



\ SCIA ENGINEER 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 reserved.

# Table of Contents

Table of Contents	
Introduction	5
Load panels	6
General properties	7
Panel type	7
Load transfer direction	7
LCS type	7
Swap orientation	7
LCS angle	7
Selection of entities	7
Load transfer method	8
Generating loads	9
Panel types	10
Load to panel nodes	10
Load to panel edges	
Load to panel edges and beams	14
Panel with parallel beams	
Free loads	1/
Properties of a free load	17
Direction	
Туре	
Distribution	
Validity	
Select	21
System	
Example: Cylindrical tank	22
Result: surface loads	26
Soil and Water pressure based on borehole	28
3D wind generator	
Traffic loads	
Example model	35
Project data	
Construction	35
Input traffic loads	36
Traffic loads generator	
Principle Influence lines	
Inductor mes	40
Influence lines	40 12
Load system	
The construction	
Input track and unit load	
Input load systems	45
Exploitation of load systems	

Generation Load cases – Enveloping Load cases	52
Train loads	57
The construction	57
Input track and unit load	59
Influence lines	62
Input load systems	63
Exploitation of the load systems	65
Generate load cases – Envelope load cases	67
Crane track	71
The construction	72
Input track and unit load	75
Input load system	79
Exploitation of the load system	80
Generation Enveloping Load Cases	83
2D wind- and snow generators (32-bit)	85
Wind generator	86
Snow generator	92
Wind & Snow generator	94

# 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.

# 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  $-10 \text{ kN/m}^2$  is used.



# **General properties**

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

## 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.

### 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

### 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**.

#### 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.

### 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.

### 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'

# Load transfer method

#### <u>Standard</u>:

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



#### 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
Generat	e loads
Table ed	dit 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'.

					1 <u>1</u>					
					115					
					Ħ					
					47					
					17					
		E	ffective width of plate							
			Structure nodes	$\bigcirc$						
			Member parameters		REOPTIONS					
			Local axes	$\bigcirc$						
			System length labels	$\bigcirc$						
			Nonlinearity labels	$\mathbf{O}$						
			Labels of local axes	$\tilde{\mathbf{O}}$		1				
			Structural shape labels	$\overline{\mathbf{O}}$						
		_	A second state of the second s							
		3	View settings for all entities	2	CHOOSE B	=				
Vie	w parameters setting - Loads/masse	5								
									Lock po	sition
						-		2000 a	View	•
٩	🔟 Structure 🛛 🔛 Labels 🛛 📥 M	lodel 🛃 Loads	s/masses 👕 Composite	Modellin	g/Drawing	🍠 Attrib	utes	MISC.		
4	TStructure 🔮 Labels 👗 M Check / Uncheck all	lodel 🛃 Loads	s/masses T Composite	Modellin Modellin	g/Drawing	🎔 Attrib	utes	Misc.		
	Structure      Labels      A     M     Check / Uncheck all     Service	lodel 🛃 Loads	s/masses T Composite	Modellin Modellin	g/Drawing	Se Attrib	utes	Misc.		
•	Structure E Labels A M Check / Uncheck all Service Display on opening the service	lodel Loads	s/masses T Composite	Modellin Modellin	g/Drawing	Attrib	utes	Misc.		
	Structure Labels M Check / Uncheck all Service Display on opening the service Display loads	lodel Loads	s/masses T Composite	Modellin Modellin	g/Drawing	S Attrib	utes	Misc.		
	Structure Market Abels Market / Uncheck all Service Display on opening the service Display loads Display		s/masses T Composite	Modellin	g/Drawing	Y Attrib	utes	Misc.		
	Structure ELabels A M Check / Uncheck all Service Display on opening the service Display loads Display Style	lodel Loads	s/masses T Composite	Modellin	g/Drawing	Attrib	utes	Misc.		
	Structure ELabels A M Check / Uncheck all Service Display on opening the service Display loads Display Style Load case	Colour by action t	s/masses T Composite	Modellin	g/Drawing	Attrib	outes	Misc.		
	Structure ELAbels A M Check / Uncheck all Service Display on opening the service Display loads Display Style Load case Display eccentricity	Loads	s/masses T Composite	Modellin	g/Drawing	Y Attrib	utes	Misc.		
	Structure ELAbels A M Check / Uncheck all Service Display on opening the service Display loads Display Style Load case Display eccentricity Generators	Colour by action t Colour by action t C2 - Roof Generated	s/masses T Composite	Modellin	g/Drawing	Y Attrib	utes	Misc.		•
	Structure ELAbels A M Check / Uncheck all Service Display on opening the service Display Style Load case Display eccentricity Generators Point forces	Colour by action t Colour by action t LC2 - Roof Generated Original	s/masses T Composite	Modellin	g/Drawing	Y Attrib	utes	Misc.		•
	Structure ELAbels A M Check / Uncheck all Service Display on opening the service Display Style Load case Display eccentricity Generators Point forces In node	Colour by action t Colour by action t CC2 - Roof Generated Original Contracted Original Contracted	s/masses T Composite	Modellin	g/Drawing	Y Attrib	utes	Misc.		• • •
	Structure El Labels A M Check / Uncheck all Service Display on opening the service Display loads Display Style Load case Display eccentricity Generators Point forces In node Surface loads	Colour by action t Colour by action t C2 - Roof Cignal Crimated Original + Generated	s/masses T Composite	Modellin	g/Drawing	Y Attrib	utes	Misc.		• • •
	Structure El Labels A M Check / Uncheck all Service Display on opening the service Display Style Load case Display eccentricity Generators Point forces In node Surface loads On 2D member In force Interview	Colour by action t Colour by action t CC - Roof Generated Original Congrate Original Congrate Generated Original Congrate Congra	s/masses T Composite	Modellin	g/Drawing	Y Attrib	utes	Misc.		- - -
	Structure El Labels A M Check / Uncheck all Service Display on opening the service Display loads Display Style Load case Display eccentricity Generators Point forces In node Surface loads On 2D member Labels of loads	Colour by action t Colour by action t LC2 - Roof Generated Original Conginal Con	s/masses T Composite	Modellin	g/Drawing	Attrib	utes	Misc.		- - - -
	Structure Structure Structure Structure Service Uisplay on opening the service Uisplay on opening the service Uisplay and the service Uisplay accentricity Generators Point forces In node Surface loads On 2D member Labels of loads Display label Uispen	Odel     Loads       Image: Colour by action t     Image: Colour by action t       LC2 - Roof     Image: Colour by action t       Generated     Original       Original     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by actiont     Image: Colour by action t	s/masses T Composite	Modellin	g/Drawing	Attrib	utes	Misc.		• • •
	Structure  Structure  Structure  Service  Display on opening the service  Display loads  Display  Style  Load case  Display eccentricity  Generators  Point forces In node  Surface loads  On 2D member  Labels of loads  Display label Name  Name	Odel     Loads       Image: Colour by action t     Image: Colour by action t       LC2 - Roof     Image: Colour by action t       Generated     Original       Original     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by action t     Image: Colour by action t       Image: Colour by actiont     Image: Colour by action t	s/masses T Composite	Modellin	g/Drawing	Attrib	utes	Misc.		• •
	Structure  Structure  Service  Display on opening the service  Display loads  Display  Style  Load case  Display eccentricity  Generators  Point forces In node  Surface loads  On 2D member  Labels of loads  Display label Name Value  Yatu safute	Colour by action to Colour	s/masses T Composite  type ated	Modellin	g/Drawing	Attrib		Misc.		• •
	Structure  Structure  Service  Display on opening the service  Display loads  Display loads  Display eccentricity  Generators  Point forces In node  Surface loads  On 2D member  Labels of loads  Display label Name Value  Yalue Tot. value	Image: constraint of the second	s/masses T Composite  type ated	Modellin	g/Drawing	Attrib		Misc.		

# Panel types

# 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	1025.64				
D	Ø	*	(F)		۲	Ċ	ß
	5	Repeat 'l	Rectangu	ılar grid'		Ente	er
	\$	Undo				Ctrl+	۰z
		Redo				Ctrl+	Y
		Modify					×
	ß	Deletion	settings				
	*	Copy/pa	ste prop	erties	Ct	rl+Shift+	F
		Add data	а				۲
		Metadat	а				۲
		Polyline	edit				۶.
		Curves e	dit				۶.
		Solids					F
	-		-			-	

#### 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:  $-10 \text{ kN/m}^2 \text{ x} (10 \text{ m} \text{ sm}) = -500 \text{kN} \text{ and } -500 \text{kN} / 4 \text{ nodes} = -125 \text{ kN} / \text{ node}.$ 



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.

#### 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.

### 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 transfer 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.



#### 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  $-10 \text{ kN/m}^2$  is added.



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

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

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



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



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

### 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.

## 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.





#### Direction

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

#### Type

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

#### 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.

# 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.

#### 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	
Table edit geometry	
O Update 2D members selection	

#### 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.

# 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.

View parameters setting			
Check / Uncheck group		Lo	ck position
I Structure I Labels ▲ Mode	l 🛃 Loads/masses 👕 Composite 🔛 Modelling/Drawing	🚱 Attributes 🛛 🌌 Misc. 🔍 View	v Þ
Check / Uncheck all			
Rendering	transparent		- ^
Highlight supporting edges/nodes			
Load distribution symbol			
Display linked members			
Structure nodes			
Display			
Mark style	Dot		•
Member parameters			
System lengths			
Member nonlinearities	V		
FEM type			
Joists			
E Local axes			
Nodes			
Members 1D			
Members 2D			
			~
Show names in tab		🕗 🕞 ОК Арр	oly Cancel

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 V	🔁 🛃
ŧ¥γ	XY workplane		
tΥZ	YZ workplane		
ŧ <b>X</b> Ζ	XZ workplane		E
12	Move		F9
L.	From 3 points		
12	from Local Member		F10
1	Poset to GCS position	6	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 Z-axis of each wall.
- Direction is set to Z.
- Validity is set to 'all'.
- Select is set to 'Auto'.

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

Surface force free	×	<
-P Name	• FF2 ^	
Direction	Z *	
Тур	Force v	
Distribution	Dir Y v	
q1 [kN/m^2	0,00	
P	¥	
q2 [kN/m^2	-25,00	
P	Y	
Validit	All	
Selec	t Auto 🗸	
A Geometry		
System	Member LCS v	
Location	Length	
	V	
	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.



# 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.

# 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.

	×
Mesh setup	
Average number of 1D mesh elements (	
Average size of 1D mesh element on cu 0,200	
Average size of 2D mesh element [m] 0.1	
Connect members/nodes 🔽	
Setup for connection of structural entit	
Advanced mesh settings	
Solver setup	
Specify load cases for linear calculation	
Advanced solver settings	
	2
	-
Average size of 2D mesh element	
	<ul> <li>Mesh setup         <ul> <li>Average number of 1D mesh elements 1</li> <li>Average size of 1D mesh element on cu 0,200</li> <li>Average size of 2D mesh element [m] 0.1</li> <li>Connect members/nodes</li> <li>Setup for connection of structural entit</li> <li>Advanced mesh settings</li> </ul> </li> <li>Solver setup         <ul> <li>Specify load cases for linear calculatior</li> <li>Advanced solver settings</li> </ul> </li> <li>Advanced solver settings</li> </ul>

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.

# 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	Name Direction	FF2 Z		*
	Туре	Soil pressure		٣
	Distribution	Uniform		
the second second	Coeff1 [-]	1,000		
	Borehole profile	7=0		×
STUR WIN	Select	Auto		
	Geometry			
2	System	Member LCS		٧
	Location	Length		
* YUUUUUU				
			ок с	ancel

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'.





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.

Line force on beam			×
RZ	Name	LF22	
	Direction	Y	Y
	Туре	Soil pressure	v
	Angle [deg]		
-P1 (j)	Distribution	Uniform	Y
All States	Acting width [m]	1,000	
ey Tez	Coeff1	1,000	
	Coeff2	1,000	
x1 x2	Load above joint	no	
	Borehole profile	BH1	v
	4 Geometry		
	System	GCS	
	Location	Projection	
	Extent	full	
	Coord. definition	Rela	v
	Position x1	0,000	
	Position x2	1,000	
	Origin	From start	¥
	<ul> <li>Eccentricity</li> </ul>		
	Eccentricity ey [m]	0,000	
	Eccentricity ez [m]	0,000	
		OK Cancel	
			d

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

	 	0,00
}		Cool           5,00           5,00           10,00           12,59           12,60           15,19           17,79           7,78           77,78           20,38           22,57           25,57           25,57           28,16           30,75           33,36
		33,36 35,95 38,85 38,85 41,14 41,14 43,73 44,73 44,73 44,73 44,73 44,73 44,93 44,93 44,93 44,93 44,93 44,93 44,93 44,93 45,75 57,72 57,72 57,72 57,72

# 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**.

sic data Fu	unctionality Actions Unit Set Protection			
	GENERAL		DETAILED	
	Property modifiers	^	4 Subsoil	^
	Model modifiers		Soil interaction	
	Parametric input		Pad foundation check	
	Climatic loads 🗸		▲ Steel	
	Mobile loads		Fire resistance checks	
	Dynamics		Steel connections	
	Stability		Scaffolding	
	Nonlinearity		7DoF 2nd order analysis for LTB	
	Structural model		Girders with sinusoidal webs	
	IFC properties			
	Prestressing			
	Bridge design			
	Excel checks			
		~		~

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 El Setup manager ×
	Standard EN Name Standard EN
Pasis data Europianality Actions Unit Set Destaction	i Wind
basic data runcuonanty Actions Unit Set Protection	Wind pressure according to EC1  4 Wind pressure according to EC1
	<ul> <li>Internal pressure for 2D wind</li> </ul>
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
	<ul> <li>External pressure for 3D wind</li> </ul>
WIND LOAD	External pressure for 3D wind: Use overall coefficients Cpe,10 *
	Correlation between zones D and E
According to code    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
	<ul> <li>c_dir - directional factor</li> </ul>
According to code V EC 1 / Sk=1,00kN/m <sup>2</sup> Ce=1,0 Ct=1,0	Value [-] 1,00
	4 c_season - season factor
	Value [-] 1,00
	4 c_o - orography factor
SEISMIC COMBINATIONS	Value [-] 1,00
	V_b,0 - basic wind velocity [m/s] 26,290
Model factor: 1,30 Factor for concomitant components 0,30	ro - air density (kg/m*s] 1,23
	Probability     Probability     The Encoded of the building basel 50.89
	L/p - me period of the building (year) - subw
CODE COMBINATIONS	C_prob - probability include 11 Aprel
CODE COMBINATIONS	n - snape duting [1] (we we have a state of the state of
Automatic	4 Terrain
Automatic System	terrain category 0 ×
	Kr - terrain factor [-] 0.16
	z_0 - roughness length [m] 0,003
1	z_min-minimal height [m] 1,000
	k_l - turbulence factor [-] 1,00
OK	Cancel Load default N4 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 LP1	
Layer Laag1 VI	
Panel type To panel edges and beams	
Shape Flat	
Load transfer direction X (LCS panel)	
Max.angle for transfer [deg] 5,00	
LCS type Standard V	
Swap orientation	
LCS angle[deg] 0.00	
30 Wind 💽	/
Supporting members validity range (with re All V	-
Max. eccentricity of members [m] 0,200	
Load transfer method Accurate (FEM), fixed link with beams	
Selection of entities All	
TLLUSTRATION GROUP	
WIND DATA (1)	
Name WD5	
Panel LP5	
Type Wall	
Actions >>>	
Update day Joan Sketching Swap outer surface	
Table edit geometry Openings	

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



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

	3D Wind Generator	×	A	dd Wind Load C	ases					×
All workstations	Load cases			Direction	+ CPE, + CP	I + CPE, - CPI	- CPE, + CPI	- CPE, - CPI	+ CPI	- CPI
				1 0					0,20	-0,30
				2 90					0,20	-0,30
All tags				3 180			2		0,20	-0,30
🚐 Longitudinal strain on 1D				4 270					0,20	-0,30
Flexural strain on 1D										
		_	(	Calculation meth	od: Stan	dard	٧			
2D wind generator				Additional loa	d cases for du	opitch roofs				
SD wind generator	Add Load Case	s		Direction of r	idge/trough of	roofs B	oth (X and Y 💙			
▼ SPECIAL LOADS	Run generator Close			Include torsio	nal load case			ОК	) <b>C</b> a	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:

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

Zone	Α		в		C D E		C D E		D		
h/d	C <sub>pe,10</sub>	C <sub>pe,1</sub>	<b>C</b> pe,10	C <sub>pe,1</sub>	<b>C</b> pe,10	C <sub>pe,1</sub>	C <sub>pe,10</sub>	C <sub>pe,1</sub>	C <sub>pe,10</sub>	C <sub>pe,1</sub>	
5	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,7		
1	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,5		
≤ 0,25	-1,2	-1,4	-0,8	-1,1	-0,5		+0,7	+1,0	-0,3		

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

Bitch	Zone f	for wind o	direction	η <i>θ</i> = 0°							
	F		G		н		1		J		
Angle a	Cpe,10	Cpe,1	Cpe, 10	Cpe,1	Cpe,10	Cpe,1	Cpe,10	Cpe,1	Cps,10	Cpo,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	
<b>E</b> 0	-2,3	3 -2,5				10	+0,2		+0,2		
-0			-1,2	-2,0	-0,8	-1,2	-0,6		-0,6		
<b>E</b> 9	-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	
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	+0,7		+0,7		+0,6		+0,0		+0,0	
60°	+0,7		+0,7		+0,7		-0,2		-0,3		
75°	+0,8		+0,8		+0,8		-0,2	1	-0,3		

Table 7.4a — External pressure coefficients for duopitch roofs

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
E -8 C B B C Name LG1 LG2 LG2 LG2 LG2 LG2 LG2 LG2 LG2 LG2 LG3 LG3 LG3 LG3 LG3 LG3 LG3 LG3 LG3 LG3	All Y Y LG2 Exclusive V Variable V Building Wind V	Load cases 30Wind 0, + CPE, + CPI 30Wind 2, - CPE, - CPI 30Wind 3, C-CPE, - CPI 30Wind 4, C-CPE, - CPI 30Wind 5, C-CPE, - CPI 30Wind 5, C-CPE, - CPI 30Wind 5, C-CPE, - CPI 30Wind 10, - CPE, - CPI 30Wind 2, - CPE, - CPI 30Wind 2, - CPE, - CPI 30Wind 2, - CPE, - CPI 30Wind 10, - CPE, -
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 position	
٩	Structure Labels Andel	Loads/masses TComposite Modelling/Drawin	ng 🗭 Attributes 🖉 Misc. 🔍 View	P
	Check / Uncheck all			
	Sensice			
	Dirplay on opening the remine			
	Display loads	•		
115	Display	7		—
	Style	Colour by action type		GW=0.20
	Load case	3DWind1 - 0 + CPE + CPI		
	Display eccentricity		-	-
	Generators	Generated		· ·
E	Surface loads			CAN-0.60
	Free			
E	Labels of loads			
	Display label	v		
	Name			
	Value	v		
	Tot. value			
	Eccentricity label			
E	Wind data			
	Display	~		
E	Labels of wind data			
	Display label	×		
	Roof overhangs underside labels			
	Cpi			
	Cpe			
	Net Pressure			
	Peak Pressure			
	·			
	Show names in tab		OK Apply Cancel	el 👗

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

•																					
A Marie	· · · · ×	<b>8</b> - v								Report_1 [3D]	Nind generator.	sa) - Engineering	report							-	a ×
🚔 🐰 Cut	🛧 Undo -						100	E			ल्ली	6	E								
Copy	A Redo -	2	+ ×	I IX II			<b>IO IO</b>			11 K			20								
Paste		properties	sert Eat	Delete Mo	e Move Ind down	ent Outbent	selected outdated	properties	view Edit point picture par	ameters point	converter	of properties	external file								
Clipboard	Undo			Document ite	m		Regenerate		Edit pictures	Edit e	ternal pictures	Tables / Pictures	External files								
Navigator																	A Pr	operties			v = ×
Wind data		<b>£</b> 0	6		1 Win	eteb b												Representation			
			Ť		1	u uata						_	_					Neme	Wind	i data	
					Name	Туре	Roof type	Roof	Swap	Load	Region	Zones			underside			Caption	Wind	i data	
								overnangs	outer	airecuoii					Чре			Caption visible	<b>2</b>		
					WD1	Wal	_		Sal tace	0	1	4	-1 2000	-1 2000				Selection			
										-	2	B	-0.8000	-0.8000				Selection type	All		*
											3	A	-1.2000	-1.2000				Table Template			
											4	В	-0.8000	-0.8000				Template name	Defa	ult (embedded)	· ·
										90	1	D	0.7067	0.7067							
							_			100	2	D	0.7067	0.7067							
										180	1	A	-1.2000	-1.2000							
											2	4	-1.2000	-0.8000							
											4	B	-0.8000	-0.8000							
										270	1	Ē	-0.3133	-0.3133							
											2	E	-0.3133	-0.3133							
					WD2	Roof	Duopitch	×	×	0	1	F1	0.0892	-1.3432							
											2	F2	0.0892	-1.3432							
											3	G	0.0892	-1.0216							
										00	1	H C	0.0892	-0.4662							
										50	2	G	-1.3000	-1.3000							
											3	H	-0.6554	-0.6554							
											4	I	-0.5554	-0.5554							
										180	1	1	-0.3324	-0.4460							
											2	I	-0.5108	-0.5108							
										270	1	E	-1.4662	-1.4662			D	chr.			* 8 ×
											2	G	-1.3000	-1.3000				00000	0		
											4		-0.5554	-0.05554					0		
					WD3	Roof	Duonitch	x	×	0	1	1	-0.3324	-0.4460			R	equest	State		Pr
								1		-	2	I	-0.5108	-0.5108							
										90	1	F	-1.4662	-1.4662							
											2	G	-1.3000	-1.3000							
											3	H	-0.6554	-0.6554							
							_			100	4	1	-0.5554	-0.5554							
										100	2	E2	0.0892	-1.3432							
											3	G	0.0892	-1.0216							
											4	H	0.0892	-0.4662							
										270	1	F	-1.4662	-1.4662							
											2	G	-1.3000	-1.3000							
					1		1		1	1	3	H	-0.6554	-0.6554	1	E.	~				
			1.																		-
																		140			

# **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.

# **Example model**

#### **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**:



#### 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**.



# Input traffic loads

# Traffic lane

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

INPUT PANEL						
A	ll workstations	$\sim$				
- A	ll categories	$\sim$				
🧷 A	ll tags	$\sim$				
TRAF	FIC LOADS					
Sir	ngle traffic loads					
A Tra	affic lane					
K <sup>®</sup> Tra	affic 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).

# 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.

ITraffic loads generator	■ Load patterns ×	
🖶 📲 🔀 🕪 🧰 🖨 🖨 🖨 🔛 🗛 👘 🗸 🖓	🖆 🕂 🗹 🕪 💼 🐟 🗢 🥅 😤 🕒 🗛 👘 🗸 🔻	
LL1 Name LL1		
Traffic Loads Y		
I and arouse 1.62	Irain Loads	×
Load area name	6	
Size [m] 1.000		Name LP1
Validity		Description
valuty		<ul> <li>Insert point</li> </ul>
		X-coordinate [m] 0,000
		Y-coordinate [m] 0,000
		4 Validity
		Validity *
		4 Entities
		Type Point v
		Add new entity
		OK Cancel
Actions		
Generate loads >>>		U.S. C.S. C.S. C.S. C.S. C.S. C.S. C.S.
Draw validity in traffic lane >>>		
New Insert Edit Delete Close	New Insert Edit Delete OK	

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


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

Train Loads		×
	8	Validity 🗸 🗸
		<ul> <li>Entities</li> </ul>
		Type Point v
		Add new entity
		Delete entity
		4 Entity 1
	1.400	Direction Z v
		Force [kN] -1,00
		Position x1 [m] 0,000
		Position v1 [m] 0,000
		Repeat x (n) 1
	YI 🛶	Repeat v (n) 2
		Delta v [m] 1.400
	X	<
		OK Correct
		OK Cancel

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</u> <u>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.

Traffic loads generator		×	
🖬 🕂 🗹 🕼 📾 🐟 🗢 🛅 🖨 🖸	All	• <b>T</b>	Moving V
LL1 Name	LL1		1 Self Weigth
Traffic Loads	LP1	۰	Moving Load - TR1/LP10,000 m
Traffic lane	TR1	*	Moving Load1 - TR1/LP10.250 m
Load group	LG2	۰	• • • • • • • • • • • • • • • • • • •
Load case name	TR1/LP1		Moving Load2 TR1/LP10,500 m
Step [m]	0,250		Moving Loads - TR1/LP10,750 m
Validity			Moving Load4 - TRI/LP11,000 m
			Moving Load5 - TR1/LP11,250 m
			↓ Moving Load6 - TR1/LP11,500 m
			↓ Moving Load7 - TR1/LP11,750 m
			↓ Moving Load8 - TR1/LP12,000 m
			↓ Moving Load9 - TR1/LP12,250 m
			1 Moving Load10 - TR1/LP12,500 m
			Moving Load11 - TR1/LP12,750 m
Actions		_	T ■ Moving Load12 - TR1/LP13,000 m
	Generate loads Draw validity in traffic lane	>>>	Ĵ <sup>Ⅲ</sup> Moving Load13 - TR1/LP13,250 m
New Insert Edit Delete		Close	Manage load cases Ctrl+L



# Mobile loads

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.

### Principle

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

This is illustrated on the picture below:



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

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

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

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

This is illustrated on the following figure.



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

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

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

At which  $h_i$  represents the location of the influence line exactly below  $P_i$ .

The maximum of  $M_n$  is found by trial and error so the sum of the products of an axis load and the influence location below is as large as possible.

This maximum is shown on figure (b) at which the moment  $M_n$  can be determined as follows:

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

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-2025/14/01

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.

# 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.

## Input track and unit load

You can import this track through 🛛 🕅 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.

			÷	1			-
Z	A 14		_	 -	 _		110
K	X			$\vdash$	 		
	2					L	

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.

8
2
N1 [B1]
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.

After defining the track, the unit load can be imported through the menu 🛁 Unit
---

A 😳 🖉 🕷 🖢 🕾	8	🗃 🖬 🛛 All	• 9		
EHL		Name	EHL		
		Track assignment	TR1	-	
		Sections	Use sections from results	+	
		Step for 2D element [m]	1,000		
		Generate section under Load system	8		
		Add new Impulse			
	8	Impulse 1			
		Type	Concentrated	+	
		Value	-1		1
		Position [m]	0,000		
		ev[m]	0.000		
		ez [m]	0.000		
		System	Local	-	18
		7	, <u>, , , , , , , , , , , , , , , , , , </u>		

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 -I++ 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.

### Influence lines

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

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

-	🛃 Infuence lines
	- 😽 Deformations on member
	- Arriver of the second

- ~ 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.



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.

# 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.



### 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.

# 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.

	Name	TR1
1	Use for calculation	
	Used nodes	4
8	Track nodes	
	Node	N1 [B1]
	Node	N2 [B1]
	Node	N3 [B2]
	Node	N4[B3]

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

Unit Mobile Loads			
## <b>≤</b> ™k ΩΩ	8	😂 🖬 🛛 All	• 7
EHL.		Name	EHL
		Track assignment	TRI
		Sections	Use sections from results
		Step for 2D element [m]	1,000
		Generate section under Load s	system 🛛
		Add new impulse	
		Impulse 1	
		Type	Concentrated
		Value	-1
		Position [m]	0.000
		evimi	0.000
		ez [m]	0.000
		System	Local
		Direction	7
			Ny-
			I
vew Insert Edit D	Delete		Clos

### 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 decoded by the 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.

# Exploitation of load systems

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

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



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

This is illustrated for 2 different cases.

#### <u>Case 1</u>

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:

Pr	operties	¢ ×
E	xploitatie van invloedslijnen - Stav	ren (1) 🔽 🖓 🖉
	Name	Exploitatie van invloedslijnen - Staven
	Unit loads	EHL
	Load systems	[P Loads left] [P Loads right]
÷	Limited run	
ŧ	Additional	
ŧ	Load case	
	Setup report	
	Selected members	[B1]
	Values	More comp
	N	
	Vz	
	My	8
	ux	
	uz	
	fiy	
A	tions	
S	ingle Check	>>>
Ρ	review	>>>

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:

Pr	operties	Ф ×
E	xploitatie van invloedslijnen - Staven (1)	- Va V/ /
Γ	Name	Exploitatie van invloedslijnen - Staven
	Unit loads	EHL
	Load systems	[P Loads left] [P Loads right] [Q Load]
ŧ	Limited run	
ŧ	Additional	
ŧ	Load case	
	Setup report	
	Selected members	[B1]
	Values	More comp
	N	
	Vz	
	My	
	ux	
	uz	
	fiy	
A	tions	
S	ingle Check	>>>
Ρ	review	>>>

### 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.



## 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:

	ipernes		ф.)
Ex	ploitatie van invloedslijnen - Staven (1)	- Va	84 0
	Name	Exploitatie van invloedslijnen - Staven	
	Unit loads	EHL	
	Load systems	[P Loads left] [P Loads right] [Q Load]	
Ð	Limited run		
Ð	Additional		
8	Load case		
	Generate		
	Load group	Mobile	·
	Setup report		
	Selected members	[B1]	
1	Values	More comp	
	N		
1	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	X
1. Description of the influence line +	
Influence line:	
Member B1, Position : 24.00[m], Type : My	
Considered load systems: P Loads left P Loads right O Load	
Unit Load : EHL	-
13	4
Section : 24.000  Unit Generate load cases : EHL Max My, B1, P11 To Document Convert EHL Min, My, B1, P11 Next O	Close
Update	

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 - Jet Setup generated load cases is used in the menu Mobile Loads.

• 2 <b>? </b> <u>*</u> •	<b>ux   12</b> 53	=   😅   🖉 🖬   Alles	• 1	
CA	N	aam	CA	
	G	ebruik voor berekening		
	Se	electeer eenheidslast	[EHL]	
	S	electeer lastsystemen	[P Lasten Links] [P Lasten Rechts] [Q Last]	
		Eenheidslast: EHL		
		Naam	EHL	
	1	Belastingsgeval		
		Groep van belastinggevallen		
		Genereer namen		
	±	Beperkte looplengte		
	ΞI	Extra		
		Selectie van staven		
	1	Alle staven		
		Componenten		
	1	Selecteer componenten		
		Staven		
		N		
		Vy		
		Vz		
		Mx		
		Му		
		Mz		
		ux		
		uy		
		uz		
		біх		
		fiy		
		fiz		

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.

<b>v</b> N	EW	₩ Vz	ГM		My [	Mz		Select All
▼ ux	Γvy	🔽 uz	T for		fy 🗆	fiz		Unselect All
utput of	component	ts on suppo	orts					
▼ Rx	☐ Ry	🔽 Rz	ГM	~	My [	Mz		Select All
								Unselect All
utput of	component	ts on 2D ele	ements					
w mx	₩ my	₩ mxy	$\overleftarrow{\mathbb{M}} \lor $	Vy.	💌 nx	M	Pr qxy	Select All
v. ux	₩ oy	🗹 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:

# 3+ ≤ 16 1 ≤ 12 2 4 4 12 12 14 14 14 14 14 14 14 14 14 14 14 14 14	All	• 9
LC1 - Eigengewicht EHL-P Loads left, P Loads right-Min Vz EHL-P Loads left, P Loads right-Min My EHL-P Loads left, P Loads right-Min uz	Nome Description Action type LoadGroup	EHL-P Loads left, P Loads right-Mir
EHL-P Loads left, P Loads right-Min fiy EHL-P Loads left, P Loads right-Max Vz EHL-P Loads left, P Loads right-Max My	Load type Specification Master load case	Static Mobile envelope None
EHL-P Loads left, P Loads right-Max dz EHL-P Loads left, P Loads right-Max fly EHL-P Loads left, P Loads right-Min Rz EHL-P Loads left, P Loads right-Max Rz		
EHL-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.

# 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.



# The construction

	Data				Structure :	
KON-2					Filler	<u> </u>
-	Name	Project M3	1		Material :	
		Projectino	·		Material	C25/30 -
ALL DO	Part	-			Steel	0
	Description	In case of			Timber	0
50.5	Description	Treinloads	8		Other	
	Author	PVT			Aluminium	0
- 3	Date	14 10 200	c			
	1.000	14.10.200				
Pita						
WY/	Project Level :		Model :			
	Advanced	•	One	•		
mill /	National Code	2 ( ) ( ) 10 ( ) 10 ( )				
行漫畫		EC.E	-			
1.1.10		EC-E	-N			

Dynamics		Concrete	
Initial stress		Fire resistance	
Subsoil			
Nonlinearity			
Stability			
Climatic loads	0		
Prestressing	0		
Pipelines			
Structural model			
Parameters			
Mobile loads	0		
Overview drawings			
LTA-load cases			

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

	V2500	1
	1	1
T		
2000		

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.

						Missierpuré	
Ī						54	
ļ							
			,				

Using Support simply selected by drawing a rectangle from right to left:



Then we have the following structure:



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

### 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
k < 7 x
New track action - Polyline - Start point >0;1;8
Command line
R - C 7 2
New track action - Polyline - End point >25;1,8

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

Lise for calculation	
Ose for calculation	
Used nodes	2
Track nodes	
Node	K7 [-]
Node	K8 [-]
Node	K8 [-]

After defining the track, the unit load can be entered through the menu **Unit loads**. As a **Name** of the unit load, **Train** is entered for a simple reference.

With Sections the option Use step according 2D element can be chosen and as step, 0.25m is entered. Unit Mobile Loads . 8 🎜 🏦 🛃 🕷 🕰 🕰 😹 🗛 🖉 Train Train Name Track assignment TR1 \* Sections Use Step according 2D element • 0,250 Step for 2D element [m] Generate section under Load system Ø Add new Impulse .... 🛛 Impulse 1 Concentrated -Туре Value -1 0,000 Position [m] 0,000 ey[m] 0,000 ez [m] Local System - -I1 (-1) I New Insert Edit Delete Close

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



Both impulses are displayed on the bridge deck:



### Influence lines

After defining the train track and the unit load that represent both rails, the linear calculation can be started. To do this it is not necessary to exit the menu Mobile Loads, but you can use the button Through He 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:

- 🖃 🛗 Infuence lines
  - → Displacement of nodes
  - Deformation on slab
  - -↔ Internal forces on slab

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:

Selection manager			X
	0	S1	
	1	Pt1 [m]	5,000, 2,500, 0,000
		Add new point	
	> </th <th>fion</th> <th></th>	fion	
	Select	Deselect	Deselect all
_		ОК	Cancel

## Influence lines - Deformation on member 2D

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

vloedslijn	
Waardes werden vermenigvuldigd met +1.000	20 macro: 1 Positie: 5 m 2.5 m m 0 m
00.0 00.0 00.0 00.0	Type C ux C uy C fix C fix C fix C fix
and the second s	Vermenigvuldiging 1 Herteken Naar docume
C Externen C Alle waarden C Elke: 1 waarde	OK

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

# Input load systems

Through the option — Load System Database a load system can be entered in the project. In this project a predefined load system is used; namely VOSB 150.

That is why the window Load system is cancelled so the Load system Manager is displayed.

Mobile load systems	🔀
利 : · · · · · · · · · · · · · · · · · ·	• 8
System database	

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

Project database	System database	
VOS8 150	KLAS 45R KLAS 60L KLAS 60L Load model 1 Lane 1 Load model 1 Lane 2 Load model 1 Lane 3 Load model 1 Other lane Model 71 Model SW/0 Model SW/2 UIC 71 Unloaded train VBS 170 VBS 170 VBS 270 VOSB 1938 VOSB 250 VOSB 1938 VOSB 250 UIC 71 - HSL 600 E CSN CSD 7 CSN CSD 7 CSN CSD 7 CSN TEM NS	
	CSN TRM 4N	~
	<< Copy to project	

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



Through the button **Properties** sthe 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.

#### 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:

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	2
	Load group	Train 💌
	Setup report	
	Selected 2D members	[S1]
	Values	mx

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

1. Description The selected load sy	of the influence li stems for which the expl	ne + oitation is done:	
Influence line:			
2D macro S1, Global position : x :5.00[m], y :0.01 Type : mx	(m), z :0.00(m)		
Considered load syste	ns:		
VUSB 150			
Unit Load . Hain			
2 Co. ordinator	of the nodes of	the the loadtrack	and their
2. Co-ordinates	of the nodes of	the the loadtrac	k and their
2. Co-ordinates	of the nodes of	the the loadtrac	c and their
2. Co-ordinates	of the nodes of	the the loadtrac	and their
2. Co-ordinates	of the nodes of	the the loadtrack	and their
2. Co-ordinates	of the nodes of	the the loadtract	and their
2. Co-ordinates	of the nodes of	the the loadtrac	x and their
2. Co-ordinates	of the nodes of	the the loadtrac	x and their

### 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 Urbling (m. 1	X1 [ma]	X2
[KNM/M]	[KNM/M]	լայ	[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.

# 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 - It Setup generated load cases is used.

cuses		
🖗 🖙 🖬 🛛 All	- 7	
Name	CA	1
Use for calculation		
Select unit loads	[Train]	
Select load systems	[VOSB 150]	
🗉 Unit Load: Train		
Name	Train	
Load case		
Group of load cases	Train	▼
Limited running length		
Enable		
Start [m]	0,000	
Finish [m]	0,000	
Additional		
Mult. factor results except deformatio	1	
Mobile factor	1	
□ Selection of members		
All members		
Components		
Select components		
Members		
N		
Vy		
Vz		
Mx		
My		
Mz		
ux		
uy		
uz		
fix	0	

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	$\Box \lor$	\_ ∨z	∏ Mx	∏ Му	☐ Mz		Select All
ux 🗌	L av	☐ uz	☐ fix	∏ fiy	E fiz	Ī	Unselect All
utput of c	component	s on suppo	rts				
E Rx	E By	∏ Rz	∏ Mx	∏ My	∏ Mz		Select All
						Ē	Unselect All
utput of c	component	s on 2D ele	ments				
🕶 mx	∏ my	∏ mxy	I▼ vx ∏	w F	nx 🗌 ny	∏ фу	Select All
Ux .	E uy	∏ uz	∏ fix ∏	fiy □	fiz	Ē	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 H 🖌 🖬 🔛 의	🗠 🎒 😂 🖬 🛛 All	- 7
LC1 - Self weight	Neme	Train-VOSB 150-max mx
Train-VOSB 150-max mx	Description	
Train-VOSB 150-min m×	Action type	Variable
Train-VOSB 150-max vx	LoadGroup	Train
Train-VOSB 150-min vx	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:



#### 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.

Pr	roperties			ά×
E	xploitatie van invloedslijnen - 2	D elementen (1)	-	Va 84 0
	Name	Exploitatie van	invloeds	lijnen - 2D
	Unit loads	Train		-
	Load systems	[VOSB 150]		
Ð	Limited run			
ŧ	Additional			
Ξ	Load case			
	Generate			
	Load group	Train		·
	Setup report			
	Selected 2D members	[S1]		
	Values	mx		-
A	ctions			
S	ingle Check			>>>
P	review			>>>

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:	0
Influence line: 2D macro S1, Global position : x :10.00[m], y :2.50[m], z :0.00[m] Type : mx Considered load systems: VOSB 150	
Generate lood cases :         Tran. Mox. mx. S1.P10825.         To Document         i           Point:         x [10         y [25         z [0         Tran. Min. mx. S1.P10825.0         To Document         i	Prev Dase

### 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		
🎜 🕃 🖋 🕼 🔛 🗅 🗅	🖞 🎒 🚅 🖬 🛛 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	<b>▼</b>
Train-VOSB 150-min VX	Load type	Static
Train, Min, mx, S.1, P.1	Specification	Standard
Hail), Max, IIIX, 3.1, F	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:



Results:



# 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       Frame X/2         Name       Project M4         Part       -         Description       Crane track.         Author       PVT         Date       15.10.2005         Project Level:       Model:         Advanced       One         Advanced       One         Mational Code:       One	Data						
Name     Project M4       Part	Name       Project M4         Part       -         Description       Crane track         Author       PVT         Date       [15.10.2005         Project Level :       Model :         Advanced       One         National Code :						Frame XYZ	
Name     Project M4       Part     F       Description     Crane track       Author     PVT       Date     15.10.2005         Project Level :     Model ::       Advanced     Image: Concerting in the image: Concering in the image: Concerting in the im	Name       Project M4         Part       F         Description       Crane track:         Author       PVT         Date       15.10.2005         Project Lavel :       Model :         Advanced       One         National Code :       Concertion         Conserver       One         National Code :       Concertion         Motional Code :       Concertion         Conserver       OK         Orality       Loads         Commission       Protection         National Code :       Concertion         Concertion       Concertion         Dynamics       Concertion         Initial stress       Concertion         Subsoil       Connection modeller         Frame pined connections       Babelity         Character in model       Connection modeller         Frame pined connections       Babelid diagonal connections         Babelid condel       Connection modeller         Praemetress       Connection modeller         Parameters       Connection modeller         Connection modeller       Connection modeller         Connection modeller       Connection constrenons         Depared						Matorial	
	Part       -       -       Statel       Metarial       S235         Description       Crane track       -       Oher       0	Name	Droject M4			_	Concrete	
Part       Image: Crane track         Description       Crane track         Author       PVT         Date       15.10.2005         Project Level:       Model:         Advanced       Image: Complexity         National Code:       Image: Complexity         Image: Complexity       Image: Complexity	Port  Percentrack  Author  PVT  Date  IS 10.2005  Project Level:  Model:  Advanced  IS 10.2005  Project Level:  Model:  Advanced  Cone  Co	( during	Project M4				Steel	
	Description       Crane track         Author       PvT         Date       15.10.2005         Project Level :       Model :         Advanced       One         Volter       Image: Comparison of the second of t	Part	-				Material	S 235
Create track  Author  PVT  Date  I5 10 2005   Project Level:  Advanced  Project Level:  Advanced  Cone  Con	Create Pack       Other Pack         Author       PVT         Date       15.10.2005         Project Level :       Model :         Advanced       One         Advanced       One         National Code :	Description	1			_	Timber	
Author PVT Author PVT Author PVT Author PVT Author PVT Author Author Author PVT Author P	Author PVT Date PVT Date IS 10.2005 Project Level: Model: Advanced One One  Retional Code: EC - EN  Total Stability Loads Combinations Protection National Annexes  Dynamics Initial stress Subsoil OK  Stability OK	Description	Crane track				Other	
Defe       15.10.2005         Project Level :       Model :         Advanced           Advanced           National Code :	Date       15 10 2005         Project Level :       Model :         Advanced       One         National Code :       Image: Comparison of the second se	Author	PVT				Aluminium	
	Date       [15:10:2005         Project Level :       Model :         Advanced       One         National Code :       Image: Comparison of the state of the sta	Data						
	Project Level : Model : Advanced  One Cone Cone Cone Cone Cone Cone Cone Co	Date	15.10.2005					
Project Level : Model : Advanced  Gene Cone  Cone Combinations Protection Netional Annexes Connection Cone Conection Subsol Conection Stability Stability Conection Stability Stability Conection Stability S	Project Level :       Model :         Advanced · One ·							
Project Level : Model : Advanced  National Code :  Concertion Research and the second	Project Level : Model : Advanced  Project Level : One Project Level : One National Code : Come Composition EC-EN OK ON ONE Subsol Committee  Provements Subsol Commettee  Provements Subsol Commettee Subsol Commette							
Project Level : Model : Advanced   One  One  National Code :  EC-EN  tionality Loads Combinations Protection National Annexes  OK  tionality Loads Combinations Protection National Annexes  Dynamics Initial stress  Dynamics Initial stress Initial	Project Level : Model : Advanced  One							
Advanced	Advanced       One         National Code :	Project Level :		Model:				
	National Code :  Company Loads Combinations Protection National Annexes  Dynamics Dynamics Dynamics Subsoil Connection modeller Firer sistance Connection modeller Firer pinde connections Bobled diagonal connections Bobled diag	Advanced	•	One		-		
Ionality Loads Combinations Protection National Annexes           Dynamics         Bitel           Initial stress         Bitel           Subsol         Cinnetic loads           Climatic loads         Bitel           Prestnessing         Bitel diaponections	Connection modeller     Connection monotones     Connection monotones     Connection monotones     Connection modeller     Frame pinted connections     Connection monotones     Student     Connection monotones     Connection monotones     Connection monotones     Connection monotrewings     Student     Mobile loads     Connection monotrewings     Connecton monotrewings     Connection monotrewings     Connecton     Connect	Mational Code		10 million 10 million				
Conection Mational Annexes      Dynamics     Initial stress     Subsoi     Subsoi     Sobility     Climatic loads     Climatic loads     Destination	Concercion monodravings     Connection							
Itionality         Loads         Combinations         Protection         National Annexes           Dynamics         Imilal stress         Imilal stres         Imilal	Bit         Combinations         Protection         National Annexes           Dynamics         Initial stress         Initial s							ОК
Dynamics     Imidial stress     Imidial stress     Imidial stress       Subsoli     Imidial stress     Imidial stress     Imidial stress       Nonlinearity     Imidial stress     Imidial stress       Stability     Imidial stress     Imidial stress       Stability     Imidial stress     Imidial stress       Climatic loads     Imidial stress     Imidial stress       Prestressing     Imidial stress     Imidial stress	Dynamics         Imital stress         Imital strese strestrestres         Imital stress <td< th=""><th></th><th></th><th></th><th></th><th></th><th></th><th>OK.</th></td<>							OK.
Initial stress     Image: Connection modeller       Subsoil     Image: Connection modeller       Nonlinemity     Image: Frame rigid connections       Stability     Image: Connection modeller       Cirmatic loads     Image: Connection modeller       Prestressing     Image: Connection modeller	Initial stress     Image: Stress     Image: Stress     Image: Stress       Subsoil     Image: Stress     Image: Stress     Image: Stress       Stability     Image: Stress     Image: Stress     Image: Stress       Cimatic loads     Image: Stress     Image: Stress     Image: Stress       Prestressing     Image: Stress     Image: Stress     Image: Stress       Structural model     Image: Stress     Image: Stress     Image: Stress       Mobile loads     Image: Stress     Image: Stress     Image: Stress       Mobile loads     Image: Stress     Image: Stress     Image: Stress       Mobile loads     Image: Stress     Image: Stress     Image: Stress	onality  Loads	Combinations   F	Protection   N	lational Ann	nexes		ок
Subsoil         Connection modeller           Nonlinearly         Frame figid connections           Stability         Frame pigit connections           Climatic loads         Grid pinned connections           Prestressing         Botted diagonal connections	Subsoli     Connection modeller       Nonlinearity     Frame rigid connections       Sability     Frame rigid connections       Climatic loads     Gird pinned connections       Prestressing     Bolked diagonal connections       Studural model     Connection modeller       Parameters     Studural model       Mobile loads     Studural model       Venniew drawnings     Parameters	onality Loads	Combinations   F	Protection   N	lational An	nexes	Steel	ОК
Nonlinearity         Image: Constraint of the second s	Nonlinearity     Imate loads     Frame gingd connections       Stability     Imate loads     Grid pinned connections       Otimate loads     Imate loads     Bolted diagonal connections       Prestressing     Imate loads     Expert system       Studural model     Imate loads     Scatfolding       Parameters     Imate loads     LTB 2nd Order       Mobile loads     Imate loads     LTB 2nd Order	onality Loads	Combinations   F	Protection   N	lational Ann	nexes	Steel Tire resistance	ОК
Stability         Image: Climatic loads         Image:	Stability     Image: Connections       Climatic loads     Girl dipmed connections       Climatic loads     Girl dipmed connections       Prestressing     Bolled diagonal connections       Pipelines     Expert system       Structural model     Connection monodrawings       Parameters     Scaffolding       Mobile loads     B       LTB 2nd Order     AccelorMimil	onality Loads Dynamics Initial stress Subsoil	Combinations   F	Protection   N	lational Ani	nexes	Steel Tire resistance connection modeller	ОК
Climatic loads D Grid pinned connections Bolted diagonal connections	Climetic loads     Gid pinned connections       Prestressing     Bolked diagonal connections       Pipelines     Expert system       Studtural model     Connection monodrewings       Parameters     Scottoling       Mobile loads     LTB 2nd Order       Ovenriew dimensa     ArceInd/Mini	onality Loads Dynamics Initial stress Subsoil Nonlinearity	Combinations   F	Protection   N	National Ann	nexes	Steel Iire resistance Jonnection modeller Tame rigid connection	ОК
Prestressing  Bolted diagonal connections	Prestressing     Bolted diagonal connections       Pipelines     Expert system       Structural model     Connection monodrawings       Parameters     Scatfolding       Mobile loads     LTB 2nd Order       Ovendew drawings     ArcelorMired	Dynamics Initial stress Subsoil Nonlinearity Stability	Combinations   F	Protection   N	lational Ant	nexes	Steel ire resistance Connection modeller rame rigid connection	OK
	Pipplines         Expert system           Structural model         Connection monodrawings           Parameters         Scafolding           Mobile loads         B           LTB 2nd Order         ArcelorMimil	onality Loads Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads	Combinations   F	Protection   N	National Ant	nexes	Steel ire resistance Somection modeller trame rigid connection trame pined connection trame injuned connection	OK 15 ions 15
Pipelines  Expert system	Structural model     Connection monodrawings       Parameters     Scaffolding       Mobile loads     LTB 2nd Order       Oxeneider drewings     ArcelorMitml	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing	Combinations   F	Protection   N	lational Ann	rexes	Steel Tree resistance onnection modeller frame figid connection trame pinned connection died diagonal conne	OK Sions Ins
Structural model Connection monodrawings	Perameters Scaffolding Scaffolding UTB 2nd Order Arrelotvillal	Dynamics Initial stress Subsoil Nonlinearity Stability Climatic loads Prestressing Pipelines	Combinations   F	Protection   N	lational Ann	nexxes	iteel ire resistance onnection modeller farme ripinde connection rid pinned connection tolted diagonal come tolted diagonal come	OK Is ions is cctions
Parameters Scaffolding	Mobile londs  Querview drawings Querview drawings Querview drawings	onality Loads   Dynamics Initial stress Subsoil Nonlinearity Stability Orimatic loads Prestressing Pipelines Structural mod	Combinations   F	Protection   N		nexces	iteel iire resistance onnection modeller rame rigid connection rame junied connection dig planed connection scipert system onnection monodraw	OK 18 ions 18 rctions ings
	Overview drawings	onality Loads Dynamics Initial stress Subsol Nonlinearity Stability Cirratic Loads Prostressing Pipelines Structural mod	Combinations   F	Protection   N		nexces	Steel irre resistance onnection modeller rame pinale connect aid pinale connection diaded diagonale conne scpert system onnection monodraw confolding	OK OK Issued to the second sec
Mobile loads   LTB 2nd Order		onality Loads Dynamics Initial stress Subsoil Nonlinearity Stability Girnetic loads Prestressing Preptienss Structural mod Parameters Mobile floads	Combinations   F	Protection   N		rexes	Steel Terre sistance Somection modeller Tame pinned connection tame pinned	OK Sons ss cctions ings
Parameters Scaffolding	Mobile londs  Querview drawings Querview drawings Querview drawings	onality Loads   Dynamics Initial stress Subsoil Noninearity Stability Climatic loads Prestressing Pipplines Structural mod	Combinations   F	Protection   N		nexces	Steel ire resistance Somection modeller tame rigid connection tame rigid connection tame ginal connection tame donnection connection connection monodraw	OK IS ions is iccions ings
	Overview drawings	Dynamics Initial stress Initial stress Subsoil Nonlineently Stability Climatic loads Prestressing Pipelines Structural mod	Combinations   F	Protection   N		nexces	Steel ire resistance onnection modeller frame injuit connection rame pinned connection ind pinned connection loaded diagonal conne spert system connection monodraw confolding	OK
Mobile loads   LTB 2nd Order		onality Loads Dynamics Initial stress Subsoil Nonlinearity Stability Girnetic loads Prestressing Preptienss Structural mod Parameters Mobile floads	Combinations   F	Protection   N		rexes	Steel Terre sistance Somection modeller Tame pinned connection tame pinned	OK Sons ss cctions ings

# The construction

The first portal of the hall can be entered through  $-\frac{1}{20}$  Catalogue blocks .



Subsequently the haunch beams on which the rail support, can be entered through <sup>45</sup> Beam . The beams have a length **1m**, type **IPE 180** and move across **¾** of the length of the column.

Ł

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




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



The window More copies can be set:

Numb	er of copies	÷.	Connect selected nodes with new beams	~
✓ Ins	ert the very last copy	1	Copy additional data	~
Distanc	e vector		How to define the distance ?	
Define	distance by cursor	Г	between two copies	
x	0,000	m	C from original to the last co	ру
v	5.000	- m	How to define the rotation ?	
,	0.000		between two copies	
e Detetia	10,000		C from original to the last co	ру
Notatio	n Izraz	-	Current LICS	
rx	0,00	deg	C distance under	
ry	0,00	deg	t distance vector	
17	0.00	dea		

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



The geometry input is ended by entering the rigid supports to the column bases and by executing the Check structure data and Connect members/nodes to connect the various members.



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

## 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:

Pr	operties	а :	×
Т	rip train (mobile load) (1)	▼ Va V/ Ø	
	Name	TR1	
	Use for calculation	8	
	Used nodes	4	
Ξ	Track nodes		
	Node	K8 [B36]	
	Node	K14 [B36]	
	Node	K23 [B37]	
	Node	K32 [B38]	
A	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  $\stackrel{\text{\tiny $22$}}{\longrightarrow}$  In this project three unit loads are entered:

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

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

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

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

Unit Mobile Loads			X
#3# <b>≤</b> ™ k 22	8	🗃 🖬 🛛 All	• 9
Midpoint	T	Name	Midpoint
Left		Track assignment	TRI 👻
Right		Sections	Use sections from results
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1		Step for 2D element [m]	1,000
		Generate section under Load system	0
		Add new impulse	
		Delete impulse	
	B	Impulse 1	
	1.0	Type	Concentrated ·
		Value	-0.5
		Position [m]	0.000
		ey[m]	0,000
		ez [m]	0,000
		System	Local
		Direction	Ζ
	Θ	Impulse 2	
		Type	Concentrated -
		Value	-0,5
		Position [m]	8,000
		ev[m]	0,000
		ez [m]	0.000
		System	Local
		Direction	Ζ
		TI (-0.5)	
New Insert Edit	plate		-
	Aren		Close

Unit Mobile Loads			X
## <b>#</b> #₩ 요요	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	Ζ 🗸
	Θ	Impulse 2	
		Туре	Concentrated -
		Value	-0,2
		Position [m]	8,000
		ey[m]	0,000
		ez [m]	0,000
		System	Local 🗸
		Direction	Ζ 🗸
		II (-0.3)	00 12 (-0.2)
New Insert Edit D	elete		Close

Unit Mobile Loads			X
## <b>#</b> #₩ 요요	8	🗃 🖬 🛛 All	• 7
Midpoint		Name	Left
Right		Track assignment	TR1 👻
Left		Sections	Use sections from results
		Step for 2D element [m]	1,000
		Generate section under Load system	0
		Add new Impulse	
		Delete impulse	
	Θ	Impulse 1	
		Туре	Concentrated 💌
		Value	-0,2
		Position [m]	0,000
		ey[m]	0,000
		ez [m]	0,000
		System	Local
		Direction	Ζ 🔹
	Θ	Impulse 2	
		Туре	Concentrated 💌
		Value	-0,8
		Position [m]	8,000
		ey[m]	0,000
		ez [m]	0,000
		System	Local
		Direction	Z 🗸
		11 (-0.2)	00 I
New Insert Edit D	elete		Close

## Input load system

The input of the load system for the crane track happens through the option **Load System Database**. For the crane track a total weight of **40 kN** is taken. If the crane track is in the middle, it means **20 kN** per rail. On every rail there are two wheels so a weight of **10 kN** is calculated. The interval between the wheels is **0,8m**. However, the defined Unit loads are entered with a factor lower than 1. For the unit load Centre a factor of **0,5** is entered per rail. Because of this the loads of the load system have to be doubled to come to the total weight of **40kN**. The single load system can be entered as two point loads of **20kN** with a mutual distance of **0,8m**.

Load Sys	tem		X
Simply lo	ad system Advar	nced load system	
Name			
Crane	e Track		
I Ne	eglect point load wh	it opposite influence	8 8
Distrib	outed load		
0,00		kN/m	
	Point load [kN]	Offset	
1	-20.00	-0,40	
2	-20,00	0.40	
-	0,00	0.00	
		D.U.	0.300
		Del All	OK Cencel Apply

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

### 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:

🛱 Detail analysis

- Member force, deformation
- \*5 Reaction
- Aember stress

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

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

In the Property window these loads can be set:

- 1		Exploitatie van invioedslijne.
- 1	Unit loads	All
	Load systems	[Crane Track]
ŧ	Limited run	
Ð	Additional	
	Load case	
	Generate	
-[	Load group	Mobile .
	Setup report	
- [	Selected members	[B33]
	Values	My
	values	iny

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	Х	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:

<u>Under *Title 6. and 7.*</u> 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:

		$\downarrow \downarrow$		
9	18		27	36

#### **Generation Enveloping Load Cases**

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

Setup generated load	cases		
# # <b>∠</b> til k   Ω α   4	🗟 😂 🖬 🛛 All	• 7	
CA	Name	CA	^
	Use for calculation	Ø	
	Select unit loads	[Midpoint] [Right] [Left]	
	Select load systems	[Crane Track]	
	🗆 Unit Load: Midpoi	int	
	Name	Midpoint	
	Load case		
	Group of load cases	Mobile	·
	🗉 Unit Load: Right		
	Name	Right	
	Load case		
	Group of load cases	Mobile	▼ 目
	🗉 Unit Load: Left		
	Name	Left	
	Load case		
	Group of load cases	Mobile	▼
	Limited running le	ength	
	Additional		
	Selection of mem	bers	
	All members		
	Selection		
	Components		
	Select components		
	Members		
	N	D	
	Vy		
	Vz		
	Mx		
	My		
	Mz	D	
	ux		~
New Insert Edit Del	ete		Close

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.

ΓN	Γvy	\ □ ∨z	Гм	× 🔽	My 🔽	Mz		Select All
∏ ux	∏ uy	☐ uz	∏ fix	E	fiy 🗌	fiz		Unselect All
utput of	componen	ts on suppo	orts					
∏ Rx	∏ Ry	∏ Rz	Гм	× Г	му Г	Mz		Select All
								Unselect All
utput of	componen	ts on 2D ele	ements					
₩ mx	🗹 my	₩ mxy	VX VX	$\boxtimes \bigtriangledown$	₩ mx	M ny	₩ cpy	Select All
UX UX	🗹 uy	₩ U2	🔽 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:

🛯 🤮 🗶 🕼 🕵 🔄 🕰 🖉	All	- 7
LC1 - Eigengewicht	Name	Left-Crane Track-Max My
Midpoint-Crane Track-Min My	Description	
Midpoint-Crane Track-Max My	Action type	Variable
Left-Crane Track-Min My	LoadGroup	Mobile
Left-Crane Track-Max My	Load type	Static
Right-Crane Track-Min My Bight-Crane Track May My	Specification	Mobile envelope
Righteer and mackenax 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

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

Member	Case	dx [m]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
B33	Left-Crane Track-Max My	0,313	-0,05	-0,02	24,89	0,00	8,17	0,04
B33	Left-Crane Track-Max My	4,688	0,14	0,05	-28,75	-0.01	2,77	0,12
B33	Left-Crane Track-Max My	0,000	0,11	0.04	-2,22	0,00	0,01	-0,10
B33	Left-Crane Track-Max My	2,188	0,07	0,03	11,35	0,00	27,69	0,00

# 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.

o The second	Dynamics	10	~	Steel	
Scia	Initial stress			Fire resistance	
- genou	Subsoil			Connection modeller	
	Nonlinearity			Frame rigid connections	
	Stability			Frame pinned connections	
	Climatic loads			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 Fr	unctionality Loads Protection		
Scia	Acceleration of gravity	9,810 m/s^2	
	Wind Load		٦
	According to code	EC 1 / 26,200m/s / 0	
	Snow Load		5
	According to code	EC 1 / Sk=1,00kN/m <sup>2</sup> Ce=1,0 Ct=1,0	
			-
		OK Cano	el

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

- Wind generator
- Snow generator
- Wind & snow generator

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

Example: 2D climatic generators.esa

# Wind generator

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

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

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

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

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]:

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

Zone	A B			с		D		E		
h/d	C <sub>pe,10</sub>	Cpe,1	C <sub>pe,10</sub>	C <sub>pe,1</sub>						
5	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,7	
1	-1,2	-1,4	-0,8	-1,1	-0,5		+0,8	+1,0	-0,5	
≤ 0,25	-1,2	-1,4	-0,8	-1,1	-0,5		+0,7	+1,0	-0,3	

In this example, h = 6m and d = 10m, so h/d = 0.6.

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



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

Dital	Zone	for wind	directio	n θ = 0°								
Angle a	F		G	G		н			J			
Angle a	Cpa,10	Cps, 1	Cps, 10	Cpe,1	Cps.10	Cpa.1	Cpe,10	Cpe,1	Cps,10	Cps,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					-0,8 -1,2			+0,2		+0,2 +0,2		
-5*	-2,3	-2,5	-1,2	-2,0			-0,6		-0,6			
50	-1,7	-2,5	-1,2	-2,0	-0,6	-1,2	-0,6		+0,2			
5-	+0,0		+0,0		+0,0				-0,6			
150	-0,9	-2,0	-0,8	-1,5	-0,3		-0,4		-1,0	-1,5		
15-	+0,2		+0,2		+0,2	HH H	+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			
459	-0,0		-0,0	)`	-0,0		-0,2 🐾		-0,3			
40"	+0,7	11	+0,7		+0,6		+0,0		+0,0			
60°	+0,7	i.	+0,7		+0,7		-0,2	1	-0,3			
75°	+0,8	1	+0,8		+0,8		-0,2	1	-0,3			

Table 7.4a — External pressure coefficients for duopitch roofs

The angle  $\alpha$  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	Ê.
La	ayers			
U	ser define	d seled	tions	
c	leaner			
c	oordinate	s info	_	
x	ML IO Doc	ument	C	E.
E	dit profile	library		

This means that an interpolation has to be done between the values for angle  $\alpha$  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:

	P	roject data		×			Manager for National	annexes	×
Basic data Functionality Lo	ads Protection				🚚 🤮 🗶 📸	k ାର ଜା <i>କି</i> ।	😂 🔜 🛛 Al		*
Scia Functionality Lo Cota Engineer Part: Description: Author: Date: Structure: Frame XZ Project Level: Advanced	ads Protection	Model	Material Concrete Material Steel Material Steel Material Coher Masony Auminium Coher Code National Code: EC - EN National Auminet: Standard EN	\$ 235 • • • •	Standard EN Austrian ONORM- Belgian NBN-EN British BS-EN NA Crech CSN-EN NZ Dutch NEN-EN NZ Dutch NEN-EN NZ Name National annex References EN 1990: Basis EN 1990: Basis EN 1991: Acti EN 1991: Acti EN 1992: Des EN 1992: Acti EN 1993: Acti EN	EN NA IA s of structural design) ms of structural estillation of structuras and actions - Snow la estillation - Snow la	gn ads) 55(5) bucktures buildings) re design) nd detaing rules) holdow core alab res Frei buildings) fre design)	Standard EN Standard EN	
					EN 1993-1-3 (Ger EN 1993-1-5 (Plat EN 1993-1-8 (Der EN 1994: Derei	eral rules - Suppleme ed structural elements ign of joints)	ntary rules for cold-formed me	•	
			UK.	Cance	New Insert	Edit Delete			Close
				Setup mar	vager				
			according to ECT sure for 2D wind et face for 2D wind vet face for 2D wind user face for 2D wind user face for 2D wind user face for 3D wind et for 3D wind et for 3D wind et for 3D wind et for for any factor eason factor only factor of the building [year] with factor [] r [] sign factor [] r [] sign factor []		re inter least two int Use o Vence 10,000 0,000 1,00 1,00 1,00 1,00 1,00	mal pressure res verall coefficients al walls or rectang	Cpe. 10 Aler buildings (EC1-1-4, 1	72.2)	
		t_min + minimal k_l +turbulence	height (m) factor (f)	Load o	1.000 1.00 Jefault NA parameter	s OK	Cancel		

## 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.

	Snow	load - EC 1		×
00 0	0.80	0.80	0.00	Coefficients Begin 0 End 0 Set coeff Next Previous Divide Connect Regenerate
Snow weight		Frame distance	5 m	Undo Redo
C default	° m			OK Cancel

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 $\alpha$	$0^\circ \le \alpha \le 30^\circ$	$30^\circ < \alpha < 60^\circ$	$\alpha \ge 60^{\circ}$		
$\mu_1$	0,8	$0,8(60 - \alpha)/30$	0,0		
μ2	0,8 + 0,8 a/30	1,6			

Since the angle  $\alpha$  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	
<pre>(•) ○ □ ○ □</pre>	

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



Figure 5.3: Snow load shape coefficients - pitched roofs

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



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

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 <sub>/4</sub> C <sub>e</sub> C <sub>1</sub> s <sub>k</sub>	[1] undrifted $\mu_i C_e C_1 a_k$	[1] undrifted $\mu_i C_e C_1 s_k$	[1] undrifted $\mu C_e C_1 s_k$		
[2] drifted <sub>µi</sub> , C <sub>e</sub> C <sub>i</sub> s <sub>k</sub>	[2] drifted <sub>JA</sub> C <sub>6</sub> C <sub>1</sub> a <sub>6</sub>	[2] drifted µ <sub>i</sub> C <sub>k</sub> C <sub>i</sub> s <sub>k</sub> (except for roof shapes in AnnexB)	[2] drifted µ <sub>i</sub> C <sub>e</sub> C <sub>t</sub> s <sub>k</sub> (except for roof shapes in AnnexB)		
D <sub>2</sub>	Accidental design situation (where snow is the accidental action)	Accidental design situation (where snow is the accidental action)	Accidental design situation (where snow is the accidenta action)		
	[3] undrifted $\mu_k C_e C_t C_{est} s_k$	<li>[3] drifted in sk (for roof shapes in AnnexB)</li>	[3] undrifted /4 CeCt Ceal Sk		
	[4] drined // CeCi Ceal Sk		[4] drifted $\mu_i$ $s_k$ (for root shapes in AnnexB)		

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

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

# Wind & Snow generator

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

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

•		Setup manager	×
E Standard EN	Flame	Bandard EN	
Wind pressure according to EC1	S Wind pressure according to EC1		
	meteria prisone ne so wee	and destined large	
	Dealers, Assessed Provider, Thursd	in Plana man	
	Operating destined face for 20 with	downart face	
	Estand sense to far 10 and	An diministratification	
	Education provided that and and	Use sweet coefficients Coe 15	
	Defense in bearing (r. e)		
	Tops of the desition	Method with or writerio darlo ddree (EC114-727)	
	h and of the state at all	100.000	
	Defensiona level of terrary lev	0.000	
	C. e. do - developed factor		
	Uniter 11	1.00	
	U o shake - shake tater		
	Weber 51	1.00	
	U. e. e. propriate factor		
	Value 11	1.00	
	V b.Dbasic word value by lock?	36,300	
	m - ar denativ \$co./m/18	13	
	Producteday		
	1/b - He period of the hubbro heart	90.00	
	e nub pubability factor []	1.00	
	E-state twite H	0.20	
	n - separat 11	0.50	
	iii. Termin		
	Isman-calacony		1.0
	R - termin factor (-)	0.11	
	a 0-mustressiandh tri	6.903	
	r ein-ninissi teistr [s]	1.000	
	k I - turbulence factor 11	1.00	
		Load default NA parameters DK Cancel	
			1