track is on **2.1**m from the begin point of the track.
The values X1 and X2 are the same since a single load system was used.

This result is also shown graphically:



g) Generation Enveloping Load Cases

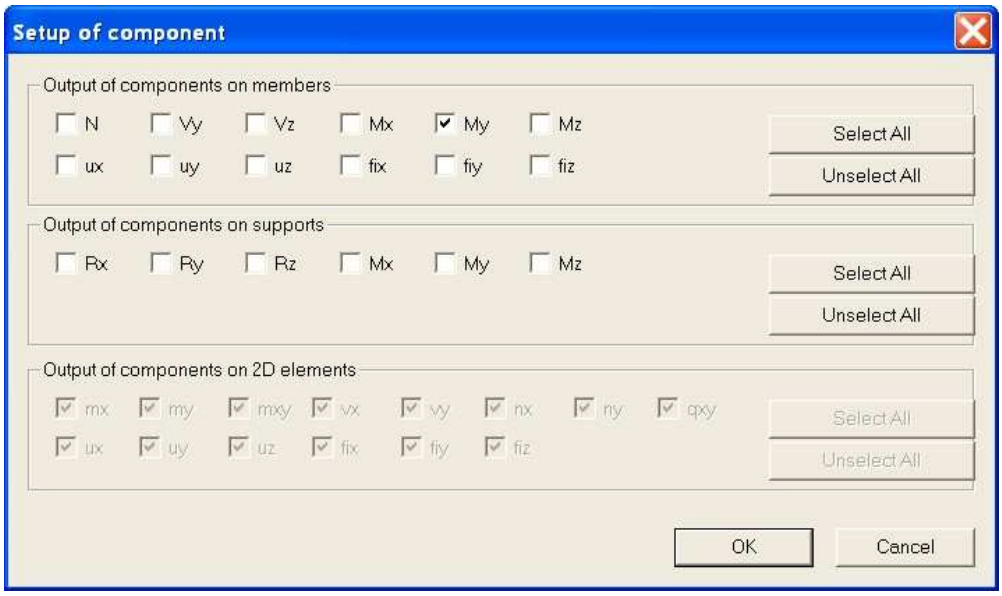
For the component My the enveloping load cases are generated through the option
Setup generated load cases .



First of all you have to indicate which unit loads and which load systems have to be taken into account. In this example all unit loads are selected.

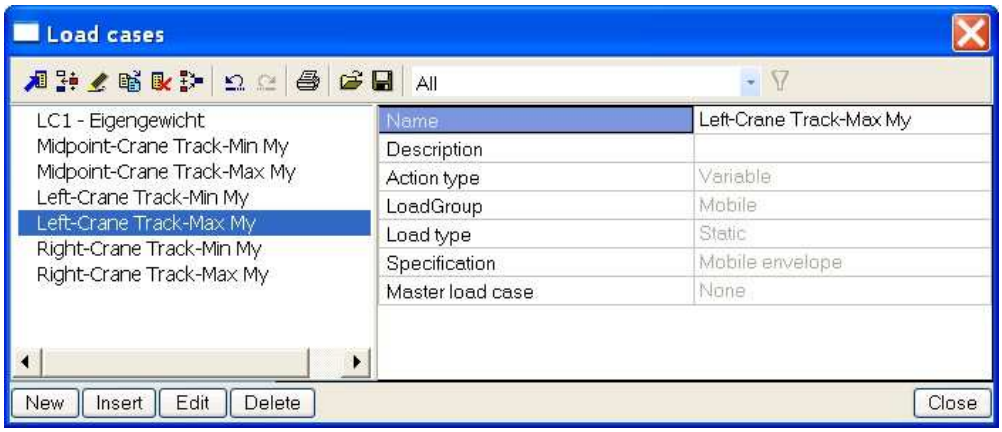
Subsequently you can enter the name through the option **Name Load case.** This is not necessary. For a load group the name **Mobile** is chosen, this load group has been created before in the Detailed analysis.

With **Selection of members** the option **All members** is deselected and the member **B33** is indicated. Through **Select components** you can indicate for which components an envelope has to be generated. In this example, only the component **My** is considered.



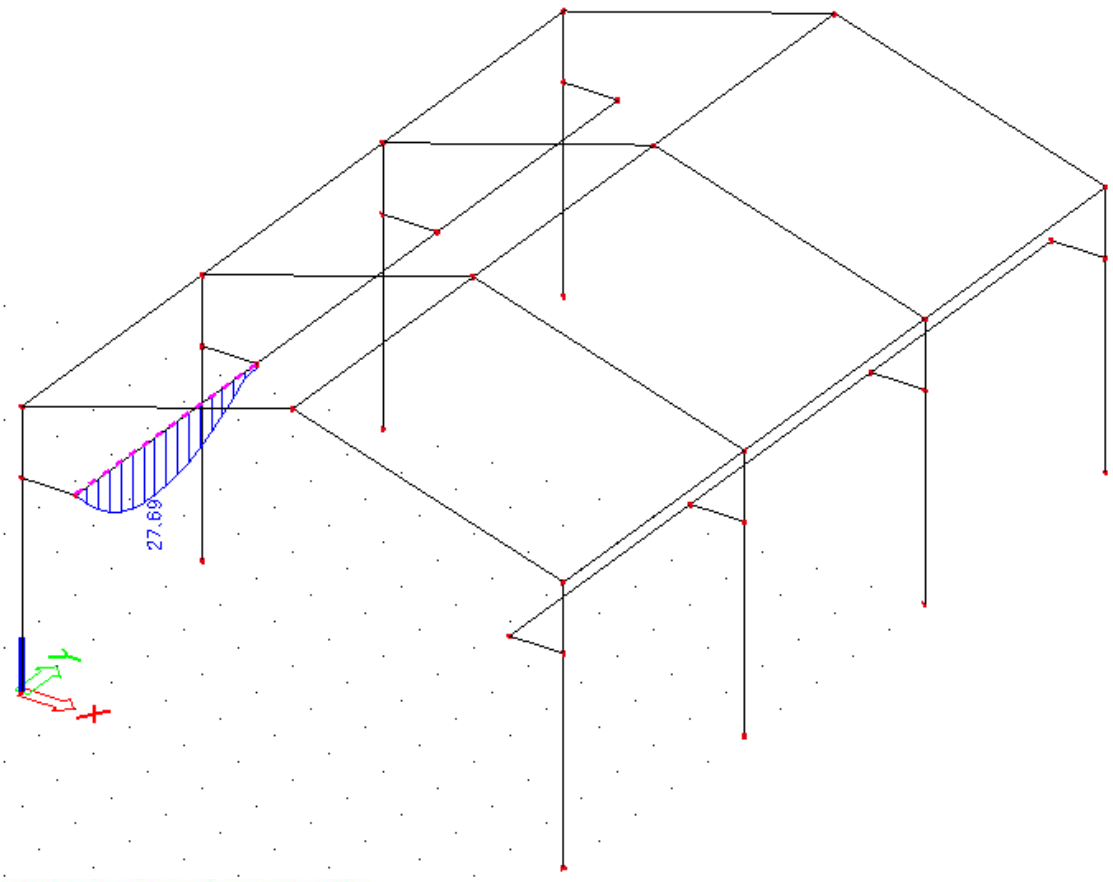
After entering these data, a linear calculation can be performed so the enveloping load cases are made.

After the calculation the Load cases manager displays the following:



The load cases have Mobile envelope as a description and are in an exclusive load group. If required, the load group can be adjusted, e.g. to set a moment factor according to NEN or a Load Type according to EC1991.

Subsequently, the results of these envelopes can be viewed. The moment course **My** on member **B33** for load case **Left – Crane track – Max My** shows the following:



Internal forces on member

Linear calculation, Extreme : Global, System : Principal
Selection : B33
Load cases : Left-Crane Track-Max My

Member	Case	dx [m]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
B33	Left-Crane Track-Max My	0,313	**-0,05**	**-0,02**	**24,89**	**0,00**	8,17	0,04
B33	Left-Crane Track-Max My	4,688	**0,14**	**0,05**	**-28,75**	**-0,01**	2,77	**0,12**
B33	Left-Crane Track-Max My	0,000	0,11	0,04	-2,22	0,00	**0,01**	**-0,10**
B33	Left-Crane Track-Max My	2,188	0,07	0,03	11,35	0,00	**27,69**	0,00