



TUTORIAL

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 2024 SCIA nv. All rights

Table of Contents

Table of Cor	ntent	S	1
Introduction			3
Chapter 1:	Lo	ad panels	4
1.1.		General properties	5
	1.1.1.	Panel type	5
,	1.1.2.	Load transfer direction	5
	1.1.3.	LCS type	5
	1.1.4.	Swap orientation	5
	1.1.5.	LCS angle	5
	1.1.6.	Selection of entities	5
	1.1.7.	Load transfer method	6
,	1.1.8.	Generating loads	7
1.2.		Panel types	
,	1.2.1.	Load to panel nodes	
,	1.2.2.	Load to panel edges	10
	1.2.3.	Load to panel edges and beams	12
,	1.2.4.	Panel with parallel beams	13
Chapter 2:	Fr	ee loads	15
2.1.		Properties of a free load	15
2	2.1.1.	Direction	16
2	2.1.2.	Туре	16
2	2.1.3.	Distribution	
2	2.1.4.	Validity	16
2	2.1.5.	Select	19
2	2.1.6.	System	19
2.2.		Example: Rectangular swimming pool	20
2.3.		Example: Cylindrical tank	23
2.4.		Result: surface loads	
2.5.		Soil and Water pressure based on borehole	
Chapter 3:	3D) wind generator	29
Chapter 4:	Tra	affic loads	33
4.1.		Example model	33
2	4.1.1.	Project data	33
2	1.1.2.	Construction	33
4.2.		Input traffic loads	
2	1.2.1.	Traffic lane	34
4	4.2.1.	Traffic loads generator	

Chapter 5	: M	obile loads (32-bit)	
5	.1.	Principle	36
5	.2.	Influence lines	38
	5.2.1	. Input track and unit load	38
	5.2.2	Influence lines	40
5	.3.	Load system	41
	5.3.1	. The construction	41
	5.3.2	. Input track and unit load	42
	5.3.3	. Input load systems	43
	5.3.4	Exploitation of load systems	44
	5.3.5	. Generation Load cases – Enveloping Load cases	50
5	.4.	Train loads	55
	5.4.1	. The construction	55
	5.4.2	. Input track and unit load	57
	5.4.3	Influence lines	60
	5.4.4	. Input load systems	61
	5.4.5	Exploitation of the load systems	63
	5.4.6	. Generate load cases – Envelope load cases	65
5	.5.	Crane track	69
	5.5.1	The construction	70
	5.5.2	. Input track and unit load	73
	5.5.3	. Input load system	77
	5.5.4	. Exploitation of the load system	78
	5.5.5	Generation Enveloping Load Cases	81
Chapter 6	;: 2C	D wind- and snow generators (32-bit)83	
6	.1.	Wind generator	84
6	.2.	Snow generator	90
6	.3.	Wind & Snow generator	92

Introduction

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

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

Chapter 1: Load panels

Load panels are entities which are not taken into account in the FEM calculation (Finite Element Method Calculation). Load panels have no self-weight, have a certain stiffness to distribute loads to underlying members but this stiffness is not taken into account in the stiffness of the structure.

All types of loads can be applied to the load panels and will be distributed to the underlying members of the load panel.

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



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



1.1. General properties

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

1.1.1. Panel type

There are 3 types of load panels:

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

Each of these types will be demonstrated in paragraph 1.2.

1.1.2. Load transfer direction

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

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

1.1.3. LCS type

This property adapts the local coordinate system of the load panel, this section is completely optional, because you will be able to turn the LCS with the property 'LCS angle'.

Note: The load panel does not have mesh elements, therefore most of the types will behave similar to each other. More information for 2D members and the LCS type can be found here.

1.1.4. Swap orientation

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

1.1.5. **LCS angle**

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

1.1.6. Selection of entities

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

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

1.1.7. Load transfer method

Standard:

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

Tributary area

The beams are loaded based on the tributary area of the particular beam.

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

Accurate (fixed link with beams/hinged link with beams):

The finite element method is used to recalculate the applied load to individual beams.

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



• Hinged link with beams



1.1.8. Generating loads

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

ACTIONS >>>>	
Update edge/beam selection	
Update all load panels	
Generate loads	
Table edit geometry	

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

					F					
			Effective width of plate Structure nodes Member parameters Local axes System length labels Nonlinearity labels Labels of local axes Structural shape labels			7				
	w parameters setting - Loads/masses heck / Uncheck gro Structure Labels Amo	_	View settings for all entities	Modellin	g/Drawing	Attribu	ites	🧭 Misc.	Lock po	sition
	heck / Uncheck all									
	Service									
	Display on opening the service	•								
E	Display loads									
	Display	I								
	Style	Colour by ad	tion type							•
			uon type							
	Load case	LC2 - Roof								•
	Display eccentricity									
	Generators	Generated								-
E	Point forces	Original Generated	ď							
	In node	Original + G	enerated							
Ξ	Surface loads									
	On 2D member	~								
Ξ	Labels of loads									
	Display label	~								
	Name									
	Value									
	Tot. value	I								
	Eccentricity label									
	Show names in tab							ОК	Apply	Cancel

1.2. Panel types

1.2.1. Load to panel nodes

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



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

🏘 S	5Ci	A 22.1.1	025.64					
D	Ø	*	(i)		۲	Ö	Ø2	
	4	Repeat 'F	Rectangu	ılar grid'		Ente	r	
	\$	Undo				Ctrl+	z	
		Redo				Ctrl+	r	
		Modify					•	
	٤°	Deletion	settings					
	*	Copy/pa	ste prope	erties	Ctr	rl+Shift+	F	
		Add data	1				•	
		Metadata	а					
	1	Polyline	edit				- Ic	± Add node
		Curves e	dit				•	🞽 Delete rode
		Solids					c	D Break into curves
							- c	Connect curves
	-		-				5	Fillet

Load transfer method 'standard'

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

In the image below, the weight distribution factor is set on 1 for each node, which means that each node will have the exact same point load: $-10 \text{ kN/m}^2 \text{ x} (10 \text{ m} \cdot 5 \text{m}) = -500 \text{kN}$ and -500 kN / 4 nodes = -125 kN / node.



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



Excluding a node can be done by changing the weight factor to zero or by changing the selection of entities as explained in paragraph 1.1.7.

1.2.2. Load to panel edges

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

Load transfer method 'standard'

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

Load transfer direction X:



Load transfer direction Y:



Load transfer direction All:



Load transfer method 'Tributary area'

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



Load transfer direction All:

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



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

1.2.3. Load to panel edges and beams

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

Max. angle for transfer

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

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

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

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



Max. angle for transfer = 45 deg.



1.2.4. Panel with parallel beams

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



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

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

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



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



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

Chapter 2: Free loads

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

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

2.1. **Properties of a free load**

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





2.1.1. Direction

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

2.1.2. **Type**

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

2.1.3. **Distribution**

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

<u>Uniform</u>: one constant value for the complete surface load.

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

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

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

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

2.1.4. Validity

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



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

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





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

2.1.5. Select

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

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

ACTIONS >>>>	
Senerate loads	
Move UCS	
S Edit plane load geometry	
O Update 2D members selection	

2.1.6. **System**

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

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

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



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



The LCS has the same direction as the GCS

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





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

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

2.2. Example: Rectangular swimming pool

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



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

	heck / Uncheck group							Lock position	
0	🔲 Structure 🔷 Labels 👗 Mode	el 🛃 l	Loads/masses	T Composite	Modelling/Drawing	😚 Attributes	Misc.	C View	
	Check / Uncheck all								
	Rendering	transpi	arent						•
	Highlight supporting edges/nodes								
	Load distribution symbol	~							
	Display linked members	~							
Ξ	Structure nodes								
	Display	V							
	Mark style	Dot							•
-	Member parameters								
	System lengths								
	Member nonlinearities	1							
	FEM type	4							
	Joists	Γ							
	Local axes								
	Nodes								
	Members 1D								
	Members 2D	~							

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

		LC2 🗸	🔁 🔛
ŧ X Υ ×	(Y workplane		
įγz γ	/Z workplane		
<mark>≜X</mark> Ζ Χ	Z workplane		E
12 N	love		F9
t. F	From 3 points		
1 🖵 f	rom Local Member		F10
A 1 6	Pacet to GCS position	N3	Ctrl+E11

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

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

To create a triangular load, the distribution is modified from 'uniform' to Dir Y and the values are set. The value is 0 kN/m^2 on the top and for example -25kN/m^2 on the bottom of the wall.

Surface force free		×
P. I.I.I.I.I.P	Name FF2 Direction Z	*
	Type Force	~
	Distribution Dir Y	~
ALL ALL	q1[kN/m^2] 0,00	
Thereit	P1	×
	q2 [kN/m^2] -25,00	
	P2	×
	Validity All	*
	Select Auto	*
	Geometry	
STRUE MAN	System MemberLCS	*
	Location Length	
		~
		OK Cancel

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



Generate the free load. The loads will be generated on each wall AND the floor based on the LCS. The load on the floor is a uniform surface load.

Notice that some loads are pointed inwards, this is caused by the direction of the LCS of the 2D element. You can simply modify the LCS of these 2D elements by selecting them and toggle on 'swap orientation' in the 2D properties.

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



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



2.3. Example: Cylindrical tank

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

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



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

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



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

2.4. Result: surface loads

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



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



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

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

FE analysis		×
Calculations	Mesh setup	
🔽 Linear analysis	Average number of 1D mesh elements (
Load cases: 2	Average size of 1D mesh element on cu 0,200	
Other processes	Average size of 2D mesh element [m] 0.1	
	Connect members/nodes 💙	
Test input of data	Setup for connection of structural entit	
	Advanced mesh settings	
Save project after analysis	Solver setup	
	Specify load cases for linear calculation	
	Advanced solver settings	
		*0
Calculate	Average size of 2D mesh element	

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



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

2.5. Soil and Water pressure based on borehole

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

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

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





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

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

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

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

Surface force free		×
P C C C C C C C C C C C C C C C C C C C	Name FF2 Direction Z Type Soil pressure Distribution Uniform Coeff1 [-] 1,000 Borehole profile BH1 Validity Z=0 Select Auto	× • •
	Geometry System Member LCS Location Length	×
		OK Cancel

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

	qz [kN/m^2] 45.03	(ip)		
	42.44	RESULTS	5 (1)	∧ ×
	39.84 37.25	Name	Surface loads	
	34.65 32.06	▼ SELECTION		
	29.46 26.87	Type of selection	All	\sim
	24.27 21.68	Filter	No	\sim
	19.08 16.49	RESULT CASE		
		Type of load	Load cases	\sim
		Load case	LC2 - Soil and Wate	r p 🗸
		▼ EXTREME		
		Extreme	Member	\sim
		Values	q_z	\sim
		▼ SYSTEM		
		System	LCS mesh element	\sim
		▼ ERRORS, WARNINGS AND I	NOTES SETTINGS	
		Show Information about w		
		Show errors	All	\sim
		Show warnings	All	\sim
	٦,	Show notes	None	\sim
100		Show table with explanatio	\bigcirc	
		ACTIONS >>>>		
		🔁 Refresh		F5
		Drawing setup 2D		
		Results table		
		Report preview		



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



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

III Line force on beam			×
RZ)	Name		
Rx RyP2	Direction		٧
	Туре	Soil pressure	٧
	Angle [deg]		
-P1	Distribution	Uniform	~
	Acting width [m]	1,000	
ey tez	Coeff1	1,000	
	Coeff2	1,000	
x1 x2	Load above joint	no	
	Borehole profile	BH1	¥
	4 Geometry		
	System	GCS	
	Location	Projection	
	Extent	full	
	Coord. definition	Rela	~
	Position x1	0,000	
	Position x2	1,000	
	Origin	From start	¥
	Eccentricity		
	Eccentricity ey [m]	0,000	
	Eccentricity ez [m]	0,000	
		OK Cance	4

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



Chapter 3: 3D wind generator

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

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

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

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



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

Project data	X Setup manager X
	Standard EN Name Standard EN
Basic data Functionality Actions Unit Set Protection	é Wind • Wind
base data i functionality Actions officient i foreculon	Wind pressure according to EC1 Wind pressure according to EC1
	✓ Internal pressure for 2D wind
Acceleration of gravity 9,810 m/s^2	Internal pressure for 2D wind no internal pressure *
	Position of dominant face for 2D wind front
	Openings dominant face for 2D wind two times
	 External pressure for 3D wind
WIND LOAD	External pressure for 3D wind Use overall coefficients Cpe, 10 *
	Correlation between zones D and E
According to code v EC 1/26,200m/s / 0	 Reference height (z_e)
	Type of the structure Vertical walls or rectangular buildings (EC: *
	b - width of the structure [m] 100,000
SNOW LOAD	Reference level of terrain [m] 0,000
SNOW LOAD	✓ c_dir - directional factor
According to code v EC 1 / Sk=1,00kN/m^2 Ce=1,0 Ct=1,0	Value [-] 1,00
	✓ c_season - season factor
	Value [:] 1,00
	✓ c_o - orography factor
POND LOAD SEISMIC COMBINATIONS	Value [-] 1,00
	V_b,0 - basic wind velocity [m/s] 26,200
Model factor: 1,30 Factor for concomitant components 0,30	ro - air density [kg/m*3] 1,25
	 Probability
	1/p - life period of the building [year] 50,00
	c_prob - probability factor [-] 1,00
CODE COMBINATIONS	K-shapefactor[-] 0,20
	n - exponent [-] 0,50
Automatic C	* Terrain
	terrain category 0 ×
	Kr - terrain factor [-] 0,16
	z_0-roughness length [m] 0,003
	z_min - minimal height [m] 1,000
	k_l - turbulence factor [-] 1,00
OK Can	Load default NA parameters OK Cancel

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

-			
PANEL ((1)		
🌶 🛃 🛃 🌈			
Name L	.P1		
Layer L	.aag1 🗸 🕂		
Panel type T	To panel edges and beams \sim		1
Shape F	lat		
Load transfer direction X	((LCS panel) V		
Max.angle for transfer [deg] 5	5,00		
LCS type S			
Swap orientation			
LCS angle [deg] 0),00		
3D Wind			
Supporting members validity range (with re A	ui ~		
Max. eccentricity of members [m] 0),200		
Load transfer method A	Accurate(FEM), fixed link with beams		
Selection of entities A	ui ~	(
▼ ILLUSTRATION GROUP		• • •	
		WIND	DATA (1)
	Real Provide American Street	-	
	J. Line	Name	WD5
		Pane	LP5
		Туре	Wall 🗸 🗸
ACTIONS >>>>			
Update edge/beam selection Update all load panels		Swap outer surface	
Table edit geometry		Opening	

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

-			
=	PANEL (2) > 1	WIND DATA (2)	2
≡ ≠			
	Panel	MIXED	\sim
	Туре	Roof	~
	Roof type	Duopitch	\sim
	Roof overhangs	0	
	Swap outer surface	\bigcirc	
▼ ILLUSTRATION GROU	P		
#A:	#4	#B	

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

INPUT PANEL	3D Wind Generator	х	Ad	d Wind Load Ca	ses					×
💼 All workstations 🗸	Load cases				+ CPE, + CP	+ CPE, - CPI	- CPE, + CPI	- CPE, - CPI		
📑 All categories 🗸 🗸				0 90					0,20 0,20	
All tags				180					0,20	
➡ Longitudinal strain on 1D ➡ Flexural strain on 1D			4	270					0,20	-0,30
▼ CLIMATIC LOADS	Add Load Cases			lculation metho Additional loa Direction of rio	d cases for due	opitch roofs	¥ Both (X and Y ¥			
▼ SPECIAL LOADS	Run generator Close			Include torsion	nal load case			ОК	Ca	ancel

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

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

Zone	Α		в		С		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	

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

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

Table 7.4a — External pressure coefficients for duopitch roofs

Ditab	Zone	for wind	directio	n θ = 0°						
Pitch Angle α	F		G		н		1		J	
Aligie a	Cpe,10	Cps, 1	Cps, 10	Cpe,1	Cpe,10	Cpe,1	Cpe,10	Cpe,1	Cps,10	Cpe,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°							+0,2		+0,2	
-5*	-2,3	-2,5	-1,2	-2,0	-0,8	-1,2	-0,6		-0,6	
5°	-1,7	-2,5	-1,2	-2,0	-0,6	-1,2			+0,2	
5-	+0,0		+0,0		+0,0		-0,6		-0,6	
4.50	-0,9	-2,0	-0,8	-1,5	-0,3		-0,4		-1,0	-1,5
15°	+0,2		+0,2		+0,2		+0,0		+0,0	+0,0
	-0,5	-1,5	-0,5	-1,5	-0,2		-0,4		-0,5	
30°	+0,7		+0,7	-	+0,4		+0,0		+0,0	
450	-0,0		-0,0)`	-0,0		-0,2		-0,3	
45°	+0,7	1	+0,7		+0,6		+0,0		+0,0	
60°	+0,7		+0,7		+0,7		-0.2		-0,3	
75°	+0,8		+0,8		+0,8		-0,2		-0,3	

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

Load groups	3D Wind Generator X
Image: Second	3D Wind Generator X Load cases 3DWind1 0, + CPE, + CPI 3DWind2 0, + CPE, + CPI 3DWind3 0, -CPE, + CPI 3DWind3 0, -CPE, + CPI 3DWind3 0, -CPE, + CPI 3DWind5 0, -CPE, + CPI 3DWind5 0, -CPE, -CPI 3DWind5 0, -CPE, -CPI 3DWind5 0, -CPE, -CPI 3DWind5 0, -CPE, -CPI 3DWind5 0, -CPE, -CPI 3DWind1 100, -CPE, -CPI 3DWind1 100, -CPE
	Add Load Cases
New Insert Edit Delete Close	Run generator Close

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



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

Vi	ew parameters setting - Loads/masses				
	Check / Uncheck group		Lock pos	ition	
4	The Structure I abels A Model	🛃 Loads/masses 🛛 🍸 Composite 🛛 🔛 Modelling/Drawin			
	Check / Uncheck all				
	Service				Sa Maria
	Display on opening the service				Mm/r=12100
	Display loads				
	Display				
	Style	Colour by action type			Gaing 30
	Load case	3DWind1 - 0, + CPE, + CPI		<u>-</u>	
	Display eccentricity	Downing to , + crc, + cri		<u> </u>	0
	Generators	Generated			
	Surface loads	ochentee		<u> </u>	Cam-a.a. a.
	Free	7			
l c	Labels of loads	1			
11	Display label	7			
	Name	E Contraction of the second se			
	Value	~			
	Tot, value				
	Eccentricity label				
I F	Wind data				
11-	Display	V			
l e	Labels of wind data				
	Display label				
	Roof overhangs underside labels				
	Cpi				
	Cpe	v			
	Net Pressure				
	Peak Pressure				
	1				
	Show names in tab		🕑 🕒 ОК Арріу	Cancel	

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



Chapter 4: Traffic loads

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

Note: Traffic loads are not compatible with 1D elements. Mobile loads on 1D members is currently only available in the 32-bit version of SCIA Engineer. Influence lines are also not available in the 64-bit version. In order to use this functionality you will also need to use the 32-bit version and change the post-processing environment to 'v16 and older'. This is further explained in chapter 5.

4.1. **Example model**

4.1.1. Project data

A new project is created with the following parameters:

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

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



4.1.2. Construction

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



4.2. Input traffic loads

4.2.1. Traffic lane

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

INPUT PANEL	
All workstations	\sim
All categories	\sim
All tags	\sim
 TRAFFIC LOADS 	
Single traffic loads	
Traffic lane	
Traffic loads generator	

This track will consist of two rails with a distance of 1,4m between them. To make sure that the train drives on two rails at the same time, 1 traffic lane is entered. The 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).

4.2.1. Traffic loads generator

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

Traffic loads

In the traffic loads generator you define the traffic loads.

Traffic loads generate	or		×	🔳 Loa	d patterns					×		
et -: 🖸 🕩 🖬 🔹	* 🗢 🗖 🕞 🖸	All 👻	T	# -8	🗷 🕩 🖬	٠,	* 🖬 위 🕞 🛛	All	¥	T		
Ш1	Name	111										
	Traffic Loads		×									
	Traffic lane		*				Train Loads					×
	Load group	LG2	×							3		
	Load case name										Name LP1	
	Step [m]	1,000									Description	
	Validity										Insert point	
											X-coordinate [m] 0,000	
											Y-coordinate [m] 0,000	
				_							✓ Validity	
											Validity	*
											4 Entities	_
											Type Point	*
											Add new entity	
												_
											ок с	ancel
	Actions						L			_		10
		Generate loads	>>>									9
		Draw validity in traffic lane	>>>							_		
New Insert Edi	t Delete		Close	New	Insert	Edit	Delete			ок		

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

X Name LP1 Description Insert point X-coordinate [m] 0,000 Y-coordinate [m] 0,000 Validity Validity Validity Validity Validity Validity Add new entity Point Add new entity Point Line Rectangle Turning joint		
Description Insert point X-coordinate [m] 0,000 Y-coordinate [m] 0,000 Validity Validity Validity Validity Validity Add new entity Point Add new entity Point Rectangle	×	
Description Insert point X-coordinate [m] 0,000 Y-coordinate [m] 0,000 Validity Validity Validity Validity Validity Add new entity Point Add new entity Point Rectangle		
Insert point X-coordinate [m] 0,000 Y-coordinate [m] 0,000 Validity Validity Validity Validity Type Point Add new entity Point Rectangle	Name LP1	Name
X-coordinate [m] 0,000 Y-coordinate [m] 0,000 Validity Validity V Entities Type Point Add new entity Point Rectangle	ription	Description
Y-coordinate [m] 0,000 Validity Validity Validity V	int	Insert point
Validity Validity Validity Entities Type Point Add new entity Point Rectangle	ate [m] 0,000	X-coordinate [m]
Validity * Entities Type Point * Add new entity Point * Line Rectangle	ate [m] 0,000	Y-coordinate [m]
Entities Type Point Add new entity Point Line Rectangle		Validity
Type Point A Add new entity Point Rectangle	Validity 💙	Validity
Add new entity Point Line Rectangle		
Rectangle		
Rectangle	w entity Point	Add new entity
Add an entity, define the Force and the Repetition of the point load. In this case, there will be two train tracks with a distance in between of 1.4m so we repeat the point load twice according to the y-direction and set the Delty y to 1.4m.

Train Loads					×
		Validity		¥	^
		Entities			
		Туре	Point	۷	
		Add new entity			
		Delete entity			
	1	Entity 1			
1.400		Direction	Z	٣	
		Force [kN]			
		Position x1 [m]			
		Position y1 [m]			
x .		Repeat x (n)		_	
		Repeat y (n)		_	
x	<	Delta y [m]	1,400	Ļ	~
			ок с	ance	el

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

Generate loads

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

-1 🗹 🕩 🖬	🐟 🗢 🔲 📄 🖌 All		• T	*	Moving \checkmark
	Name LL1			J ^{III} Self Weigth	[
	Traffic Loads LP1		×	Moving Load - TR1/LP10,000	m
	Traffic lane TR1		~	I [™] Moving Load1 - TR1/LP10,25	
	Load group LG2		×	1 [™] Moving Load2 - TR1/LP10,50	
	Load case name TR1/LI	P1			
	Step [m] 0,250			Moving Load3 - TR1/LP10,75	
	Validity			Moving Load4 - TR1/LP11,00	
				Moving Load5 - TR1/LP11,25	
				Moving Load6 - TR1/LP11,50	0 m
				Moving Load7 - TR1/LP11,75	0 m
				Moving Load8 - TR1/LP12,00	0 m
				Moving Load9 - TR1/LP12,25	0 m
				Moving Load10 - TR1/LP12,5	
				I ^{III} Moving Load11 - TR1/LP12,7	n
	Actions			↓ Moving Load12 - TR1/LP13,0	in a
		Generate loads		1.	
		Draw validity in traffic lane	>>>	IIII Moving Load13 - TR1/LP13,2	50 m
		1			
ew Insert Ed	dit Delete		Close	Manage load cases	Ctrl+L
	001		Close	Manage load cases	Ctrl+L

Chapter 5: Mobile loads (32-bit)

In this chapter 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.

5.1. **Principle**

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

This is illustrated on the picture below:



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

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

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

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

This is illustrated on the following figure.



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

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

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

At which 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.

MC - 2024/03/06

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.

5.2. Influence lines

In this first project **Influence lines.esa** 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.



Open the menu - Mobile loads

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.

5.2.1. Input track and unit load

You can import this track through 7 Traffic Lane

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.

Name	TRI
Use for calculation	
Used nodes	2
Track nodes	
Node	N1 [B1]
Node	N2 [B1]

The option 'Use for calculation' shows that this track is taken into account in the calculation. If more tracks are defined, this 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.

EHL		Name	EHL	
		Track assignment	TRI	-
		Sections	Use sections from results	
		Step for 2D element [m]	1,000	
		Generate section under Load system	0	
		Add new Impulse		
	8	Impulse 1		
		Туре	Concentrated	
		Value	-1	
		Position [m]	0.000	
		ey[m]	0,000	
		ez [m]	0,000	
		System	Local	
	-		11 (-1)	
	1	7	II (-1)	

After defining the track, the unit load can be imported through the menu ¹ Unit loads

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

The option <u>Sections</u> 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 <u>Generate section under Load system</u>, 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.

5.2.2. 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 $-rac{120}{3}$ Calculation in the project toolbar.

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

🛃 Infuence lines
- 🗢 Deformations on member
- 😽 Internal forces on member
- 🗢 Displacement of nodes
- ∽ Supports
↔ Member stresses

Note: Influence lines are only available in post-processing environment v16 and older.

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.

	8	B1 0.000 1.000 2.000			EHL ov		beam B1	- section	
		3.000			poz	N	Vz	My	
		4.000 5.000			0.00	0.000	0.000	0.000	
		5.000			0.00	0.000	0.000	0.000	
>		6.000 7.000			1.00	0.000	-0,100	+0.500	
Toracio di Companya di Comp		8.000			2.00	0.000	-0.200	+1.000	
>>		9.000			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		
					5.00	0.000	+0.417		
					5.00	0.000	+0.500	0.0000000	
					6.00	0.000	14120000000000	+2.000	
					7.00	0.000	+0.300		
	L				8.00	0.000	+0.200	THE REAL PROPERTY.	
Group	select	tion Deselec		Deselect all	9.00	0.000	+0.100	0.0000000	
	neu	Deselec	<u> </u>	Deselect all	10.00	0.000	0.000	0.000	
		ОК		Cancel	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.

beam

5.3. 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.



5.3.1. The 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 $\xrightarrow{2}$ Beam, at which the begin node is imposed hinged and the other nodes are rolled.



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

5.3.2. 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.

se for calculation sed nodes	0
and nodes	
seu noues	4
ack nodes	
ode	N1 [B1]
ode	N2 [B1]
ode	N3 [B2]
ode	N4 [B3]
	ode ode

After defining the track, a unit load can be inserted through the menu $\stackrel{\text{\tiny 22 Unit loads}}{=}$.



5.3.3. 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.

5.3.4. 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:

🛱 Detail analysis

Member force, deformation

🎋 Reaction

A Member stress

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:

	Jnit loads	Exploitatie van invloedslijnen - Staven EHL
L	n anna a status anna anna anna	
	.oad systems	[P Loads left] [P Loads right]
B L	imited run	
1	Additional	
1	.oad case	
S	Setup report	
S	Selected members	[B1] . More comp
1	/alues	More comp
Ν	4	
1	/z	
N	hy	
U	IX.	
U	Z	
fi	iv .	

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.

<u>Under *Title 1*</u>. 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:

	Name	Exploitatie van invloedslijnen - Staven	
	Unit loads	EHL	•
	Load systems	[P Loads left] [P Loads right] [Q Load]	
Ŧ	Limited run		
÷	Additional		
ŧ	Load case		
	Setup report		
	Selected members	[B1]	
	Values	More comp	-
	N		
	Vz		
	My		
	ux		
	uz		
	fiy		

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 is 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.



5.3.5. Generation Load cases – Enveloping Load cases

SCIA Engineer allows making both single and enveloping load cases.

Generate 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:

	Name	Exploitatie van invloedslijnen - Staven
	Unit loads	EHL
	Load systems	[P Loads left] [P Loads right] [Q Load]
Ŧ	Limited run	
÷	Additional	
Ξ	Load case	
	Generate	
	Load group	Mobile
	Setup report	
	Selected members	[B1]
	Values	More comp
	N	
	Vz	
	My	
	ux	
	uz	
	fiy	

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

Numerical and graphical output - 1/1	×
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	-
1 2 3 4	
Section : 24.000 Unit Generate load cases : EHL Mox. My, B1, P.11 To Document On Discussion : EHL, Min, My, B1, P.11	150
Updeta	

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:



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

Generate 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 - ¹/₄ Setup generated load cases is used in the menu Mobile Loads.

CA		Naam	CA	_
CA .		Gebruik voor berekening	8	
		Selecteer eenheidslast	IEHL]	
		Selecteer lastsystemen	[P Lasten Links] [P Lasten Rechts] [Q Last]	
		Eenheidslast: EHL		_
		Naam	EHL	
		Belastingsgeval		
		Groep van belastinggevallen		
		Genereer namen		
	+	Beperkte looplengte		_
	÷	Extra		
	Ξ	Selectie van staven		
		Alle staven		
		Componenten		
		Selecteer componenten		
	6	3 Staven		
		N		
		Vy		
		Vz		
		Mx		
		My		
		Mz		
		ux		
		uy		
		uz		
		fix		
		fiy		

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.

₹ N	EW	I▼ ∨z	E M	 I 	му Г	Ma		Select All
🔽 ux	E vy	🔽 uz	T for		iy F	fiz		Unselect All
utput of	componen	ts on suppo	orts					
Rx Rx	☐ Ry	🔽 Rz	Гм	v (v)	му Г	Mz		Select All
								Unselect All
utput of	component	ts on 2D ele	ements					
🖂 mx	₩ my	₩ mxy	₩ v×	Vy	⊡ nx	Miny	₩ qxy	Select All
w ux	₩ uy	₩ uz	₩ fix	🗹 fiy	🗹 fiz			Unselect All

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:

_C1 - Eigengewicht	Nome	EHL-P Loads left, P Loads right-Min
EHL-P Loads left, P Loads right-Min Vz	Description	
BHL-P Loads left, P Loads right-Min My	Action type	Variable
BHL-P Loads left, P Loads right-Min uz	LoadGroup	Mobile
BHL-P Loads left, P Loads right-Min fiy	Load type	Static
EHL-P Loads left, P Loads right-Max Vz EHL-P Loads left, P Loads right-Max My	Specification	Mobile envelope
HL-P Loads left, P Loads right-Max uz	Master load case	None
3HL-P Loads left, P Loads right-Max fly 3HL-P Loads left, P Loads right-Min Rz 3HL-P Loads left, P Loads right-Max Rz		

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:



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.

5.4. 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.



5.4.1. **The construction**

	Data		Structure : Plate XY	•
ten			Material :	
	Name	Project M3	Concrete	13
			Material	C25/30
en or	Part	-	Steel	0
19TH	Description	Treinloads	Timber	
222.5	Coscipation	Tremodas	Other	
EX.	Author	PVT	Aluminium	0
- 8	Dette			
	Date	14.10.2005		
the second				
100				
ANY A	Project Level :	Model :		
A day	Advanced	• One	+	
		and the second	<u> </u>	
ALC: NO.	National Code	<u>R </u>		
	1.120	EC-EN		

	amics		8	Concrete	
	al stress			Fire resistance	0
and the second se	isoil				
	linearity				
Stat	bility				
Clim	natic loads				
Pre:	stressing				
Pipe	elines				
Stru	ctural model				
Par	ameters				
Mok	oile loads	0			
Ove	erview drawings				
LTA	-load cases	0			

The bridge deck can be entered as *Plate* with thickness **500mm**.

The length of the bridge deck is 25m, the width 5m.

		25000	×
	1		1
Ì	1		1
			l
			l
			l
2000			l
40			l
			l
	ł		l
1	.1	X	

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 40 Internal edge.

				,	,		,	(Missimpurk)
								Le .
						,		
Î								
\downarrow								
	,			,			,	
▙	⇒×	 	 		 			

Using Support since 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 from right to left:

2						 	

Then we have the following structure:



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

5.4.2. 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:

Command line	
₽ < 7 ×	
New track action - Polyline - Start point >0	;1;8
Command line	
A - C7 R	
New track action - Polyline - End point >25	5;1,8

As Name of the track, TR1 is entered.

<u></u>

After defining the track, the unit load can be entered through the menu $\stackrel{\text{\tiny 2}}{\longrightarrow}$ 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.

🔊 🔆 🖋 🕷 💺 🕰 🕰	6	6	All	• 9		
Train		Name		Train	-	
		Track a	ssignment	TR1	-	
		Sections	\$	Use Step according 2D element	-	
		Step for	2D element [m]	0,250		
			te section under Load system			
			vImpulse			
	8	Impuls				
	1.23	Туре		Concentrated	-	
		Value		-1		
		Position	ſm]	0.000		
		ey[m]		0,000		
		ez [m]		0,000		
		System		Local	-	
			7			
				I		

🖉 🏦 🏒 📸 🔣 🕰 🖸	- 6	🗃 🖬 🛛 All	• 9	
Train		Delete impulse		^
	Ξ	Impulse 1		
	1	Туре	Concentrated	-
		Value	-1	
		Position [m]	0,000	
		ey [m]	0,000	
		ez [m]	0,000	
		System	Local	-
		Direction	Z	-
	8	Impulse 2		
	100	Туре	Concentrated	-
		Value	-1	
		Position [m]	1,400	
		ey[m]	0.000	
		ez [m]	0,000	
		System	Local	-
		Direction	Z	-
		No.	1,400	2

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

Both impulses are displayed on the bridge deck:



5.4.3. 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 - We 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:

election manager			<u> </u>
e		3 S1	
		Pt.1 [m]	5,000, 2,500, 0,000
		Add new point	
	> >> <		
	Select	Deselect	Deselect all
		ОК	Cancel

Influence lines - Deformation on member 2D

nuicipiy	factor : 1.00	nacro 1	- Position: x=5.0,y=2.5	,z=0
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.

5.4.4. Input load systems

Through the option - 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.

Mobile load syste	ms		
🖉 💱 🗶 📽 🔛 😟 😂 🐼 🎯 🕼 🔒 🗛		2 4	
	System database		

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

ad from database		E C
Project database	System database	
VOS8 150	KLAS 45R KLAS 60L KLAS 60R Load model 1 Lane 1 Load model 1 Lane 2 Load model 1 Lane 3 Load model 1 There 3 Model 5W/2 UIC 71 Unice 3 VOS8 100 VOS8 100 VOS8 250 VOS8 270 UIC 71 - HSL 600 E CSN CSD 7 CSN CSD 7 CSN CSD 7 CSN TRM NS CSN TRM NS CSN TRM NS	
	Copy to project	
Close	<< Copy all	

With the button <u>Copy to project</u> 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** I 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.

5.4.5. **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.

	Name	Exploitatie van invloedslijnen - 2D el.
	Unit loads	Train
	Load systems	[VOSB 150]
÷	Limited run	
÷	Additional	
Ξ	Load case	
	Generate	8
	Load group	Train 💌
	Setup report	
	Selected 2D members	[S1]
	Values	[S1]

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

1. Description of the i	
The selected load systems for	which the exploitation is done:
Influence line:	
2D macro S1.	
Global position :	[m]00.0;
x :5.00[m], y :0.00[m], z Type : mx	so.oo[m]
Considered load systems:	
VOSB 150	
Linit Load - Train	
Unit Load : Train	
	nodes of the the loadtrack and their
2. Co-ordinates of the	nodes of the the loadtrack and their
	nodes of the the loadtrack and their
2. Co-ordinates of the	nodes of the the loadtrack and their
2. Co-ordinates of the	nodes of the the loadtrack and their
2. Co-ordinates of the	nodes of the the loadtrack and their
2. Co-ordinates of the	nodes of the the loadtrack and their
2. Co-ordinates of the	nodes of the the loadtrack and their
2. Co-ordinates of the	nodes of the the loadtrack and their
2. Co-ordinates of the	Concept bootcopes

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	Sum Q	X1	X2
[kNm/m]	[kNm/m]	[m]	[m]
-83.093	-125.154	1.000	18.000

Positive maximum position : VOSB 150

Sum P	Sum Q	X1	X2
[kNm/m]	[kNm/m]	[m]	[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.

5.4.6. 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.

Generate Envelope Load Cases

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

Setup generated load c	ases		X
🔊 🗄 🗶 📸 💺 🖭 😅 🎒	🚔 🖬 🛛 All	- 7	
CA	Name	CA	^
	Use for calculation	0	
	Select unit loads	[Train]	
	Select load systems	[VOSB 150]	
8	Unit Load: Train		
	Name	Train	
	Load case		
	Group of load cases	Train	▼
8	Limited running length		
	Enable		
	Start [m]	0,000	
	Finish [m]	0,000	
8	Additional		
	Mult. factor results except deformatio	1	
	Mobile factor	1	
8	Selection of members		
	All members	8	
8	Components		
	Select components		
	Members		
	N		
	Vy		
	Vz		
	Mx		
	My		
	Mz		
	ux		
	uy		
	uz		
	fix		
	fiy	0	~
New Insert Edit Delet	e		Close

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.

utput of c	component	s on memb	ers				
ΓN	$\sqcap \lor$	∏ ∨z	∏ Mx	∏ My	🗖 Mz		Select All
L ax	UV.	☐ uz	∏ fix	☐ fiy	E fiz		Unselect All
utput of c	component	s on suppo	orts				
F Rx	E By	∏ Rz	∏ Mx	∏ My	☐ Mz		Select All
							Unselect All
utput of c	component	s on 2D ele	ements				
🔽 mx	∏ my	∏ mxy	I▼ vx I	Ty F	īnx ∏īny	Г фу	Select All
□ ux	⊑ uy	☐ uz	∏ fix [fiy Γ	fiz		Unselect All

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:

1 🕂 🛃 🖬 💽 😒 😒	🖻 🍯 🖬 🛛 Ali	- Y
.C1 - Self weight	Name	Train-VOSB 150-max mx
Frain-VOSB 150-max mx	Description	
Train-VOSB 150-min mx Train-VOSB 150-max vx	Action type	Variable
	LoadGroup	Train
Train-VOSB 150-min v×	Load type	Static
	Specification	Mobile envelope
	Master load case	None

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.

Name		Exploitatie van invloedslijnen -	2D .
Unit load	ds	Train	
Load sy	stems	[VOSB 150]	
E Limited	run		
Additio	nal		
E Load c	ase		
Generat	e		
Load gr	oup	Train	▼.
Setup re	port		
Selecter	d 2D members	[S1]	
Values		mx	
Values		mx	

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.

Numerical and graphical output - 1/1	×
1. Description of the influence line + The selected load systems for which the exploitation is done:	1
Influence line: 2D macro S1, Global position : x :10.00[m], y :2.50[m], z :0.00[m] Type : mx Considered load systems:	
Generate kod ceses : Tran. Mox. mx. S1,P1902.5. To Document	Prov. Gase

Through Generate Load Cases the load cases are generated.

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

Load cases			
🎜 lik 🏒 🖬 🗽 🗠 🗅	🗉 🎯 🗃 🖬 🛛 Ali	- 7	
LC1 - Self weight	Name	Train, Max, mx, S:1, P:10.0,2.5,0.0	
Train-VOSB 150-max mx	Description		
Train-VOSB 150-min m×	Action type	Variable	-
Train-VOSB 150-max vx	LoadGroup		·
Train-VOSB 150-min vx	Load type	Static	-
Train, Min, mx, S:1, P:1 Train, Max, mx, S:1, P:	Specification	Standard	-
IT all, Max, IIIX, S. 1, P	Duration	Short	-
	Master load case	None	-
New Insert Edit De	lete	[Close

After re-running the linear calculation, the results for these generated load cases can be viewed. Load case Max, mx:





5.5. 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.



Data Firstene X/Z Name Project M4 Part - Description Crane track. Author Pv/T Date [15.10.2005 Project Level : Model : Advanced One National Code :					Structure :	
Name Project M4 Part	Data					
Name Project M4 Part - Description Crane track Author PVT Date 15.10.2005 Project Level : Model : Advanced One National Code :					1	-
Part - Part - Description Crane track Author PVT Date 15.10.2005 Project Level : Model : Advanced One National Code : -	Name	le				100
Part - Description Crane track: Author Pv/T Date 15.10.2005 Project Level : Model : Advanced One National Code : -	Name	Project M4			The second se	
	Part	-		_		
Author P Author P Date 15.10.2005 Project Level: Model: Advanced One Advanced One National Code: Ctionality Loads Combinations Protection National Annexes Dymamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Prestressing Piesters Studuel model Prestressing Piesters Subcold Ownerview drawings Advanced Nonlinearity Stability Connection modeller Farme piloned connections Frame piloned connections Scafiolding Little All order Overview drawings	Description	,		_		
	Description	Crane track				
	Author	PVT			Aluminium	
	Date	15 10 2005		_		
Advanced One National Code: Image: Complexity of the construction of th	0.000	115.10.2005				
Advanced One National Code: Image: Complexity of the construction of th						
Advanced One National Code: EC-EN Combinations Protection National Annexes Committy Loads Combinations Protection National Annexes Committee Ses Dymamics Dymamics Dymamics Description Dymamics Description Dymamics Description Des						
Advanced One National Code: Image: Complexity of the construction of th	Project even		Model -			
National Code : EC - EN Combinations EC - EN Clionality Loads Combinations Dynamics Imited stress Imited stress Subsoil Imited stress Imited stress Subsoil Imited stress Imited stress Staboil Imited stress Imited stress Prestressing Imited connections Prestressing Imited stress Studuel model Imited connections Prestressing Imited stress Studuel model Imited stress Parameters Imited stress Mobile loads Imited stress Overview drewings Imited stress						
Cionality Loads Combinations Protection National Annexes Dynamics Dynamics Dynamics Dimetess Disboil Climatic loads Climatic loads Disboil Climatic loads Disbo	Comparison de la company		love	-		
	National Code					
tionality Loads Combinations Protection National Annexes Dynamics Image Steel Initial stress Image Fire resistance Subsoil Image Connection modeller Nonlinemity Image Frame nigit connections Stability Image Frame nigit connections Stability Image Image Prestressing Image Expert system Structural model Image Connection monodrawings Parameters Image Image Mobile loads Image Image	111	LOTEN				
Dynamics Initial stress Initial stress Initial stress Subsoil Initial stress Nonlinearity Initial stress Stability Initial stress Climatic loads Initial stress Prestressing Initial stress Stuctural model Initial stress Parameters Initial stress Mublic loads Initial stress Overview drewings Initial stress	and the set					
Initial stress Imitial stress Imitial stress Imitial stress Subsoil Connection modeller Nonlinearity Imitial connections Stability Frame rigit connections Climatic loads Imitial connections Prestressing Imitial connections Stuckural model Connection monodrawings Parameters Imitial cond Mobile loadi Imitial connections Overview drewings Imitial cond	nctionality Loads	Combinations Prot	tection National Ann	exes		
Subsoil Connection modeller Nonlinearity Frame ipid connections Stability Frame ipid connections Climatic loads Grid pinned connections Prestressing Babled diagonal connections Pipelines Connection modeller Structural model Connection modeller Parameters Connection modeller Mable loads Connection modeller Overview drewings Arcelor/Mitel		Combinations Prot				
Nonlinearity Image: Connections Stability Image: Connections Climatic loads Image: Connections Prestressing Image: Connections Prestressing Image: Connections Stuctural model Image: Connections Parameters Image: Connections Mobile loads Image: Connections Overview drewings Image: Connection monodrawings	Dynamics	Combinations Pro	-	⊟ Ste		
Stability Image: Connections Climatic loads Image: Connections Prestressing Image: Connections Pipelines Image: Connections Structural model Image: Connections Parameters Image: Connection monodrawings Mobile cloads Image: Connection monodrawings Overview drewings Image: Connection monodrawings	Dynamics Initial stress	Combinations Prot		B Ste	e resistance	
Climatic loads Grid pinned connections Prestressing Balted diagonal connections Structural model Connection monodrawings Parameters Connection monodrawings Mobile loads Connection diagonal Overview drewings Arcelor/Mitel	Dynamics Initial stress Subsoil	Combinations Prot		B Ste Fin Co	e resistance nnection modeller	15
Prestressing Image: Connection of the system Pipelines Image: Connection monodrawings Structural model Image: Connection monodrawings Parameters Image: Connection monodrawings Mobile loads Image: Connection monodrawings Overview drewings Image: Connection monodrawings	Dynamics Initial stress Subsoil Nonlinearity	Combinations Prot		B Stu Fin Co Fre	e resistance nnection modeller ame rigid connection	
Pipelines Expert system Structurel model Connection monotrewings Parameters Scatfolding Mobile loads B Overview drewings Arcelor/Mitel	Dynamics Initial stress Subsoil Nonlinearity Stability	Combinations Pro		B Sta Fire Free Free	e resistance nnection modeller ame rigid connection ame pinned connect	ions
Structural model Connection monodrawings Parameters Scatioling Mobile loads LTB 2nd Order Overview drawings Arcelof/Mital	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads	Combinations Pro		B Sta Fin Co Fre Gri	e resistance nnection modeller ame rigid connection ame pinned connect d pinned connectior	ions 1s
Mobile loads Image: Control of the second seco	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing			B Sta Fin Co Fre Gri Bo	e resistance nnection modeller ame rigid connection ame pinned connection id pinned connection Ited diagonal conne	ions 1s
Overview drawings	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines			Stu Fin Co Fre Gri Bo Ex	e resistance nnection modeller ame rigid connection ame pinned connection d pinned connection (ted diagonal conne pert system	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode			Stu Fin Co Fre Gri Bo Ex Co	e resistance nnection modeller ame rigid connection ame pinned connection d pinned connection lted diagonal conne pert system nnection monodrawi	ions 1s ctions
LTA-load cases	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters			Fin Co Fre Gri Bo Ex Co So	e resistance nnection modeller ime rigid connection ime pinned connection d pinned connection thed diagonal conne pert system nnection monodrawi affolding	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile Loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions
	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines Structural mode Parameters Mobile loads Overview draw	el		Fin Co Fre Gri Bo Ex Co So LT	e resistance nnection modeller me rigid connection ame pinned connection lited diagonal conne pert system nnection monodrawi attolding B 2nd Order	ions 1s ctions

5.5.1. The construction

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



Subsequently the haunch beams on which the rail support, can be entered through ⁴⁵ Beam . The beams have a length **1m**, type **IPE 180** and move across ³/₄ of the length of the column.

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

	e grid 🔽 Dot grid Only snapped points Midpoints Endpoints / Nodes Intersections
	Midpoints Endpoints / Nodes Intersections
	Endpoints / Nodes Intersections
	Intersections
	Orthogonal points
d) [Orthogonal points
2/4 e)	Tangential points
1/4 3/4 0	Arc/Circle centre
	Points on line-curve - length
	Length[m]: 1.000
	Repeat 3 😤
	Start point. Begin 💌
h) 🔽	Points on line-curve - N-ths 4
0 [Points on line-curve - % of length 50.00 %
D F	Surface edges
к) Г	General solids 2000
	OK Cancel


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:

Tutorial – load generator

Numbe	er of copies 🚦	÷	Connect selected nodes with new beams	~
✓ Inse	ert the very last copy	/	Copy additional data	~
Distanc	e vector		How to define the distance ?	
Define	distance by cursor	Г	between two copies	
×	0,000	m	C from original to the last co	ру
у	5,000	- m	How to define the rotation ?	
7	0.000		between two copies	
- Rotatio	Jane		 from original to the last co Botation around 	ру
rx	0,00	deg	(current UCS	
000	1.42.4		C distance vector	

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 - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid supports to the column bases and by executing the - The geometry input is ended by entering the rigid support of the column bases and by executing the - The geometry input is ended by entering the rigid support of the column bases and by executing the - The column bases are column bases and by executing the - The column bases are column bare column bases are column bases are column bare column bare colu



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

5.5.2. 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:

-	operties	4 ×
T	rip train (mobile load) (1)	- Va V/ /
	Name	TR1
	Use for calculation	
	Used nodes	4
Ξ	Track nodes	
	Node	K8 [B36]
	Node	K14 [B36]
	Node	K23 [B37]
	Node	K32 [B38]
	tions	
U	pdate track definition	>>>
Т	able edit geometry	>>>

As a Name for the track TR1 is entered.

After defining the track, the Unit loads can be entered through the menu 2 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.

🛯 🔆 🗶 🖬 🗽 🕰 🖉	6	🗃 🖬 🛛 All	• 9
Midpoint		Name	Midpoint
Left		Track assignment	TRI
Right		Sections	Use sections from results
		Step for 2D element [m]	1,000
		Generate section under Load system	0
		Add new Impulse	
		Delete impulse	<u>.</u>
	8	Impulse 1	-
	(2.0)	Туре	Concentrated
		Value	-0.5
		Position [m]	0.000
		ey[m]	0,000
		ez [m]	0,000
		System	Local
		Direction	Z
	a	Impulse 2	-
	-	Туре	Concentrated
		Value	-0,5
			8,000
		Position [m]	0,000
		ey [m]	0.000
		ez [m]	
		System	Local
		Direction	Z
		11 (-0.5)	12 (-0.5)
		II (-0.5)	I2 (-0.5)
		No.	

Unit Mobile Loads			2
l 🤮 🗶 📸 🗽 🕰 🖾	8	🗃 🖬 🛛 All	- 7
Midpoint		Name	Right
Right		Track assignment	TR1
Left		Sections	Use sections from results
		Step for 2D element [m]	1,000
		Generate section under Load system	8
		Add new Impulse	
		Delete impulse	
	Θ	Impulse 1	
		Туре	Concentrated
		Value	-0,8
		Position [m]	0,000
		ey [m]	0,000
		ez [m]	0,000
		System	Local
		Direction	Z
	Θ	Impulse 2	
		Туре	Concentrated
		Value	-0,2
		Position [m]	8,000
		ey[m]	0,000
		ez [m]	0,000
		System	Local
		Direction	Z
		11 (-0.8)	I2 (-0.2)
		I	Ĩ

Midpoint Right Left		Name Track as Sections	signment	Left	
			signment	TD1	
Left		Sections		TR1	
		0660000	-	Use sections from results	
		Step for 2	2D element [m]	1,000	
			e section under Load system		
		Add new	Impulse		
		Delete in	npulse		
	Θ	Impulse	1		
		Туре		Concentrated	
		Value		-0,2	
		Position	[m]	0,000	
		ey[m]		0,000	
		ez [m]		0,000	
		System		Local	
		Direction		Z	
	8	Impulse			
		Туре		Concentrated	
		Value		-0,8	
		Position [m]		8,000	
		ey[m]		0,000	
		ez [m]		0,000	
		System		Local	
		Direction	1	Z	
			11 (-0.2)	12 (-0.8)	

5.5.3. Input load system

The input of the load system for the crane track happens through the option - the 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**.

Load System		
Simply load system Advanced load s Name Crane Tract Neglect point load whit opposite Distributed load	 0007-	00.00-
0.00 kN/m	-0.400 0.800	
	OK	Cancel Apply

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

5.5.4. **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.

E	xploitatie van invloedslijn	
	Name	Exploitatie van invloedslijne
	Unit loads	All
	Load systems	[Crane Track]
Ð	Limited run	
Ð	Additional	
Ξ	Load case	
	Generate	
	Load group	Mobile
	Setup report	
	Selected members	[B33]
	Values	My
	tions	
S	ingle Check	>>>
P	review	>>>

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	Y	Z	
	[m]	[m]	[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 2.1m 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:



5.5.5. Generation Enveloping Load Cases

For the component My the enveloping load cases are generated through the option - J++ Setup generated load cases

1 🗄 🗶 🖬 🗽 🗠 🗠	A		• 7	
CA		ame	CA	
un .		se for calculation		
		elect unit loads	[Midpoint] [Right] [Left]	
		elect load systems	[Crane Track]	
		Jnit Load: Midpoint		
		lame	Midpoint	
	L	.oad case		
	0	aroup of load cases	Mobile	·
	ΒU	Jnit Load: Right		
	N	lame	Right	
	L	.oad case		
	0	Group of load cases	Mobile	·
	Θι	Jnit Load: Left		
	N	lame	Left	
	L	.oad case		
		Group of load cases	Mobile	•
		imited running length		
		Additional		
		Selection of members		
		All members		
		Selection		
		Components		
		select components		
		Members	_	
		N		
		Vy	0	
		Vz	0	
		Mx	0	
		My		
		Mz ux		

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.

E N	Γvy	√z	Гм	× 🔽	My F	Mz		Select All
∏ ux	∏ uy	∏ uz	∏ fo	Г	fiy [fiz		Unselect All
utput of	componen	ts on supp	orts					
∏ Rx	∏ Ry	∏ Rz	Гм	× Г	My F	Mz		Select All
								Unselect All
utput of	componen	ts on 2D el	ements					
₩ mx	<mark>™</mark> my	₩ mxy	VX VX	$\boxtimes \lor$	₩ mx	M ny	₩ œv	Select All
ux 🕅	₩ uy	₩ U2	l▼ fix	₩ fy	🗹 fiz			Unselect All

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:

LC1 - Eigengewicht	The second se	
	Name	Left-Crane Track-Max My
Midpoint-Crane Track-Min My	Description	
Midpoint-Crane Track-Max My	Action type	Verieble
Left-Crane Track-Min My	LoadGroup	Mobile
Left-Crane Track-Max My	Load type	Static
Right-Crane Track-Min My Right-Crane Track-Max My	Specification	Mobile envelope
Right-chane mack-max my	Master load case	None

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

Selection : B33	on, Extreme : Global eft-Crane Track-Max		cipal
Member	Case	dx [m]	N

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

Chapter 6: 2D wind- and snow generators (32-bit)

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

Scia	Dynamics	100	^	Steel	
Scia	Initial stress	10		Fire resistance	10
-geneer	Subsoil			Connection modeller	
	Nonlinearity			Frame rigid connections	
	Stability			Frame pinned connections	
	Climatic loads	V		Grid pinned connections	
	Prestressing			Bolted diagonal connections	
	Pipelines			Expert system	
	Structural model			Connection monodrawings	
	BIM properties			Scaffolding	
	Parameters			LTB 2nd Order	
	Mobile loads			ArcelorMittal	
	Automated GA drawings				
	LTA - load cases				
	External application checks				
	Property modifiers				
	Bridge design				
	Customized design form				
	Old style document	1	~		

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

		Project data	×
Basic data F	unctionality Loads Protection		
Scia	Acceleration of gravity	9,810 m/s^2	
	Wind Load		ר
	According to code	EC 1 / 26,200m/s / 0	
	Snow Load		
	According to code	EC 1 / Sk=1,00kN/m ² Ce=1,0 Ct=1,0	
		OK Car	icel

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

6.1. Wind generator

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

Load cases:

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

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

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

Real value expected:	×
Frame distance:	
5	
Format: a	
Limits OK 0.001 - 1000000 m Cancel	

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]:

Zone	Α		в		с		D		E	
h/d	C _{pe,10}	Cpe,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	

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

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:



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

Pitch	Zone	for wind	directio	n θ = 0°										
Angle a	F		G H I		H I		I J							
Angle a	Cpa,10	Cpo, 1	Cps, 10	Cpe,1	Cpe,10	Cpa,1	Cpe,10	Cpo.1	Cps,10	Cpa,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				
50											+0,2		+0,2	
-5°	-2,3	-2,5	-1,2	-2,0	-0,8 -1,2 -0,6		-0,6							
50	-1,7	-2,5	-1,2	-2,0					+0,2					
5°	+0,0		+0,0		+0,0		-0,6		-0,6					
450	-0,9	-2,0	-0,8	-1,5	-0,3		-0,4		-1,0	-1,5				
15°	+0,2		+0,2		+0,2	INNR.	+0,0		+0,0	+0,0				
200	-0,5	-1,5	-0,5	-1,5	-0,2		0,4		-0,5					
30°	+0,7		+0,7	F	+0,4		+0,0		+0,0					
450	-0,0		-0,0)`	-0,0		-0,2		-0,3					
45°	+0,7	1	+0,7		+0,6		+00	6. 16	+0,0					
60 °	+0,7	1	+0,7		+0,7			11	-0,3					
75°	+0,8	1	+0,8		+0,8			11	-0,3					

Table 7.4a — External pressure coefficients for duopitch roofs

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

Tools	Modify	Tree	Plugins	Setu
A	ctivity			•
s	elections			•
U	CS			•
c	ursor snap	settin	9	
D	ot grid an	d tracki	ing setting	
Li	ayers			
U	ser define	d select	tions	
c	leaner			
- C	oordinate	s info		-
X	ML IO Doc	ument	[5
E	dit profile	library		

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:



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			×	•		Manager for National	annexes		
Basic data Fu	nctionality Loa	ds Protectio	'n					A 2	🖲 🖋 💽 🗠 🗠	🎒 🚅 🔒 Al		٠	
Scia	Data Name:	•			Material Concrete Steel			Austri Belgia British Czech	ard EN an ÖNORM-EN NA n NBN-EN NA BS-EN NA CSN-EN NA NEN-EN NA				^
	Part: Description:	•			Material Timber Masonry	S 235	•	Name Natio			Standard EN Standard EN		^
	Author:	-DP			Other Aluminium			EN EN	1990: Basis of structural 1990 (Basis of structural design 1991: Actions of structure 1991-1-3 (General actions - Sn)) 85			
	Date:	21. 05. 2014	1		Code			EN E EN	1991-1-4 (General actions - Wi 1992: Design of concrete 1992-1-1 (General rules and rul	nd actions) e structures			
	Structure: Frame XZ		Solver Model Nexis		National Code:	(EN EN	1992-1-2 (General rules -Struct 1992-2 (Concrete bridges - Des 1168 (Precast concrete produc	tign and detailing rules) ts - Hollow core slab)			н. 10.
	Project Level: Advanced	٠	Model: One		National annex:	EN (EN EN	1993: Design of steel stm 1993-1-1 (General rules and rul 1993-1-2 (General rules - Struct 1993-1-3 (General rules - Suppl	es for buildings)			
						ок	Cancel	EN		nents) Le steel and concrete struct		 Co	ose

Standard EN	Name	Randard EN	
B-Wind	B Wind		
Wind pressure according to D	Wind pressure according to EC1		
	E Internal pressure for 2D wind		
	Internal pressure for 2D wind	no internal pressure	
	Position dominant face for 2D wind	liont	
	Openings dominant face for 2D wind	two times.	
	External pressure for 3D wind		
	Edemal pressure for 30 wind	Use overall coefficients Cpc.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]	10,000	
	Reference level of terrain [m]	0.000	
	E c dr - drectional factor		
	Value [-]	1.00	
	B 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	
	E 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_1 - turbulence factor [-]	1,00	
		Load default NA parameters OK Cancel	

6.2. Snow generator

After generating the wind load in both directions, also the snow loads will be generated in this example.

For the snow loads, three load cases have to be created.

Extra load cases:

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

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

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



After clicking on OK in this window, the snow load 1 will be generated on the roof elements.



The same can be done for Snow load 2 and 3.

The Load Coefficients are calculated according to the code, EN 1991-1-3.

Table 5.2 [2] is used for the load coefficients.

Table 5.2: Snow load shape coefficients						
Angle of pitch of roof α	$0^\circ \le \alpha \le 30^\circ$	$30^\circ < \alpha < 60^\circ$	$\alpha \ge 60^{\circ}$			
μ_1	0,8	0,8(60 - α)/30	0,0			
μ_2	0,8 + 0,8 a/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.

Snow weight		
C default		
GI CH C	ш	

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.



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	Persistent/transient design situation	Persistent/transient design situation	Persistent/transient design situation
[1] undrifted $\mu C_e C_1 a_k$	[1] undrifted $\mu_i C_e C_1 a_k$	[1] undrifted $\mu_i C_e C_1 s_k$	[1] undrifted $\mu_k C_p C_1 a_k$
[2] drifted µ _i C _e C _i s _k	[2] drifted $\mu_i C_e C_i a_k$	[2] drifted µ _i C _e C _i s _k (except for roof shapes in AnnexB)	[2] drifted µ _i C _s C _i s _k (except fo roof shapes in AnnexB)
Þ		Accidental design situation (where snow is the accidental action)	Accidental design situation (where snow is the accidenta action)
	 [3] undrifted µ C₀C₁ C_{ust} s_k [4] drifted µ C₀C₁ C_{ust} s_k 	[3] drifted in sk (for roof shapes in AnnexB)	[3] undrifted # C _e C _t C _{ed} a _k
	[4] guilled tri Celci Cell ak		[4] drifted , A sk (for roof shapes in AnnexB)

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

6.3. Wind & Snow generator

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

This generator automatically creates:

- 2 new load groups ٠
- Wind (variable, exclusive) 0
- Snow (variable, exclusive) 0
- Without over- and underpressure taken into account •
- 3 new load cases 0
- WND – L – Wind from the left
- WND R Wind from the right
- SN - Snow loads
- With over- and underpressure taken into account •
- o 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**.

	- U		
Standard EN	Nore	Bardad EN	
Wind pressure according to EC1	Wasd		
	A much businesses recomming an error		
	Extend pressure for 20 wind		
	Entertail pressure for 20 sand	ré-deninent face	
	Position dominant face for 2D wind	downart face	
	Openings dominant face for 20 wind	no dominant face	
	Edenal pressure for 3D wind		
	Edense pressure for 3D yend	Use sveral coefficients Cpe. 10	
	Reference height (r_e)		
	Type of the structure	Vetical walk or rectangular buildings (EC1114, 7.2.2)	
	b width of the structure [n]	100,000	
	Fallemence level of terrein (m)	0.000	
	G e_dr - drectional factor		
	Value 3-1	1.00	
	O c_season - season factor		
	Value [-]	1.00	
	Cilic_o - orography factor		
	Volue [-]	1.00	
	V_b.D -basic wind velocity [m/k]	26,200	
	m - siz denaty (kg/m ¹ 2)	1.3	
	G Probability		
	1/p - We period of the building (year)	90.00	
	c_prib_pribability factor []	1.00	
	K-shapefactor[-]	0.29	
	n-exponent []	0.50	
	Terrain		
	Teman-category	0	
	R - tensin factor (H	0.46	
	a_0 - roughness langh (m)	6.003	
	r_en-minus teight [n]	1,000	
	k_1-tarbulence factor []	1.00	