\ SCIA ENGINEER

ADVANCED TRAINING

DYNAMICS

TABLE OF CONTENTS

CHAPTER 1 : INTRODUCTION

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

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

- Eigen frequency calculations

Esas.21 (Dynamics (natural frequencies) – Frames)	Professional edition
Esas.22 (Dynamics (natural frequencies) – Surfaces)	Professional edition

- Advanced dynamic calculations

Esas.23 (Dynamics (advanced) – Frames)	Professional edition
Esas.24 (Dynamics (advanced) – Surfaces)	Professional edition

- Non uniform damping characteristics

Esas.25 (Non uniform damping – Frames)	Not part of an edition

Dynamic calculations are not so frequent in civil engineering as static calculations. On the other hand, they are inevitable in certain projects. Wind effects on high-rise structures, transverse vibration of towers and chimneys, structures located in seismic regions,…
SCIA Engineer contains specialized modules covering common dynamics-related issues. In this course, the different aspects of these modules are regarded in detail.
First, the foundation of dynamic calculations is examined: the eigen frequency calculation. Eigen frequencies form the basis for all types of dynamic analysis.
In one of the last chapters, the eigen frequency calculation is extended with harmonic loads: the influence of for example vibrations due to machinery, can be calculated using these principles.
Two chapters are devoted to seismic calculations and the influence of damping on the seismic action.

All chapters are illustrated with examples. The relatively easy examples have been purposefully chosen to provide a clear understanding of what actually happens in the dynamic calculations. To this end, nearly all calculations have been verified by manual calculations to give a good insight into the application of the theory in SCIA Engineer.
When the principles are clearly understood, they can be applied to more complex structures without difficulties.

Functionalities from chapters 2 to 10 are available on the 64 bits version of SCIA Engineer.

But, for the moment, functionalities from chapters 11 to 13 are only available on the 32 bits version.

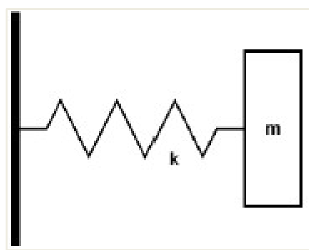
CHAPTER 2 : FREE VIBRATION : EIGEN FREQUENCIES

In this chapter, the calculation of eigen frequencies in SCIA Engineer is explained in detail.
Eigen frequencies can be required to verify comfort criteria for buildings, to analyse wind-induced resonance for bridges, to check requirements for sensitive equipment,…
First, the theory behind the calculation is discussed and illustrated with an example. The procedure is then used for both frame and slab structures. The results of all examples are compared with manual calculations to provide a clear understanding of the applied principles.

2.1 Theory

To understand what is going on during the dynamic analysis of a complex structure with frames or finite elements, the free vibration of a SDOF (Single Degree Of Freedom) system is regarded in detail. A complete overview can be found in reference [1].

Consider the following system:



A body of mass m is free to move in one direction. A spring of constant stiffness **k**, which is fixed at one end, is attached at the other end to the body.

The equation of motion can be written as:

$$m.\ddot{y}(t) + k.y(t) = 0 \qquad (2.1)$$

A solution for this differential equation is:

$$y(t) = A.\cos(\omega t)$$

Inserting this in (2.1) gives:

$$(-m.\omega^2 + k).A.\cos(\omega t) = 0 \qquad (2.2)$$

This implies that:

$$\omega = \sqrt{\frac{k}{m}} \qquad (2.3)$$

Where ω is called the natural circular frequency.
The natural period T can be written as:

$$T = \frac{2\pi}{\omega} \qquad (2.4)$$

The natural frequency (or eigen frequency) f can be written as:

$$f = \frac{1}{T} = \frac{\omega}{2\pi} \tag{2.5}$$

For a general, MDOF (Multiple Degree Of Freedom) structure, equation (2.1) can be written in matrix notation:

$$M.\ddot{U} + K.U = 0 \tag{2.6}$$

Where:

U is the vector of translations and rotations in nodes,
$\ddot{U}$ is the vector of corresponding accelerations,
K is the stiffness matrix assembled during the dynamic calculation,
M is the mass matrix assembled during the dynamic calculation.

From this equation, it is clear that the calculation model created for a static analysis needs to be completed with additional data: masses.
The solutions of (2.6) are harmonic functions in time. A possible solution can have the following form:

$$U = \Phi.\sin(\omega.(T - T_0) \tag{2.7}$$

Notice that in this solution, a separation of variables is obtained:

- The first part, (Φ), is a function of spatial co-ordinates,
- The second part, $\sin(\omega.(T - T_0)$, is a function of time.

When substituting (2.7) in (2.6), an equation is obtained which is known as the Generalized Eigenproblem Equation:

$$K.\Phi - \omega^2.M.\Phi = 0 \tag{2.8}$$

The solution of (2.8) yields as many eigenmodes as there are equations.
Each eigenmode consists of 2 parts:

- An eigenvalue: value ω_i
- An eigenvector: vector Φ_i, which is not fully determined. The deformation shape is known, but the scale factor is unknown.

This scale factor can be chosen in different ways. In the next paragraph this will be explained further.

An overview of the mathematical (matrix) approach behind the calculation of eigenvalues and eigenvectors can be found in reference [25].

2.2 Eigen frequencies in SCIA Engineer

In SCIA Engineer, as scale factor, a M-orthonormalisation has been implemented. This is shown in the following relation:

$$\Phi_i^T.M.\Phi_i = 1 \tag{2.9}$$

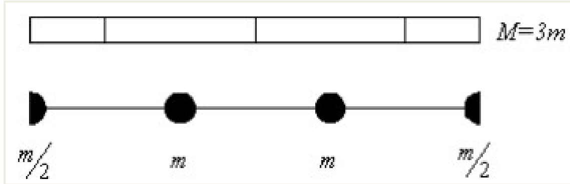
Some of the characteristics of M-orthonormalisation are :

$$\Phi_j^T.M.\Phi_i = 0 \quad \text{quand } i \neq j \tag{2.10}$$

$$\Phi_i^T.K.\Phi_i = \omega_i^2 \tag{2.11}$$

The M-matrix (the mass matrix) can be computed in different ways. SCIA Engineer uses the so-called lumped mass matrix representation of the *M*-matrix. The lumped mass matrix offers considerable advantages with respect to memory use and computational effort because in this case the *M*-matrix is a diagonal matrix. The masses are thus guided to the nodes of the Finite Element mesh.

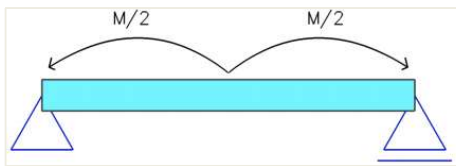
This principle is illustrated on the following figure [28]:



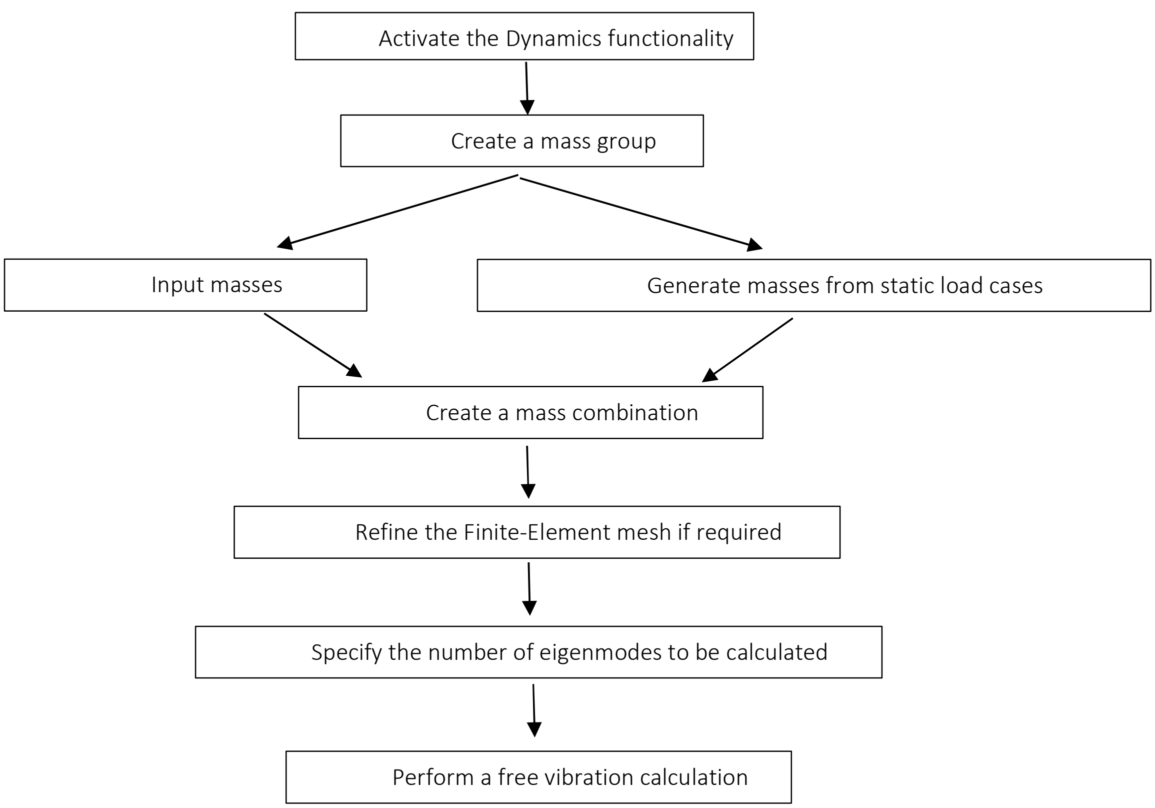
The calculation of eigenmodes and eigen frequencies is thus made on a discretised finite element model of the structure. This means that instead of a general structure with an infinite number of degrees of freedom, a calculation model with a **finite number of degrees of freedom** is analysed.
The number of degrees of freedom can generally be determined by a simple multiplication: the **number of mesh nodes** is **multiplied** by the number of **possible displacements** in the node.

It is important to know that **the accuracy of the model** is in proportion to the "precision of discretisation", i.e. to **the number of elements** of the finite element mesh. This refinement has almost no practical meaning in static calculations. However, for dynamic and non-linear analyses, it significantly affects the accuracy of the results.

Consider the following example. A beam on two supports is loaded by its self-weight. By default (for a static calculation) there is only one finite element for the beam. Taking the above into account, the mass M of the beam will be guided to the two end nodes of the beam since these correspond with the mesh nodes of the finite element mesh.



In this case, this means that the entire mass will be located in the supports so no mass can go into vibration and the dynamic calculation cannot be executed. As indicated, a mesh refinement is required here to attain results.
The following diagram shows the required steps to perform a Free Vibration calculation:



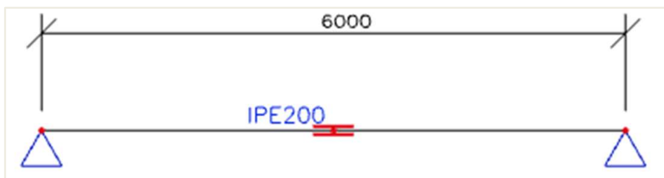
The analogy between a static and dynamic calculation is clear:

- In a static calculation, Loads are grouped in Load cases and the Load cases are used in Combinations.
- In a dynamic calculation Masses are grouped in Mass Groups and the Mass Groups are used in Mass Combinations.

The required steps from this diagram are illustrated in the following example.

Example_02-1.esa

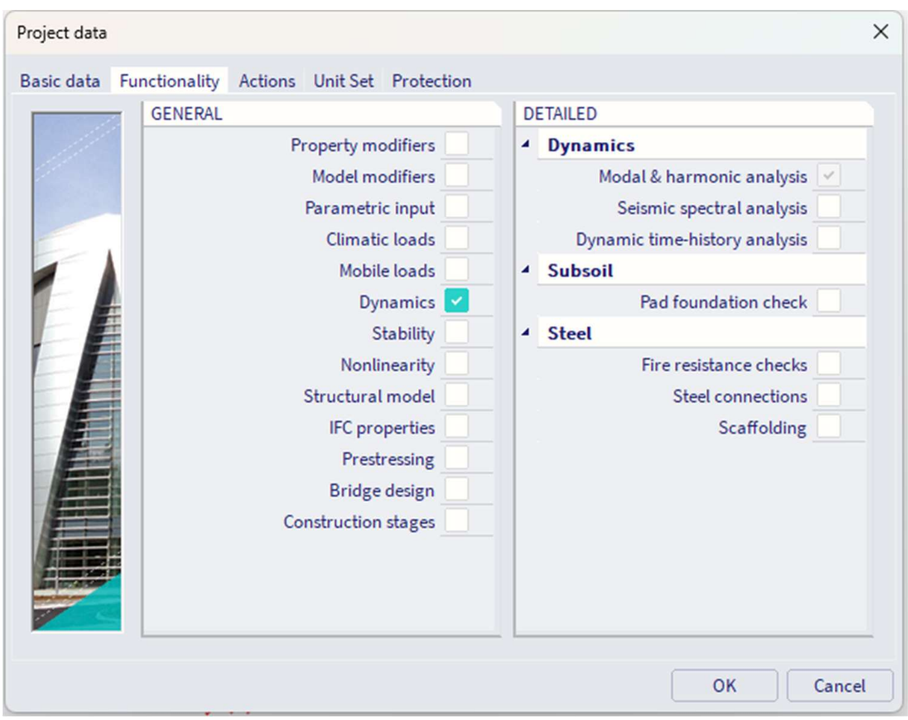
In this example, a beam on two supports is modelled. The beam has a cross-section type IPE 200, a length of 6m and is manufactured in S235 according to EC-EN. A node has been added to the middle of the beam, which will make it possible to add a nodal mass in that location.



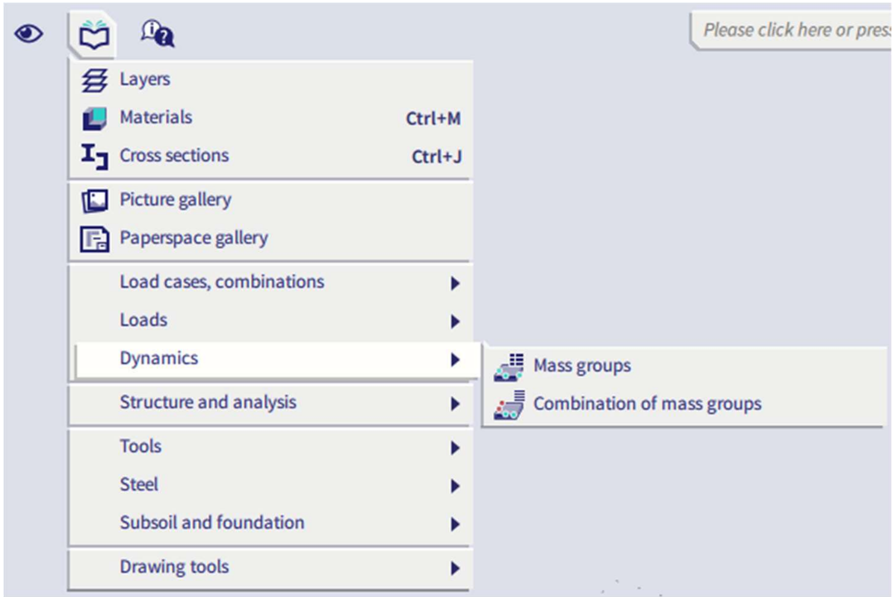
Only one static load case is created: the self-weight of the beam.

Step 1: functionality

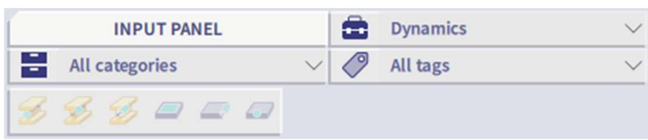
The first step in the Dynamic calculation is to activate the functionality Dynamics on the Functionality tab in the Project Data.



When this is done, a new menu, « Dynamics » will appear in the main menu “Library”:

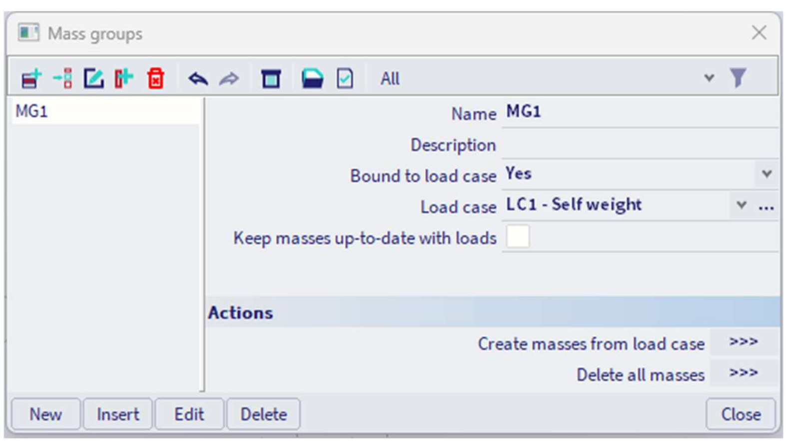


And also in the input panel:



Step 2: mass group

The second step is to create a Mass group.



As indicated in the diagram, a Mass Group is used to group masses in a same way a Load Case is used to group Loads. When a Mass Group is defined, masses can be inputted.
SCIA Engineer also allows the user to create masses from a static load case.
When for example a roof weight is inputted as line loads, the action *“Create masses from load case”* will automatically generate masses from these line loads. It is clear that this provides a quick input of necessary data. When the option *“Keep masses up-to-data with loads”* is ticked on, then the action to create masses will create masses which remain linked

to the loads of the load case. The amount of mass in a 'linked' mass is updated each time you click on the action button "Create masses from load case" or each time you perform a calculation.

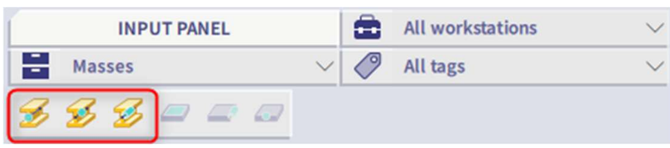
NB:

- The self-weight of a structure is always taken into account automatically for a dynamic calculation. Even if there is no mass group linked to self weight. The mass of the self-weight is not displayed.
- When specifying a load case but not using the action 'Create masses from load case' nothing will happen: no masses will be created.
- When creating masses from loads, SCIA Engineer will use the acceleration of gravity specified on the Loads tab of the Project Data. By default this value is 9,81 m/s².
- The mass remains unchanged after any modification or deletion of the original force. If the mass is intended to correspond to the new force, it is necessary to delete the mass and create it again.
- The mass is generated only from vertical force components.
- Free loads cannot be converted into masses.

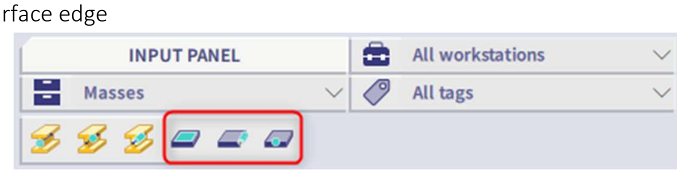
Step 3: masses

When Mass Groups are created, Masses can be inputted on the structure. SCIA Engineer allows the input of:

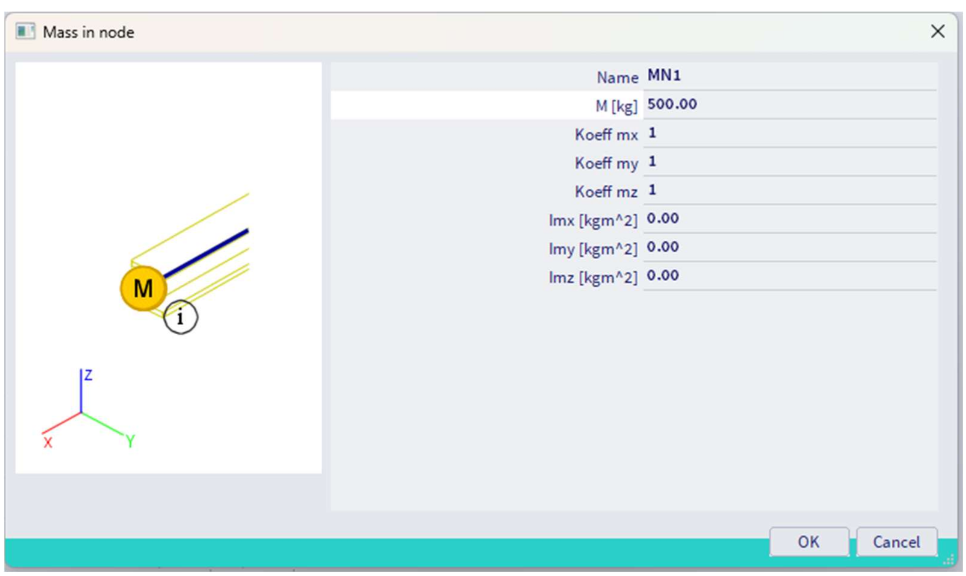
- Mass in node
- Point mass on beam
- Line masse on beam



- Surface mass
- Line mass on surface edge
- Point mass on surface edge

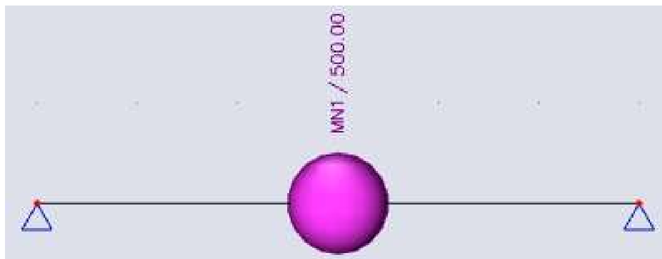


In this example, a mass of 500 kg will be inputted on the middle node of the beam using "Mass in node".



The parameters **Koeff mx, Koeff my** and **Koeff mz** specify how much of the mass will participate in the vibration according to the global X, Y or Z axis.
This can be used when calculating for example a chimney: when **Koeff mx** is put on 1 and **Koeff my** and **Koeff mz** are taken 0, then the mass can only vibrate in the global X-direction so only eigenmodes in that direction will be obtained.
Imx, Imy and **Imz** specify the moment of inertia around the global X, Y or Z axis. By default a nodal mass is concentrated so it has no inertia. When the mass represents a large machine, it is possible to input the moment of inertia of this machine.

The nodal mass of **500 kg** is inputted on the middle node of the beam:

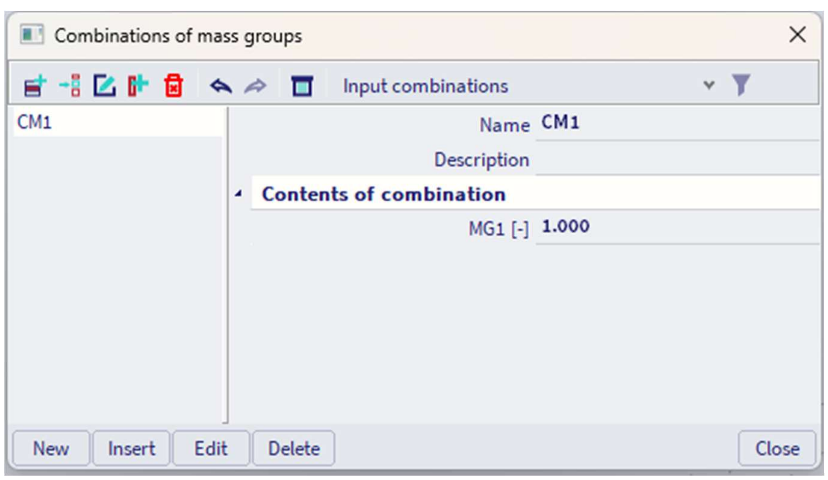


NB:

- To display masses and mass labels, make a right click on the screen and go to + « **Set view parameters for all** », and tab « **Loads / Masses** ».
- Masses are **Additional Data**, which can be moved / copied to other entities.

Step 4: mass matrix

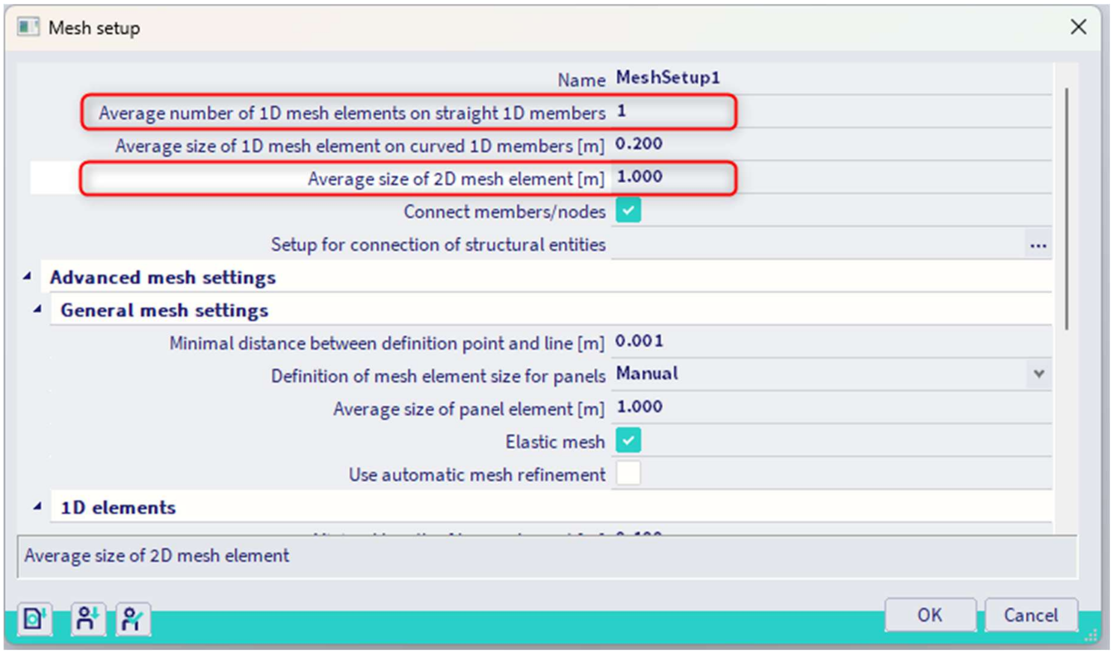
Next, the Mass groups can be combined within a **Combination of Mass Groups**.
This is actually the mass matrix M which has been mentioned in the beginning of this chapter.



The Combination of Mass Groups works in the same way as a linear Load Combination.
A multiplication coefficient can be inputted for each Mass Group. This coefficient can be used when the mass of a structure changes during its lifetime. Consider for example a water tank. One Combination of Mass Groups can be created with a coefficient 1,00 to specify a full tank and another Combination of Mass Groups can be created with a coefficient 0,50 to specify a tank, which is half-full. In this way, both cases can be calculated in one time. As stated in step 2: the self-weight is automatically taken into account for each Combination of Mass Groups.

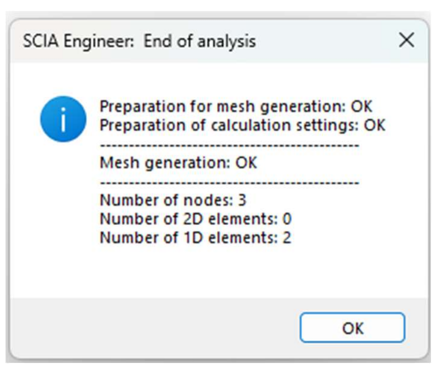
Step 5: mesh setup

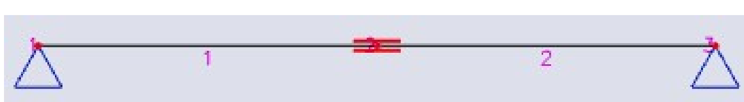
After executing the previous steps, the calculation can already be started. However, as stated previously it can be required to refine the finite element mesh.
This can be done though the main menu Tools / Calculation & Mesh / Mesh settings :



For 1D members (beams) the Average number of tiles of 1D element can be augmented. In general, 5 to 10 tiles are sufficient for a dynamic calculation. When specifying a too high amount, the calculation will take a long time to complete. For 2D elements (plates & shells) the Average size of 2D element needs to be altered.

In this example, due to the inserting of the middle node, there is already a mesh node there, so it is not required to have a denser Finite Element Mesh. This can be seen after mesh generation :



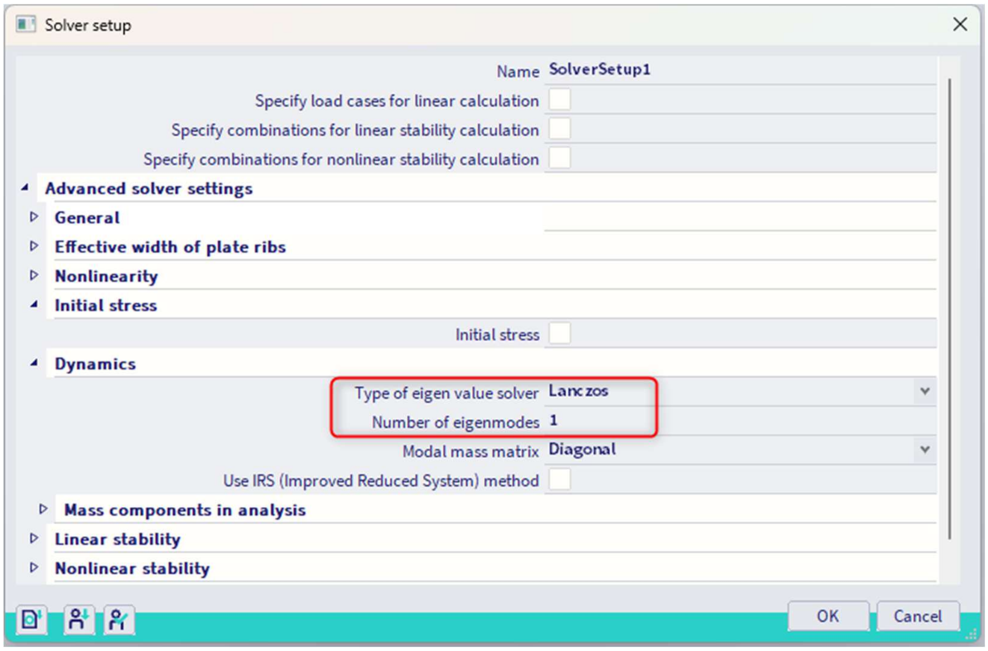


Note: To display the numbering of finite elements, make a right click on the screen + « Set view parameters for all »:

- Tab « Structure », in « Mesh », and tick « Draw mesh » on.
- Tab « Labels », in « Mesh », tick « Display label & Elements 1D » on.

Step 6: solver setup

Another important step before launching the calculation is to specify the amount of eigenmodes that need to be calculated and with which method they can be calculated.
This can be done through the main menu Tools / Calculation & Mesh / Solver settings.

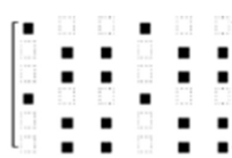


By default, the Lanczos method is used. This method is set as default even in older projects where originally another method was used. In comparison with older methods, the Lanczos method is faster and more stable.
As explained above, the number of eigen frequencies is dependent of the number of degrees of freedom of the structure which are on their turn dependent of the discretisation.

In this example, only the mesh node located in the middle of the beam can vibrate vertically. Therefore only one eigenmode needs to be calculated. The Number of Frequencies can thus be lowered to 1.

The modal mass matrix can be Diagonal or Consistent.
In the first case (diagonal matrix), masses are affected to nodes. The matrix contains only components in diagonal and in translation (not in rotation). This method is faster but less precise.
In the second case (consistent matrix), masses are distributed along the element with shape functions. The matrix contains components in translation (but no in diagonal) and also in rotation. This method is more precise but can lead to a more important calculation time.

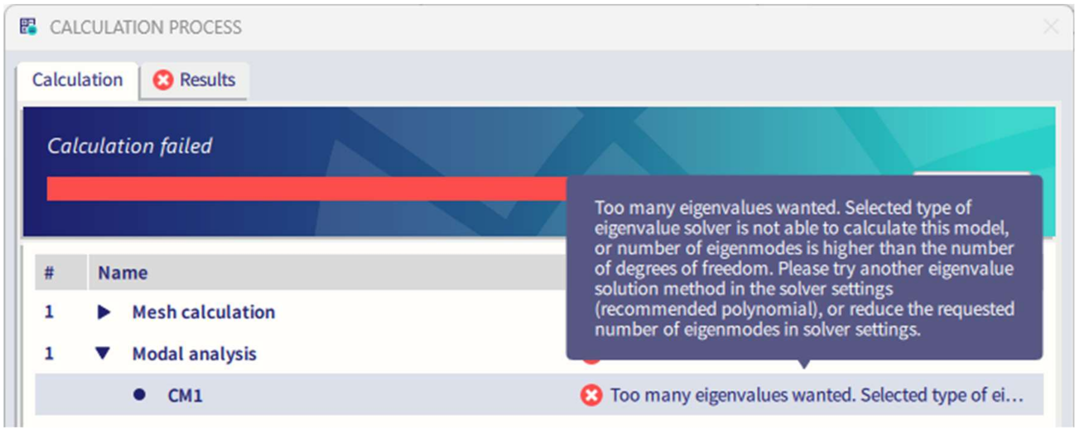
Diagonal matrix



Consistent matrix

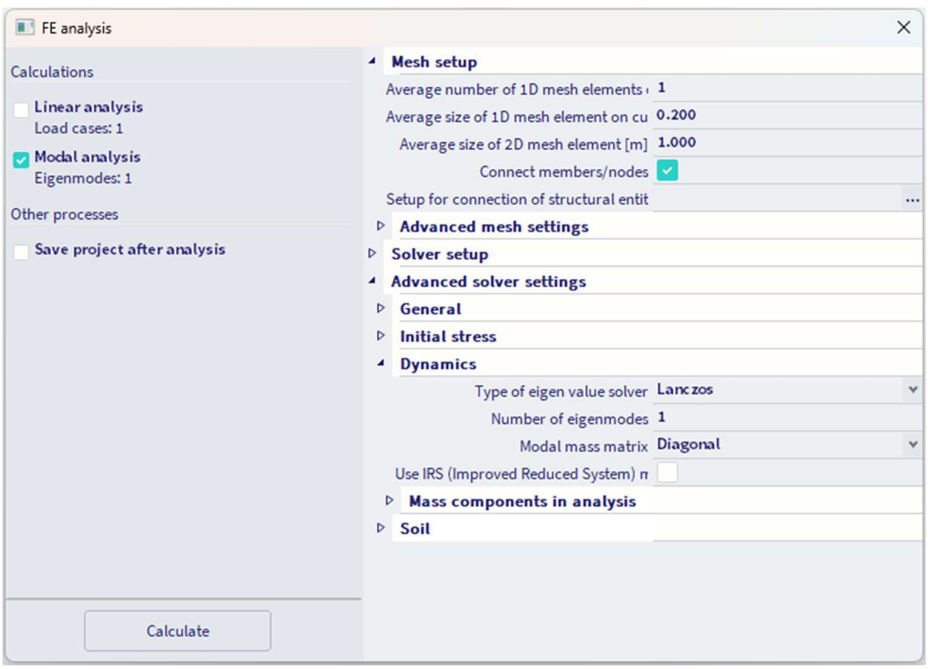
The option “Use IRS (Improved Reduced System) method” requires floors to be defined first, so this option cannot be used now. “Produce wall eigenmode results (needed for ECtools)” is only used if you are using the extra program ECtools to analyse seismic effects in masonry.

NB: When the number of frequencies is higher than the amount of degrees of freedom, a message will appear during the calculation, stating the calculation cannot be executed. The solution is to lower the number of frequencies to be calculated or to apply a mesh refinement so more degrees of freedom are created.

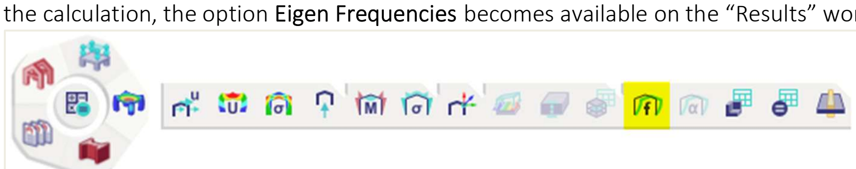


Step 7: modal analysis

The last step is to perform the Modal Analysis calculation through the main menu Tools / Calculation & Mesh / Calculate.



After performing the calculation, the option Eigen Frequencies becomes available on the "Results" workstation:



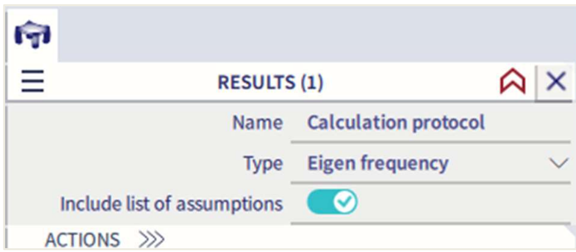
The preview shows the following result:

Eigen frequencies

N	f [Hz]	ω [1/s]	ω^2 [1/s²]	T [s]
Mass combination : CM1				
1	6.32	39.69	1575.58	0.16

Step 8: calculation protocol

According to this calculation, the natural frequency of the first mode is shown to be 6,32Hz.
To view the results in more detail, it is possible to look at the Calculation protocol for the Eigen Frequency calculation:



Calculation protocol

Solution of Free vibration

Number of 2D elements	0
Number of 1D elements	2
Number of mesh nodes	3
Number of equations	18
Combination of mass groups	CM1
Modification group	None
Number of frequencies	1
Method	Lanczos
Bending theory	Mindlin
Type of analysis model	Standard
Modal mass matrix	Diagonal

Sum of masses

	Mass type	X [kg]	Y [kg]	Z [kg]
CM1	Moving mass	567.12	634.23	567.12
CM1	Total mass	634.23	634.23	634.23

Relative modal masses

Mode	iega [rad	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	$_{xi\_R}/W_{xtot}$	$_{yi\_R}/W_{ytot}$	$_{zi\_R}/W_{ztot}$
1	39.6947	0.16	6.32	0.0000	0.0000	-23.8142	0.0000	0.0000	1.0000	0.0000	0.0000	0.0000
							0.0000	0.0000	1.0000	0.0000	0.0000	0.0000

Let's see more in detail about the results in the calculation protocol.

Solution of the free vibration:

- The model was divided in 2 finite elements, resulting in 3 mesh nodes.
- Each node has 6 degrees of freedom (X, Y, Z, Rx, Ry, Rz) resulting in 18 equations.
- The combination of mass groups for the results was CM1.
- The number of frequencies set in the solver settings is 1.
- The Lanczos method was used to perform this calculation.

The Sum of masses shows the amount of mass, which can vibrate for this Combination of Mass Groups (CM1). In this example, this is governed by the mass of 500 kg and the mass of the beam.

The mass of the beam can be calculated as follows:

- The beam is an IPE 200 with cross-section A= 0,00285 m²
- The length of the beam is 6 m
- The volumetric mass of S 235 is 7850 kg/m³.

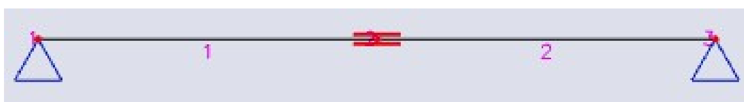
$$M = 0{,}00285\ m^2 * 7850\ kg/m^3 = 22{,}3725\ kg/m$$

Now to find the total mass, we must assign the masses to the mesh nodes and take into account the vibrations which are possible:

- For node 1: 1,5 m * 22,3725 kg/m = 33,5587 kg
  (1/4 of the beam mass goes to the left node)

- For node 2: 3,0 m * 22,3725 kg/m + 500 kg = 67,1175 kg + 500 kg = 567,1175 kg
  (1/2 of the beam mass goes to the middle node along with the nodal mass in the node)

- For node 3: 1,5 m * 22,3725 kg/m = 33,55875 kg
  (1/4 of the beam mass goes to the right node)



	Direction X	Direction Y	Direction Z
Node 1	Fixed	(Frame XZ)	Fixed
Node 2	567,1175 kg	(Frame XZ)	567,1175 kg
Node 3	Fixed	(Frame XZ)	Fixed
Total	634,2349 kg	X	634,2349 kg
Calculation Protocol	634,23 kg	0 kg	634,23 kg

As you can see, the sum of masses in the calculation protocol corresponds to the sum of masses in all mesh nodes, taking into account the degrees of freedom in each node.

It is clear that a denser mesh will provide a more accurate participation of the beam mass.

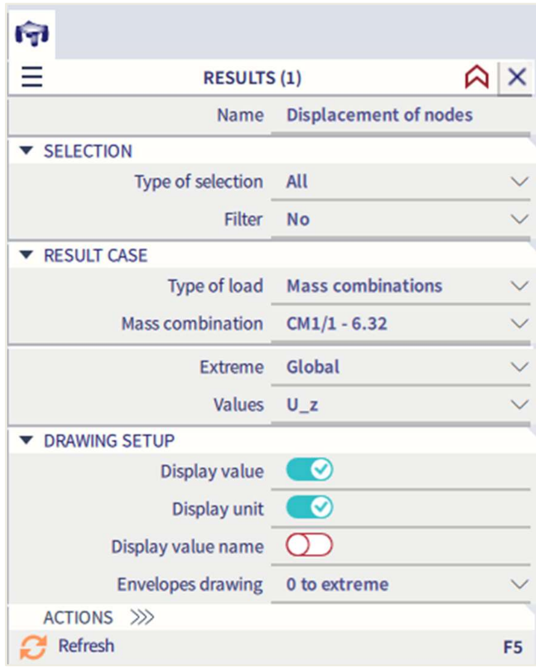
The Modal Participation Factors show the amount of mass that is vibrating in a specific eigenmode as a percentage of the total mass. In this example W_{zi}/W_{ztot} is equal to 1 which means that 100% of the mass is vibrating in the vertical direction for the first eigenmode. This means that in the other degrees of freedom, no mass will be displaced in the Z-direction.

The W_{yi_R}/W_{ytot_R} is equal to 1 means that this first eigenmode the only eigenmode in which mass can rotate around the global Y-axis.

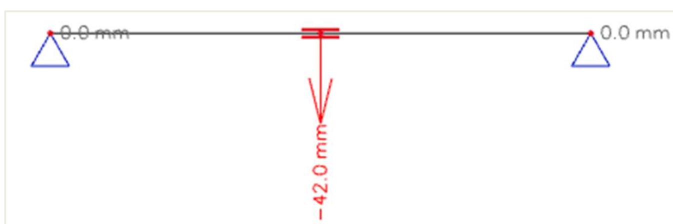
As a side note, we must indicate that these results will strongly alter once we use a finer mesh. Since more nodes will add more degrees of freedom and thus more possible eigenmodes.
These factors will be looked upon in more detail during the Seismic calculations.

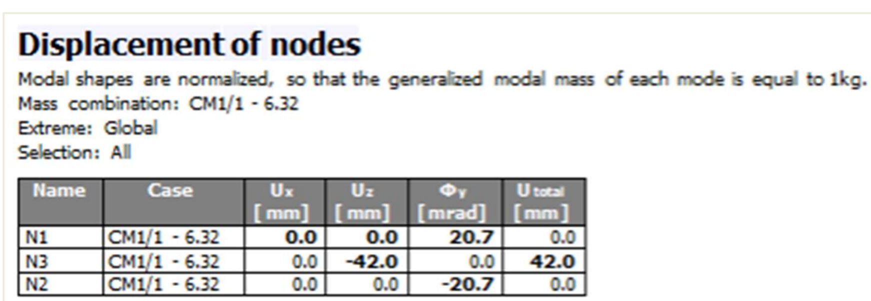
Step 9: displacement of nodes

The eigenmode can be visualized through Displacement of nodes.



- « Selection » = All
- « Type of loads » = Mass combinations
- For each eigenmode, a specific mass combination can now be shown.
- « Value » = U_z to view the displacement of nodes.





Displacement of nodes

Modal shapes are normalized, so that the generalized modal mass of each mode is equal to 1kg.
Mass combination: CM1/1 - 6.32
Extreme: Global
Selection: All

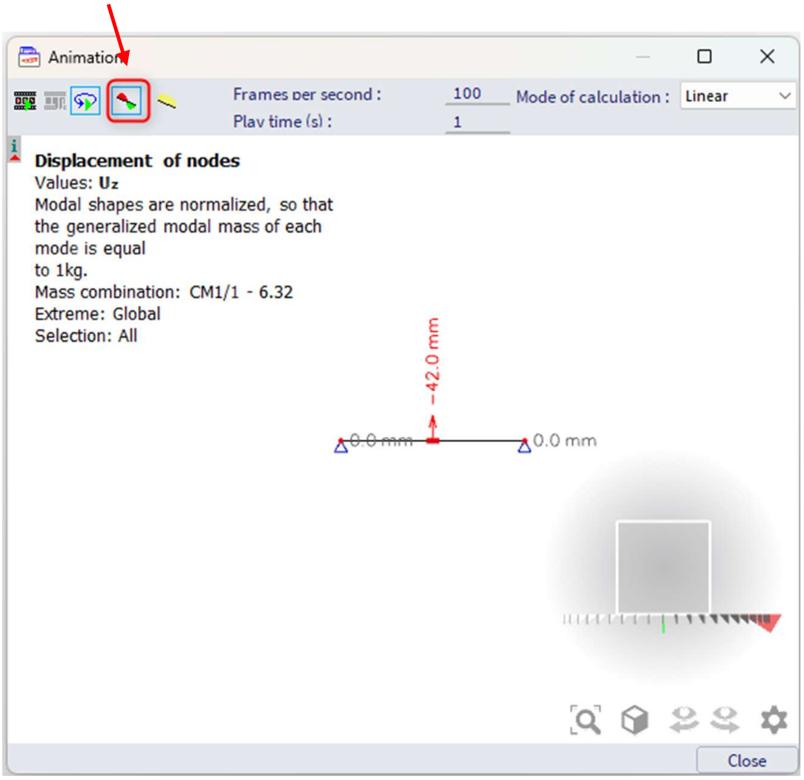
Name	Case	U_x [mm]	U_z [mm]	Φ_y [mrad]	U_{total} [mm]
N1	CM1/1 - 6.32	**0.0**	**0.0**	**20.7**	0.0
N3	CM1/1 - 6.32	0.0	**-42.0**	0.0	**42.0**
N2	CM1/1 - 6.32	0.0	0.0	**-20.7**	0.0

The result is as expected, the inner node is vibrating. A denser mesh will provide a much better representation of the Eigenmode. It is important to bear in mind that a vibration is in two directions: in this case the eigenmode is shown moving up, however half a period later it will be moving down.

Free vibration gives only the conception of structure properties and allows predicting the behaviour of the structure under time varying load conditions. In nature, each body prefers to remain in a standstill. If forced to move, it prefers the way requiring minimal energy consumption. These ways of motion are the eigenmodes.

The **eigenmodes** do not represent the actual deformation of the structure. They only show deformation that is **"natural" for the structure**. This is why the magnitudes of calculated displacements are dimensionless numbers. The numbers provided are ortho-normed, i.e. they have a particular relation to the masses in the structure. The absolute value of the individual numbers is not important. What matters is their mutual proportion.

The vibration of the structure can be shown through by the main menu **Result > Animation**.
Activating the option « Symmetry » will show the actual vibration in both directions.



NB: using CTRL + right mouse button, the structure can be rotated in the "Animation" window.

Manual calculation

In order to check the results of SCIA Engineer, the eigen frequency of this structure is calculated by a manual calculation. Following reference [1], the circular frequency of a beam on two supports with a mass in the middle can be calculated as follows:

$$\omega^2 = 48.\frac{EI}{ML^3}$$

With:

ω:	circular frequency
E:	modulus of Young
I:	moment of inertia of the beam
L:	length of the beam
M:	mass in the middle of the beam

In this example:

$E = 210000\ N/mm^2$

$I_y = 19430000\ mm^4$

$L = 6000\ m$

$M = 500\ kg$

So we have:

$$\omega^2 = \frac{48 * 210000\,{}^{\mathrm{N}}\!/_{\mathrm{mm}^2} * 19430000\mathrm{mm}^4}{500\mathrm{kg} * (6000\mathrm{mm})^3} = 1813{,}47\,{}^{\mathrm{rad}^2}\!/_{\mathrm{s}^2}$$

$$\omega = 42{,}58\,{}^{\mathrm{rad}}\!/_{\mathrm{s}}$$

$$\mathrm{f} = \frac{\omega}{2\pi} = 6{,}78\ \mathrm{Hz}$$

The result calculated by SCIA Engineer was 6,32 Hz.

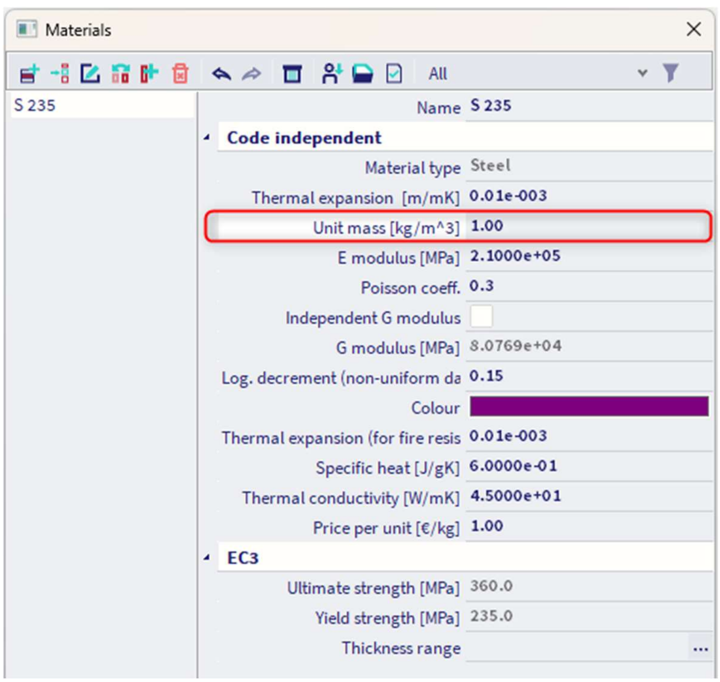
The difference in results is caused by two assumptions in the manual calculation:

- The manual calculation does not take into account the self-weight of the beam.
  Since $\omega = \sqrt{k/m}$, a lower mass will lead to a higher ω and a higher f.

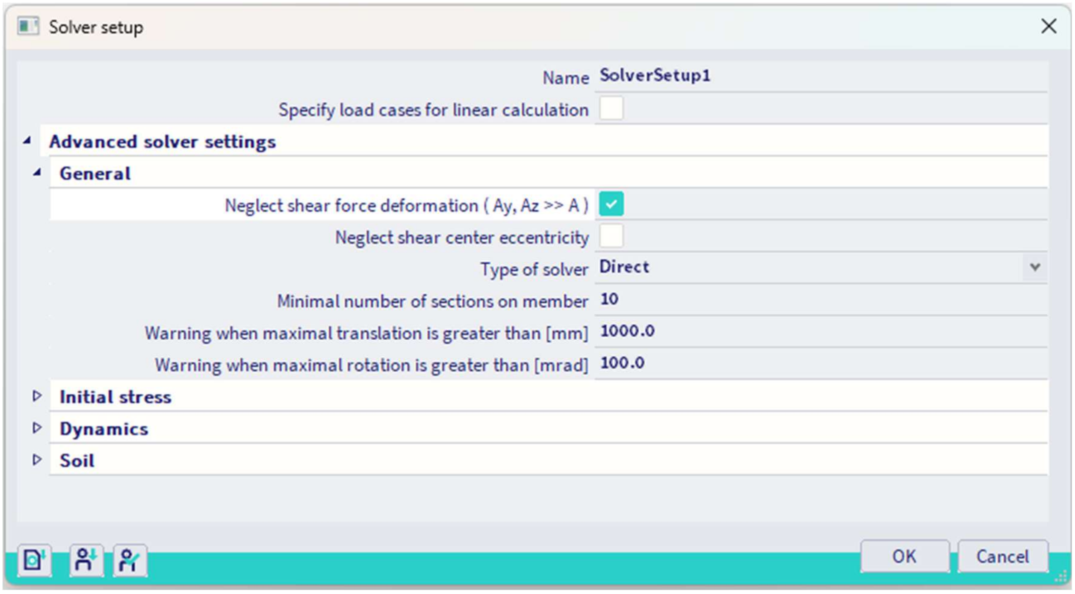
- The manual calculation does not take into account shear deformation.
  A lower deformation leads to a higher stiffness k, a higher ω and a higher f.

These two assumptions can also be implemented in the calculation model of SCIA Engineer:

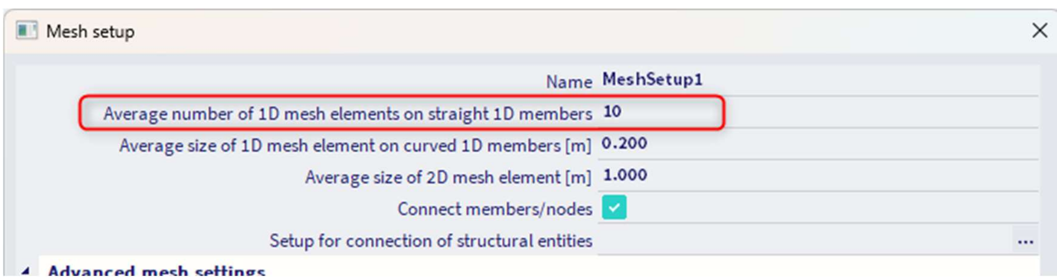
- In order not to take the self-weight into account, the volumetric mass of S 235 can be set to 1 kg/m^3 in the material library:



- To neglect the shear deformation, activate this option through the menu Tools / Calculation & Mesh / Solver settings:



- To obtain a correct and precise result, the mesh must also be refined to 10 finite elements. This can be done through the main menu Tools / Calculation & Mesh / Mesh settings.



Now when the calculation is performed again, the following results are obtained :

Eigen frequencies

N	f [Hz]	ω [1/s]	ω² [1/s²]	T [s]
Mass combination : CM1				
1	6.78	42.58	1813.32	0.15

These results correspond exactly to the manual calculation.

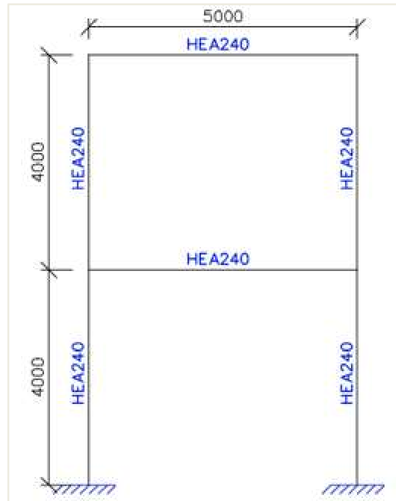
This example clearly shows the importance of checking the assumptions behind the applied theories. When comparing results between two calculations, always make sure the same assumptions/hypotheses are used.

2.3 Frames

In this paragraph, the Free Vibration calculation is illustrated for frame structures. The principles of the theory are applied in detail and verified by means of manual calculations.

Example 02-2.esa

In this example, a two-storey frame is modelled. The members have cross-section HE240A and are manufactured in S 235 according to EC-EN.



The height of each storey is 4 m.
The width of the frame is 5 m.
The column bases are inputted as fixed supports.

One static load case is created: self-weight.

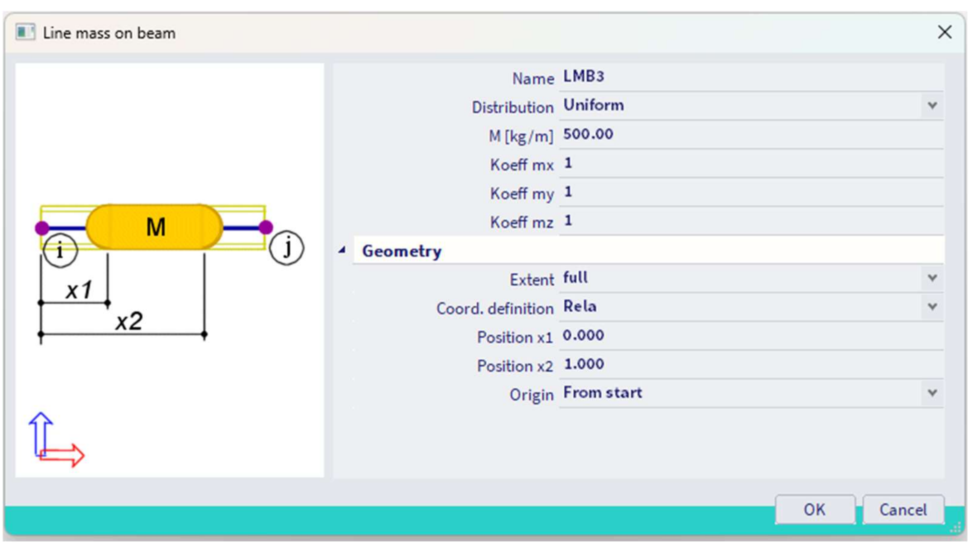
On the beams of the floor and roof level, a line mass of 500 kg/m will be introduced.

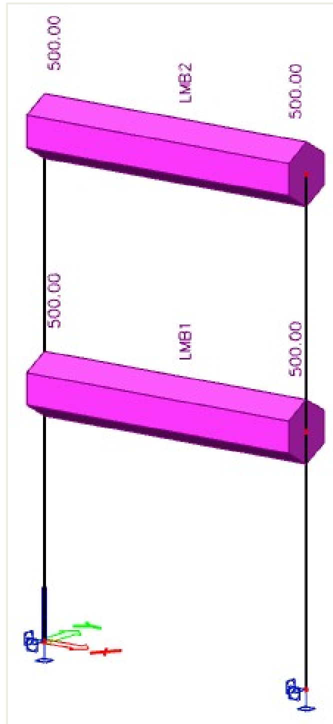
Step 1&2: functionality and mass group

The activation of the Dynamics Functionality and the creation of a Mass Group are identical to the previous example.

Step 3: masses

When the Mass Group is created, the line masses of 500 kg/m can be inputted on the roof and floor beams of the frame.

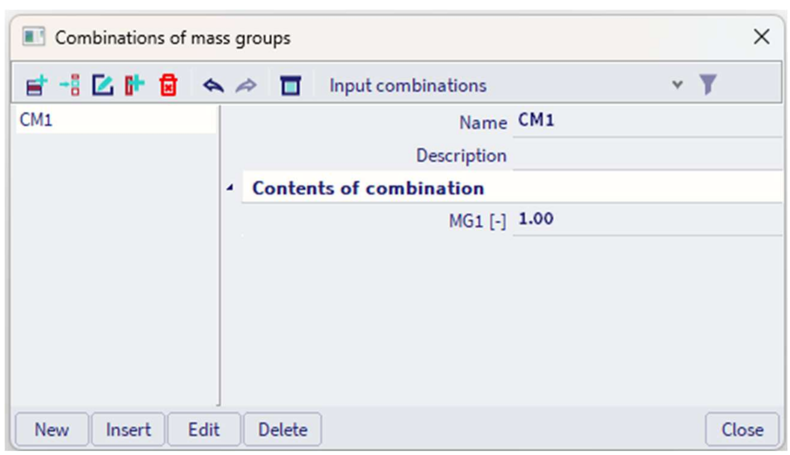




Note: to render the display of masses, go to ”Set view parameters for all” / « Loads / masses ».

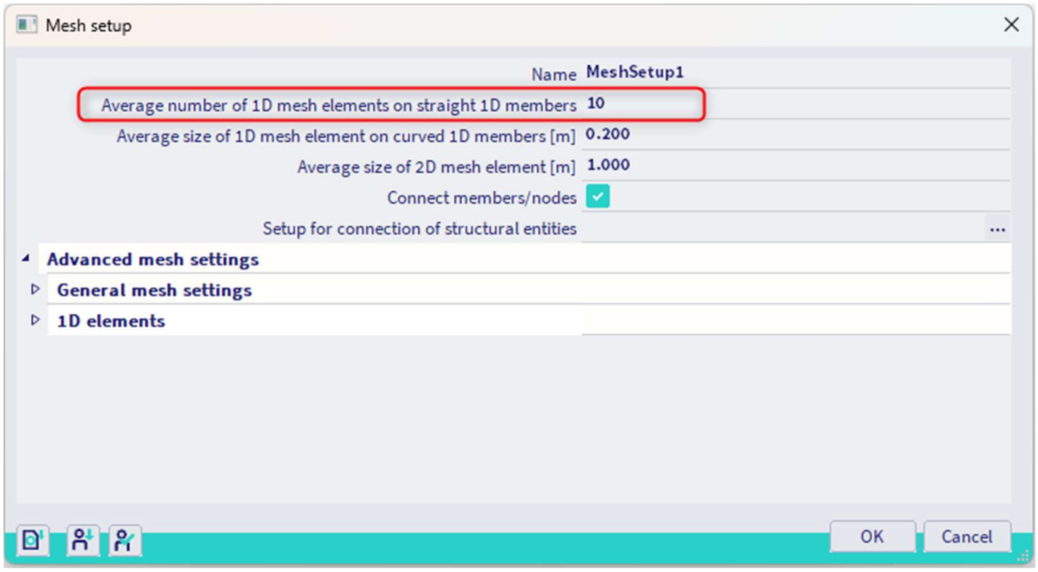
Step 4: mass matrix

Next, a Combination of Mass Groups can be created.



Step 5: mesh setup

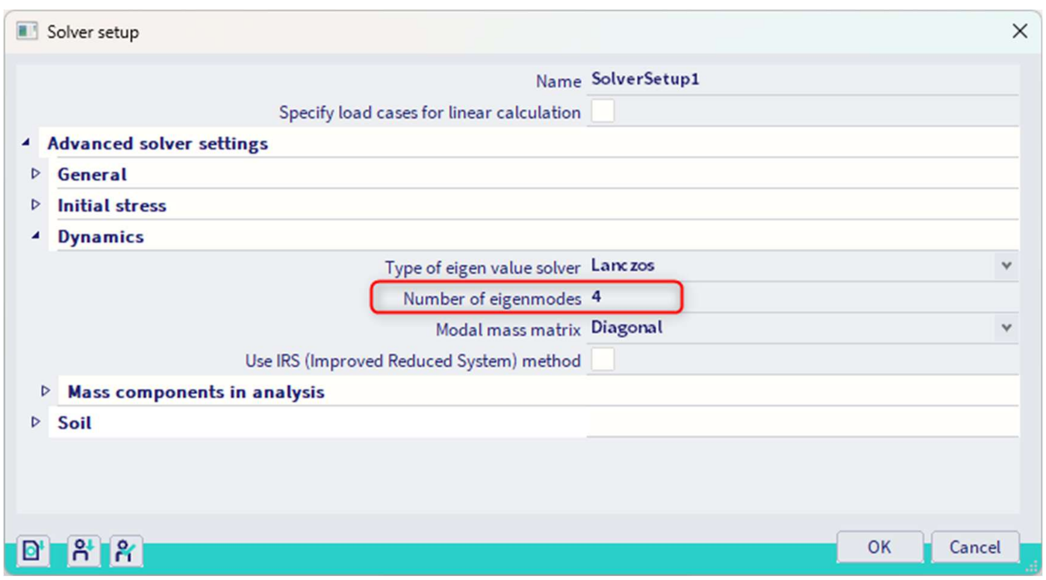
To obtain precise results for the dynamics calculation, the Finite Element Mesh is refined.
This can be done through Calculation & Mesh / Mesh Settings.



The Average number of tiles of 1D element is set to 10 to obtain a good distribution of the line masses and the mass of the members.

Step 6: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. The default value in the menu Tools / Calculation & Mesh / Solver Settings is 4. This is sufficient for this example.



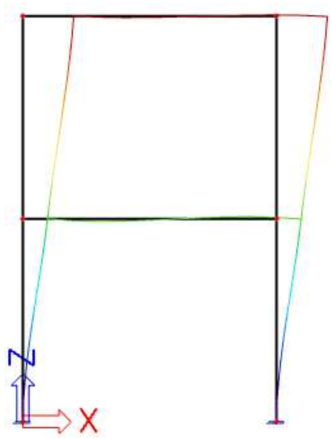
Step 7: modal analysis

The Free Vibration calculation can now be executed through the main menu Tools / Calculation & Mesh / Calculate. The following results are obtained:

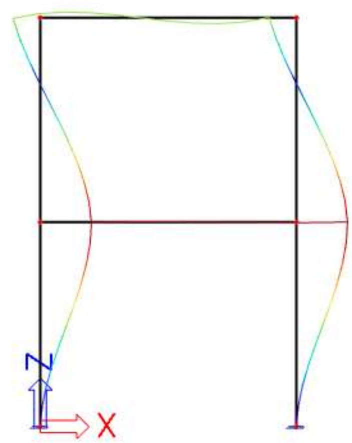
Eigen frequencies

N	f [Hz]	ω [1/s]	ω^2 [1/s²]	T [s]
Mass combination : CM1				
1	2.90	18.25	333.01	0.34
2	9.58	60.22	3626.53	0.10
3	14.64	91.99	8462.43	0.07
4	17.15	107.78	11615.85	0.06

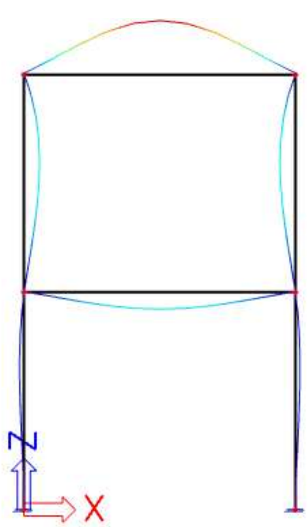
As stated in the previous example, using Deformation of Nodes, the Deformed Mesh can be shown to view the eigenmodes:



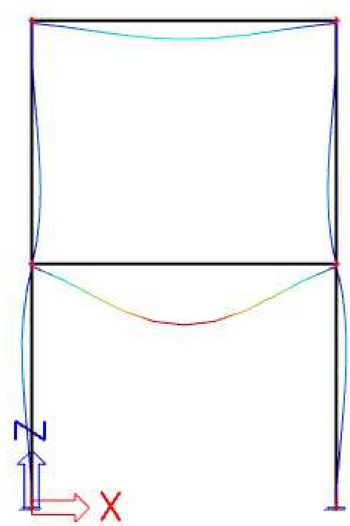
Eigenmode 1: f = 2,90Hz



Eigenmode 2: f = 9,58Hz



Eigenmode 3: f = 14,64Hz



Eigenmode 4: f = 17,15Hz

The **Calculation Protocol** for the **Eigen Frequency** calculation shows the following:

Sum of masses

	Mass type	X [kg]	Y [kg]	Z [kg]
CM1	Moving mass	6543.37	6567.49	6543.37
CM1	Total mass	6567.49	6567.49	6567.49

Relative modal masses

Mode	ıega [rad	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	$_{xi\_R}/W_{xtot}$	$_{yi\_R}/W_{ytot}$	$_{zi\_R}/W_{ztot}$
1	18.2491	0.34	2.90	75.1282	0.0000	0.0000	0.8626	0.0000	0.0000	0.0000	0.0805	0.0000
2	60.2224	0.10	9.58	-26.9398	0.0000	0.0000	0.1109	0.0000	0.0000	0.0000	0.4202	0.0000
3	91.9942	0.07	14.64	0.0000	0.0000	-31.6956	0.0000	0.0000	0.1535	0.0000	0.0000	0.0000
4	107.78	0.06	17.15	0.0000	0.0000	-59.3291	0.0000	0.0000	0.5379	0.0000	0.0000	0.0000
							0.9735	0.0000	0.6915	0.0000	0.5007	0.0000

The Sum of masses shows the amount of mass, which can vibrate for this Mass combination. In this example, this is governed by the line masses of 500 kg/m and the mass of the members.

This value can be calculated as follows:

- The members are of type HE240A with cross-section A= 0,00768 m²
- The volumetric mass of S 235 is 7850 kg/m³
- The total length of the members is 4 x 4 m + 2 x 5 m = 26 m

However, as stated in 2.2 the masses are guided to the mesh nodes. The Finite Element Mesh was refined to 10 1D elements per member.

This implies that for the two lower columns, half the mass of a 1D element is guided to a support and does not take part in the free vibration:

- The length of the columns is 4 m
- The length of a 1D element is de 4 m / 10 = 0,4 m
- The length of half a 1D element is 0,4 m / 2 = 0,2 m
    - The total length of the members taken into account for the mass is:
      26 m – 0,2 m – 0,2 m = 25,6 m
    - Total member mass = 0,00768 m² x 25,6 m x 7850 kg/m³ = 1543,37 kg

The mass is added to the line masses of 500 kg/m

- Vibrating mass = 2 x 500 kg/m x 5m + 1543,37 kg = **6543,37 kg**

The **Modal Participation Factors** show the amount of mass that is vibrating in a specific eigenmode as a percentage of the total mass.

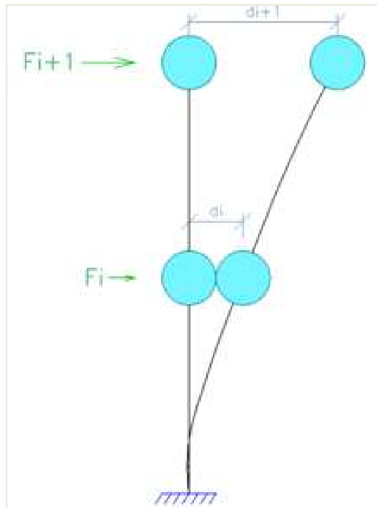
For Eigenmode 1: 86% of the total mass is vibrating in the X-direction
For Eigenmode 2: 11% of the total mass is vibrating in the X-direction
For Eigenmode 3: 16% of the total mass is vibrating in the Z-direction
For Eigenmode 4: 54% of the total mass is vibrating in the Z-direction

The lower row shows the total percentage when these four modes are combined: **97%** is taken into account for the X-direction and **69%** for the Z-direction.

These factors will be looked upon in more detail during the Seismic calculations in Chapter 4. For a seismic calculation, it is required that sufficient eigenmodes are included in the calculation so that at least 90% of the total mass is being taken into account [7].

Manual calculation

In order to check the results of SCIA Engineer, the lowest eigen frequency, or natural frequency of this structure is calculated by a manual calculation.
The method used here is described in the literature as Rayleigh's Energy Method. [1], [13].
In this method, the structure is idealized as a cantilever beam with lumped masses at each floor level:



The structure is then loaded with a set of linearly increasing horizontal loads on each floor level. Due to this loading, the structure will deform and thus the rigidity of the system is known. The eigen frequency of the structure can then be approximately calculated as follows:

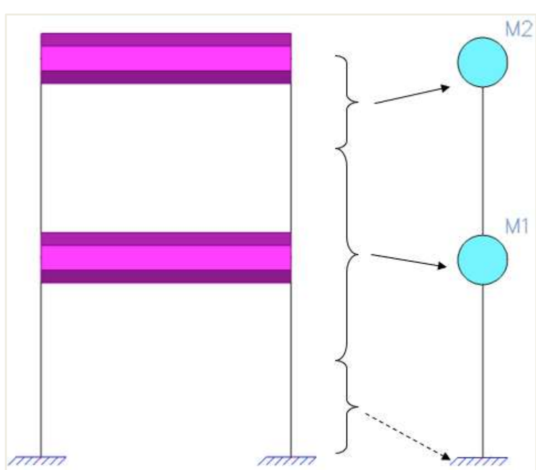
$$f = \frac{1}{2\pi} \cdot \sqrt{\frac{\sum_{i=1}^{n} F_i . d_i}{\sum_{i=1}^{n} M_i . d_i^2}} \tag{2.12}$$

With:

n: number of floors
F_i: horizontal force acting on floor level i
d_i: horizontal deformation of floor level i
M_i: idealized mass of floor level i

The analogy between this formula and $\omega = \sqrt{k/m}$ can clearly be seen.

To use this formula, the frame needs to be idealized to a cantilever:



The mass of the lower part of the structure is idealized to the support of the cantilever so it takes no part in the vibration.

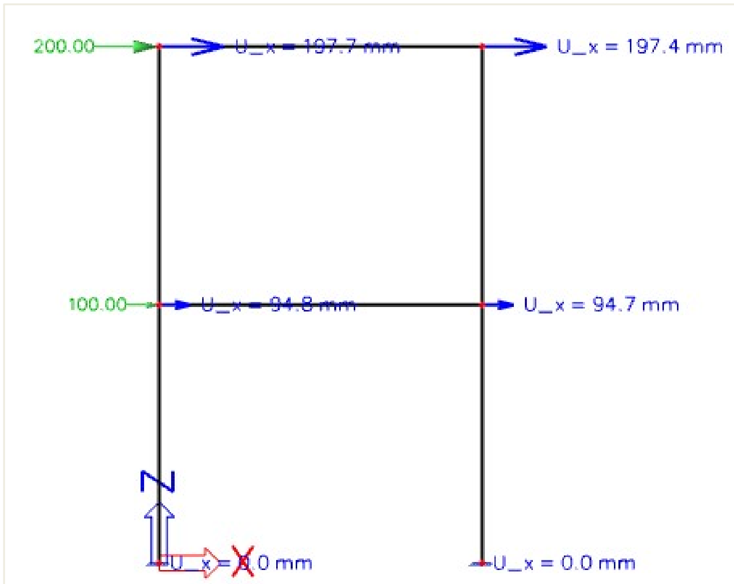
The mass M_1 can be calculated as follows:

0,00768 m² x (5 + 4 + 4) m x 7850 kg/m³ + 500 kg/m x 5 m = 3283,74 kg

The mass M_2 can be calculated as follows:

0,00768 m² x (5 + 4) m x 7850 kg/m³ + 50 kg/m x 5 m = 3042,5 9kg

In order to calculate the horizontal deformations d_i of each floor level due to a linearly increasing load F_i, a static load case is calculated with SCIA Engineer consisting of loads of 100 kN and 200 kN. The following results are obtained for the nodal deformations:



- F_1 = 100 kN = 100000 N => d_1 = 94,7 mm = 0,0947 m
- F_2 = 200 kN = 200000 N => d_2 = 197,4 mm = 0,1974 m

Applying formula (2.12):

$$f = \frac{1}{2\pi}.\sqrt{\frac{100000N * 0{,}0947m + 200000N * 0{,}1974m}{3283{,}74kg * (0{,}0947)^2 + 3042{,}59kg * (0{,}1974m)^2}} = 2{,}88 \text{ Hz}$$

This result corresponds to the 2,90 Hz calculated by SCIA Engineer.

2.4 Combining mass groups

Mass Groups are combined in a Combination of Mass Groups.

According to Eurocode 8 [7] article 3.2.4, all gravity loads appearing in the following combination of actions need to be taken into account for an eigenmode calculation:

$$\sum G_k + \sum \psi_{E,i} . Q_{k,i} \tag{2.13}$$

Where:

G_k: characteristic value of the permanent load
$Q_{k,j}$: characteristic value of the variable load i
$\psi_{E,i}$: combination coefficient for load $i = \varphi . \psi_{2,i}$

The combination coefficient $\psi_{E,i}$ takes into account the likelihood of the variable loads not being present over the entire structure during the occurrence of an earthquake.

According to Eurocode 8 [7] article 4.2.4, $\psi_{E,i}$ should be calculated in the following way:

$$\psi_{E,i} = \varphi . \psi_{2i}$$

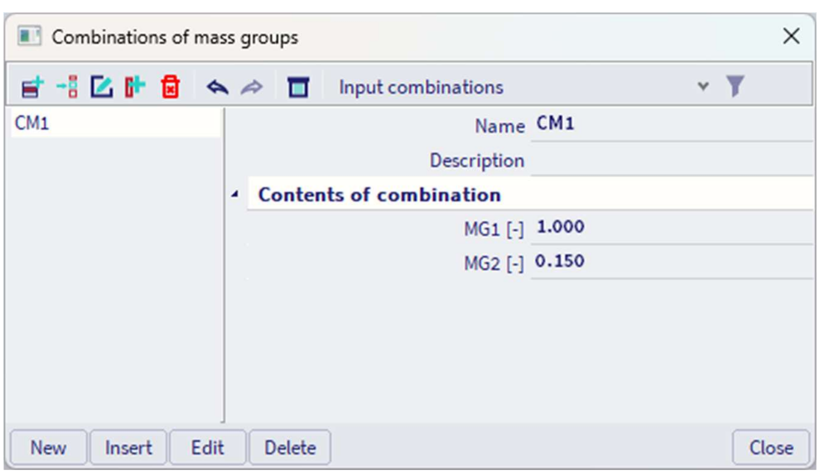
NOTE The values to be ascribed to φ for use in a country may be found in its National Annex. The recommended values for φ are listed in Table 4.2.

Table 4.2: Values of φ for calculating ψ_{Ei}

Type of variable action	Storey	φ
Categories A-C*	Roof	1,0
	Storeys with correlated occupancies	0,8
	Independently occupied storeys	0,5
Categories D-F* and Archives		1,0

* Categories as defined in EN 1991-1-1:2002.

For example, if a first mass group MG1 represents the mass of permanent loads and a second mass group MG2 represents the mass of a variable load case with a Category B imposed load and independently occupied storeys, then φ is taken as 0,5 and $\psi_{2,i}$ as 0,3.
This gives a value of 0,15 for $\psi_{E,i}$.
The Combination of Mass Groups CM1 can then be formulated as 1,00 MG1 + 0,15 MG2.

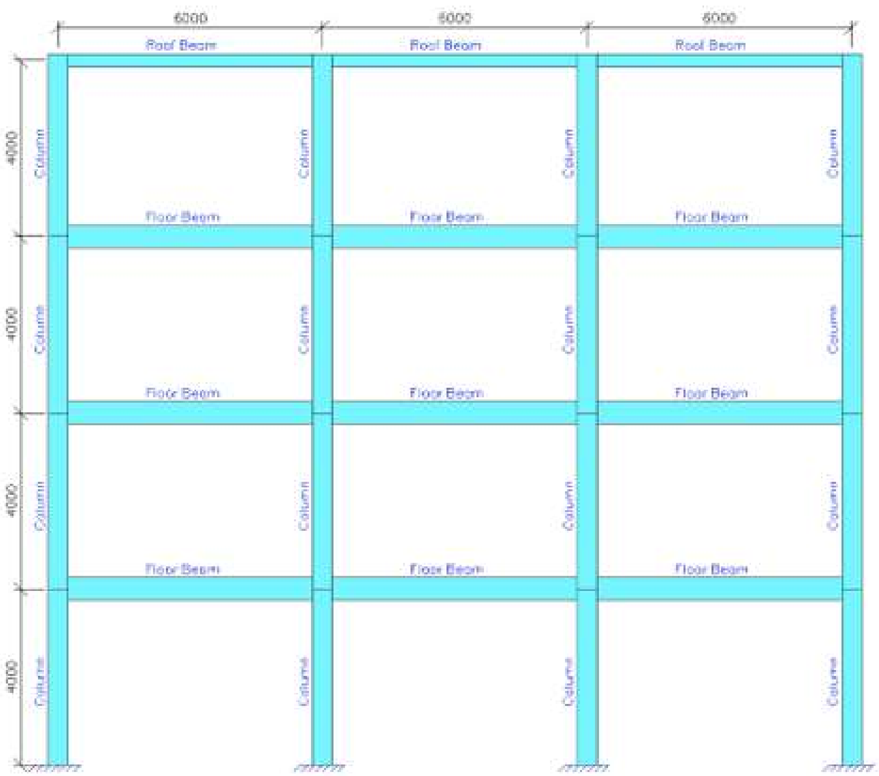


Example 02-3.esa

In this example an office building is modelled as a frame. The office is manufactured in C30/37 according to EC-EN. The building has four storeys with a storey height of 4 m. In horizontal direction, the frame is made up of four columns with a distance of 6 m between them. In the direction out of plane, the frames are spaced 5 m. The column bases are inputted as fixed supports.

The members of the frame have following cross-sections:
- Columns: Rectangular 300 x 450
- Floor Beams: Rectangular 250 x 500
- Roof Beams: Rectangular 150 x 300

The vertical loads acting on the structure are:
- The self-weight of the concrete members
- The weight of the floors: 5 kN/m²
- The weight of the roof: 2 kN/m²
- A category B (Office) imposed load of 3 kN/m²



This gives 3 static load cases:

- LC1: self-weight
- LC2: permanent load: 25 kN/m on the floor beams, 10 kN/m on the roof beams
- LC3: variable load: 15 kN/m on the floor beams

Step 1: functionality

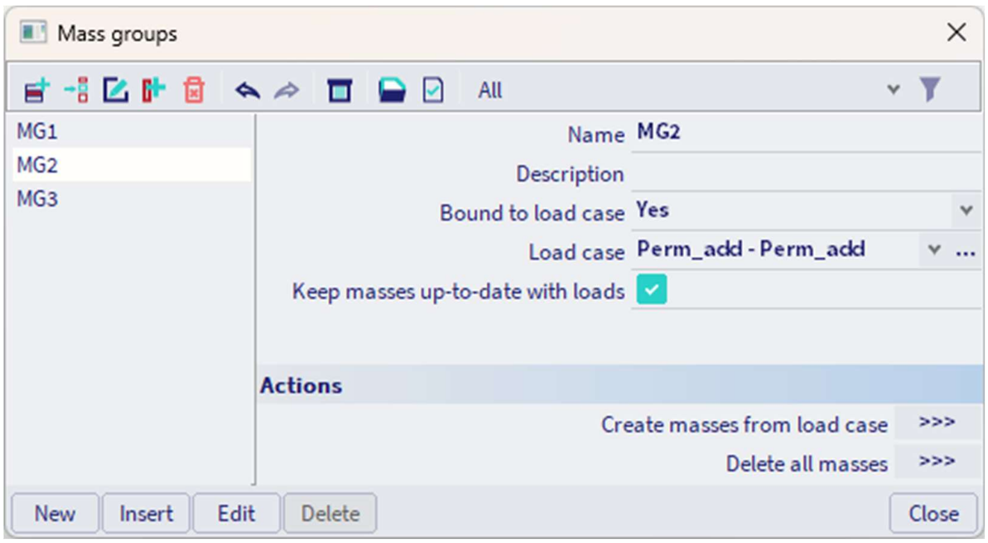
The first step in the Dynamic calculation is to activate the functionality Dynamics on the Functionality tab in the Project Data.

Step 2 & 3: mass groups

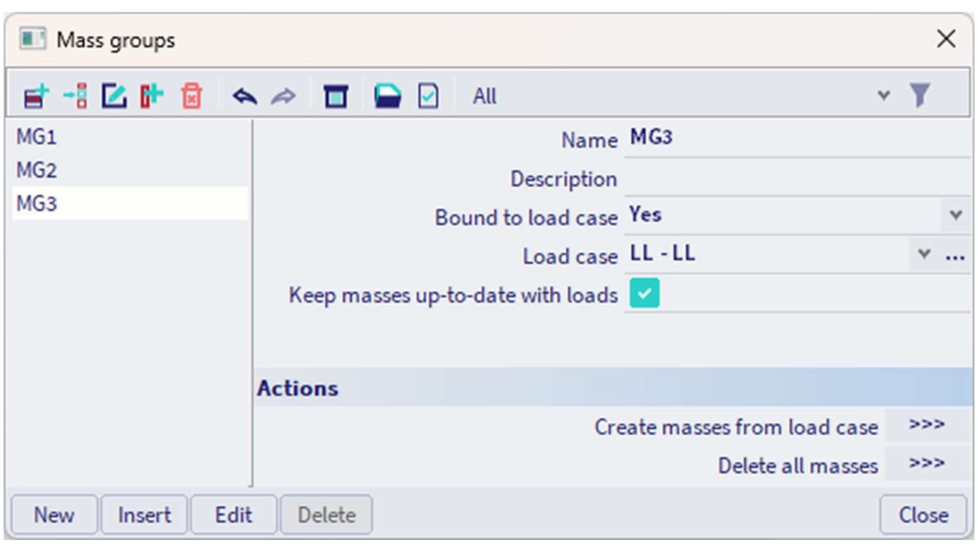
The second step is to create Mass Groups, the third step the creation of Masses.

Three Mass Groups are created, one for the dead load and one for each static load case.

For the Mass Group MG2, the load case LC2 is chosen: the weight of the floors and roof. Using the action "Create masses from load case", you can automatically generate masses from the already inputted loads which remain linked to the loads.



In the same way, the Mass Group MG2 is created in which masses are automatically created from load case LC3: the imposed load.



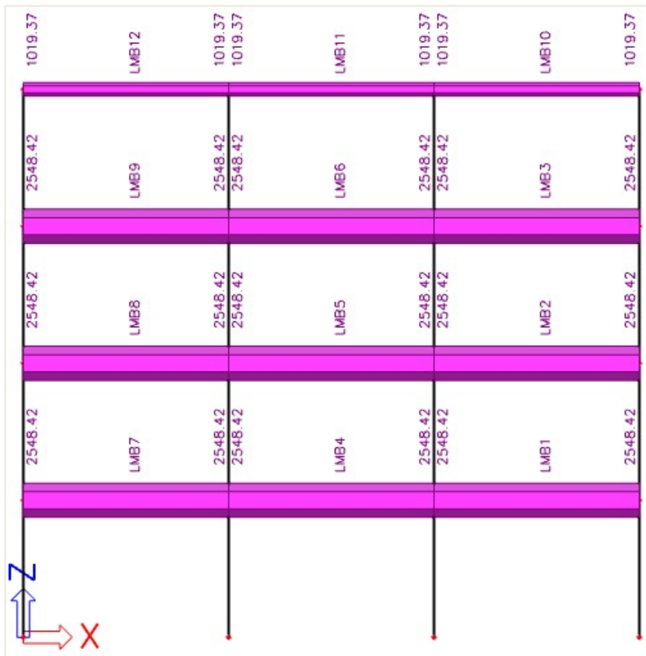
NB:

As stated in the first example: When creating masses from loads, SCIA Engineer will use the acceleration of gravity specified on the **Loads** tab of the **Project Data**. By default it is **9,81 m/s²**.

The mass, which has been created from a load case, can be automatically regenerated when the load case is modified. To link the mass to a load case, you have to activate the option **"Keep masses up-to-date with loads"**.

The contain of the two mass groups can be visualized.

Mass group MG2:



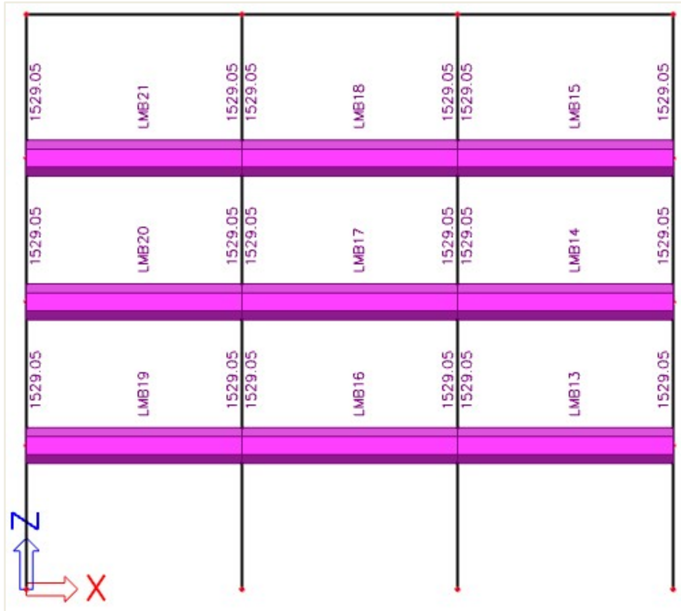
Floor mass:

$$\frac{25000\ ^{\mathrm{N}}/_{\mathrm{m}}}{9{,}81\ ^{\mathrm{m}}/_{\mathrm{s}^2}} = 2548{,}4\ ^{\mathrm{kg}}/_{\mathrm{m}}$$

Roof mass:

$$\frac{10000\ ^{\mathrm{N}}/_{\mathrm{m}}}{9{,}81\ ^{\mathrm{m}}/_{\mathrm{s}^2}} = 1019{,}4\ ^{\mathrm{kg}}/_{\mathrm{m}}$$

Mass groups MG3:



Mass of imposed load:

$$\frac{15000\ ^{\mathrm{N}}/_{\mathrm{m}}}{9{,}81\ ^{\mathrm{m}}/_{\mathrm{s}^2}} = 1529{,}1\ ^{\mathrm{kg}}/_{\mathrm{m}}$$

Step 4: mass matrix

Both Mass Groups can now be combined in a Combination of Mass Groups.
According to Eurocode 8 [7] article 3.2.4, all gravity loads appearing in the following combination of actions need to be taken into account for an eigenmode calculation:

$$\sum \mathrm{G_k} + \sum \psi_{\mathrm{E,i}} . \mathrm{Q_{k,i}} \tag{2.14}$$

With:

G_k: characteristic value of the permanent load

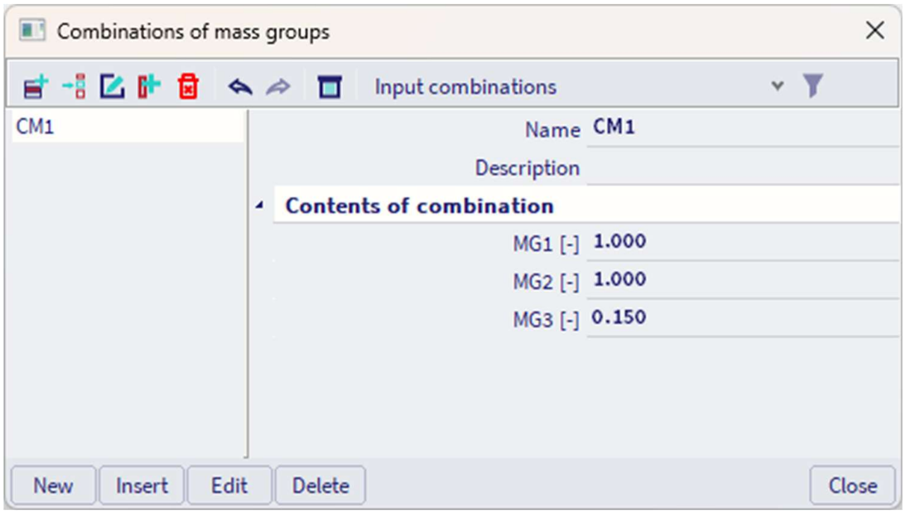
$Q_{k,j}$: characteristic value of the variable load i

$\psi_{E,i}$: combination coefficient for load $i = \varphi . \psi_{2,i}$

The combination coefficient $\psi_{E,i}$ consider the probability that variable loads may not be present on the whole structure when the earthquake happens.

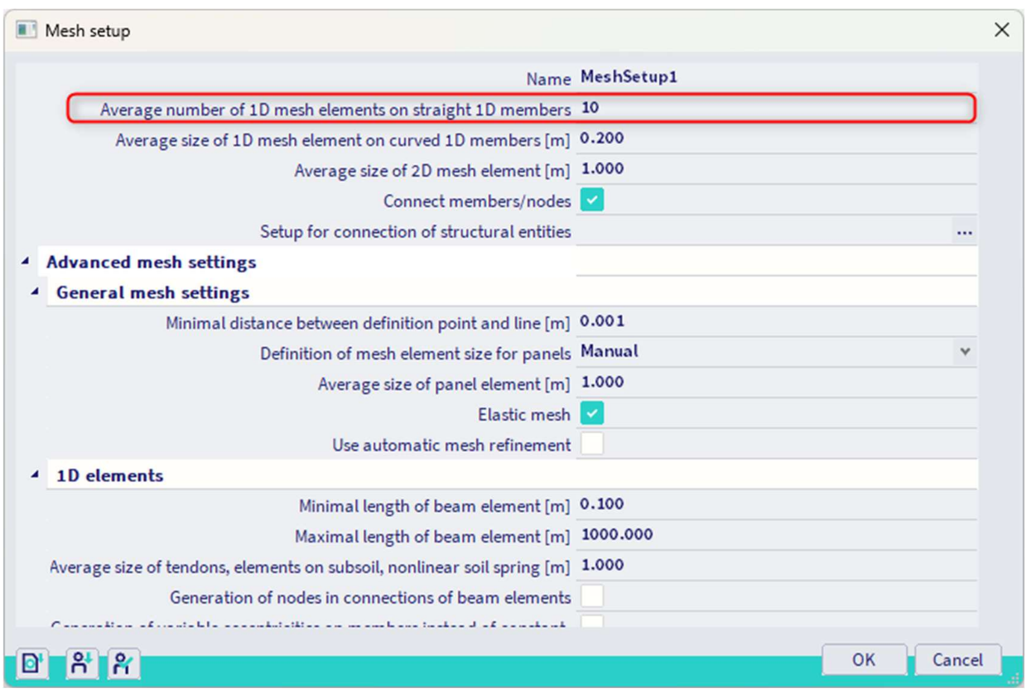
For this example, with a Category B imposed load and independently occupied storeys, φ is taken as 0,5 and $\psi_{2,i}$ as 0,3. This gives a value of 0,15 for $\psi_{E,i}$

The Combination of Mass Groups CM1 can then be formulated as 1,00 MG1 + 0,15 MG2.



Step 5: mesh setup

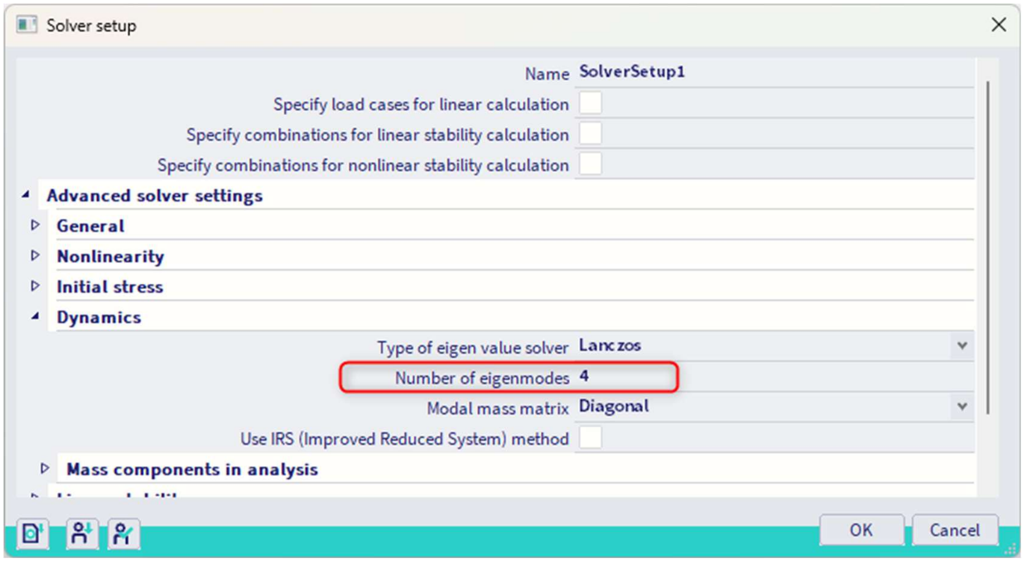
To obtain precise results for the dynamics calculation, the Finite Element Mesh is refined. This can be done through the main menu Tools / Calculation & Mesh / Mesh settings.



The Average number of tiles of 1D element is set to 10 to obtain a good distribution of the line masses and the mass of the members.

Step 6: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. The default value in the main menu Tools / Calculation & Mesh / Solver Settings is 4. This is sufficient for this example.



Step 7: calculation

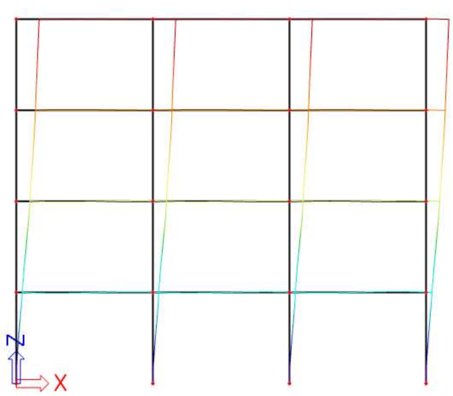
The Free Vibration calculation can now be executed through the main menu Tools / Calculation & Mesh / Calculate.

The following results are obtained:

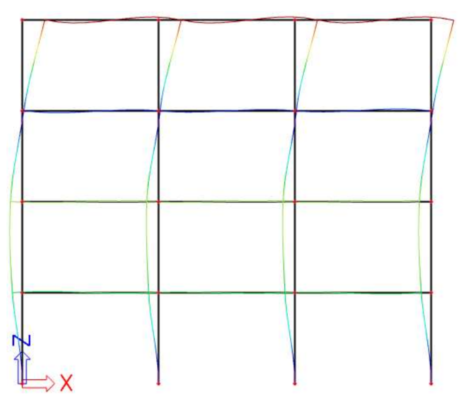
Eigen frequencies

N	f [Hz]	ω [1/s]	ω² [1/s²]	T [s]
Mass combination : CM1				
1	1.27	7.99	63.80	0.79
2	3.69	23.19	537.79	0.27
3	5.99	37.64	1417.01	0.17
4	8.23	51.73	2676.16	0.12

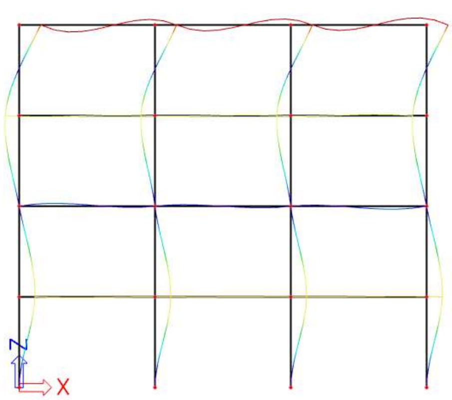
With corresponding eigenmodes:



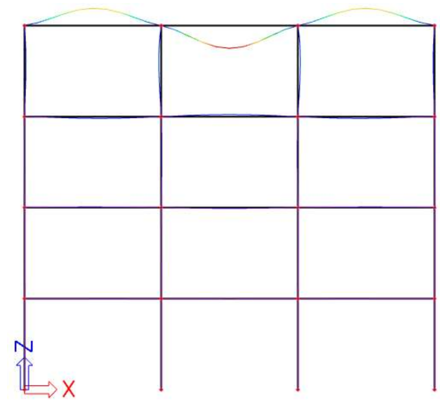
Eigenmode 1: f = 1,27 Hz



Eigenmode 2: f = 3,69 Hz



Eigenmode 3: f = 5,99 Hz



Eigenmode 4: f = 8,23 Hz

Step 8: calculation protocol

The Calculation Protocol for the Eigen Frequency calculation shows the following:

Sum of masses

	Mass type	X [kg]	Y [kg]	Z [kg]
CM1	Moving mass	208578.65	208848.65	208578.65
CM1	Total mass	208848.65	208848.65	208848.65

Relative modal masses

Mode	ιega [rad	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	$_{xi\_R}/W_{xtot}$	$_{yi\_R}/W_{ytot}$	$_{zi\_R}/W_{ztot}$
1	7.98794	0.79	1.27	417.2609	0.0000	0.0000	0.8347	0.0000	0.0000	0.0000	0.0549	0.0000
2	23.1909	0.27	3.69	-142.8630	0.0000	0.0000	0.0979	0.0000	0.0000	0.0000	0.2151	0.0000
3	37.6444	0.17	5.99	93.8642	0.0000	0.0000	0.0422	0.0000	0.0000	0.0000	0.0466	0.0000
4	51.7331	0.12	8.23	0.0000	0.0000	19.9525	0.0000	0.0000	0.0019	0.0000	0.0000	0.0000
							0.9748	0.0000	0.0019	0.0000	0.3166	0.0000

The Sum of masses can be calculated as follows:

- According to the Bill of Material, the self-weight of the frame, is 40500 kg:

Bill of material
Selection: All

Material	Mass [kg]	Surface [m²]	Volume [m³]
Concrete	40500.00	193.200	1.6200e+01
Total	40500.00	193.200	1.6200e+01

However, for the four lower columns, half the mass of a 1D element is guided to a support and does not take part in the free vibration.

- The length of the columns is 4 m
- Since 10 1D elements per member were used, the length of a 1D element is 4 m / 10 = 0,4 m
- The length of half a 1D element is 0,4 m / 2 = 0,2 m

- The columns have a cross-section of 0,135 m² and a volumetric masse of 2500 kg/m³
    - The mass of the columns not taken into account is:
    $$4 \times 0{,}135 \text{ m}^2 \times 0{,}2 \text{ m} \times 2500 \text{ kg/m}^3 = 270 \text{ kg}$$
    - The mass of the self-weight taken into account is: 40500 kg – 270 kg = 40230 kg

- For MG1 the mass of the floors is 9 x 2548,42 kg/m x 6 m = 137614,68 kg

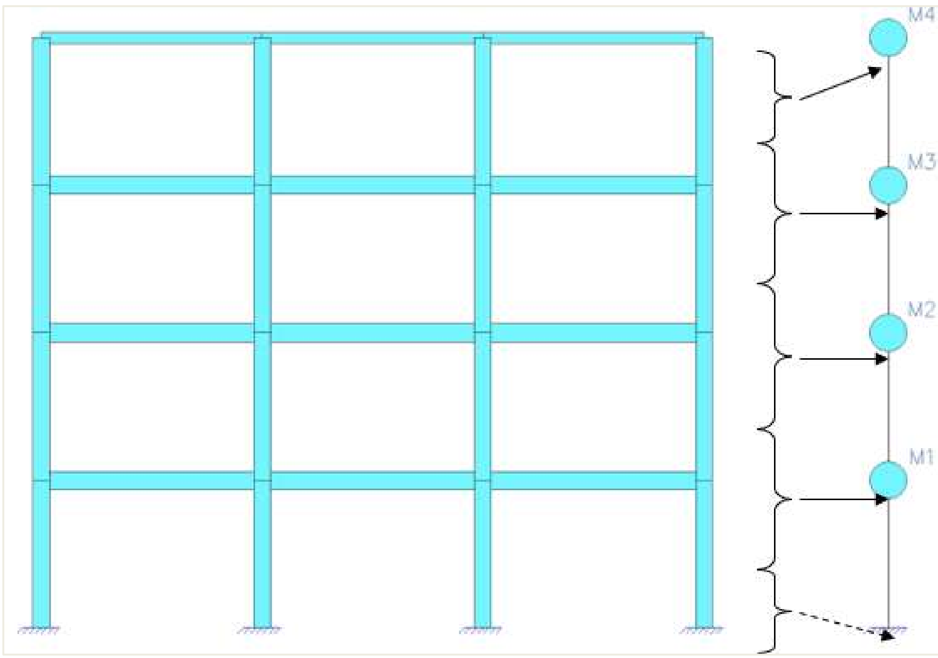
- For MG1 the mass of the roof is 3 x 1019,37 kg/m x 6 m = 18348,66 kg

- For MG2 the mass of the floors is 9 x 1529,05 kg/m x 6 m = 82568,7 kg
  However only 15% was taken into account => 0,15 x 82568,7 kg = 12385,31 kg

- Vibrating mass = 40230 kg + 137614,68 kg + 18348,66 kg + 12385,31 kg
  = 208578,65 kg

Manual calculation

In order to check the results of SCIA Engineer, the lowest eigen frequency of this structure is calculated by means of the Rayleigh Method.
As specified in the previous example, the frame is idealized as a cantilever:



The masses M_1, M_2 and M_3 can be calculated as follows:

- Self-weight of the three floor beams and four columns:
  - 3 x 0,125 m² x 2500 kg/m³ x 6 m = 5625 kg
  - 4 x 0,135 m² x 2500 kg/m³ x 4 m = 5400 kg
  - 5625 kg + 5400 kg = 11025 kg

- Floor weight of mass group MG1:
  - 3 x 2548,42 kg/m x 6 m = 45871,56 kg

- Weight of imposed load of mass group MG2 (15%)
  - 0,15 x 3 x 1529,05 kg/m x 6 m = 4128,44 kg

- Total: 11025 kg + 45871,56 kg + 4128,44 kg = **61024,995 kg**

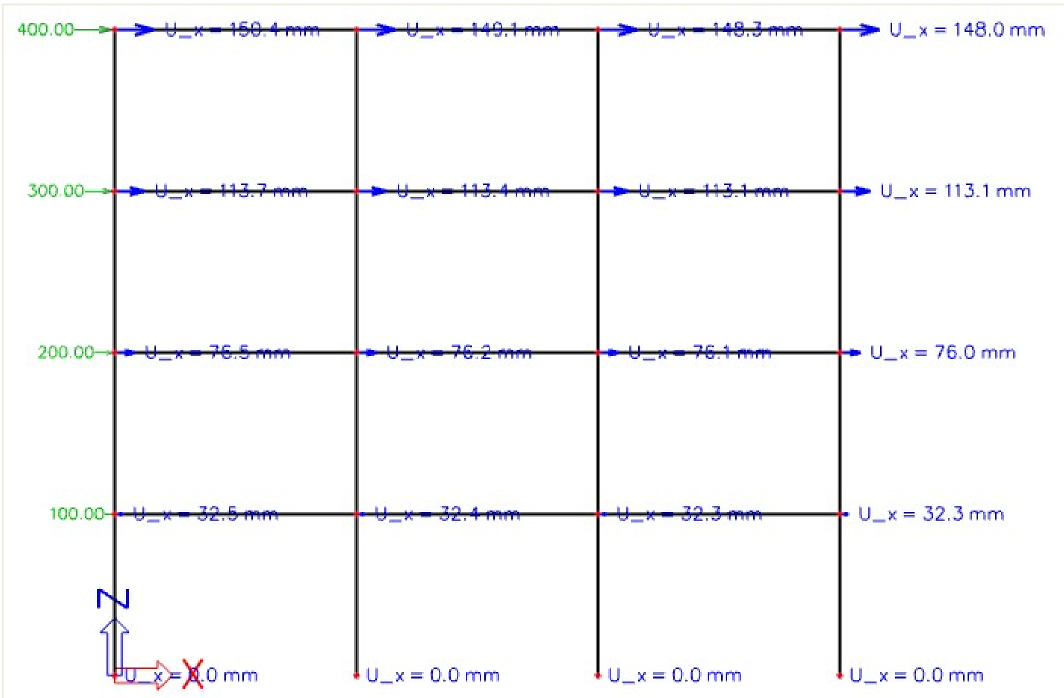
The mass M_4 can be calculated as follows:

- Self-weight of three roof beams and half of four columns:
  - 3 x 0,045 m² x 2500 kg/m³ x 6m = 2025 kg
  - 0,5 x 4 x 0,135 m² x 2500 kg/m³ x 4m = 2700 kg
  - 2025 kg + 2700 kg = 4725 kg

- Roof weight of mass group MG1:
    - 3 x 1019,37 kg/m x 6 m = 18348,66 kg

- Total: 4725 kg + 18348,66 kg = **23073,66 kg**

In order to calculate the horizontal deformations d_i of each floor level due to a linearly increasing load F_i, a static load case is calculated with SCIA Engineer consisting of loads of **100 kN, 200 kN, 300 kN** and **400 kN**. The following results are obtained for the nodal deformations:



- F_1 = 100 kN = 100000 N => d_1 = 32,3 mm = 0,0323 m
- F_2 = 200 kN = 200000 N => d_2 = 76,0 mm = 0,0760 m
- F_2 = 300 kN = 300000 N => d_2 = 113,1 mm = 0,1131 m
- F_2 = 400 kN = 400000 N => d_2 = 148,0 mm = 0,1480 m

Applying formula (2.12):

$$f = \frac{1}{2\pi} \cdot \sqrt{\frac{100000\mathrm{N} * 0{,}0323\mathrm{m} + 200000\mathrm{N} * 0{,}076\mathrm{m} + 300000\mathrm{N} * 0{,}1131\mathrm{m} + 400000\mathrm{N} * 0{,}148\mathrm{m}}{61024{,}99\mathrm{kg} \cdot (0{,}032)^2 + 61024{,}99\mathrm{kg} \cdot (0{,}076\mathrm{m})^2 + 61024{,}99\mathrm{kg} \cdot (0;113\mathrm{m})^2 + 23073{,}66\mathrm{kg} \cdot (0{,}148\mathrm{m})^2}}$$

f = 1,27 Hz

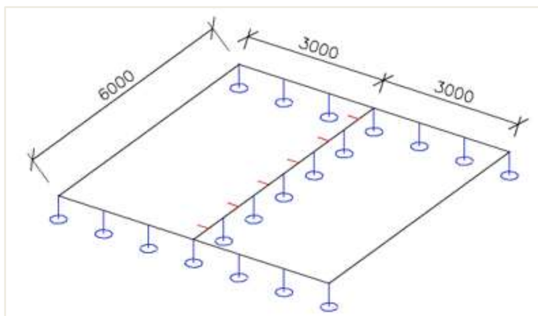
This result corresponds to the 1,27 Hz calculated by SCIA Engineer.

2.5 Slabs

This paragraph illustrates the procedure for the Free Vibration calculation of slabs. The applied method is entirely the same as for frames. This is shown in the following example.

Example 02-4.esa

In this example, a multi-span rectangular slab is modelled. The slab has a length and width of **6 m**. The slab has a thickness of **0,06 m** and is manufactured in **S 235** according to **EC-EN**. On two sides the slab is simply supported, on the other two, the slab is free. In the middle of the slab, perpendicular on both simply supported edges, a line support is introduced. One static load case is created: the **self-weight** of the slab.

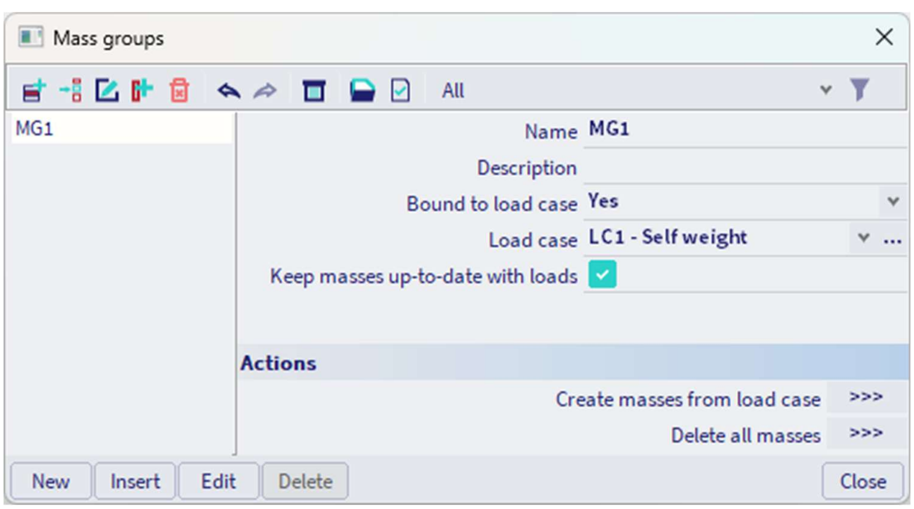


Step 1: functionality

The first step in the Dynamic calculation is to activate the functionality **Dynamics** on the **Functionality** tab in the **Project Data**.

Step 2 & 3: mass group

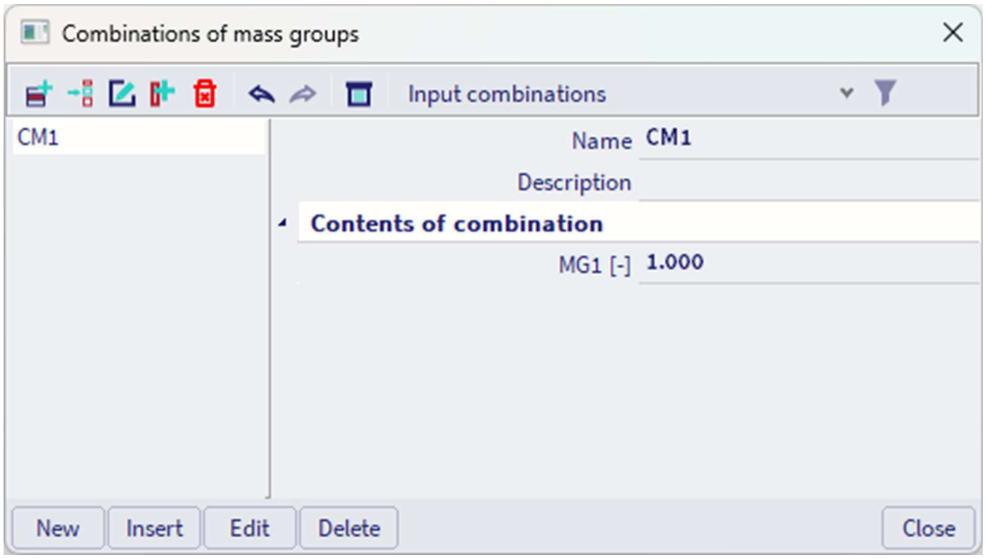
The second step is to create a **Mass Group**



Since the Free Vibration calculation will be executed for the self-weight of the slab, no additional masses need to be inputted.

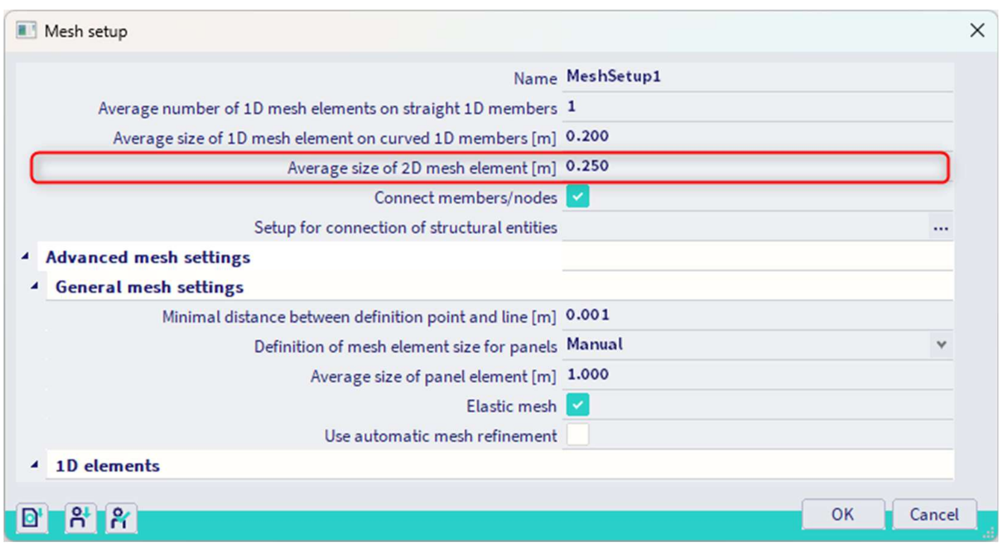
Step 4: mass matrix

Next, a Combination of Mass Groups can be created.



Step 5: mesh setup

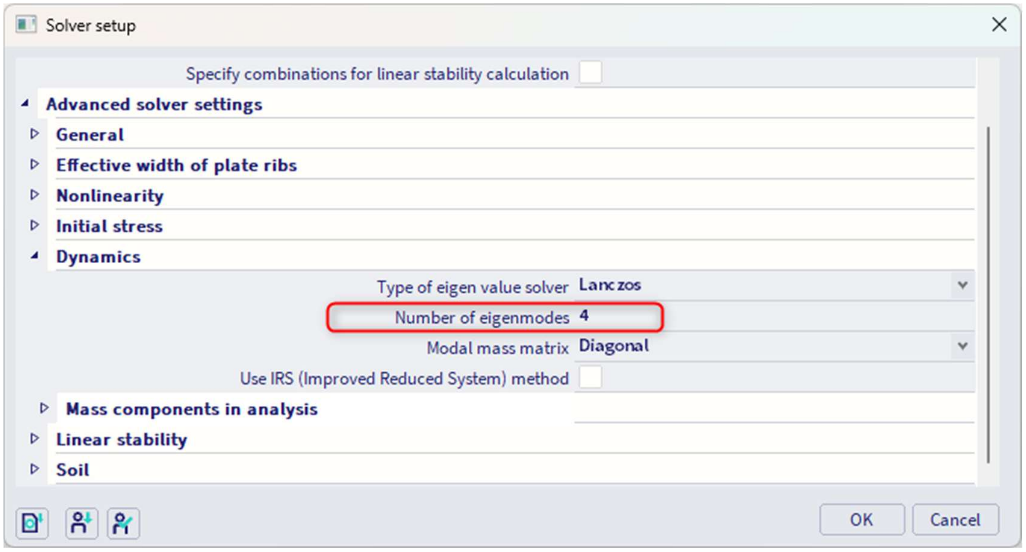
To obtain precise results for the dynamics calculation, the Finite Element Mesh is refined. Analogous as for frames, this can be done through the main menu Tools / Calculation & Mesh / Mesh settings.



The Average size of 2D elements is set to 0,25 m.

Step 6: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. The default value in the main menu Tools / Calculation & Mesh / Solver settings is 4. This is sufficient for this example.



Step 7: modal analysis

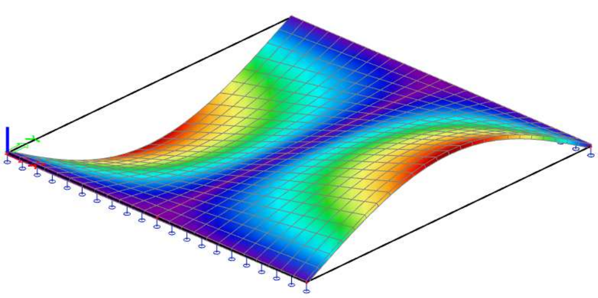
The Free Vibration calculation can now be executed through the main menu Tools / Calculation & Mesh / Calculate.

The following results are obtained:

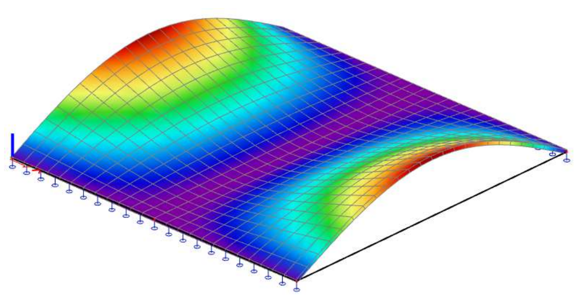
Eigen frequencies

N	f [Hz]	ω [1/s]	ω² [1/s²]	T [s]
Mass combination : CM1				
1	6.68	41.94	1759.34	0.15
2	9.43	59.24	3509.65	0.11
3	19.37	121.72	14815.37	0.05
4	21.00	131.96	17412.72	0.05

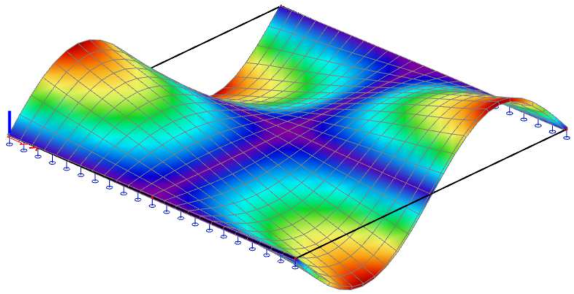
The same way as for frames, the Eigenmodes can be visualized through Deformation of nodes now under 2D Members. The Deformed structure for value Uz shows the following:



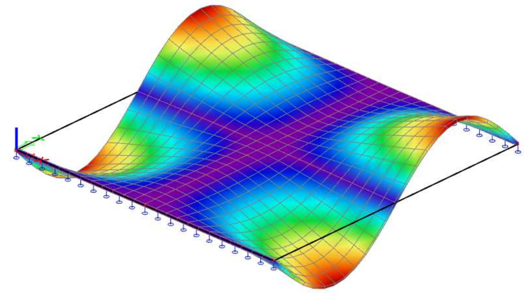
Eigenmode 1: f = 6,68Hz



Eigenmode 2: f = 9,43Hz



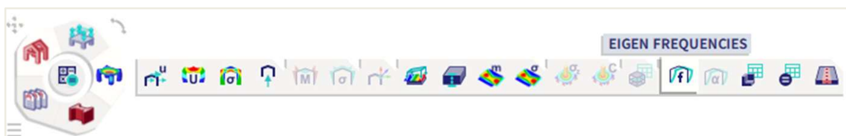
Eigenmode 3: f = 19,37Hz

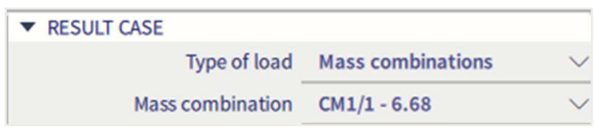


Eigenmode 4: f = 21,00Hz

NB:

- *With the option* Displacement 3D, *you display the deformation of both 2D and 1D elements. This allows seeing the complete eigenmode for a structure containing both element types i.e. General XYZ projects.*
- *To generate all eigenmodes quickly, this document can be used: the picture of one eigenmode can be set as a nested table for the Combinations of Mass Groups :*





This way, all eigenmodes are generated automatically.

Step 8: calculation protocol

The Calculation Protocol for the Eigen Frequency calculation shows that the following "Sum of masses" is accounted for:

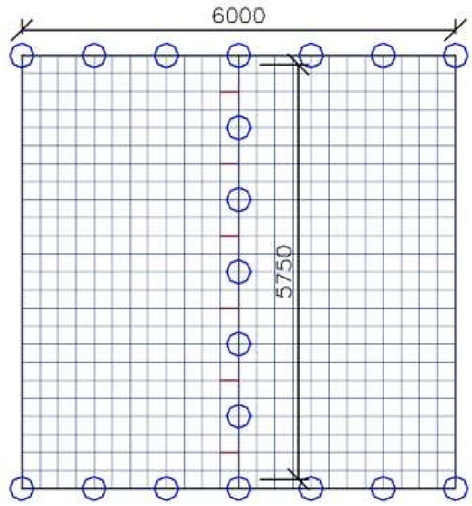
Sum of masses

	Mass type	X [kg]	Y [kg]	Z [kg]
CM1	Moving mass	16956.00	16956.00	15572.44
CM1	Total mass	16956.00	16956.00	16956.00

This value can be calculated as follows:

- The total weight of the slabs is 6 m x 6 m x 0,06 m x 7850 kg/m³ = 16956 kg
- Half of the mass of the elements near the two externally supported edges is carried to the supports and does not participate in the vibration. Since the mesh size was set to 0,25 m, half the size of a 2D element is 0,125 m.
    - 2 x 6 m x 0,125 m x 0,06 m x 7850 kg/m³ = 706,5 kg

- The same applies for the internal edge, however the mass of the two elements on the start and end nodes has already been taken into account in the above calculation for the externally supported edges. This leaves a length of 6 m – 0,125 m – 0,125 m = 5,75 m.
  The following figure illustrates this length:
    - 2 x 5,75 m x 0,125 m x 0,06 m x 7850 kg/m³ = 677,06 kg



- The total mass taken into account for the « Free Vibration » calculation is:
    - 16956 kg – 706,5 – 677,06 = **15572,44 kg**

Manual calculation

In order to check the results of SCIA Engineer, the eigen frequencies of the slab are calculated by a manual calculation. The method used here is described in reference [14] In this reference; the eigen frequency of a multi-span slab is expressed in terms of a non-dimensional parameter:

$$\lambda = \frac{\omega L^2}{\pi^2} . \sqrt{\frac{\rho h}{D}}$$

(2.15)

Where:

ω: circular frequency
L: distance between the two simply supported external edges
ρ: density of the slab material
h: slab thickness
D: flexural rigidity of the slab

$$D = \frac{Eh^3}{12.(1-\upsilon^2)}$$

(2.16)

E: modulus of Young
υ: Poisson's ratio

In this example, the material properties are the following:

L = 6 m
ρ = 7850 kg/m^3
h = 0,06 m
E = 210000 N/mm² = 2,1.e^{11} N/m²
υ = 0,3

$$D = \frac{\left(2{,}1e^{11}\, {}^{N}/_{m^2}\right) * (0{,}06m)^3}{12 * (1 - 0{,}3^2)} = 4153846{,}15N.m$$

The values for λ for the first four modes, for a slab with two edges simply supported and two edges free, a h/L ratio of 0,01 and an internal edge on position 0,5L are given in reference [14]:

Mode 1: λ = 1,6309
Mode 2: λ = 2,3050
Mode 3: λ = 4,7253
Mode 4: λ = 5,1271

Using these parameters in formula (2.15), the circular frequencies can be calculated:

Mode 1: ω = 41,99rad/s => f = 6,68Hz
Mode 2: ω = 59,34rad/s => f = 9,45Hz
Mode 3: ω = 121,66rad/s => f = 19,36Hz
Mode 4: ω = 132rad/s => f = 21,01Hz

The results correspond perfectly to the results calculated by SCIA Engineer:

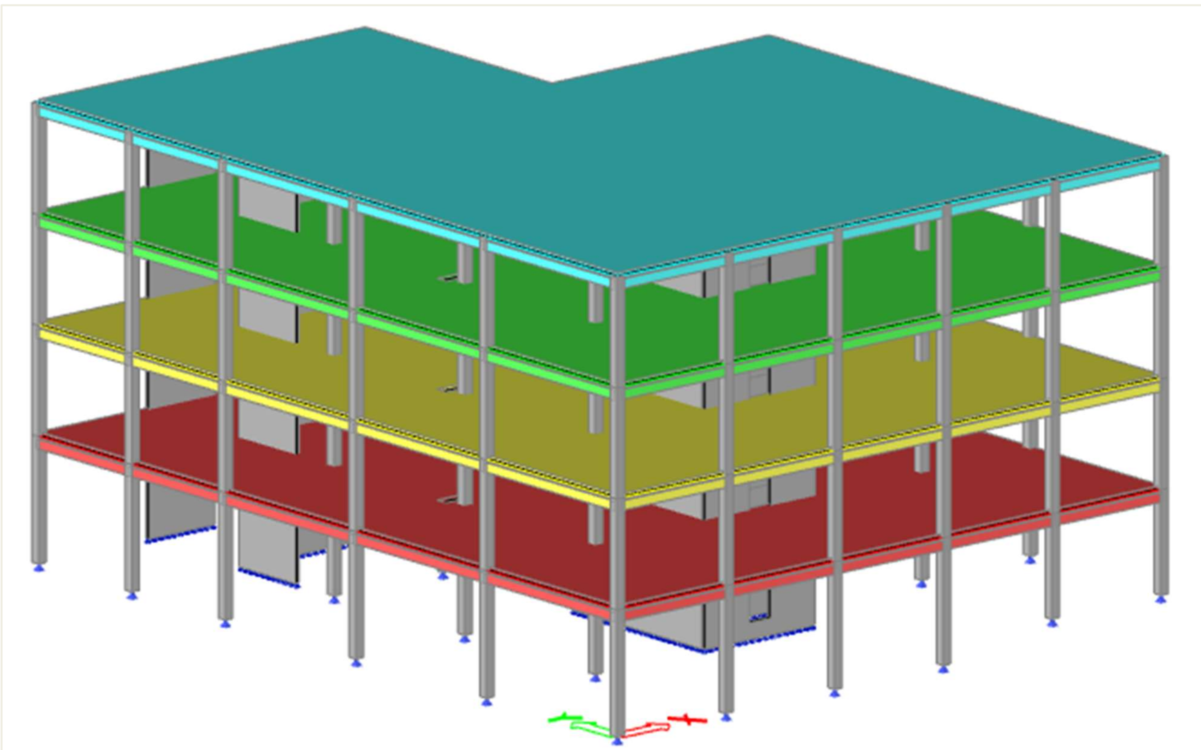
Mode 1: f = 6,68Hz
Mode 2: f = 9,44Hz
Mode 3: f = 19,39Hz
Mode 4: f = 21,05Hz

2.6 Floor of a concrete building

The last paragraph of this chapter illustrates the procedure for the Free Vibration calculation of a floor of a multi-storey concrete building. This is shown in the following example.

Example 02-5.esa

In this example, a multi-storey concrete building was modeled. It consists of a first floor, three upper floors and a flat roof.



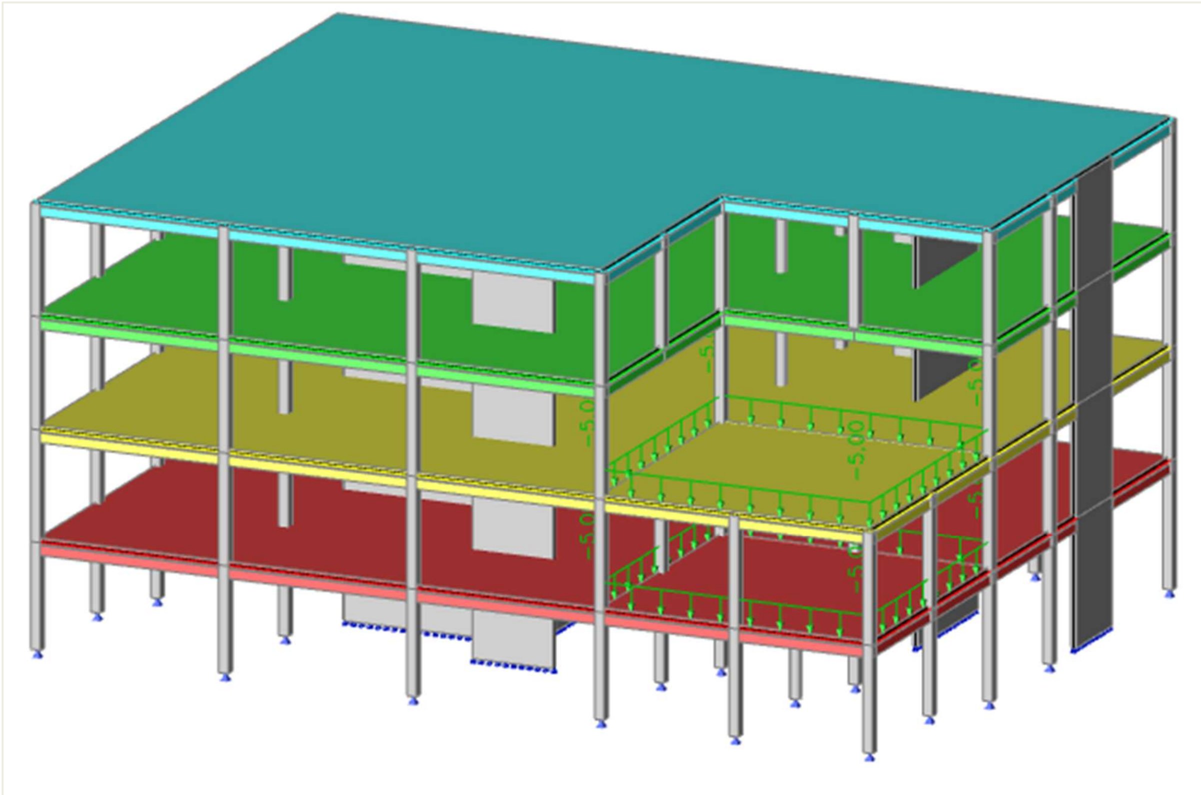
Several load cases are created :

- Self-weight.

- Other permanent loads, which are -2,00kN/m² on all floors of all storeys and -1,50kN/m² on the roof slab.

- Live loads :
  - On all floors of all storeys, in the office areas (cat B) : -2,50kN/m²
  - On the office area of storeys 1 and 2 (cat C): -5,00kN/m²
  - On the roof slab (cat H) : -0,6kN/m²

The two office are located on storeys 1 and 2, and are in the corner non present on the floor 3:



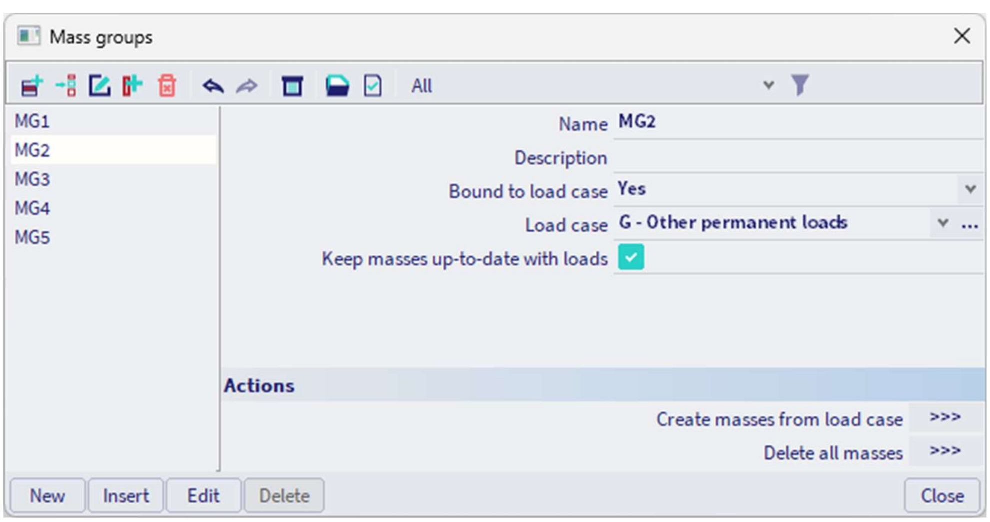
The study will focus on the small office floor of the 2nd floor (in yellow on the images above).

Step 1 : functionality

The first step in dynamic calculation is to activate the "Dynamic" functionality in the "Functionality" tab of the "Project settings".

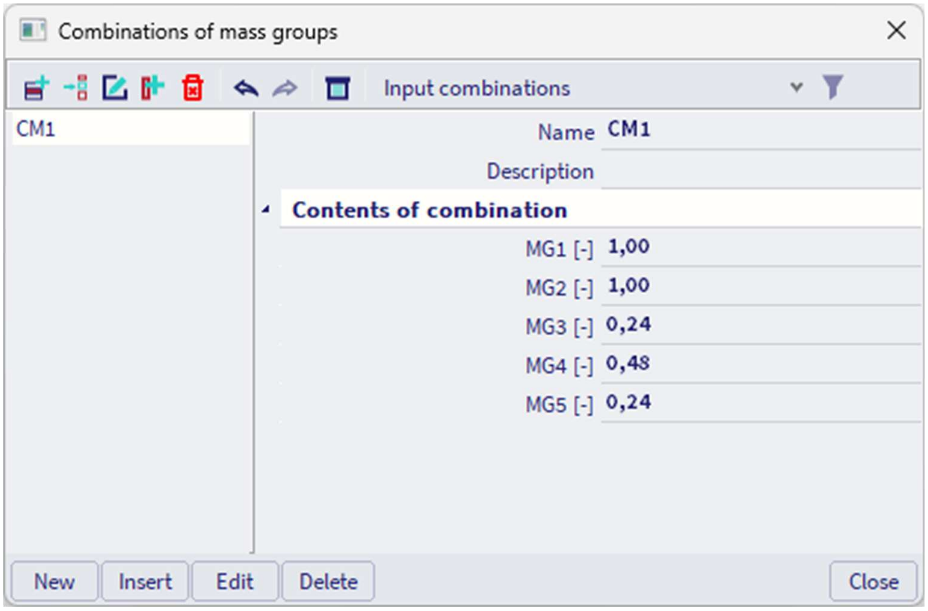
Step 2 & 3 : mass groups

Then, you have to create the "Mass groups" corresponding to the 5 local cases.



Step 4 : mass matrix

A « combination of mass groups » has to be now created.



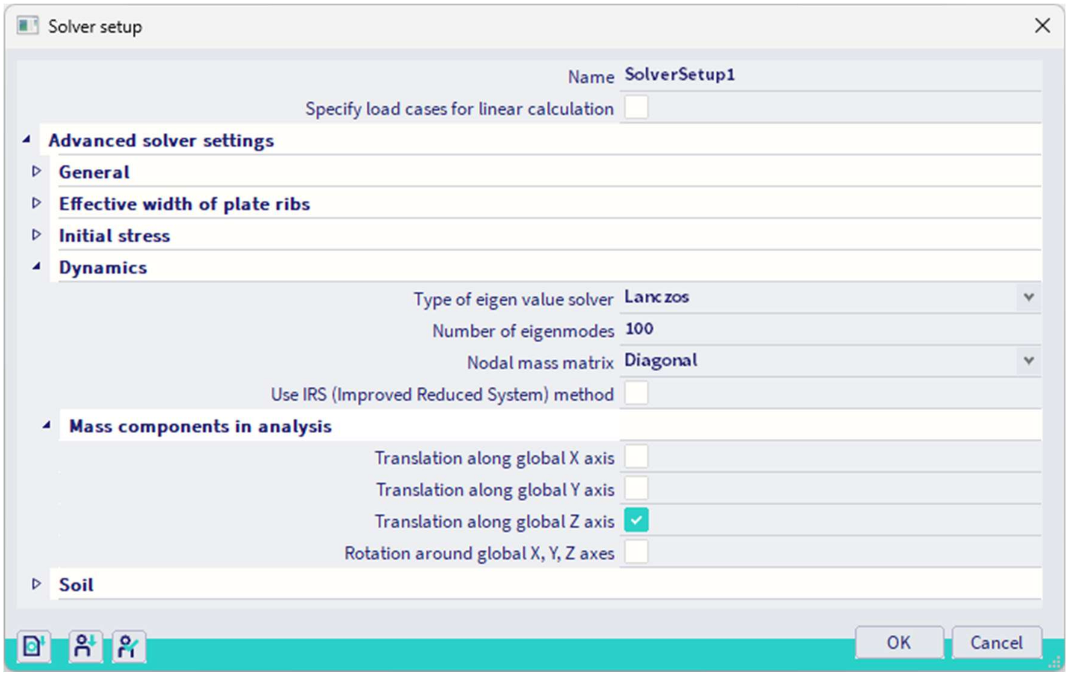
Step 5 : mesh setup

To obtain accurate results in the case of dynamic analysis, the mesh must be refined.
As before, this can be achieved via the menu Tools / Calculation and mesh / Mesh setup.

The average size of 2D elements is taken equal to 0,50m.

Step 6 : solver setup

Before starting the analysis, you must define the number of eigenmodes to be calculated via the menu Tools / Calculation and Mesh / Solver setup. Here, the number of eigenmodes is defined to 100.



Since we are looking for the vibration mode of a floor, therefore a vertical vibration, we can click on "Translation along global Z axis" only in the "Mass components in analysis".

Step 7 : modal analysis

The eigenfrequency calculation can be run via the menu Tools / Calculation and Mesh / Calculate.

The « Calculation protocol » for the calculation of "Eigenfrequencies » shows that the sum of the following masses is taken into account :

Sum of masses

	Mass type	X [kg]	Y [kg]	Z [kg]
CM1	Moving mass	0,0	0,0	2894714,4
CM1	Total mass	0,0	0,0	2904573,2

The following results are obtained :

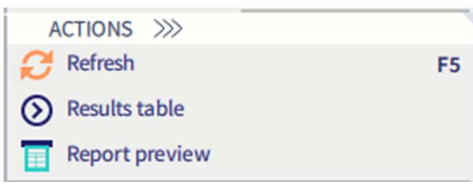
Relative modal masses

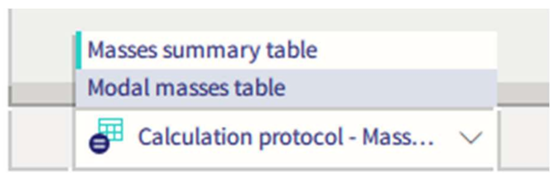
Mode	ega [rad	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	$_{xi\_R}/W_{xtot}$	$_{yi\_R}/W_{ytot}$	$_{zi\_R}/W_{ztot}$
1	41.9674	0,15	6,68	0,0000	0,0000	266,3834	0,0000	0,0000	0,0245	0,0386	0,0328	0,0000
2	54.3134	0,12	8,64	0,0000	0,0000	-668,7291	0,0000	0,0000	0,1545	0,3078	0,0032	0,0000
3	55.262	0,11	8,80	0,0000	0,0000	641,6486	0,0000	0,0000	0,1422	0,0007	0,1484	0,0000
4	55.8229	0,11	8,88	0,0000	0,0000	-262,8334	0,0000	0,0000	0,0239	0,0088	0,1598	0,0000
5	57.3681	0,11	9,13	0,0000	0,0000	-418,6475	0,0000	0,0000	0,0605	0,0547	0,0016	0,0000
6	57.555	0,11	9,16	0,0000	0,0000	-94,5695	0,0000	0,0000	0,0031	0,0000	0,0002	0,0000
7	58.7545	0,11	9,35	0,0000	0,0000	-70,7243	0,0000	0,0000	0,0017	0,0041	0,0000	0,0000
8	59.8468	0,10	9,52	0,0000	0,0000	78,6799	0,0000	0,0000	0,0021	0,0000	0,0084	0,0000
9	60.0128	0,10	9,55	0,0000	0,0000	-1,2636	0,0000	0,0000	0,0000	0,0011	0,0005	0,0000
10	60.2342	0,10	9,59	0,0000	0,0000	-181,9411	0,0000	0,0000	0,0114	0,0156	0,0172	0,0000
11	61.1156	0,10	9,73	0,0000	0,0000	14,5231	0,0000	0,0000	0,0001	0,0007	0,0011	0,0000
12	62.2596	0,10	9,91	0,0000	0,0000	-25,7164	0,0000	0,0000	0,0002	0,0000	0,0005	0,0000
13	63.3424	0,10	10,08	0,0000	0,0000	-424,9598	0,0000	0,0000	0,0624	0,0009	0,0873	0,0000
14	63.8622	0,10	10,16	0,0000	0,0000	194,6623	0,0000	0,0000	0,0131	0,0040	0,0474	0,0000
15	64.3958	0,10	10,25	0,0000	0,0000	-22,2598	0,0000	0,0000	0,0002	0,0010	0,0001	0,0000
16	64.6661	0,10	10,29	0,0000	0,0000	-51,9993	0,0000	0,0000	0,0009	0,0003	0,0005	0,0000
17	66.0166	0,10	10,51	0,0000	0,0000	-51,8242	0,0000	0,0000	0,0009	0,0000	0,0002	0,0000
18	67.1947	0,09	10,69	0,0000	0,0000	-149,3491	0,0000	0,0000	0,0077	0,0005	0,0067	0,0000
19	68.5599	0,09	10,91	0,0000	0,0000	-129,1487	0,0000	0,0000	0,0058	0,0041	0,0024	0,0000
20	71.0073	0,09	11,30	0,0000	0,0000	-37,8632	0,0000	0,0000	0,0005	0,0000	0,0000	0,0000
21	71.82	0,09	11,43	0,0000	0,0000	-137,5871	0,0000	0,0000	0,0065	0,0019	0,0000	0,0000
22	73.2783	0,09	11,66	0,0000	0,0000	59,8042	0,0000	0,0000	0,0012	0,0009	0,0001	0,0000
23	73.5419	0,09	11,70	0,0000	0,0000	-21,3090	0,0000	0,0000	0,0002	0,0000	0,0109	0,0000
24	73.5546	0,09	11,71	0,0000	0,0000	-7,3191	0,0000	0,0000	0,0000	0,0000	0,0001	0,0000
25	74.9694	0,08	11,93	0,0000	0,0000	-3,5927	0,0000	0,0000	0,0000	0,0000	0,0009	0,0000
26	75.6779	0,08	12,04	0,0000	0,0000	-74,7289	0,0000	0,0000	0,0019	0,0004	0,0003	0,0000
27	75.8944	0,08	12,08	0,0000	0,0000	99,2571	0,0000	0,0000	0,0034	0,0036	0,0108	0,0000
28	76.1823	0,08	12,12	0,0000	0,0000	116,8175	0,0000	0,0000	0,0047	0,0012	0,0085	0,0000
29	76.4986	0,08	12,18	0,0000	0,0000	-11,3429	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
30	77.1147	0,08	12,27	0,0000	0,0000	20,8764	0,0000	0,0000	0,0002	0,0000	0,0044	0,0000
31	77.2441	0,08	12,29	0,0000	0,0000	-40,9979	0,0000	0,0000	0,0006	0,0003	0,0007	0,0000
32	77.4664	0,08	12,33	0,0000	0,0000	43,2528	0,0000	0,0000	0,0006	0,0000	0,0255	0,0000
33	78.112	0,08	12,43	0,0000	0,0000	-16,8235	0,0000	0,0000	0,0001	0,0000	0,0024	0,0000
34	82.6373	0,08	13,15	0,0000	0,0000	6,2441	0,0000	0,0000	0,0000	0,0000	0,0002	0,0000
35	83.5422	0,08	13,30	0,0000	0,0000	28,8061	0,0000	0,0000	0,0003	0,0004	0,0001	0,0000
36	83.6766	0,08	13,32	0,0000	0,0000	-27,2851	0,0000	0,0000	0,0003	0,0007	0,0030	0,0000
37	84.1903	0,07	13,40	0,0000	0,0000	-36,2254	0,0000	0,0000	0,0005	0,0013	0,0018	0,0000
38	84.7678	0,07	13,49	0,0000	0,0000	-7,7140	0,0000	0,0000	0,0000	0,0009	0,0000	0,0000
39	84.9764	0,07	13,52	0,0000	0,0000	-59,2812	0,0000	0,0000	0,0012	0,0011	0,0000	0,0000
40	85.1194	0,07	13,55	0,0000	0,0000	8,1443	0,0000	0,0000	0,0000	0,0004	0,0009	0,0000
41	86.1139	0,07	13,71	0,0000	0,0000	-109,4010	0,0000	0,0000	0,0041	0,0002	0,0019	0,0000
42	87.4186	0,07	13,91	0,0000	0,0000	-185,2523	0,0000	0,0000	0,0119	0,0068	0,0011	0,0000
43	89.6912	0,07	14,27	0,0000	0,0000	385,3972	0,0000	0,0000	0,0513	0,0863	0,0523	0,0000
44	91.0681	0,07	14,49	0,0000	0,0000	-110,6600	0,0000	0,0000	0,0042	0,0017	0,0156	0,0000
45	91.3773	0,07	14,54	0,0000	0,0000	-171,1027	0,0000	0,0000	0,0101	0,0188	0,0492	0,0000
46	92.3074	0,07	14,69	0,0000	0,0000	-78,6983	0,0000	0,0000	0,0021	0,0190	0,0349	0,0000
47	93.6186	0,07	14,90	0,0000	0,0000	127,5199	0,0000	0,0000	0,0056	0,0157	0,0000	0,0000
48	93.7281	0,07	14,92	0,0000	0,0000	1,9988	0,0000	0,0000	0,0000	0,0010	0,0029	0,0000
49	95.0979	0,07	15,14	0,0000	0,0000	-36,1657	0,0000	0,0000	0,0005	0,0000	0,0088	0,0000
50	95.3945	0,07	15,18	0,0000	0,0000	-182,5232	0,0000	0,0000	0,0115	0,0282	0,0008	0,0000
51	97.3788	0,06	15,50	0,0000	0,0000	210,1729	0,0000	0,0000	0,0153	0,0193	0,0005	0,0000
52	98.2836	0,06	15,64	0,0000	0,0000	-165,9295	0,0000	0,0000	0,0095	0,0199	0,0000	0,0000
53	98.6612	0,06	15,70	0,0000	0,0000	57,3023	0,0000	0,0000	0,0011	0,0122	0,0028	0,0000
54	99.3024	0,06	15,80	0,0000	0,0000	74,5058	0,0000	0,0000	0,0019	0,0007	0,0029	0,0000
55	99.8652	0,06	15,89	0,0000	0,0000	204,7373	0,0000	0,0000	0,0145	0,0004	0,0161	0,0000
56	100.167	0,06	15,94	0,0000	0,0000	-186,6645	0,0000	0,0000	0,0120	0,0052	0,0003	0,0000
57	100.98	0,06	16,07	0,0000	0,0000	39,4151	0,0000	0,0000	0,0005	0,0005	0,0001	0,0000
58	102.183	0,06	16,26	0,0000	0,0000	42,2693	0,0000	0,0000	0,0006	0,0003	0,0005	0,0000
59	102.453	0,06	16,31	0,0000	0,0000	147,6349	0,0000	0,0000	0,0075	0,0010	0,0008	0,0000
60	103.075	0,06	16,40	0,0000	0,0000	-58,3704	0,0000	0,0000	0,0012	0,0017	0,0012	0,0000
61	103.621	0,06	16,49	0,0000	0,0000	12,7865	0,0000	0,0000	0,0001	0,0003	0,0000	0,0000
62	103.754	0,06	16,51	0,0000	0,0000	29,3705	0,0000	0,0000	0,0003	0,0005	0,0001	0,0000
63	104.404	0,06	16,62	0,0000	0,0000	14,3538	0,0000	0,0000	0,0001	0,0000	0,0016	0,0000
64	104.634	0,06	16,65	0,0000	0,0000	159,1444	0,0000	0,0000	0,0087	0,0057	0,0087	0,0000
65	104.728	0,06	16,67	0,0000	0,0000	-32,4966	0,0000	0,0000	0,0004	0,0000	0,0010	0,0000
66	105.071	0,06	16,72	0,0000	0,0000	83,5451	0,0000	0,0000	0,0024	0,0070	0,0067	0,0000
67	105.635	0,06	16,81	0,0000	0,0000	98,2224	0,0000	0,0000	0,0033	0,0085	0,0026	0,0000
68	106.542	0,06	16,96	0,0000	0,0000	-24,2524	0,0000	0,0000	0,0002	0,0011	0,0007	0,0000
69	107.349	0,06	17,09	0,0000	0,0000	-1,1032	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
70	107.862	0,06	17,17	0,0000	0,0000	-12,5517	0,0000	0,0000	0,0001	0,0001	0,0001	0,0000
71	108.633	0,06	17,29	0,0000	0,0000	8,9453	0,0000	0,0000	0,0000	0,0001	0,0001	0,0000
72	109.235	0,06	17,39	0,0000	0,0000	17,5911	0,0000	0,0000	0,0001	0,0001	0,0023	0,0000
73	109.764	0,06	17,47	0,0000	0,0000	-3,6578	0,0000	0,0000	0,0000	0,0016	0,0000	0,0000
74	110.934	0,06	17,66	0,0000	0,0000	29,3405	0,0000	0,0000	0,0003	0,0009	0,0001	0,0000
75	111.351	0,06	17,72	0,0000	0,0000	68,3217	0,0000	0,0000	0,0016	0,0006	0,0002	0,0000
76	112.842	0,06	17,96	0,0000	0,0000	95,8355	0,0000	0,0000	0,0032	0,0096	0,0009	0,0000
77	113.454	0,06	18,06	0,0000	0,0000	-33,2396	0,0000	0,0000	0,0004	0,0039	0,0000	0,0000

78	114.192	0,06	18,17	0,0000	0,0000	29,6693	0,0000	0,0000	0,0003	0,0031	0,0086	0,0000
79	114.837	0,05	18,28	0,0000	0,0000	226,1462	0,0000	0,0000	0,0177	0,0047	0,0000	0,0000
80	116.251	0,05	18,50	0,0000	0,0000	-4,9848	0,0000	0,0000	0,0000	0,0075	0,0030	0,0000
81	119.393	0,05	19,00	0,0000	0,0000	43,9370	0,0000	0,0000	0,0007	0,0017	0,0004	0,0000
82	120.547	0,05	19,19	0,0000	0,0000	17,5131	0,0000	0,0000	0,0001	0,0000	0,0003	0,0000
83	121.374	0,05	19,32	0,0000	0,0000	-6,8623	0,0000	0,0000	0,0000	0,0001	0,0111	0,0000
84	122.165	0,05	19,44	0,0000	0,0000	-10,1242	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
85	123.64	0,05	19,68	0,0000	0,0000	11,2590	0,0000	0,0000	0,0000	0,0004	0,0000	0,0000
86	125.246	0,05	19,93	0,0000	0,0000	41,2422	0,0000	0,0000	0,0006	0,0025	0,0009	0,0000
87	127.066	0,05	20,22	0,0000	0,0000	121,0938	0,0000	0,0000	0,0051	0,0039	0,0017	0,0000
88	130.446	0,05	20,76	0,0000	0,0000	81,5248	0,0000	0,0000	0,0023	0,0002	0,0004	0,0000
89	132.754	0,05	21,13	0,0000	0,0000	-4,4427	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
90	133.873	0,05	21,31	0,0000	0,0000	20,7770	0,0000	0,0000	0,0001	0,0000	0,0002	0,0000
91	134.286	0,05	21,37	0,0000	0,0000	79,4732	0,0000	0,0000	0,0022	0,0004	0,0003	0,0000
92	136.399	0,05	21,71	0,0000	0,0000	14,6707	0,0000	0,0000	0,0001	0,0000	0,0000	0,0000
93	137.193	0,05	21,83	0,0000	0,0000	-160,0057	0,0000	0,0000	0,0088	0,0002	0,0002	0,0000
94	137.448	0,05	21,88	0,0000	0,0000	-86,2200	0,0000	0,0000	0,0026	0,0000	0,0007	0,0000
95	137.805	0,05	21,93	0,0000	0,0000	-46,8241	0,0000	0,0000	0,0008	0,0020	0,0004	0,0000
96	138.475	0,05	22,04	0,0000	0,0000	-32,7979	0,0000	0,0000	0,0004	0,0029	0,0028	0,0000
97	139.39	0,05	22,18	0,0000	0,0000	-14,1175	0,0000	0,0000	0,0001	0,0002	0,0006	0,0000
98	139.855	0,04	22,26	0,0000	0,0000	-5,7041	0,0000	0,0000	0,0000	0,0014	0,0004	0,0000
99	140.798	0,04	22,41	0,0000	0,0000	29,4275	0,0000	0,0000	0,0003	0,0002	0,0002	0,0000
100	141.054	0,04	22,45	0,0000	0,0000	98,1215	0,0000	0,0000	0,0033	0,0049	0,0065	0,0000
							0,0000	0,0000	0,7698	0,7713	0,8460	0,0000

It is noted that modes exciting the vertical structure the most are modes 2 (15,45%), 3 (14,22%), 13 (6,44%), 5 (6,05%), 43 (5,13%) et 1 (2,45%) … for a total of 76%.

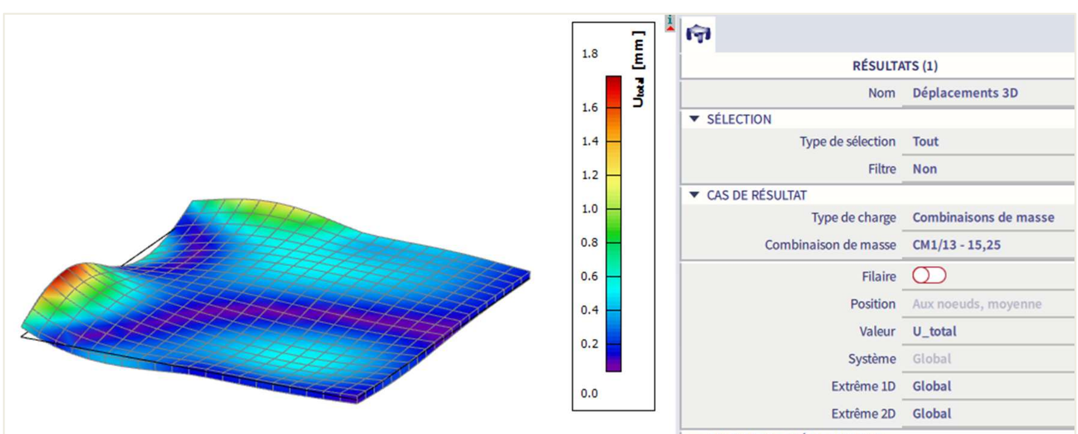
NB : Results can be displayed in the « Results table ». By clicking on the head of the « W_{zi}/W_{ztot} » column, results are displayed in ascending or descending order :



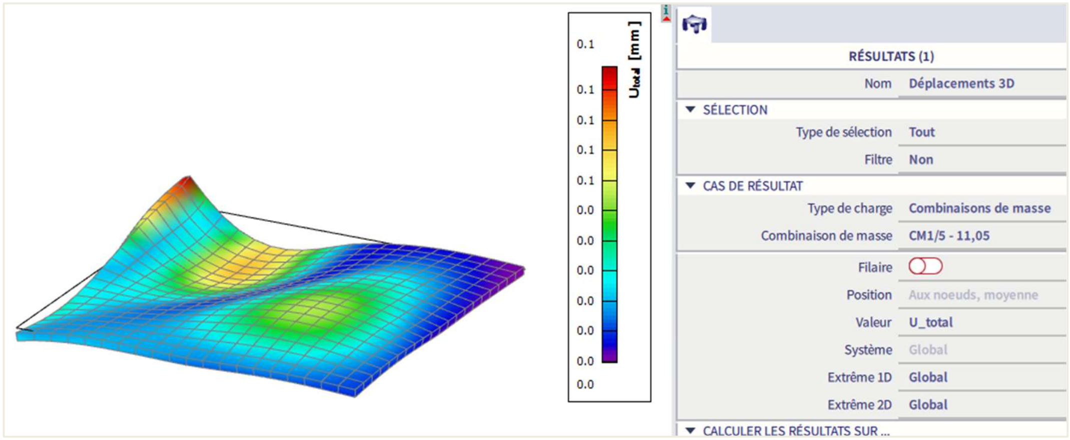


As for the frames or the slab, the eigenmodes can be visualized through the « 3D displacements » for the surfaces. Here it is necessary to visually search which mode excites the studied floor the most (the indentation in the calculation report can here have a major interest to obtain the complete visualization in all modes very quickly). Among the main modes mentioned above :

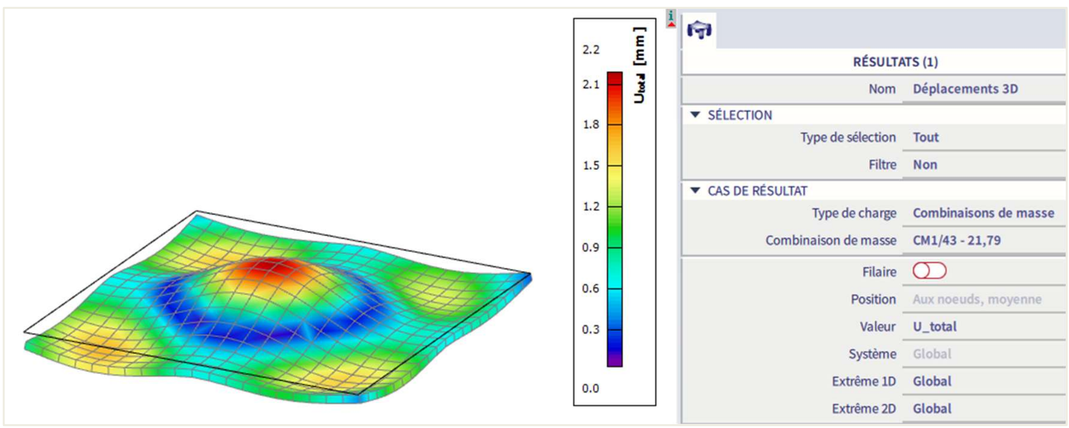
- The mode 13 with a frequency of 10,08Hz excites a little bit the part of the studied floor :



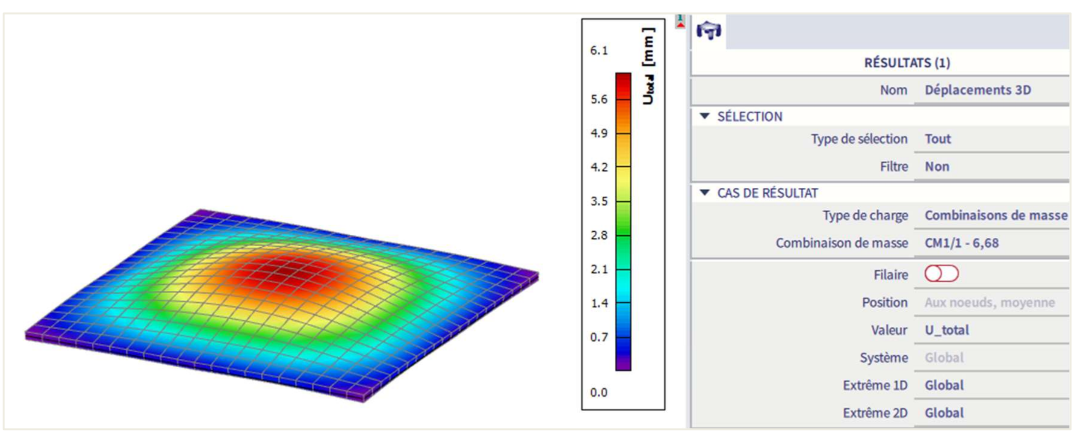
- Also the mode 5 with a frequency of 9,13Hz :



- The mode 43 with a frequency of 14,27Hz :



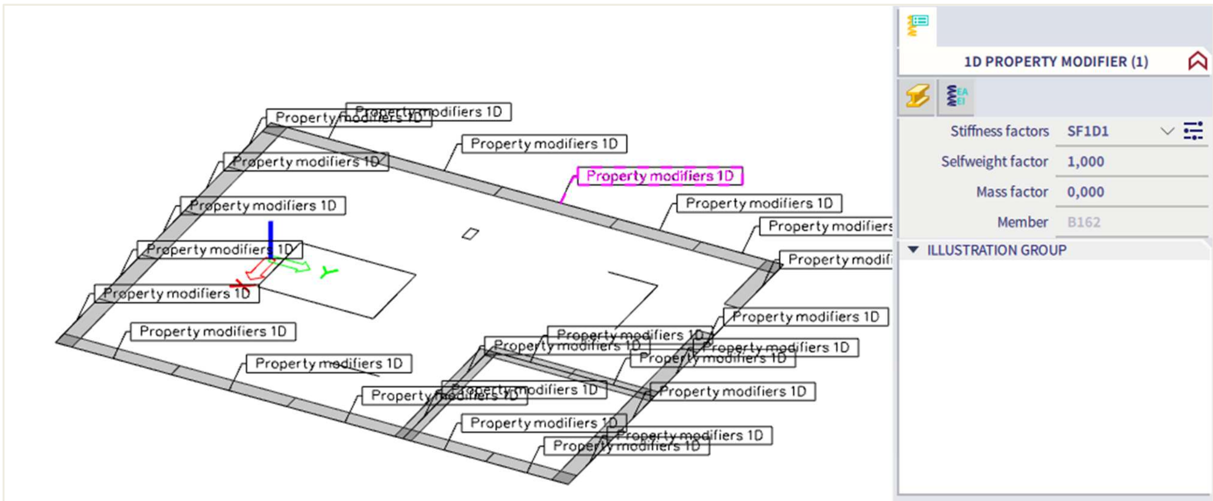
- But it is especially the mode 1 with a frequency of 6,68Hz that causes the biggest vibration of this part of the floor :



The total of the excited masses in vertical is 76%. It is not indicated in the standard any particular limits, but it is always better to approach 100%. One solution might be to use property or model modifiers.

Model modifiers impact only the case on which the modifier group is applied. This may be interesting to use it if two analyses need to be performed for example: seismic and vibration (floor). The modifier can then be applied to the mass combination related to the vibration analysis only.

Property modifiers impact all cases. Let's test by putting property modifiers with zero mass, on all 1D elements of the project, on all walls, and on all slabs other than the studied floor.



The sum of the masses taken into account is much smaller than without the use of modifiers :

Sum of masses

	Mass type	X [kg]	Y [kg]	Z [kg]
CM1	Masse en mouvement	0,0	0,0	1137457,0
CM1	Masse totale	0,0	0,0	1137457,0

And the mass participation results slightly differ :

Relative modal masses

Mode	ıega [rad	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	$_{xi\_R}/W_{xtot}$	$_{yi\_R}/W_{ytot}$	$_{zi\_R}/W_{ztot}$
1	41.9988	0,15	6,68	0,0000	0,0000	258,6900	0,0000	0,0000	0,0588	0,0813	0,0674	0,0000
2	58.1717	0,11	9,26	0,0000	0,0000	-304,7927	0,0000	0,0000	0,0817	0,1508	0,0010	0,0000
3	60.5555	0,10	9,64	0,0000	0,0000	-21,7388	0,0000	0,0000	0,0004	0,0065	0,0529	0,0000
4	62.5453	0,10	9,95	0,0000	0,0000	-170,0642	0,0000	0,0000	0,0254	0,0216	0,0034	0,0000
5	69.4477	0,09	11,05	0,0000	0,0000	397,6239	0,0000	0,0000	0,1390	0,0354	0,1212	0,0000
6	70.8711	0,09	11,28	0,0000	0,0000	184,0821	0,0000	0,0000	0,0298	0,0210	0,0946	0,0000
7	72.8822	0,09	11,60	0,0000	0,0000	142,9251	0,0000	0,0000	0,0180	0,0256	0,0005	0,0000
8	73.3058	0,09	11,67	0,0000	0,0000	-187,0106	0,0000	0,0000	0,0307	0,0025	0,0597	0,0000
9	82.8687	0,08	13,19	0,0000	0,0000	-29,3766	0,0000	0,0000	0,0008	0,0003	0,0153	0,0000
10	83.4038	0,08	13,27	0,0000	0,0000	99,8025	0,0000	0,0000	0,0088	0,0049	0,0003	0,0000
11	84.8194	0,07	13,50	0,0000	0,0000	-39,4336	0,0000	0,0000	0,0014	0,0000	0,0064	0,0000
12	85.4311	0,07	13,60	0,0000	0,0000	-47,7919	0,0000	0,0000	0,0020	0,0023	0,0005	0,0000
13	95.7949	0,07	15,25	0,0000	0,0000	192,7382	0,0000	0,0000	0,0327	0,0317	0,0112	0,0000
14	98.3924	0,06	15,66	0,0000	0,0000	194,3439	0,0000	0,0000	0,0332	0,0049	0,0003	0,0000
15	99.2397	0,06	15,79	0,0000	0,0000	122,3700	0,0000	0,0000	0,0132	0,0073	0,0049	0,0000
16	100.467	0,06	15,99	0,0000	0,0000	219,9564	0,0000	0,0000	0,0425	0,0972	0,0009	0,0000
17	102.262	0,06	16,28	0,0000	0,0000	-160,2201	0,0000	0,0000	0,0226	0,1010	0,0531	0,0000
18	103.057	0,06	16,40	0,0000	0,0000	73,6967	0,0000	0,0000	0,0048	0,0000	0,0283	0,0000
19	104.228	0,06	16,59	0,0000	0,0000	-34,3869	0,0000	0,0000	0,0010	0,0010	0,0089	0,0000
20	104.465	0,06	16,63	0,0000	0,0000	19,4899	0,0000	0,0000	0,0003	0,0013	0,0121	0,0000
21	104.815	0,06	16,68	0,0000	0,0000	-254,3964	0,0000	0,0000	0,0569	0,0002	0,0233	0,0000
22	105.056	0,06	16,72	0,0000	0,0000	-77,9974	0,0000	0,0000	0,0053	0,0114	0,0112	0,0000
23	107.405	0,06	17,09	0,0000	0,0000	-137,4492	0,0000	0,0000	0,0166	0,0195	0,0010	0,0000
24	108.154	0,06	17,21	0,0000	0,0000	166,9742	0,0000	0,0000	0,0245	0,0101	0,0180	0,0000
25	108.858	0,06	17,33	0,0000	0,0000	11,0641	0,0000	0,0000	0,0001	0,0007	0,0014	0,0000
26	113.484	0,06	18,06	0,0000	0,0000	-61,7485	0,0000	0,0000	0,0034	0,0004	0,0029	0,0000
27	114.322	0,05	18,19	0,0000	0,0000	16,3965	0,0000	0,0000	0,0002	0,0065	0,0017	0,0000
28	115.535	0,05	18,39	0,0000	0,0000	179,7413	0,0000	0,0000	0,0284	0,0123	0,0475	0,0000
29	118.296	0,05	18,83	0,0000	0,0000	-170,5151	0,0000	0,0000	0,0256	0,0166	0,0184	0,0000
30	119.906	0,05	19,08	0,0000	0,0000	65,0044	0,0000	0,0000	0,0037	0,0104	0,0027	0,0000
31	120.573	0,05	19,19	0,0000	0,0000	56,2047	0,0000	0,0000	0,0028	0,0027	0,0048	0,0000
32	122.629	0,05	19,52	0,0000	0,0000	-6,1150	0,0000	0,0000	0,0000	0,0003	0,0000	0,0000
33	124.076	0,05	19,75	0,0000	0,0000	3,6234	0,0000	0,0000	0,0000	0,0003	0,0032	0,0000
34	125.65	0,05	20,00	0,0000	0,0000	94,8246	0,0000	0,0000	0,0079	0,0000	0,0165	0,0000
35	127.483	0,05	20,29	0,0000	0,0000	35,8400	0,0000	0,0000	0,0011	0,0015	0,0009	0,0000
36	127.825	0,05	20,34	0,0000	0,0000	72,6578	0,0000	0,0000	0,0046	0,0024	0,0000	0,0000
37	130.339	0,05	20,74	0,0000	0,0000	4,8351	0,0000	0,0000	0,0000	0,0015	0,0221	0,0000
38	130.903	0,05	20,83	0,0000	0,0000	21,9384	0,0000	0,0000	0,0004	0,0034	0,0001	0,0000
39	132.385	0,05	21,07	0,0000	0,0000	83,6439	0,0000	0,0000	0,0062	0,0027	0,0000	0,0000
40	134.14	0,05	21,35	0,0000	0,0000	31,1361	0,0000	0,0000	0,0009	0,0000	0,0006	0,0000
41	134.856	0,05	21,46	0,0000	0,0000	-10,4315	0,0000	0,0000	0,0001	0,0028	0,0076	0,0000
42	135.23	0,05	21,52	0,0000	0,0000	-103,8178	0,0000	0,0000	0,0095	0,0006	0,0011	0,0000
43	136.89	0,05	21,79	0,0000	0,0000	-195,2748	0,0000	0,0000	0,0335	0,0379	0,0267	0,0000
44	137.883	0,05	21,94	0,0000	0,0000	-86,5457	0,0000	0,0000	0,0066	0,0019	0,0000	0,0000
45	138.221	0,05	22,00	0,0000	0,0000	-6,7349	0,0000	0,0000	0,0000	0,0173	0,0130	0,0000
46	138.847	0,05	22,10	0,0000	0,0000	37,2509	0,0000	0,0000	0,0012	0,0000	0,0077	0,0000
47	142.492	0,04	22,68	0,0000	0,0000	-18,4329	0,0000	0,0000	0,0003	0,0025	0,0043	0,0000
48	142.808	0,04	22,73	0,0000	0,0000	46,7587	0,0000	0,0000	0,0019	0,0005	0,0105	0,0000
49	143.365	0,04	22,82	0,0000	0,0000	31,8664	0,0000	0,0000	0,0009	0,0002	0,0234	0,0000
50	144.589	0,04	23,01	0,0000	0,0000	-6,6883	0,0000	0,0000	0,0000	0,0003	0,0032	0,0000
51	145.808	0,04	23,21	0,0000	0,0000	-23,4047	0,0000	0,0000	0,0005	0,0001	0,0013	0,0000
52	148.193	0,04	23,59	0,0000	0,0000	4,0377	0,0000	0,0000	0,0000	0,0002	0,0003	0,0000
53	148.515	0,04	23,64	0,0000	0,0000	-6,2798	0,0000	0,0000	0,0000	0,0006	0,0000	0,0000
54	149.34	0,04	23,77	0,0000	0,0000	-14,7344	0,0000	0,0000	0,0002	0,0000	0,0003	0,0000
55	149.961	0,04	23,87	0,0000	0,0000	-9,8483	0,0000	0,0000	0,0001	0,0001	0,0001	0,0000
56	150.095	0,04	23,89	0,0000	0,0000	78,3152	0,0000	0,0000	0,0054	0,0001	0,0010	0,0000
57	151.049	0,04	24,04	0,0000	0,0000	-33,1724	0,0000	0,0000	0,0010	0,0008	0,0025	0,0000
58	151.917	0,04	24,18	0,0000	0,0000	-25,0014	0,0000	0,0000	0,0005	0,0002	0,0051	0,0000
59	153.921	0,04	24,50	0,0000	0,0000	6,6520	0,0000	0,0000	0,0000	0,0002	0,0001	0,0000
60	156.226	0,04	24,86	0,0000	0,0000	-97,4799	0,0000	0,0000	0,0084	0,0076	0,0005	0,0000
61	156.952	0,04	24,98	0,0000	0,0000	-28,9554	0,0000	0,0000	0,0007	0,0020	0,0007	0,0000
62	158.226	0,04	25,18	0,0000	0,0000	-11,4373	0,0000	0,0000	0,0001	0,0016	0,0003	0,0000
63	158.669	0,04	25,25	0,0000	0,0000	42,2142	0,0000	0,0000	0,0016	0,0036	0,0001	0,0000
64	159.653	0,04	25,41	0,0000	0,0000	-91,9403	0,0000	0,0000	0,0074	0,0075	0,0001	0,0000
65	161.803	0,04	25,75	0,0000	0,0000	34,9771	0,0000	0,0000	0,0011	0,0022	0,0035	0,0000
66	162.568	0,04	25,87	0,0000	0,0000	-34,3451	0,0000	0,0000	0,0010	0,0006	0,0022	0,0000
67	165.08	0,04	26,27	0,0000	0,0000	3,4962	0,0000	0,0000	0,0000	0,0004	0,0000	0,0000
68	166.88	0,04	26,56	0,0000	0,0000	-28,3536	0,0000	0,0000	0,0007	0,0007	0,0004	0,0000
69	170.536	0,04	27,14	0,0000	0,0000	-49,4541	0,0000	0,0000	0,0022	0,0018	0,0015	0,0000
70	172.202	0,04	27,41	0,0000	0,0000	-52,4781	0,0000	0,0000	0,0024	0,0001	0,0049	0,0000

71	173.153	0,04	27,56	0,0000	0,0000	-29,0121	0,0000	0,0000	0,0007	0,0010	0,0035	0,0000
72	173.82	0,04	27,66	0,0000	0,0000	128,7780	0,0000	0,0000	0,0146	0,0217	0,0205	0,0000
73	174.284	0,04	27,74	0,0000	0,0000	62,3312	0,0000	0,0000	0,0034	0,0008	0,0001	0,0000
74	174.865	0,04	27,83	0,0000	0,0000	-38,6841	0,0000	0,0000	0,0013	0,0010	0,0067	0,0000
75	176.184	0,04	28,04	0,0000	0,0000	-7,8160	0,0000	0,0000	0,0001	0,0006	0,0000	0,0000
76	176.88	0,04	28,15	0,0000	0,0000	-35,7974	0,0000	0,0000	0,0011	0,0075	0,0014	0,0000
77	177.606	0,04	28,27	0,0000	0,0000	-14,0595	0,0000	0,0000	0,0002	0,0049	0,0039	0,0000
78	177.966	0,04	28,32	0,0000	0,0000	15,9927	0,0000	0,0000	0,0002	0,0009	0,0000	0,0000
79	179.723	0,03	28,60	0,0000	0,0000	-20,8254	0,0000	0,0000	0,0004	0,0014	0,0025	0,0000
80	180.122	0,03	28,67	0,0000	0,0000	30,1837	0,0000	0,0000	0,0008	0,0022	0,0000	0,0000
81	183.354	0,03	29,18	0,0000	0,0000	9,5160	0,0000	0,0000	0,0001	0,0003	0,0008	0,0000
82	183.498	0,03	29,20	0,0000	0,0000	38,9774	0,0000	0,0000	0,0013	0,0000	0,0000	0,0000
83	185.141	0,03	29,47	0,0000	0,0000	-74,1924	0,0000	0,0000	0,0048	0,0012	0,0011	0,0000
84	186.636	0,03	29,70	0,0000	0,0000	-0,0966	0,0000	0,0000	0,0000	0,0058	0,0060	0,0000
85	186.886	0,03	29,74	0,0000	0,0000	-1,0720	0,0000	0,0000	0,0000	0,0015	0,0006	0,0000
86	187.228	0,03	29,80	0,0000	0,0000	65,8925	0,0000	0,0000	0,0038	0,0026	0,0008	0,0000
87	188.178	0,03	29,95	0,0000	0,0000	25,7472	0,0000	0,0000	0,0006	0,0001	0,0003	0,0000
88	189.422	0,03	30,15	0,0000	0,0000	-63,7938	0,0000	0,0000	0,0036	0,0001	0,0003	0,0000
89	190.398	0,03	30,30	0,0000	0,0000	-28,2344	0,0000	0,0000	0,0007	0,0046	0,0063	0,0000
90	190.969	0,03	30,39	0,0000	0,0000	-45,8403	0,0000	0,0000	0,0018	0,0002	0,0017	0,0000
91	191.64	0,03	30,50	0,0000	0,0000	1,0584	0,0000	0,0000	0,0000	0,0024	0,0003	0,0000
92	194.17	0,03	30,90	0,0000	0,0000	61,7315	0,0000	0,0000	0,0034	0,0000	0,0004	0,0000
93	195.549	0,03	31,12	0,0000	0,0000	-71,0361	0,0000	0,0000	0,0044	0,0051	0,0021	0,0000
94	196.862	0,03	31,33	0,0000	0,0000	-24,6712	0,0000	0,0000	0,0005	0,0006	0,0001	0,0000
95	197.781	0,03	31,48	0,0000	0,0000	15,2131	0,0000	0,0000	0,0002	0,0004	0,0000	0,0000
96	199.172	0,03	31,70	0,0000	0,0000	59,3103	0,0000	0,0000	0,0031	0,0013	0,0001	0,0000
97	200.105	0,03	31,85	0,0000	0,0000	-73,0078	0,0000	0,0000	0,0047	0,0033	0,0008	0,0000
98	203.853	0,03	32,44	0,0000	0,0000	-21,3945	0,0000	0,0000	0,0004	0,0010	0,0002	0,0000
99	204.493	0,03	32,55	0,0000	0,0000	-32,8742	0,0000	0,0000	0,0010	0,0003	0,0000	0,0000
100	206.231	0,03	32,82	0,0000	0,0000	-33,1376	0,0000	0,0000	0,0010	0,0002	0,0001	0,0000
							0,0000	0,0000	0,8814	0,8671	0,9018	0,0000

We note that the modes that excite the vertical structure the most are a little bit different from without modifiers : modes 5 (13,9%), 2 (8,17%), 21 (5,69%), 1 (5,88%) et 16 (4,25%) … for a total of 88%. In this example, only 12% of total mass participation is gained, but in other examples, the jump to 100% might be more pronounced.

CHAPTER 3 : SPECTRAL ANALYSIS : SEISMIC LOAD

In this chapter, the seismic analysis in SCIA Engineer is explained in detail.

During an earthquake, the subsoil bearing a structure moves. The structure tries to follow this movement and as a result, the masses in the structure begin to move. Subsequently, these masses subject the structure to inertial forces. When these forces are determined, they can be applied to the structure and thus, like with the harmonic load, the dynamic calculation is transformed to an equivalent static.

In the first part of the chapter, the theory will be explained. The theory will then be illustrated by examples, which are again verified by manual calculations.

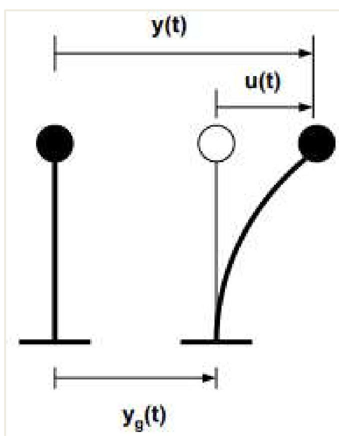
3.1 Theory

General

Analogous to the previous chapters, before examining the dynamic analysis of a complex structure, the Seismic analysis of a SDOF (Single Degree Of Freedom) system is regarded in detail. A complete overview can be found in references [2], [3].

Generally, this paragraph deals with the analysis of structures that are submitted to a harmonic ground motion. The most important harmonic ground motions are earthquakes (seismic loads), but this calculation method can also be applied to the analysis of underground or surface explosions and vibrations generated by heavy traffic or machinery.

The following figure illustrates the displacement of a system that is submitted to a ground motion:



Where:

$y_g(t)$ is the ground displacement
$y(t)$ is the total displacement of the mass
$u(t)$ is the relative displacement of the mass

The total displacement can thus be expressed as follows:

$$y(t) = y_g(t) + u(t) \tag{4.1}$$

Since y_g is assumed to be harmonic, it can be written as:

$$y_g(t) = Y_g . \sin(\nu . t) \tag{4.2}$$

The equilibrium equation of motion can now be written as:

$$m.\ddot{y}(t) + c.\dot{u}(t) + k.u(t) = 0 \tag{4.3}$$

Since the inertia force is related to the total displacement (y) of the mass and the damping and spring reactions are related to the relative displacements (u) of the mass.

When (4.1) is substituted in (4.3), the following is obtained:

$$m.\left(\ddot{u}(t) + \ddot{y_g}(t)\right) + c.\dot{u}(t) + k.u(t) = 0$$

or

$$m.\ddot{u}(t) + c.\dot{u}(t) + k.u(t) = -m.\ddot{y_g}(t) \tag{4.4}$$

This equation is known as the General Seismic Equation of Motion. This equation can be used to illustrate the behaviour of structures that are loaded with a seismic load.

Substituting (4.2) in (4.4) gives the following:

$$m.\ddot{u}(t) + c.\dot{u}(t) + k.u(t) = -m.Y_g.\nu^2.\sin(\nu.t)$$

This equation can be compared with equation (3.2) of the previous chapter. As a conclusion, the ground motion can also be replaced by an external harmonic force with amplitude:

$$F = -m.Y_g.\nu^2$$

But an earthquake will be a combination of many harmonic loads acting on different frequencies simultaneously. The load represented in these harmonic loads is the acceleration of the structure multiplied with the mass of the structure. The frequencies of these harmonic loads are the frequencies on which this acceleration occurs in the earthquake.

The combination of all the accelerations over the different frequencies in the earthquake will be given by a response spectrum. A response spectrum is therefore nothing more than a list of accelerations and the frequencies on which they occur.

- *Response spectra*

When a structure has to be designed for earthquakes, in most cases spectral analysis is used because the earthquake loading is often described as a response spectrum. This response spectrum can either be a displacement, velocity or acceleration spectrum.

The relation of an earthquake (given by an acceleration time-history) and the corresponding displacement response spectrum is given by [16]:

$$S_d(\xi,\omega) = \frac{1}{\omega}.\left[\int \ddot{y_g}(\tau).e^{-\xi\omega(T-\tau)}.\sin(\omega.(T-\tau)).d\tau\right]_{max} \tag{4.5}$$

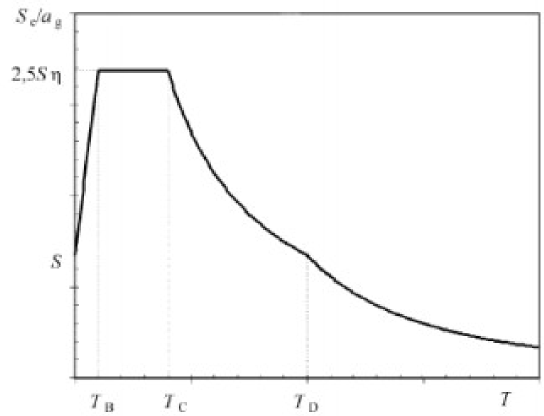
Where:

- $\ddot{y_g}(\tau)$: the ground acceleration in function of time
- ξ: the damping factor
- T: the period $2\pi/\omega$

Instead of the displacement response spectrum S_d, also the velocity response spectrum S_v or the acceleration response spectrum S_a can be used. These three spectra are related by ω:

$$S_a = \omega.S_v = \omega^2.s_d \tag{4.6}$$

In Eurocode 8 [7] the earthquake motion at a given point on the surface is represented by an elastic ground acceleration response spectrum or "Elastic Response Spectrum S_e" This spectrum is illustrated in the following figure:



A commonly used way of describing an earthquake magnitude is the so-called Richter scale. Annex A gives a relation between the magnitude on the Richter scale and the Peak Ground Acceleration.

- *Spectral analysis*

For MDOF (Multiple Degree Of Freedom) systems, equation (4.4) can be written in matrix notation as a set of coupled differential equations:

$$\mathrm{M}.\ddot{\mathrm{U}} + \mathrm{C}.\dot{\mathrm{U}} + \mathrm{KU} = -\mathrm{M}.\{\mathrm{l}\}.\ddot{\mathrm{Y}}_{\mathrm{g}} \tag{4.7}$$

The matrix {1} is used to indicate the direction of the earthquake. For example, for a two-dimensional structure (three degrees of freedom) with an earthquake that acts in the x-direction, the matrix is a sequence like {1,0,0,1,0,0,1,0,0,…}.

The resulting set of coupled differential equations is reduced to a set of uncoupled differential equation by a transformation U = Z.Q, where Z is a subset of Φ (the eigenvectors) and Q is a vector, which is time-dependent.

$$\mathrm{M}.\mathrm{Z}.\ddot{\mathrm{Q}} + \mathrm{C}.\mathrm{Z}.\dot{\mathrm{Q}} + \mathrm{K}.\mathrm{Z}.\mathrm{Q} = -\mathrm{M}.\{\mathrm{l}\}.\ddot{\mathrm{Y}}_{\mathrm{g}}$$

Or

$$Z^T.\mathrm{M}.\mathrm{Z}.\ddot{\mathrm{Q}} + Z^T.\mathrm{C}.\mathrm{Z}.\dot{\mathrm{Q}} + Z^T.\mathrm{K}.\mathrm{Z}.\mathrm{Q} = -Z^T.\mathrm{M}.\{\mathrm{l}\}.\ddot{\mathrm{Y}}_{\mathrm{g}}$$

This can be simplified to a set of uncoupled differential equations:

$$\ddot{\mathrm{Q}} + C^*.\dot{\mathrm{Q}} + \Omega^2.\mathrm{Q} = -Z^T.\mathrm{M}.\{\mathrm{l}\}.\ddot{\mathrm{Y}}_{\mathrm{g}} \tag{4.8}$$

where C^* is a diagonal matrix containing terms like $2.\omega_i.\xi_i$

Each equation j has a solution of the form:

$$\mathrm{Q}_j = -\mathrm{Z}^{\mathrm{T}}.\mathrm{M}.\{\mathrm{l}\}.\frac{1}{\omega}.\int_0^t \ddot{\mathrm{Y}}_{\mathrm{g}}(\tau).\mathrm{e}^{-\xi\omega_i(\mathrm{T}-\tau)}.\sin\left(\omega_j.(\mathrm{T}-\tau)\right).\mathrm{d}\tau \tag{4.9}$$

To obtain the maximum displacements, the displacement response spectrum S_d of equation (4.5) can be substituted:

$$\mathrm{Q}_{j,\max} = -\mathrm{Z}^{\mathrm{T}}.\mathrm{M}.\{\mathrm{l}\}.\mathrm{S}_{\mathrm{d}}\left(\xi_j, \omega_j\right) \tag{4.10}$$

And:

$$\mathrm{U}_{j,\max} = -\mathrm{Z}.\mathrm{Z}^{\mathrm{T}}.\mathrm{M}.\{\mathrm{l}\}.\mathrm{S}_{\mathrm{d}}\left(\xi_j, \omega_j\right)$$

Or

$$U_{j,max} = -Z.\Psi.S_d(\xi_j, \omega_j) \tag{4.11}$$

Where Ψ: modal participation factor:

$$\Psi = Z^T.M.\{l\}$$

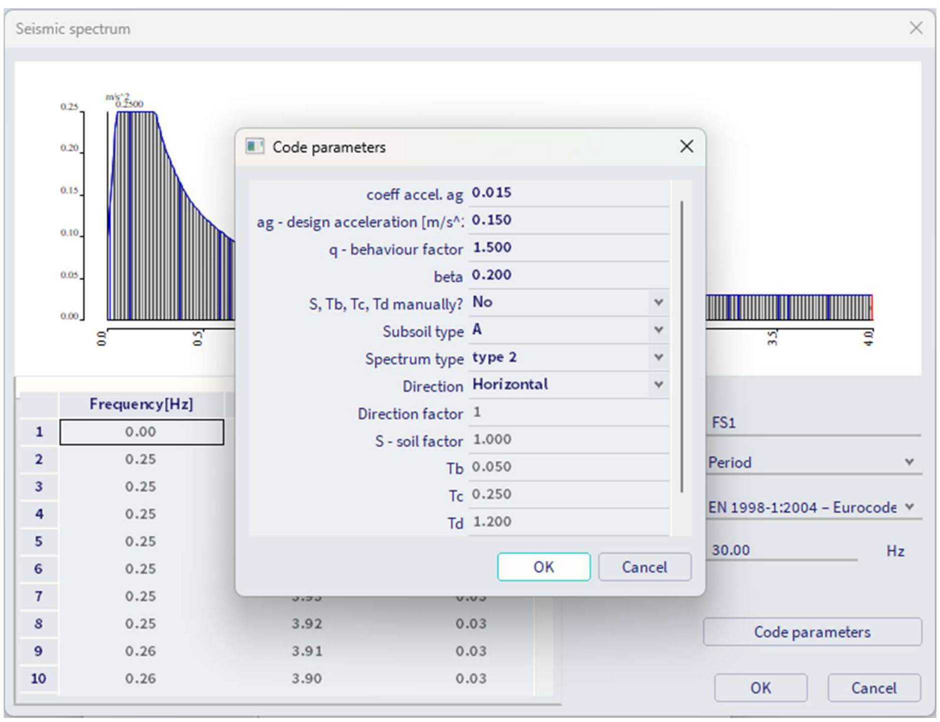
3.2 Seismic load in SCIA Engineer

- *Response spectra*

In SCIA Engineer, a Seismic Load can be inputted after creating a Combination of Mass Groups. This implies that the steps used to perform a Free Vibration calculation still apply here and are expanded by the properties of the Seismic Load.

As specified in the theory, Eurocode 8 [7] specifies an Elastic Response Spectrum S_e. For design purposes, this spectrum is reduced to a **Design Spectrum** S_d. This Design Spectrum is dependent on several parameters: the **Ground Type**, the **Ground Acceleration**, the **Behaviour Factor** and the **Damping**.

When defining a spectrum in SCIA Engineer, the spectrum can be defined either by combinations of Frequencies & accelerations, or Periods & accelerations, or by simply inputting the parameters that define this spectrum according to Eurocode 8. If the user wishes to compose the spectrum based on the parameters in Eurocode 8, then he will have the next input window:



For a detailed description of these parameters, reference is made to Eurocode 8 [7]. The following is a brief overview for understanding the input needed for SCIA Engineer.

- **Damping**: The Design Spectra of Eurocode 8 are defined for a damping ratio of **5%**. If the structure has another damping ratio, the spectrum has to be adapted with a damping correction factor **η**. This will be looked upon in more detail in Chapter 11.

- **Ground acceleration**: The ground acceleration a_g or the coefficient of acceleration α can be calculated by means of an importance factor.

⇨ The **ground acceleration** a_g can be calculated from the importance factor and the peak ground acceleration (PGA) a_{gr}:

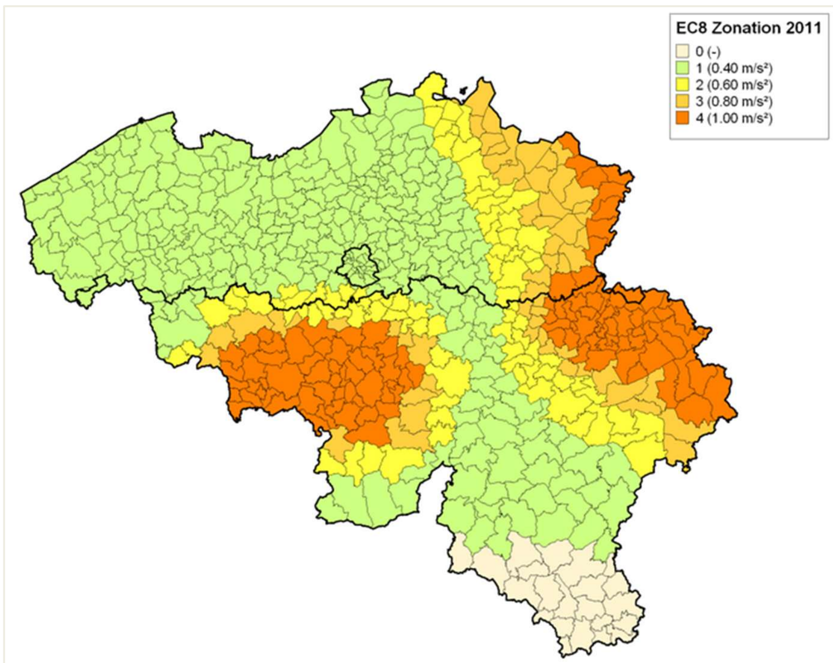
$$a_g = \gamma_I * a_{gr} \tag{4.12}$$

⇨ The **coefficient of acceleration** α is defined as the ground acceleration divided by the acceleration of gravity g:

$$\alpha = \frac{a_g}{g} \tag{4.13}$$

⇨ The **importance factor** is derived from the return period of the seismic action and the importance of the structure. An importance factor γ_I equal to 1 is assigned to the reference return period.

⇨ The **peak ground acceleration (PGA)** a_{gr} can be found from the seismic zones in which a country is divided. By definition, the seismic hazard within each zone is assumed to be constant. The hazard is described by a single parameter: the peak ground acceleration (PGA) a_{gr}. The following figure illustrates the division in seismic zones for the map of Belgium [9]:



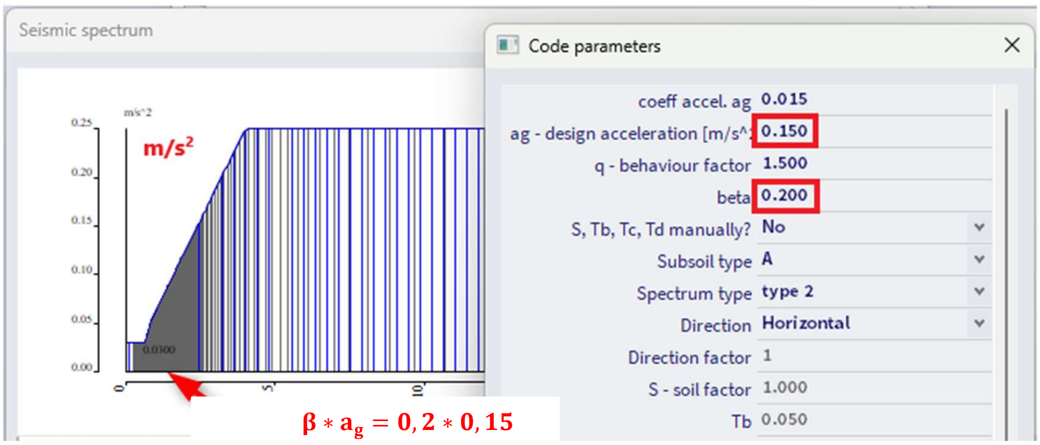
Zones	0	1	2	3	4
A_{gr} (for A ground) in m/s²	/	0,39	0,59	0,78	0,98

- **Behaviour factor** (EN1998, 3.2.2.5): To avoid explicit inelastic structural analysis in design, the capacity of the structure to dissipate energy, mainly through ductile behaviour of its elements, is taken into account by performing an elastic analysis based on a response spectrum reduced with respect to the elastic one. This reduction is accomplished by introducing the behaviour factor **q**.

⇨ For the vertical component of the seismic action a behaviour factor q up to 1,5 should generally be adopted for all materials and structural systems. The adoption of values of q greater than 1,5 in the vertical direction should be justified through proper analysis.

⇨ The values of the behaviour factor q, which also account for the influence of the viscous damping being different from 5%, are given for various materials and structural systems according to the relevant ductility classes in the various Parts of EN 1998. The value of the behaviour factor q may be different in different horizontal directions of the structure, although the ductility classification shall be the same in all directions.

- **Beta (β)**: the coefficient corresponds to the lowest limit (asymptote) for the horizontal design spectrum. The recommended value for (β) is 0,2 but can be overruled by the relevant national annex.
If you plot the spectrum as acceleration to frequency, then the most left value would be the lower bound factor β multiplied with the ground acceleration.



- **S, T_b, T_c, T_d manually?**: If you set this to "No", then the values to compose the spectrum are calculated automatically from the other properties in this window.

- **Ground type**: the Ground Type is dependent on the soil characteristics and is specified by letters A to E.

Ground type	Description of stratigraphic profile	Parameters		
		$v_{s,30}$ (m/s)	N_{SPT} (blows/30cm)	c_u (kPa)
A	Rock or other rock-like geological formation, including at most 5 m of weaker material at the surface.	> 800	_	_
B	Deposits of very dense sand, gravel, or very stiff clay, at least several tens of metres in thickness, characterised by a gradual increase of mechanical properties with depth.	360 – 800	> 50	> 250
C	Deep deposits of dense or medium-dense sand, gravel or stiff clay with thickness from several tens to many hundreds of metres.	180 – 360	15 - 50	70 - 250
D	Deposits of loose-to-medium cohesionless soil (with or without some soft cohesive layers), or of predominantly soft-to-firm cohesive soil.	< 180	< 15	< 70
E	A soil profile consisting of a surface alluvium layer with v_s values of type C or D and thickness varying between about 5 m and 20 m, underlain by stiffer material with v_s > 800 m/s.			

- **Type of spectrum**: If the earthquakes that contribute most to the seismic hazard defined for the site for the purpose of probabilistic hazard assessment have a surface-wave magnitude, Ms, not greater than 5,5, it is recommended that the Type 2 spectrum is adopted. A simple formula to find the surface wave magnitude from the Richter magnitude scale ([29]) is:

$$M_S = -3{,}2 + 1{,}45.M_L \tag{4.14}$$

- **Direction**: If the spectrum is applied in X or Y direction, then this must be set to 'Horizontal'. If the spectrum is to be applied in the Z direction, then this property must be set to 'Vertical'.

Calculation protocol

In the calculation protocol of SCIA Engineer the intermediate results that were determined while calculating the global effect of a spectral loading can be found.
This paragraph describes the formulas that have been used to determine those intermediate results.

Natural circular frequency and modal shape

Mass matrix	$[\mathrm{M}]_\mathrm{D}$
Mass vector	$\{m\} = [M]_D \cdot \{1\}$
Natural circular frequency of mode shape j	$\omega_{(j)}$
Natural normalized modal shape	$\{\phi\}_{(j)}$, Avec $\{\phi\}^T_{(j)} \cdot [M]_D \cdot \{\phi\}_{(j)} = M_{(j)} = 1$
Total mass in k^th direction	$M_{k,tot}$
Acceleration response spectrum	$S_{a,k,(j)}$
Direction	k
Total number of directions	NK

Participation factor of the mode shape j in direction k

Participation factor	$\gamma_{k,(j)} = \dfrac{\{\phi_k\}^T \cdot \{m\}}{M_{(j)}} = \{\phi_k\}^T \cdot \{m\}$
Effective mass	$M_{k,ef,(j)} = \gamma^2_{k,(j)} \cdot M_{(j)} = \gamma^2_{k,(j)}$
Participation mass ratio	$L_{k,(j)} = \dfrac{M_{k,ef,(j)}}{M_{k,tot}}$

__Mode coefficient for mode j__

Mode coefficient in k[th] direction	$G_{k,(j)} = \dfrac{S_{a,k,(j)} \cdot \gamma_{k,(j)}}{\omega_{(j)}^2}$
Total mode coefficient	$G_{(j)} = \dfrac{\sum_{k=1}^{NK} S_{a,k,(j)} \cdot \gamma_{k,(j)}}{\omega_{(j)}^2}$

__Response of mode shape j__

Displacement	$\{u\}_{(j)} = G_{(j)} \cdot \{\phi\}_{(j)}$ $\{u_k\}_{(j)} = G_{k,(j)} \cdot \{\phi_k\}_{(j)}$
Acceleration	$\{\ddot{u}\}_{(j)} = \omega_{(j)}^2 \cdot G_{(j)} \cdot \{\phi\}_{(j)}$ $\{\ddot{u}_k\}_{(j)} = \omega_{(j)}^2 \cdot G_{k,(j)} \cdot \{\phi_k\}_{(j)} = S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \{\phi_k$
Lateral force in node i for k direction	$F_{i,k,(j)} = m_{i,k,(j)} \cdot S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \phi_{i,k,(j)}$
Shear force in direction k	$F_{k,(j)} = \sum_i F_{i,k,(j)} = \{\ddot{u}_k\}_{(j)}^T \cdot \{m\} = S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \{\phi_k\}_{(j)}^T \cdot$ $F_{k,(j)} = S_{a,k,(j)} \cdot \gamma_{k,(j)}^2$
Overturning moment in node i for direction k	$M_{i,k,(j)} = m_{i,k} \cdot S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \phi_{i,k,(j)} \cdot z_i$
Overturning moment in direction k	$M_{k,(j)} = \sum_i M_{i,k,(j)} = \sum_i \left(m_{i,k} \cdot S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \phi_{i,k,(j)} \cdot z_i \right.$ $M_{k,(j)} = S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \sum_i \left(m_{i,k} \cdot \phi_{i,k,(j)} \cdot z_i \right.$

The calculation of these parameters will be illustrated with an example further in this chapter.

Modal combination methods

Modal combination methods are used to calculate the response R of a seismic analysis. The term "response" (R) refers to the results obtained by a seismic analysis, i.e. displacements, velocities, accelerations, member forces and stresses. Because the differential equations were uncoupled, a result will be obtained for each mode j.

To obtain the global response R_{tot} of the structure, the individual modal responses $R_{(j)}$ have to be combined.

The modal combination methods that are used in SCIA Engineer are:

- SRSS method (Square Root of Sum of Squares)

$$R_{tot} = \sqrt{\sum_{j=1}^{N} R_{(j)}^2}$$

  Where $R_{(j)}$ the response of mode j.

- CQC method (Complete Quadratic Combination)

$$R_{tot} = \sqrt{\sum_{i=1}^{N} \sum_{j=1}^{N} R_{(i)} . \rho_{i,j} . R_{(j)}}$$

  Where:

  $R_{(i)}$, $R_{(j)}$ the response of mode i and j
  $\rho_{i,j}$: modal cross correlation coefficients

$$\rho_{i,j} = \frac{8.\sqrt{\xi_i\xi_j}.(\xi_i + r\xi_j).r^{\frac{3}{2}}}{(1-r^2)^2 + 4.\xi_i\xi_j r(1+r^2) + 4.(\xi_i^2+\xi_j^2).r^2}$$

r:

$$r = \frac{\omega_j}{\omega_i}$$

ξ_i, ξ_j: damping ratio for mode i and j

This method is based on both modal frequency and modal damping. The CQC-method thus requires the input of additional data: a Damping Spectrum to define the damping ratio for each mode.
In many cases however, there is no procedure to calculate the damping ratio for the higher modes. Most of the time, the same damping ratio is then used for all modes [17].

- MAX method

$$R_{tot} = \sqrt{R^2_{(jMAX)} + \sum_{j=1}^{N} R^2_{(j)}}$$

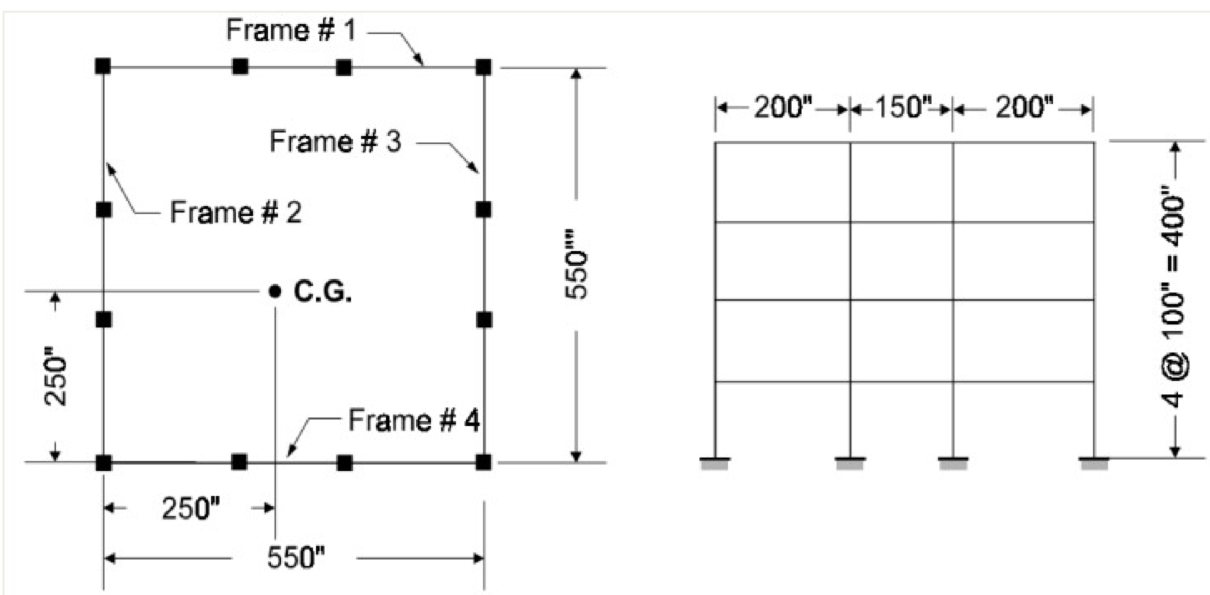
where:

$R_{(j)}$ the response of mode j.
$R_{(jMAX)}$ the maximum response of all modes.

Eurocode 8 [7] prescribes the SRSS-method. However this method may only be applied if all relevant modal responses are independent of each other. This is met if the period of mode j is smaller or equal to 90% of the period of mode i.
If modal responses are not independent of each other a more accurate procedure like the CQC-method needs to be used.
The following numerical example shows this difference between SRSS and CQC.

Example 03-1.esa

A four-storey symmetrical building is modelled in a 3D analysis (from [18], p.15-9). The building is symmetrical; however, the centre of mass of all floors is located 25 inches from the geometric centre of the building.



The structure has the following natural frequencies for the first 5 modes:

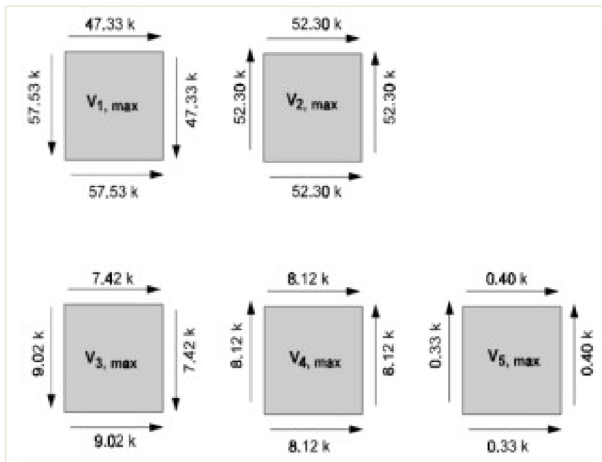
Mode 1: f = 13,87 Hz
Mode 2: f = 13,93 Hz

Mode 3: f = 43,99 Hz
Mode 4: f = 44,19 Hz
Mode 5: f = 54,42 Hz

It is clear that modes 1 & 2 and 3 & 4 are very closely spaced. It is typical for most three-dimensional building structures that they are designed to resist earthquakes from both directions equally. Therefore the similar eigenmodes in X and Y-direction have almost the same natural frequencies.

Because of the small mass eccentricity, which is normal in real structures, the fundamental mode shape has x, y, as well as torsion components. Therefore, the model represents a very common three-dimensional building system.

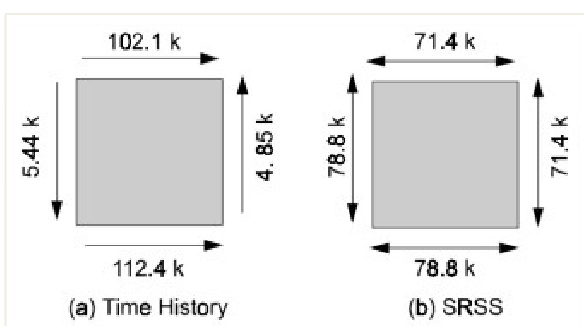
The building was subjected to one component of the Taft 1952 earthquake. An exact time history analysis using all 12 modes and a response spectrum analysis were conducted. The maximum modal base shears in the four frames for the first five modes are shown in the figure below.

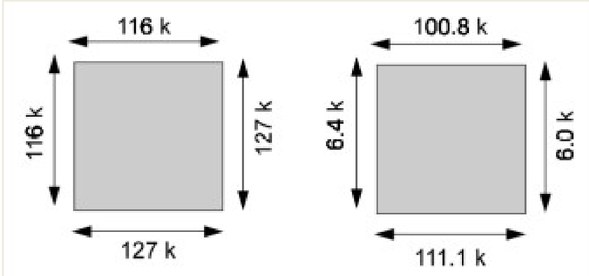


The maximal base shear forces are:
Mode 1: F = -57,53 kN
Mode 2: F = 52,30 kN
Mode 3: F = -9,02 kN
Mode 4: F = 8,12 kN
Mode 5: F = 0,33 kN

To obtain the Global Response, these modal responses are combined using both the SRSS-method and the CQC-method as well as a sum of the Absolute Values.

Now the maximum total base shears using different methods are compared:
- The time history base shears are exact.
- The SRSS method produces base shears that under-estimate the exact values in the direction of the loads by approximately 30 percent and overestimate the base shears normal to the loads by a factor of 10.
- The sum of the absolute values grossly over-estimates all results.
- The CQC-method produces very realistic values that are close to the exact time history solution.





(c) Sum of Absolute Values (d) CQC

Results for the global Base Shear:

	Lateral	Transversal
Exact solution using Time-History Analysis	112,4kN	5,44kN
Global Base Shear using SRSS	78,8kN	78,8kN
Global Base Shear using CQC	111,1kN	6,37kN

In this example, the SRSS-method overestimates the Base Shear by a factor of 10.

For the **CQC-method**, the following Modal Cross Correlation coefficients $\boldsymbol{\rho_{i,j}}$ are used with a damping ratio $\boldsymbol{\xi_{i,j}}$ of **5%**.

Mode	1	2	3	4	5
1	1,000	0,998	0,006	0,006	0,004
2	0,998	1,000	0,006	0,006	0,004
3	0,006	0,006	1,000	0,998	0,180
4	0,006	0,006	0,998	1,000	0,186
5	0,004	0,004	0,180	0,186	1,000

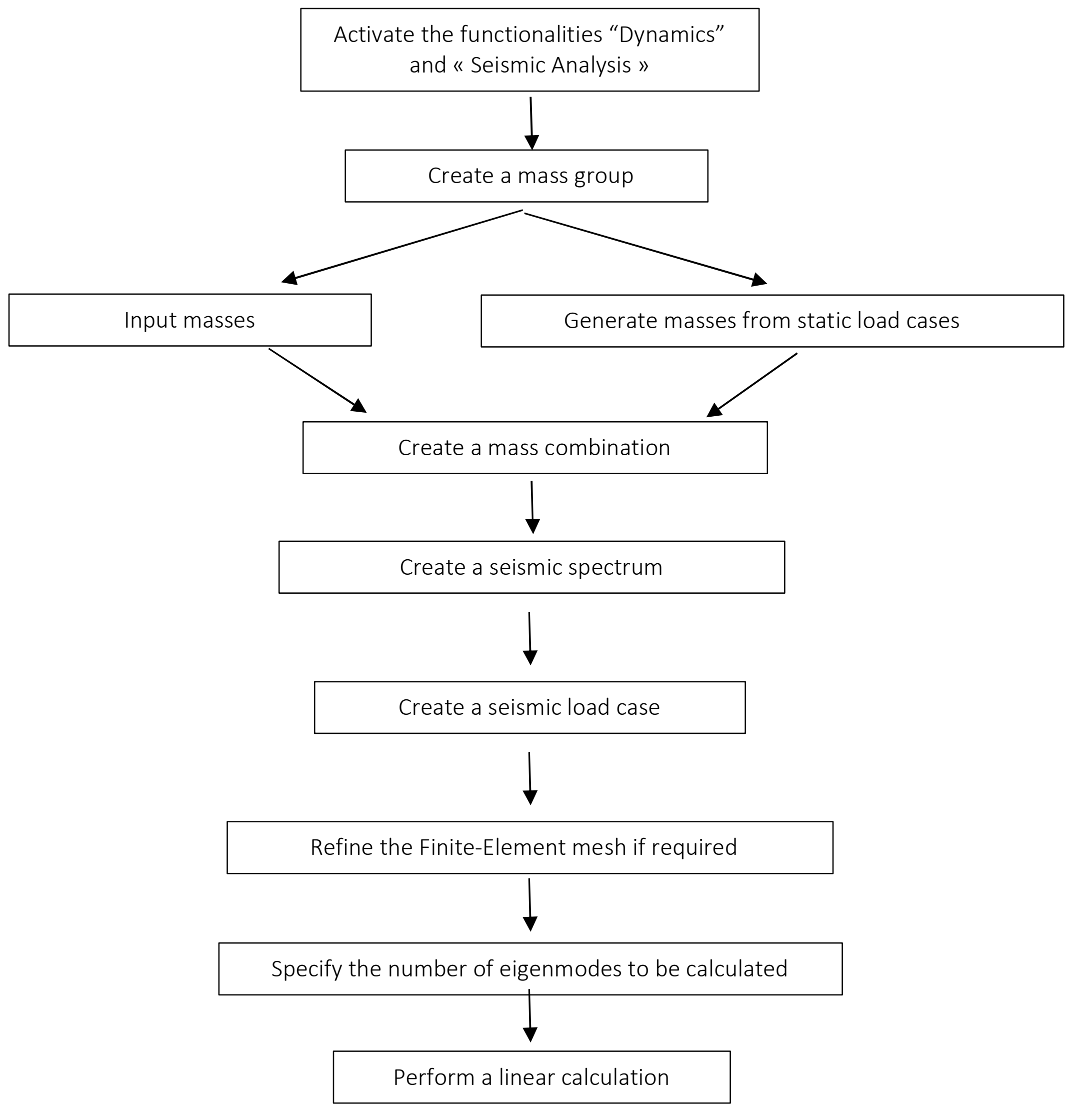
It is of importance to note the existence of the relatively large off-diagonal terms that indicate which modes are coupled.

If one notes the signs of the modal base shears shown on the previous page, it is apparent how the application of the CQC method allows the sum of the base shears in the direction of the external motion to be added directly. In addition, the sum of the base shears, normal to the external motion, tend to cancel.

The ability of the **CQC-method** to recognize the relative sign of the terms in the modal response is the key to the elimination of errors in the **SRSS-method**.

3.3 Seismic calculation in SCIA Engineer

The following diagram show the required steps to perform a Spectral Analysis calculation:

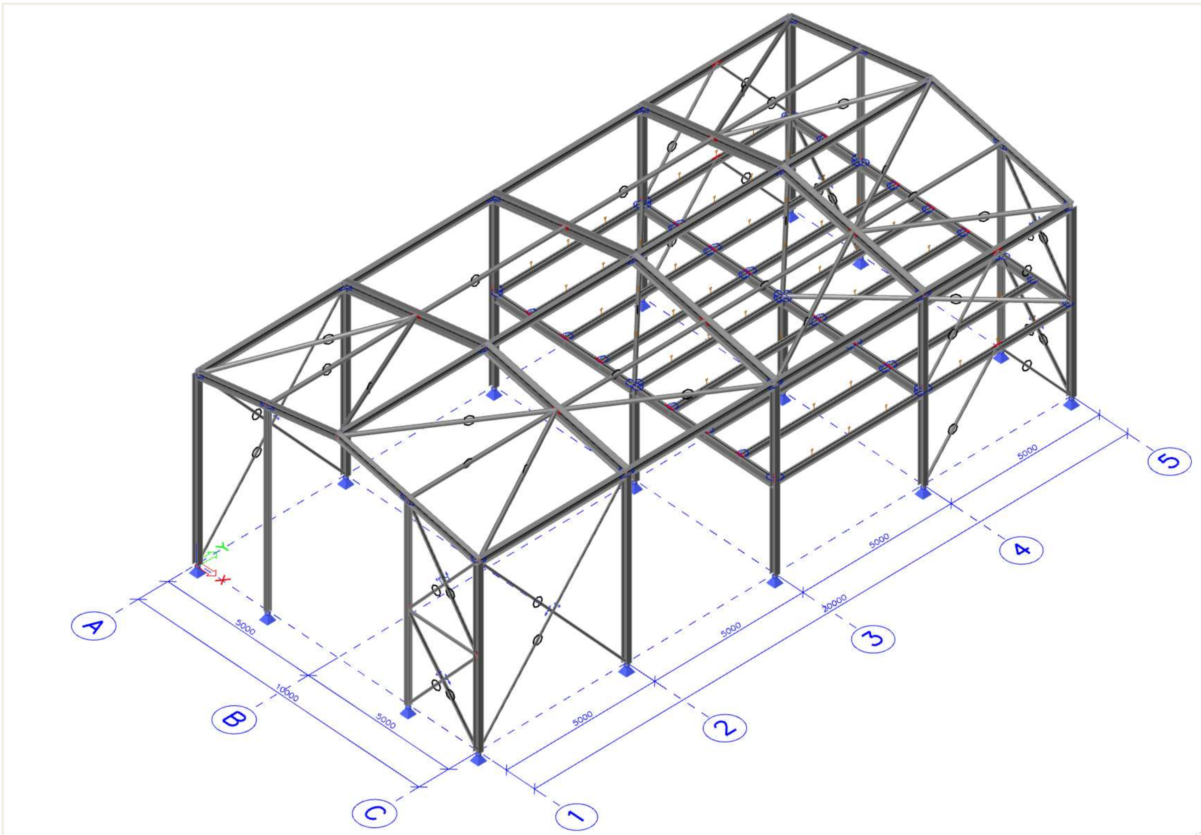


As specified in the theory, the dynamic calculation is transformed to an equivalent static calculation. Therefore, a Linear Calculation needs to be executed. During this calculation, the Free Vibration Calculation will also be performed since this data is needed for the Seismic results.

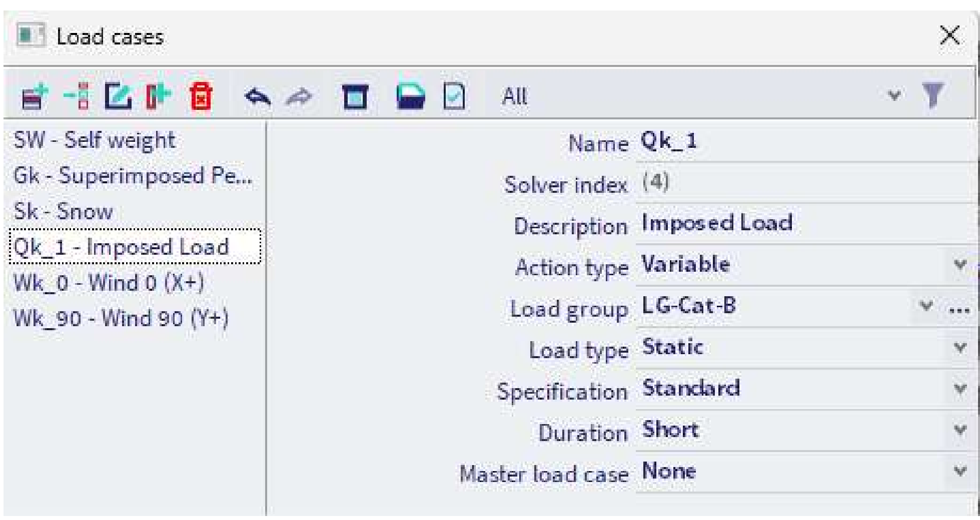
The diagram is illustrated in the following examples.

Example 03-2.esa

In this example a steel industrial structure is modelled.



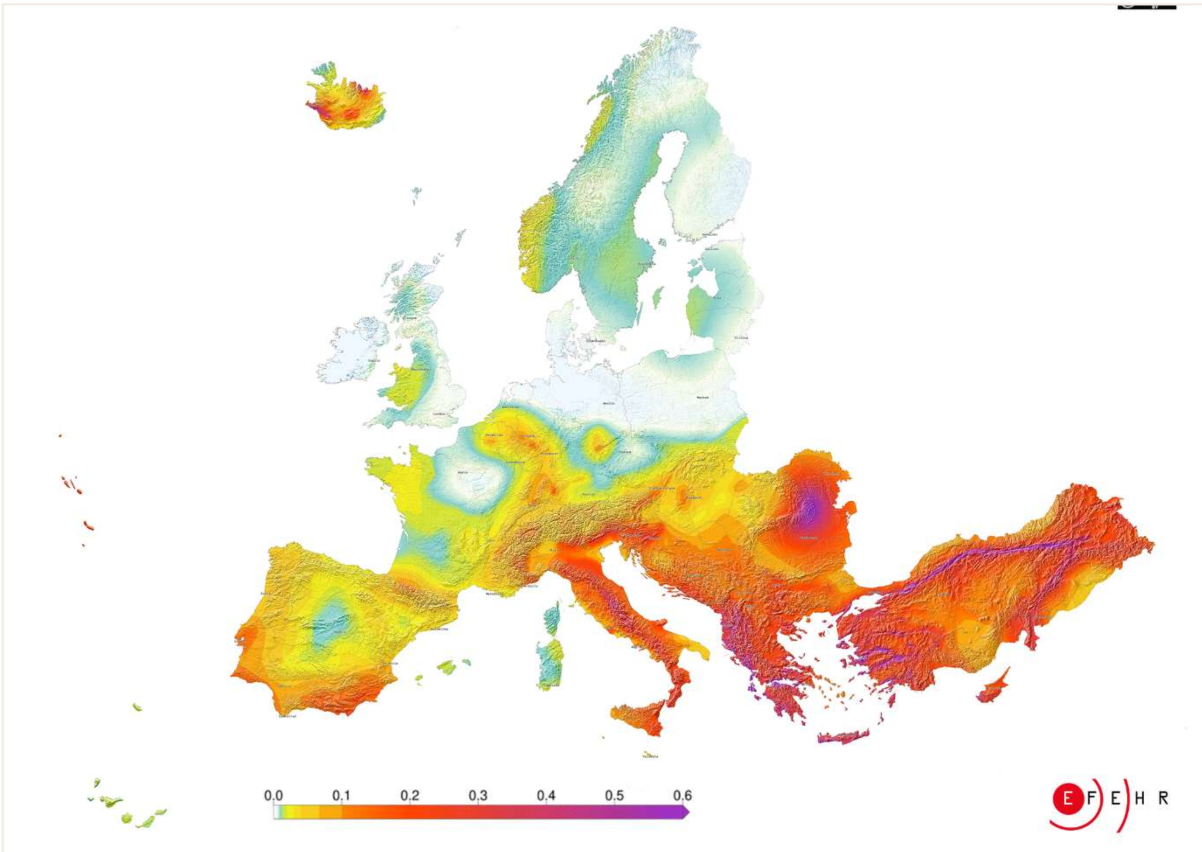
Static load cases are created: the imposed load is CAT-B, offices.



The structure will be subjected to an earthquake in X and Y-direction according to Eurocode 8, using a Design Response Spectrum for Ground Type C with a behaviour factor of 1,5. The coefficient of acceleration is 0,20g.
Below we reported a map of Europe to show how this acceleration relates to the values in the different countries.

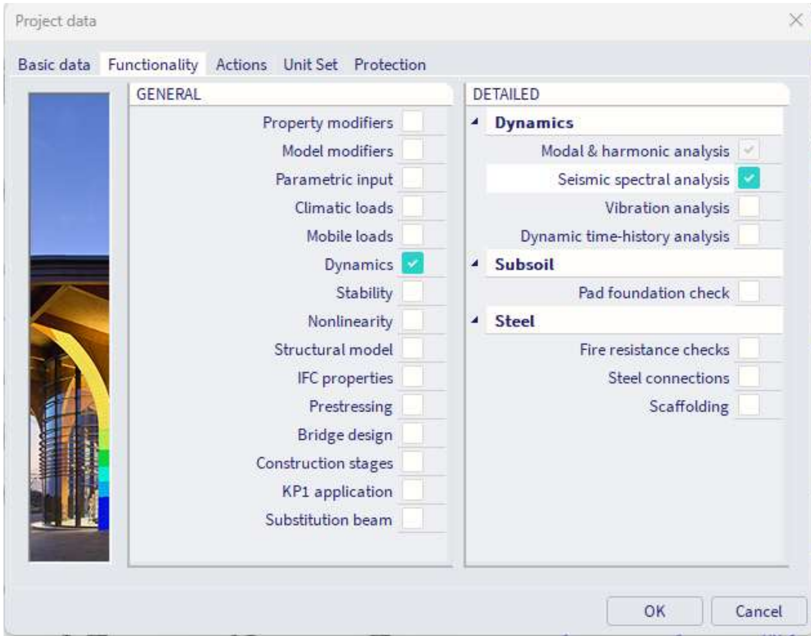
The earthquake hazard map of Europe

The earthquake hazard map shows the expected level of ground shaking at a specific location due to future potential earthquakes that might occur locally or at a greater distance. Ground shaking is expressed as Peak Ground Acceleration (PGA), normally given in the percentage of "g", the Earth's gravitational acceleration. The values displayed on the earthquake hazard map of Europe are based on the calculations of Europe's updated earthquake hazard model (ESHM20).



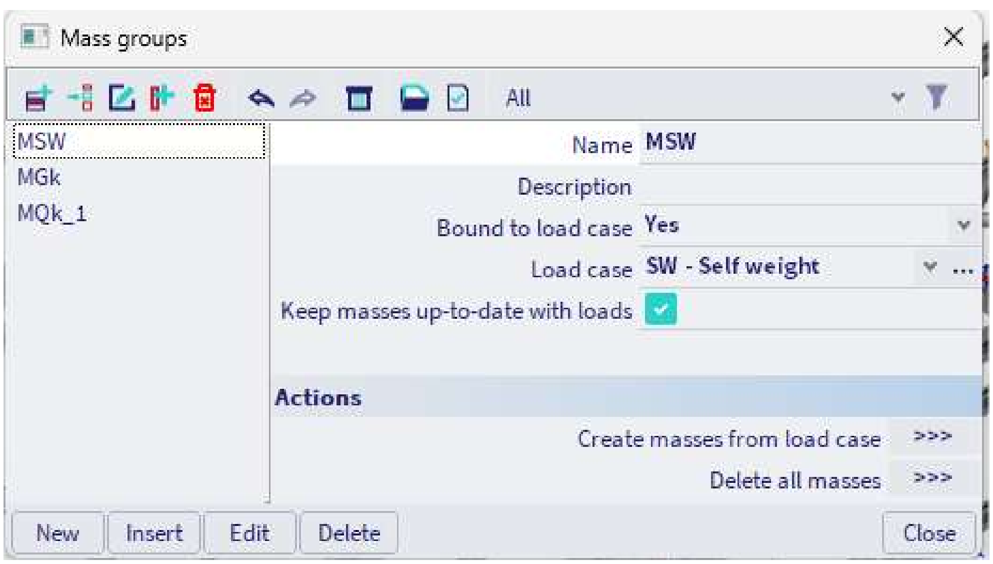
Step 1: functionality

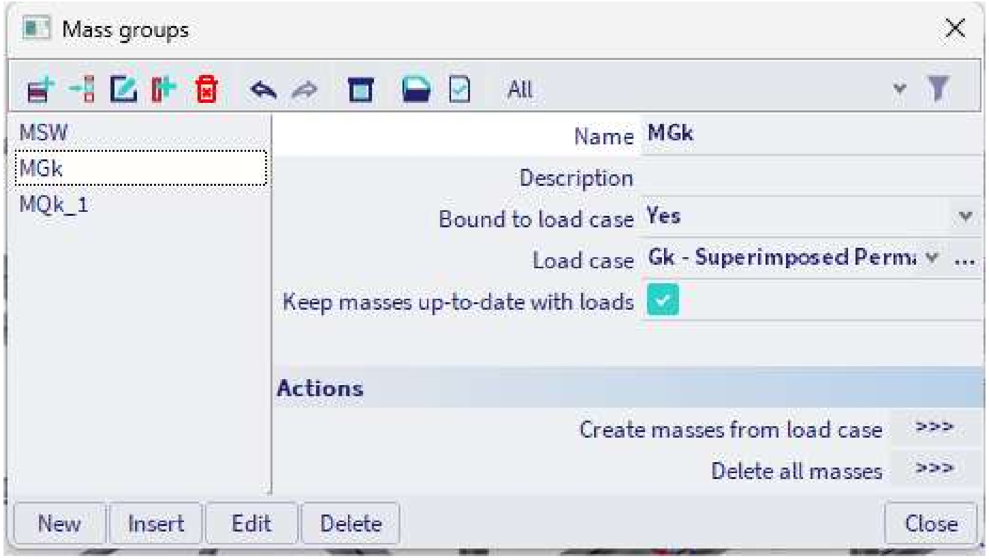
The first step in the Dynamic calculation is to activate the functionality Dynamics on the Functionality tab in the Project Data. In order to execute a Seismic calculation, also the Seismic spectral analysis functionality needs to be activated:

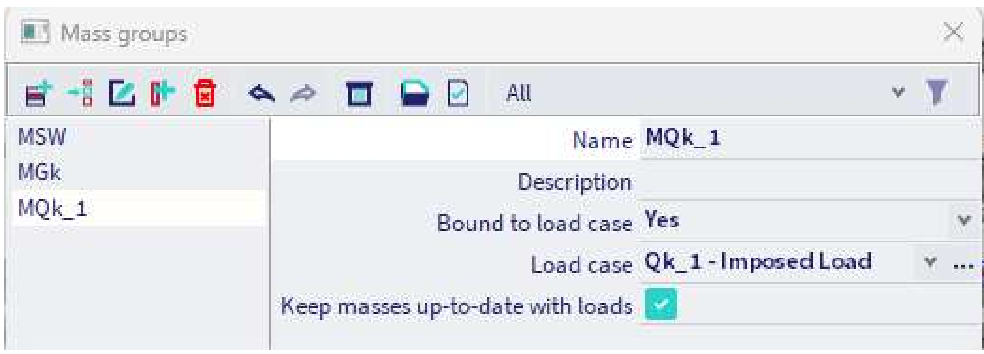


Step 2: mass group

The second step is to create a "Mass Group":







NB:

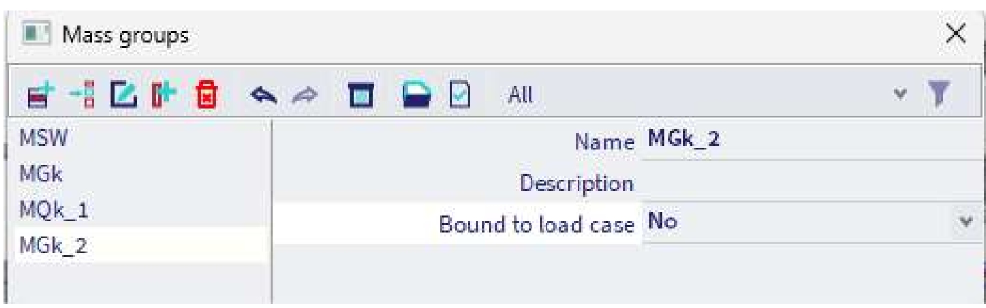
- It is recommended to use the option: "Keep masses up-to date with loads": this makes sure that all changes in the load case are taken into account in the transformation to masses.
- These masses are NOT directly applied to the analysis, it is only a transformation from load to mass. For example, we can transform the 100% of the imposed load Q1, but we will apply only a percentage of it for the analysis.
- The transformation of the masses of snow load and wind load is not needed for this case, please check the psi2 values of your structure.
- For additional masses, even if they're related to a specific load case (ex. Permanent load) we suggest to add a separated mass group, as for Step 3

Step 3: additional masses

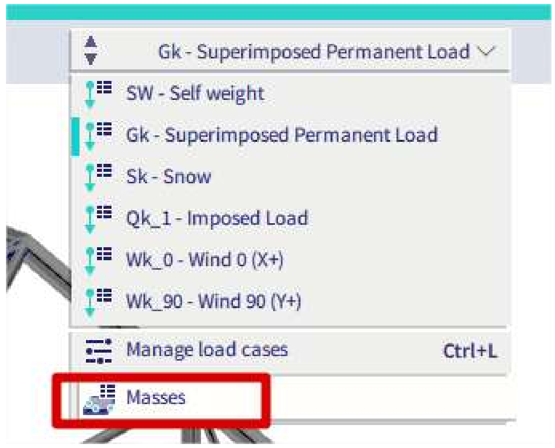
Imagine to take into account an additional acceleration of panels which are often used as cladding:

The static load goes to the foundation, we don't take it into consideration in the analysis.

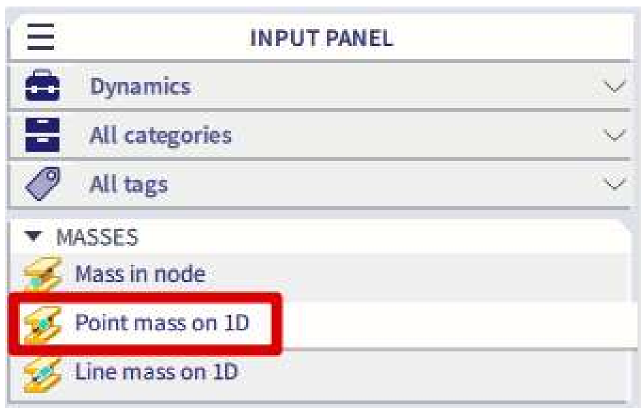
For seismic purposes, it needs to be applied instead a mass acting out of the plane, considering half of the mass of the panel. For this reason, a separate mass group is created:



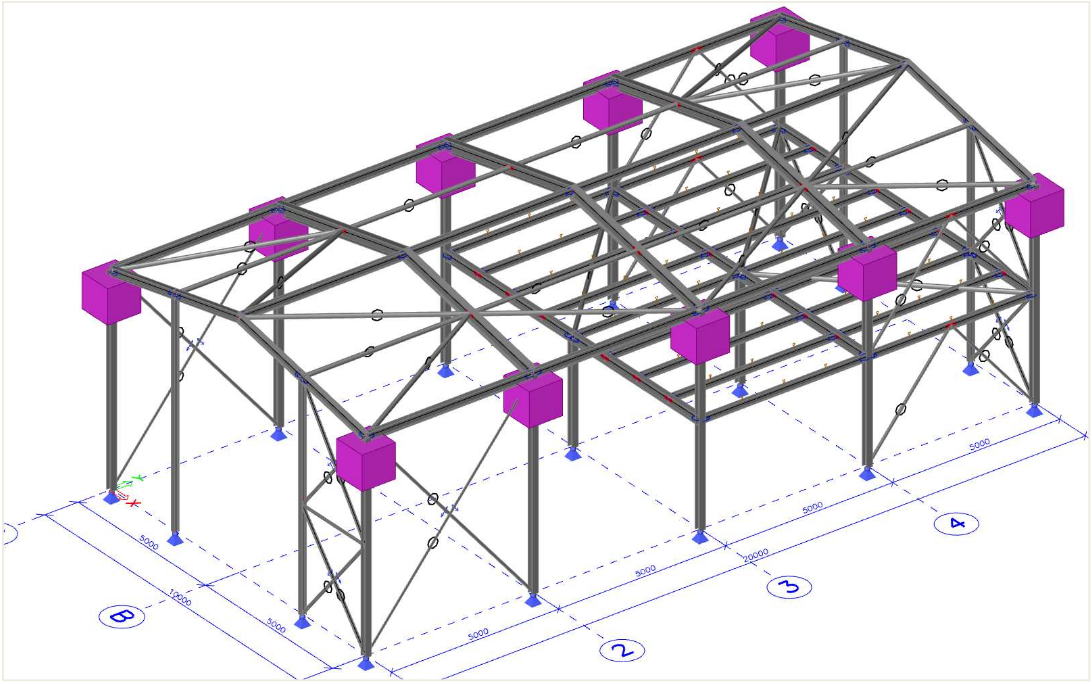
Go to the option "masses" and select MGk_2:



Apply a “Point in mass on 1D”:

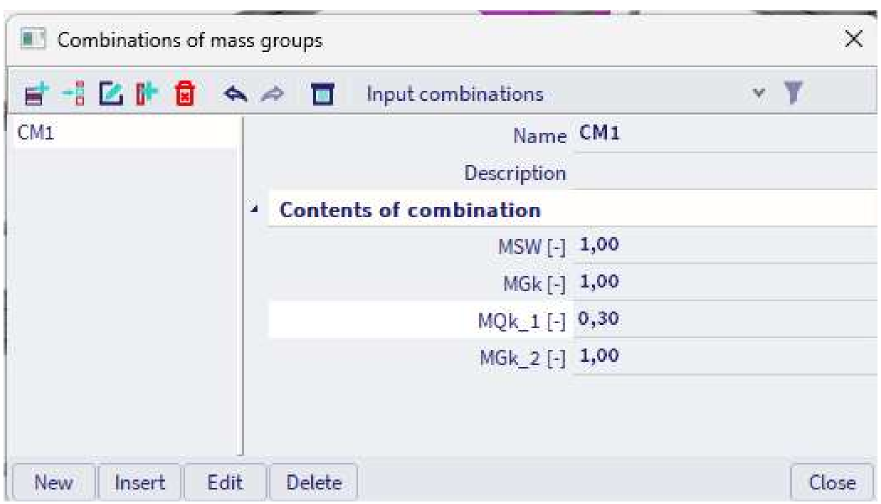


The panel could be 20cm thick, and considered of being in concrete material, then a mass of 2500(kg/m^3)x0.2(m)x6(m)x5(m)/2=7500 kg can be applied to the central columns, and half of it to the sides. It will activate an acceleration only in the X direction:

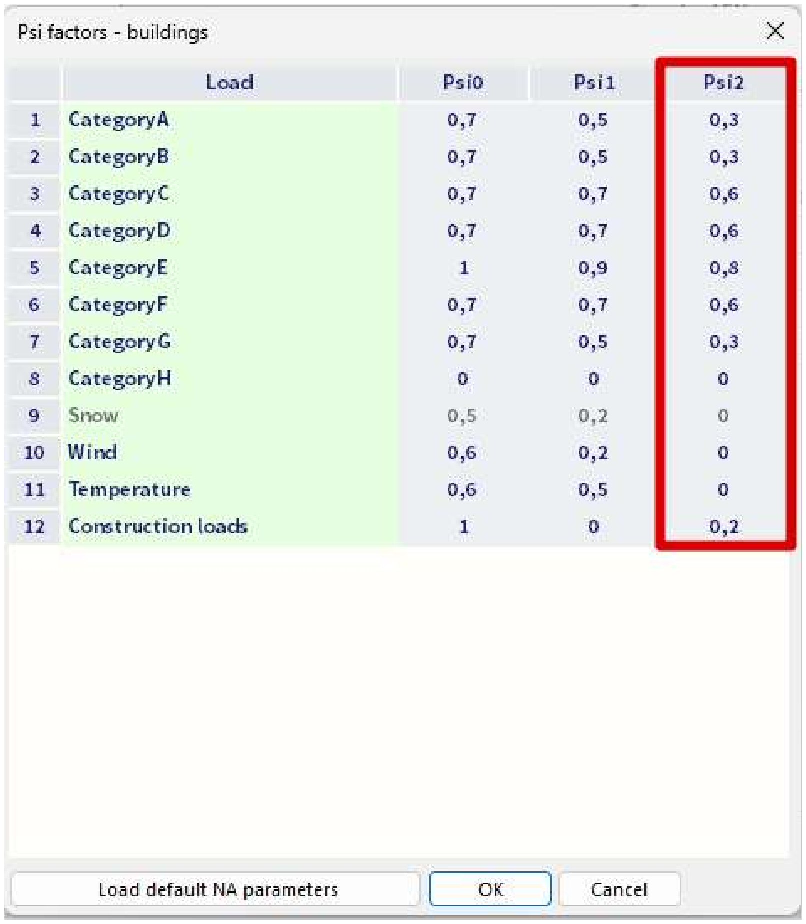


Step 4: combination of mass groups

Next, the Mass Group is put within a Combination of Mass Groups, which can be used for defining the Seismic load case.



The coefficient 0,3 is the psi2 factor that you can find in the code; this is also the reason why snow and wind load are not taken into account in this specific case, as the coefficient is null:



Psi factors - buildings

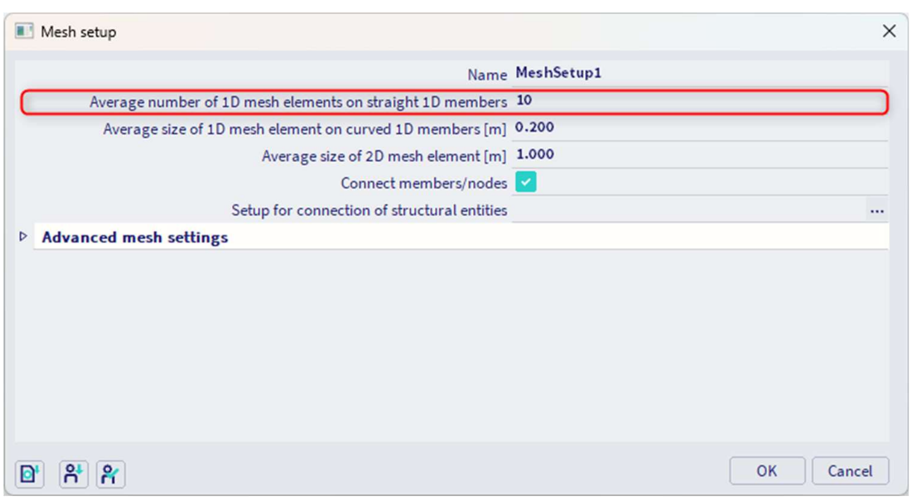
	Load	Psi0	Psi1	Psi2
1	CategoryA	0,7	0,5	0,3
2	CategoryB	0,7	0,5	0,3
3	CategoryC	0,7	0,7	0,6
4	CategoryD	0,7	0,7	0,6
5	CategoryE	1	0,9	0,8
6	CategoryF	0,7	0,7	0,6
7	CategoryG	0,7	0,5	0,3
8	CategoryH	0	0	0
9	Snow	0,5	0,2	0
10	Wind	0,6	0,2	0
11	Temperature	0,6	0,5	0
12	Construction loads	1	0	0,2

Load default NA parameters OK Cancel

Step 5: finite element mesh

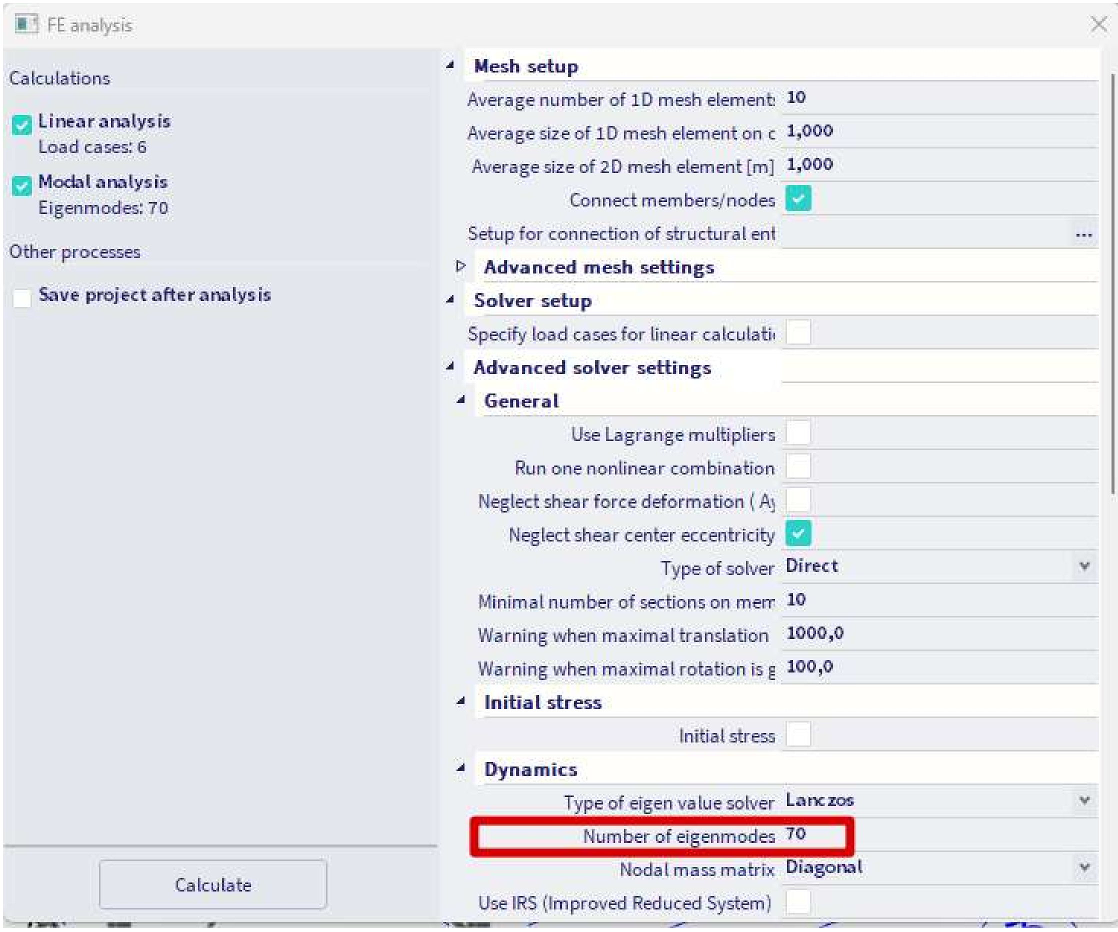
As specified in the previous chapters, the finite element mesh needs to be refined to obtain precise results. This can be done through the main menu Tools / Calculation & Mesh / Mesh settings.

For this example it will be shown the difference between the default mesh, not refined, which will position the masses on the nodes of the structure, and a mesh set to 10 finite elements.



Step 6: number of frequencies

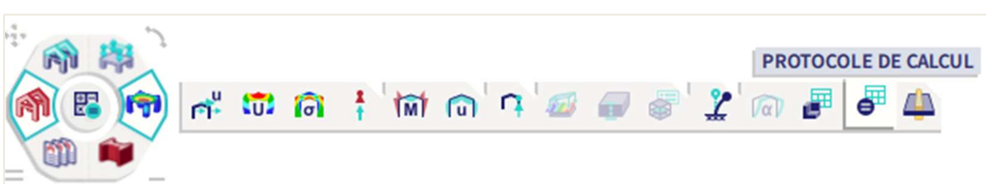
The last step before the seismic results can be checked, is setting a sufficient amount of eigenmodes to be calculated. For this example, 70 eigenmodes are chosen.
In the main menu Tools / Calculation & Mesh / Solver settings, the number of frequencies is thus set to 70.



Step 7: results of the modal analysis

A linear calculation and eigenmodes have been performed.

Open the calculation protocol:



Select eigen frequency, and open the Result Table.

For the case with a default mesh with 1 finite element:

- Modes in X direction: mode number 1 is predominant, with a period of 0,52s

RESULTS TABLE | Calculation protocol

	Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W
×								>0,01		
1	1	12.1396	0,52	1,93	-326,9118	-5,6241	-15,7843	0,7469	0,0004	0,
2	4	16.6976	0,38	2,66	113,4267	60,9072	-30,2348	0,0899	0,0527	0,
3	5	16.7299	0,38	2,66	46,0474	-172,8662	-6,0539	0,0148	0,4249	0,
4	9	22.6535	0,28	3,61	-83,1426	20,9254	-9,0844	0,0483	0,0062	0,
5	13	35.0602	0,18	5,58	55,9018	-8,8693	-7,6229	0,0218	0,0011	0,
6	15	39.6978	0,16	6,32	46,0119	-85,7665	1,7761	0,0148	0,1046	0,
7	34	92.8945	0,07	14,78	-45,3179	-2,4346	-3,0679	0,0144	0,0001	0,
8								0,9999	0,9982	0,

- Modes in Y direction: mode number 5 is predominant

RESULTS TABLE | Calculation protocol

	Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	W
×									>0,01		
1	3	15.2484	0,41	2,43	-36,2105	-40,1936	-5,4103	0,0092	0,0230	0,0004	0
2	4	16.6976	0,38	2,66	113,4267	60,9072	-30,2348	0,0899	0,0527	0,0130	0
3	5	16.7299	0,38	2,66	46,0474	-172,8662	-6,0539	0,0148	0,4249	0,0005	0
4	14	38.6746	0,16	6,16	-28,0011	-63,3527	2,3267	0,0055	0,0571	0,0001	0
5	15	39.6978	0,16	6,32	46,0119	-85,7665	1,7761	0,0148	0,1046	0,0000	0
6	20	45.8752	0,14	7,30	-14,8875	-39,9853	-0,2487	0,0015	0,0227	0,0000	0
7	22	48.4711	0,13	7,71	-26,3707	-83,1424	4,2267	0,0049	0,0983	0,0003	0
8	23	51.4654	0,12	8,19	9,5043	-65,3542	4,9401	0,0006	0,0607	0,0003	0
9	25	54.667	0,11	8,70	1,3549	54,8131	3,2270	0,0000	0,0427	0,0001	0
10	26	56.1314	0,11	8,93	-1,3843	-55,4590	-1,4799	0,0000	0,0437	0,0000	0
11	32	78.1794	0,08	12,44	-7,2250	-28,6146	-1,1677	0,0004	0,0116	0,0000	0
12	36	110.924	0,06	17,65	-2,2364	27,2012	-0,2901	0,0000	0,0105	0,0000	0
13								0,9999	0,9982	0,7029	0

- Torsional modes: mode number 2 is predominant

RESULTS TABLE | Calculation protocol

	Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	W_{xi_R}/W_{...	W_{yi_R}/W_{...	W_{zi_R}/W_{...
×													>0,01
1	1	12.1396	0,52	1,93	-326,9118	-5,6241	-15,7843	0,7469	0,0004	0,0035	0,0006	0,0908	0,0926
2	2	14.4675	0,43	2,30	-0,7815	15,5669	-0,2560	0,0000	0,0034	0,0000	0,0010	0,0008	0,0114
3	3	15.2484	0,41	2,43	-36,2105	-40,1936	-5,4103	0,0092	0,0230	0,0004	0,0284	0,0142	0,3395
4	9	22.6535	0,28	3,61	-83,1426	20,9254	-9,0844	0,0483	0,0062	0,0012	0,0358	0,0926	0,0681
5	10	24.2666	0,26	3,86	-35,3562	-2,7927	-6,0097	0,0087	0,0001	0,0005	0,0089	0,0483	0,2254
6	15	39.6978	0,16	6,32	46,0119	-85,7665	1,7761	0,0148	0,1046	0,0000	0,0292	0,0219	0,0372
7	20	45.8752	0,14	7,30	-14,8875	-39,9853	-0,2487	0,0015	0,0227	0,0000	0,0012	0,0013	0,0347
8	22	48.4711	0,13	7,71	-26,3707	-83,1424	4,2267	0,0049	0,0983	0,0003	0,0010	0,0026	0,0529
9	24	52.9929	0,12	8,43	-12,4268	9,7083	14,3717	0,0011	0,0013	0,0029	0,0027	0,0013	0,0280
10	25	54.667	0,11	8,70	1,3549	54,8131	3,2270	0,0000	0,0427	0,0001	0,0028	0,0002	0,0166
11	26	56.1314	0,11	8,93	-1,3843	-55,4590	-1,4799	0,0000	0,0437	0,0000	0,0008	0,0001	0,0197
12	34	92.8945	0,07	14,78	-45,3179	-2,4346	-3,0679	0,0144	0,0001	0,0001	0,0001	0,0213	0,0287
13	43	159.31	0,04	25,36	22,9509	2,9168	5,0506	0,0037	0,0001	0,0004	0,0028	0,0078	0,0110
14								0,9999	0,9982	0,7029	0,6001	0,6255	0,9994

For the case with a division in 10 finite elements:

- Modes in X direction: mode number 2 is predominant, with period 0,52

RESULTS TABLE

Calculation protocol

	Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	W_{xi_R}/W_{...	W_{yi_R}/W_{...	W_{zi_R}/W_{...
×								>0,01					
1	1	8.57378	0,73	1,36	-72,1868	-1,7348	-0,5181	0,0356	0,0000	0,0000	0,0000	0,0052	0,0000
2	2	12.0156	0,52	1,91	-350,2376	-8,4999	-13,7209	0,8389	0,0010	0,0026	0,0002	0,0271	0,0512
3	8	16.0543	0,39	2,56	-40,5614	36,8988	0,8144	0,0113	0,0191	0,0000	0,0003	0,0001	0,0298
4	11	16.3134	0,39	2,60	45,8444	-32,9905	-23,7056	0,0144	0,0153	0,0079	0,0050	0,0267	0,0154
5	19	17.6595	0,36	2,81	69,4231	17,2837	-25,2146	0,0330	0,0042	0,0089	0,0288	0,0429	0,0047
6								0,9760	0,9077	0,6831	0,4598	0,4960	0,9371

- Modes in Y direction: mode number 18 is predominant

RESULTS TABLE

Calculation protocol

	Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_
×									>0,01	
1	6	15.8983	0,40	2,53	-26,9480	31,3679	2,4649	0,0050	0,0138	0,0001
2	8	16.0543	0,39	2,56	-40,5614	36,8988	0,8144	0,0113	0,0191	0,0000
3	11	16.3134	0,39	2,60	45,8444	-32,9905	-23,7056	0,0144	0,0153	0,0079
4	14	16.4551	0,38	2,62	8,9419	-46,6983	-0,3852	0,0005	0,0306	0,0000
5	18	16.7406	0,38	2,66	-4,8171	-164,6009	6,7860	0,0002	0,3804	0,0006
6	44	38.1416	0,16	6,07	9,6817	-96,1653	-2,0412	0,0006	0,1298	0,0001
7	48	42.2775	0,15	6,73	5,7650	66,3097	2,8578	0,0002	0,0617	0,0001
8	49	42.5598	0,15	6,77	0,2347	-39,6422	-1,8910	0,0000	0,0221	0,0001
9	50	43.2096	0,15	6,88	9,8754	46,4412	1,5344	0,0007	0,0303	0,0000
10	54	44.5481	0,14	7,09	-4,0485	40,6216	-2,3540	0,0001	0,0232	0,0001
11	56	45.2	0,14	7,19	3,3812	38,9549	1,3930	0,0001	0,0213	0,0000
12	65	47.1087	0,13	7,50	22,4855	-74,8635	-4,9955	0,0035	0,0787	0,0004
13								0,9760	0,9077	0,6831

- Torsional modes: mode number 4 is predominant

RESULTS TABLE

Calculation protocol

	Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	W_{xi_R}/W_{...	W_{yi_R}/W_{...	W_{zi_R}/W_{...	Da
×													>0,01	
1	2	12.0156	0,52	1,91	-350,2376	-8,4999	-13,7209	0,8389	0,0010	0,0026	0,0002	0,0271	0,0512	
2	3	14.236	0,44	2,27	2,2677	-23,1369	4,6742	0,0000	0,0075	0,0003	0,0086	0,0418	0,0917	
3	4	14.4053	0,44	2,29	13,4858	12,4810	-4,1178	0,0012	0,0022	0,0002	0,0069	0,0423	0,3011	
4	5	15.3223	0,41	2,44	-27,2163	6,5662	-0,8281	0,0051	0,0006	0,0000	0,0002	0,0003	0,0384	
5	8	16.0543	0,39	2,56	-40,5614	36,8988	0,8144	0,0113	0,0191	0,0000	0,0003	0,0001	0,0298	
6	11	16.3134	0,39	2,60	45,8444	-32,9905	-23,7056	0,0144	0,0153	0,0079	0,0050	0,0267	0,0154	
7	23	21.0359	0,30	3,35	-19,5717	5,3432	-6,5302	0,0026	0,0004	0,0006	0,0056	0,0107	0,0111	
8	25	23.0955	0,27	3,68	-17,8634	12,6151	16,4079	0,0022	0,0022	0,0038	0,0231	0,0069	0,2095	
9	28	28.618	0,22	4,55	-0,1034	14,9154	-15,4163	0,0000	0,0031	0,0033	0,0251	0,0000	0,0265	
10	34	31.2999	0,20	4,98	-24,7325	8,7435	6,0523	0,0042	0,0011	0,0005	0,0024	0,0015	0,0177	
11	45	39.9308	0,16	6,36	3,3508	8,0842	3,7743	0,0001	0,0009	0,0002	0,0097	0,0000	0,0244	
12	46	40.1965	0,16	6,40	23,0954	13,2263	-3,3446	0,0036	0,0025	0,0002	0,0030	0,0001	0,0288	
13	47	40.6764	0,15	6,47	14,7487	23,9743	1,4819	0,0015	0,0081	0,0000	0,0072	0,0000	0,0209	
14								0,9760	0,9077	0,6831	0,4598	0,4960	0,9371	

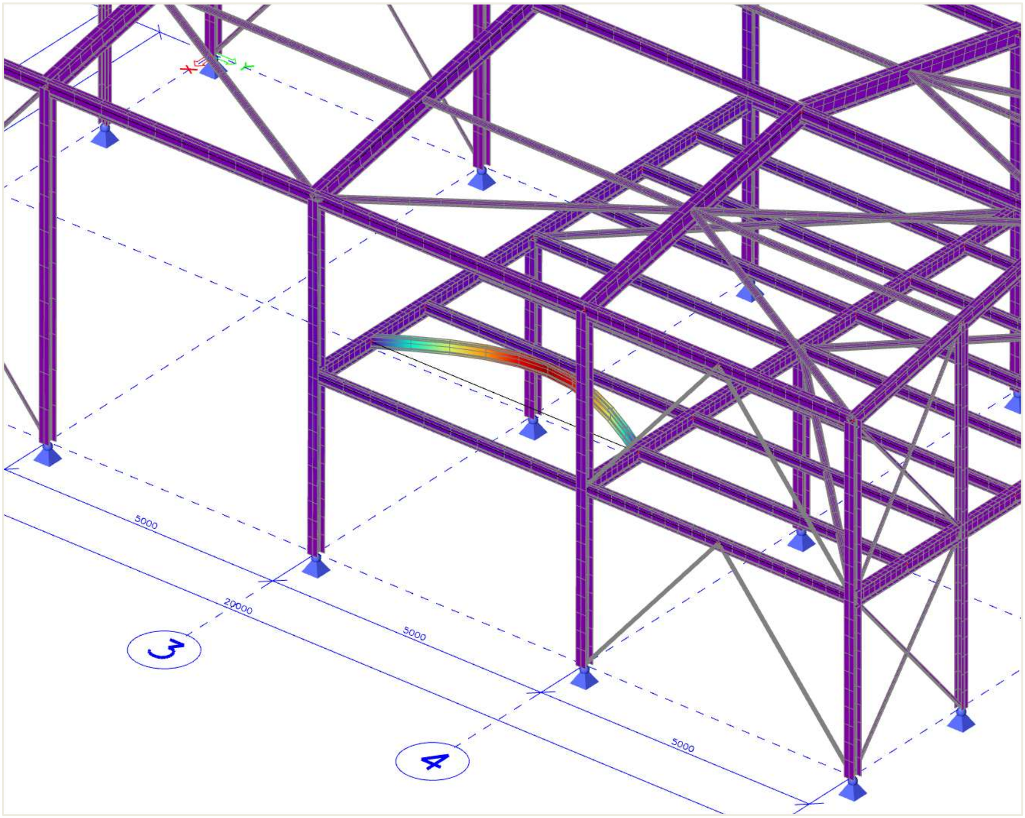
Conclusion:

With the masses in the nodes (1 finite element), we avoid local modes, and we need fewer modes to reach 90% of the activated mass, while the results remain quite similar. This approach could be used in specific cases, but it is suggested always to compare the results with a more detailed analysis.

By reading further, you will notice that there was a mistake in the model, that was detected only thanks to a more detailed analysis.

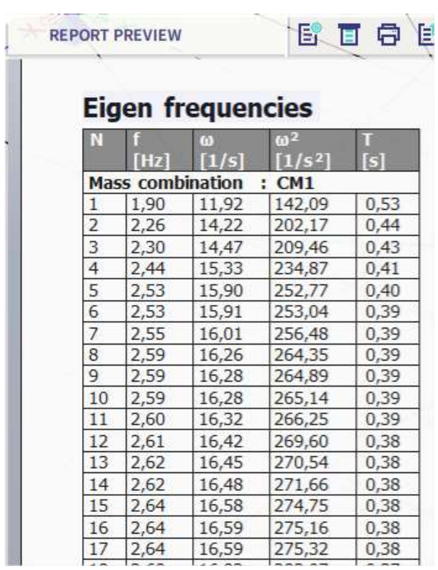
Notice that the mass of the wall is applied to the nodes of the columns in the first case, which produces the difference in the activated mass between the two cases.

Always check if there are any local modes before a global mode. In this case there is mode n.1:



The hinge was not defined correctly, as for the others. By changing it, the local mode disappear:

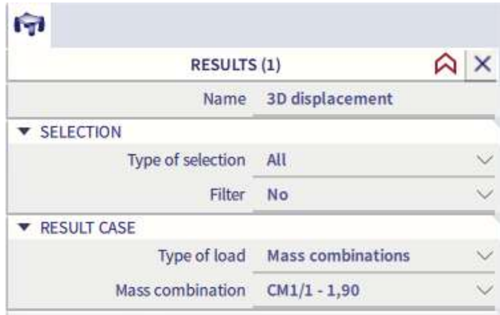
The eigenmodes can be displayed by this option:

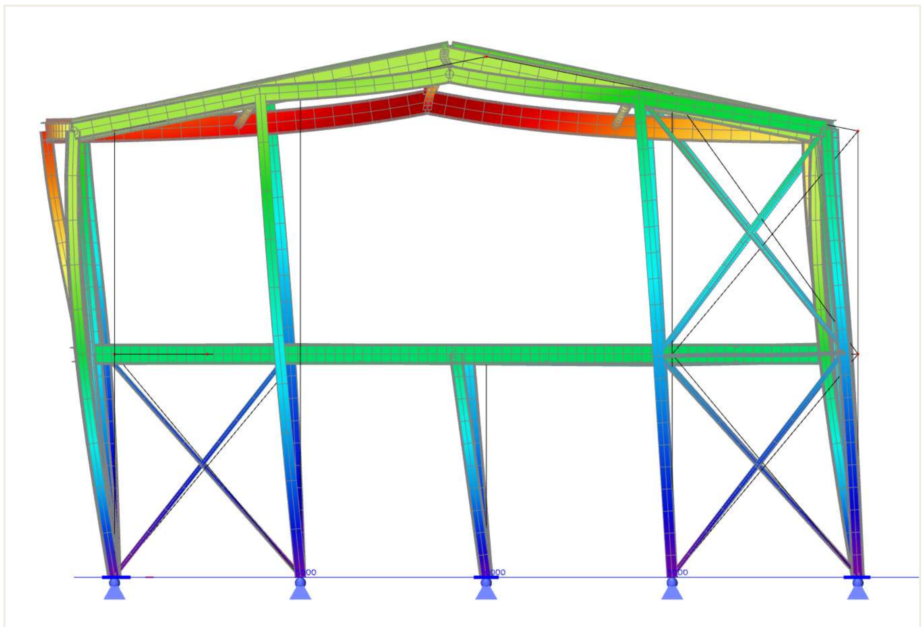


Eigen frequencies

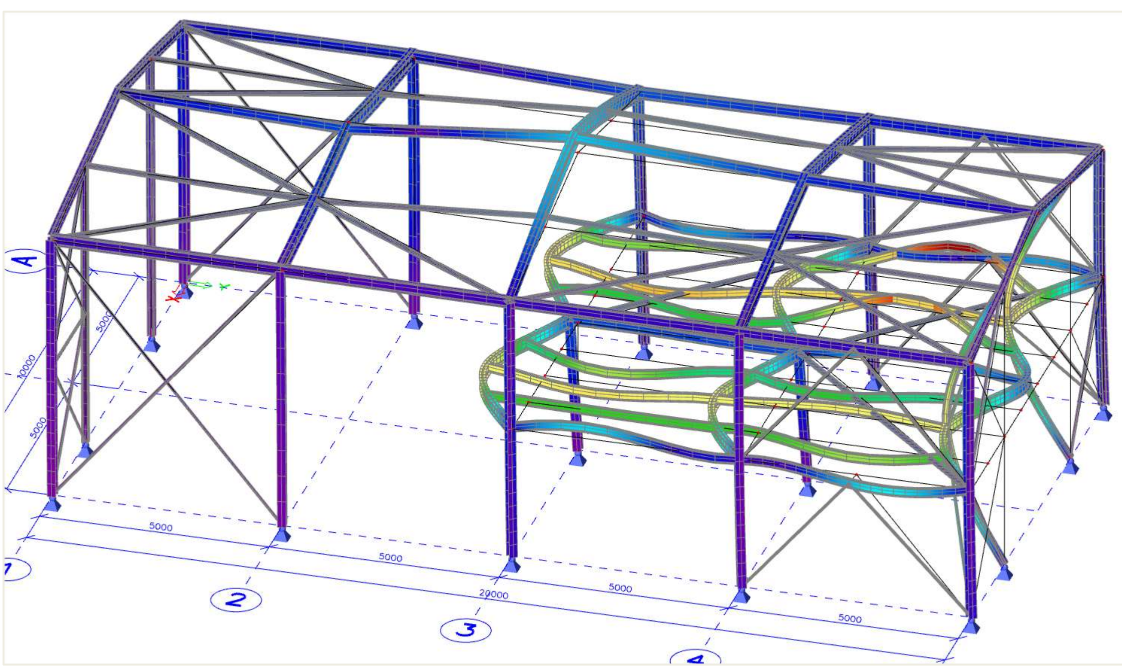
N	f [Hz]	ω [1/s]	ω² [1/s²]	T [s]
Mass combination : CM1				
1	1,90	11,92	142,09	0,53
2	2,26	14,22	202,17	0,44
3	2,30	14,47	209,46	0,43
4	2,44	15,33	234,87	0,41
5	2,53	15,90	252,77	0,40
6	2,53	15,91	253,04	0,39
7	2,55	16,01	256,48	0,39
8	2,59	16,26	264,35	0,39
9	2,59	16,28	264,89	0,39
10	2,59	16,28	265,14	0,39
11	2,60	16,32	266,25	0,39
12	2,61	16,42	269,60	0,38
13	2,62	16,45	270,54	0,38
14	2,62	16,48	271,66	0,38
15	2,64	16,58	274,75	0,38
16	2,64	16,59	275,16	0,38
17	2,64	16,59	275,32	0,38

The **deformed structure** can be shown to view the eigenmodes using the **3D displacements**:

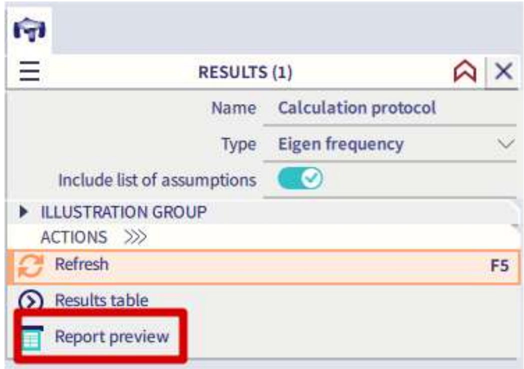




Mode in Y direction:



To check if the number of modes is sufficient, we have to have a look at the **calculation protocol** for the **eigen frequencies**.



Calculation protocol

Solution of Free vibration

Number of 2D elements	0
Number of 1D elements	1185
Number of mesh nodes	1111
Number of equations	6666
Combination of mass groups	CM1
Modification group	None
Number of frequencies	70
Method	Lanczos
Type of analysis model	Standard
Nodal mass matrix	Diagonal

Sum of masses

	Mass type	X [kg]	Y [kg]	Z [kg]
CM1	Moving mass	146221,6	71221,6	71221,6
CM1	Total mass	146543,7	71543,7	71543,7

Relative modal masses

Mode	ιega [rad	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	xi_R/W_{xtot}	yi_R/W_{ytot}	zi_R/W_{ztot}
1	11.9203	0,53	1,90	357,7570	8,8339	13,2560	0,8753	0,0011	0,0025	0,0002	0,0191	0,0428
2	14.2192	0,44	2,26	-3,0020	22,6920	-6,9358	0,0001	0,0072	0,0007	0,0136	0,0873	0,2762
3	14.473	0,43	2,30	-7,4044	-8,9325	1,8476	0,0004	0,0011	0,0000	0,0014	0,0085	0,1153
4	15.3258	0,41	2,44	26,2151	-3,9002	0,7785	0,0047	0,0002	0,0000	0,0003	0,0003	0,0378
5	15.8991	0,40	2,53	19,5570	-24,1377	-2,1601	0,0026	0,0082	0,0001	0,0006	0,0003	0,0025
6	15.9075	0,39	2,53	-16,7579	8,0649	2,2840	0,0019	0,0009	0,0001	0,0001	0,0003	0,0012
7	16.0154	0,39	2,55	41,8579	-31,6370	0,7170	0,0120	0,0141	0,0000	0,0001	0,0000	0,0439
8	16.2594	0,39	2,59	-8,9632	19,2502	4,8090	0,0005	0,0052	0,0003	0,0008	0,0014	0,0012
9	16.276	0,39	2,59	-10,2867	6,2790	6,5799	0,0007	0,0006	0,0006	0,0004	0,0022	0,0018
10	16.2837	0,39	2,59	3,4193	-11,8240	-1,5513	0,0001	0,0020	0,0000	0,0002	0,0002	0,0001
11	16.3176	0,39	2,60	-46,3858	31,4412	24,2324	0,0147	0,0139	0,0082	0,0050	0,0280	0,0164
12	16.4199	0,38	2,61	-7,8625	2,9167	5,3664	0,0004	0,0001	0,0004	0,0001	0,0012	0,0017
13	16.4487	0,38	2,62	-14,6124	25,9451	1,5741	0,0015	0,0095	0,0000	0,0004	0,0000	0,0017
14	16.4827	0,38	2,62	7,2012	18,7645	-2,9070	0,0004	0,0049	0,0001	0,0002	0,0002	0,0000
15	16.5759	0,38	2,64	2,0945	22,0456	-1,5529	0,0000	0,0068	0,0000	0,0004	0,0000	0,0000
16	16.5886	0,38	2,64	-2,6208	4,1960	0,4779	0,0000	0,0002	0,0000	0,0000	0,0000	0,0000
17	16.5931	0,38	2,64	2,0179	18,5868	-1,3154	0,0000	0,0049	0,0000	0,0003	0,0000	0,0000
18	16.8253	0,37	2,68	1,6893	171,4976	-5,9377	0,0000	0,4130	0,0005	0,0234	0,0007	0,0001

At the top it is possible to read the total moving mass and total mass; the difference gives the mass that is set on the nodes where a support is. It is possible to see the mass related to the cladding, which is activated only in the X direction.

It follows the list of modes, where it is possible to remark that now the first mode is not a local mode anymore.

Mode	ıega [rad	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	W_{xi_R}/W_{xtot}	W_{yi_R}/W_{ytot}	W_{zi_R}/W_{ztot}
54	44.706	0,14	7,12	-11,3217	-9,8131	0,2548	0,0009	0,0014	0,0000	0,0000	0,0002	0,0016
55	45.1827	0,14	7,19	2,6628	37,6508	1,5935	0,0000	0,0199	0,0000	0,0010	0,0002	0,0006
56	45.681	0,14	7,27	-3,9078	-5,1730	0,6013	0,0001	0,0004	0,0000	0,0002	0,0000	0,0002
57	45.7493	0,14	7,28	-11,8056	-0,2146	1,9662	0,0010	0,0000	0,0001	0,0023	0,0000	0,0031
58	45.9601	0,14	7,31	14,4971	10,5423	-1,6447	0,0014	0,0016	0,0000	0,0023	0,0000	0,0022
59	46.0432	0,14	7,33	7,4607	4,1686	-0,8543	0,0004	0,0002	0,0000	0,0009	0,0000	0,0006
60	46.0984	0,14	7,34	-0,8655	-4,9790	-0,0021	0,0000	0,0003	0,0000	0,0000	0,0000	0,0001
61	46.2104	0,14	7,35	-8,6428	-6,4593	0,8063	0,0005	0,0006	0,0000	0,0004	0,0001	0,0006
62	46.4087	0,14	7,39	-0,3554	-5,4156	-0,2788	0,0000	0,0004	0,0000	0,0001	0,0000	0,0000
63	46.4867	0,14	7,40	0,7497	-0,6366	-0,0677	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
64	46.4979	0,14	7,40	-0,5640	1,4590	0,1196	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
65	47.1627	0,13	7,51	22,1288	-74,1547	-4,9202	0,0033	0,0772	0,0003	0,0056	0,0005	0,0075
66	49.3363	0,13	7,85	-14,7802	-7,8638	0,2377	0,0015	0,0009	0,0000	0,0008	0,0042	0,0002
67	50.3785	0,12	8,02	7,0912	4,5535	0,6178	0,0003	0,0003	0,0000	0,0023	0,0009	0,0003
68	50.4426	0,12	8,03	2,5056	2,8861	-0,4730	0,0000	0,0001	0,0000	0,0000	0,0003	0,0001
69	50.5584	0,12	8,05	0,1417	-1,0553	0,1669	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
70	50.5652	0,12	8,05	1,8517	1,1214	-0,1071	0,0000	0,0000	0,0000	0,0001	0,0002	0,0002
							0,9760	0,9080	0,6831	0,4599	0,4961	0,9370

As specified in the first example of this course, the Modal Participation Factors show the amount of mass that is vibrating in a specific eigenmode as a percentage of the total mass.

According to Eurocode 8 [7] the sum of the effective modal masses for the modes taken into account must amount to at least 90% of the total mass of the structure.

This criterion is fulfilled which indicates the Eigen modes are sufficient for this example.
However it is important to see that the number of eigenmodes taken into account is sufficient in the X-direction to evaluate a dynamic load working in the X-direction. If the total is under 90%, the number of eigenmodes in the solver setup would have to be augmented and the calculation protocol for the Eigen Frequency would have to be checked again.

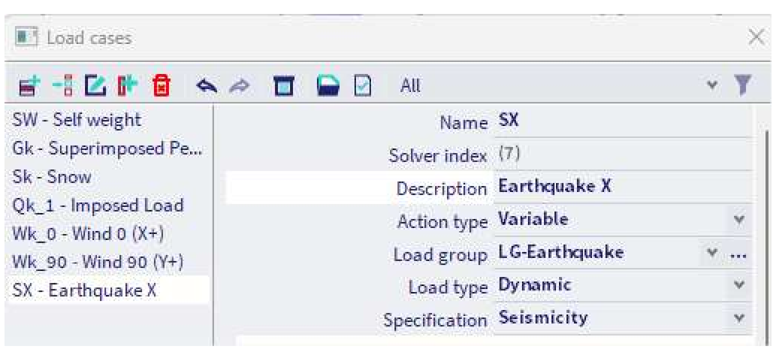
The Damping ratio shows the manually inputted damping ratio for the respective Eigenmodes.

It is important to keep in mind that the Seismic Spectra of Eurocode 8 have been calculated with a damping ratio of 5% as specified in the theory. When a damping ratio is manually inputted, the spectra need to be adapted. This is done through the Damping Coefficient.

Step 8: seismic load case

After creating a Mass Combination, a Seismic load case can be defined through the workstation « Loads » and « Load Cases ».

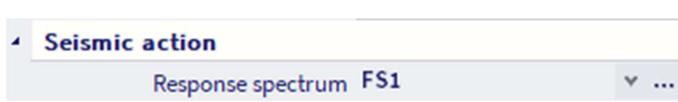
The action type is « Variable ».
The load type is « Dynamic ».
The specification is « Seismicity ».



Now the parameters for the seismic load case will become visible. These parameters will now be explained (going from top to bottom).

⇨ Response Spectrum:

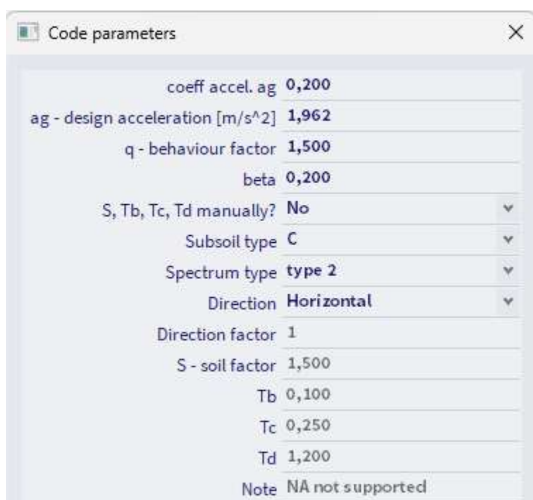
- After choosing the « Load type » as « Dynamic », you will see the different spectrums which are already composed in the project (FS1 by default). You can tick on the three points on the line "Response Spectrum » to go to the list with spectrums, and then choose "New" to create a new spectrum.



- It is also possible that there is no spectrum in the project yet. Then, after choosing the « Load type » as « Dynamic », the software will automatically open the list with spectrums and click on "New" for you. The next window will pop up. Choose « Input type = Eurocode » and tick on « Code Parameters ».

- In « **Code Parameters** », the spectrum will be defined:

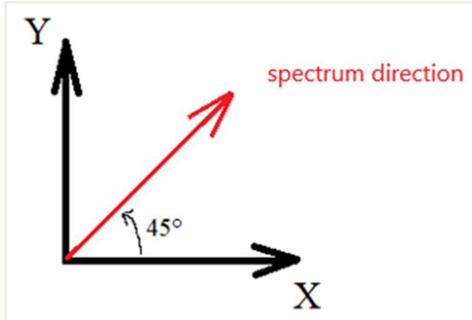
  The **coefficient of acceleration a_g** is **0,2**. Note that a_g is automatically calculated after changing coefficient of acceleration a_g.
  - The behaviour factor **q** is **1,5**.
  - The subsoil type is **type C**.
  - The spectrum type is **type 2**.
  - The spectre is used in X (and Y) direction, so the **horizontal** direction.



- After changing the parameters, click on « OK » until you get back to the load case.

⇨ **Direction**: you need to choose a direction (X, Y or Z) to apply a spectrum in this global direction. We advise to use one direction by load case, and to combine the different load cases in one seismic combination.

⇨ **Rotation around Z axis [deg]**: if you decide to apply a spectrum in an inclined direction from X, Y or Z axis, you can define a rotation angle. For example if you define 45° in the X direction, the spectrum will applied in the following direction:



⇨ **X, Y Z coefficients**: this is used to modify the accelerations in the spectrum without changing the spectrum parameters. We advise to **set this to 1**.

⇨ **Acceleration factor**: this factor is multiplied with the factor X, Y, Z (all of them). This factor should be **set to 1** since the acceleration factor is already used in the parameters of the spectrum.

⇨ **Overturning**: this parameter is used when the supports of the structure are above ground level. By **default**, this value equals **0**.

⇨ **Equivalent lateral forces**: the analysis method by default in the software is the 4.3.3.3 article « Modal analysis using response spectrum ». But by ticking this option, the software will apply the method of 4.3.3.2 article « Analysis method using lateral forces ».

⇨ **Accidental eccentricity**: most of the seismic codes require that structures are checked for torsion due to mass eccentricity including an additional eccentricity, which is the "accidental eccentricity". Please note, that "accidental eccentricity" may be used only together with the **reduced model analysis**. We will explain the reduced model analysis and accidental eccentricity later on.

⇨ **Modal superposition**:

- **Type of superposition** : here the type of modal superposition can be chosen. In this example, the SRSS method is used. The use of the CQC method will be illustrated later on.
  - **SRSS:** Square Root of Sum of Squares. Because of the square root in the formulas of the modal combination methods, the results are **always positive**.

$$R = \sqrt{R_{(1)}^2 + R_{(2)}^2 + R_{(3)}^2 + R_{(4)}^2 + R_{(5)}^2 + \cdots}$$

  - **Max:** modified SRSS method (method not included or described in Eurocode 8)

$$R_{tot} = \sqrt{R_{(jmax)}^2 + \sum_{j=1}^{N} R_{(j)}^2}$$

  - **CQC:** Complete Quadratic Combination

$$R_{tot} = \sqrt{\sum_{i=1}^{N}\sum_{j=1}^{N} R_{(i)}.\rho_{i,j}.R_{(j)}}$$

- **Unify eigenshapes**: this option can be used in the seismic analysis in the case of the method SRSS. Classical the following formula is used for SRSS:

$$R = \sqrt{R_{(1)}^2 + R_{(2)}^2 + R_{(3)}^2 + R_{(4)}^2 + R_{(5)}^2 + \cdots}$$

If the option unify eigenshapes is checked, then the following condition is verified:

$$1 - \frac{\omega_i}{\omega_j} \leq \text{precision } \% \quad (\text{or } i < j \text{ and } \omega_i \leq \omega_j)$$

If the check is fulfilled and mode (i) and (j) are multiple, then the superposition will be modified:

$$R = \sqrt{R_{(1)}^2 + \left(R_{(2)} + R_{(3)}\right)^2 + R_{(4)}^2 + R_{(5)}^2 + \cdots}$$

Note:

The options under Unify Eigen Shapes can be used to avoid the errors in the SRSS-method for closely spaced modes. As specified in the theory however, it is advised to use the CQC-method in such cases (Eurocode 8 article 4.3.3.3.2).

- After choosing « CQC » for the type of superposition, an option « Damping » displays below. The user has to define a constant damping ratio which will be used for all eigenmodes. By default, the displayed ratio is equal to 5% because this is the ratio used in the seismic spectrums definition of the Eurocode 8. But in this example, the ratio will be equal to **4%**.

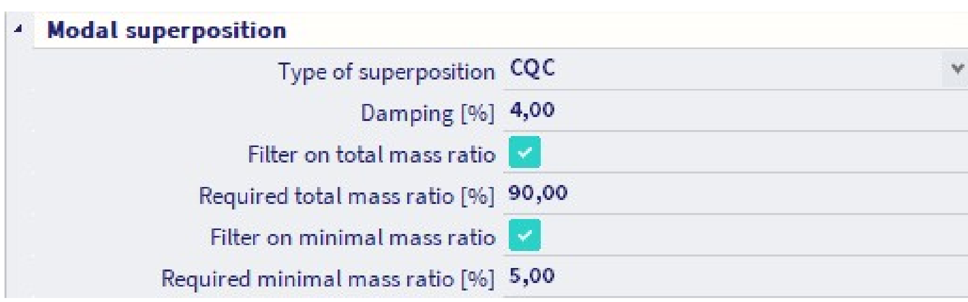
  This damping spectrum will be used for the calculation of the **Modal Cross Correlation coefficients** of the **CQC-method** and will also be used to calculate the **Damping Coefficient** for each mode as specified in the previous example.

- **Filter on total mass ratio**: Only modes with the highest modal mass ratio are taken into account for modal superposition. Modes are sorted in decreasing order of their modal mass ratio and superposed until the specified cumulated mass ratio is reached.

  The ratio to reach should be at least 90% to respect the article 4.3.3.3.1 from EN 1998-1-1.

- **Filter on minimal mass ratio** : Only modes with a modal mass ratio higher than the specified value are taken into account for modal superposition.

  The minimal mass ratio should be at least 5% to respect the article 4.3.3.3.1 from EN 1998-1-1.



Note: if the two previous filter options are not ticked, all modes asked by the user will be displayed and considered in the modal superposition.

For additional options, check paragraph 3.5 and 3.6 for more information

- **Use residual mode**: you have to verify if 90% of the total mass is included in de modal masses (EN 1998-1-1 art.4.3.3.3.1). This will be checked later on in the calculation protocol. If the number of total participating mass is under 90%, the number of eigen frequencies has to be increased.

  To avoid this check, it is possible to choose **missing mass in modes** or **residual mode**.

⇨ Signed results / Dominant mode : you can select the mode shape which will be used to define the sign. If automatic is chosen as mode shape, the mode shape with the biggest mass participation is used (sum of direction X, Y and Z). This option can be used for example for shear walls.

This result only makes sense if this single eigenmode is clearly the most dominant for that spectrum, and all other modes have almost no significance for that spectrum. But since this option manipulates the results, we advise you not to use it, unless you have a very good knowledge of SCIA Engineer and of seismic calculations.

Step 9: reading results

The following results are obtained through the Calculation protocol of the Linear Calculation:

Dynamic load case 7 : SX

Mode	Freq. [Hz]	Damp ratio	Damp coef.	Wi/Wtot [-]	Sax [m/s²]	Say [m/s²]	Saz [m/s²]	G(j) [-]	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	1,90	0.04	1.05409	0,8753	2,452	0,000	0,000	6,1747	313,89	7,75	-24,45	-1560,06
19	2,81	0.04	1.05409	0,0326	3,635	0,000	0,000	-0,8037	17,31	4,50	-25,14	-131,22
Level=	0,00			0,9079					315,03	9,11	35,73	1570,59

- Sax, Say and Saz represent the spectral accelerations.
- G(j) is the mode coefficient for mode j.
- Fx and Fy are the Base Shears for each mode.
- Mx and My are the Overturning Moments for each mode.

The results show that for each mode, the Damping Ratio is equal to **4%.**

As specified in the theory, the Seismic Spectra of Eurocode 8 have been defined using a Damping Ratio of 5%. Since now another value is used for the damping, the spectrum needs to be corrected using a Damping Coefficient η.
Following Eurocode 8 [6], this coefficient is calculated as follows:

$$\eta = \sqrt{\frac{10}{(5+\xi)}} \geq 0{,}55 \tag{4.13}$$

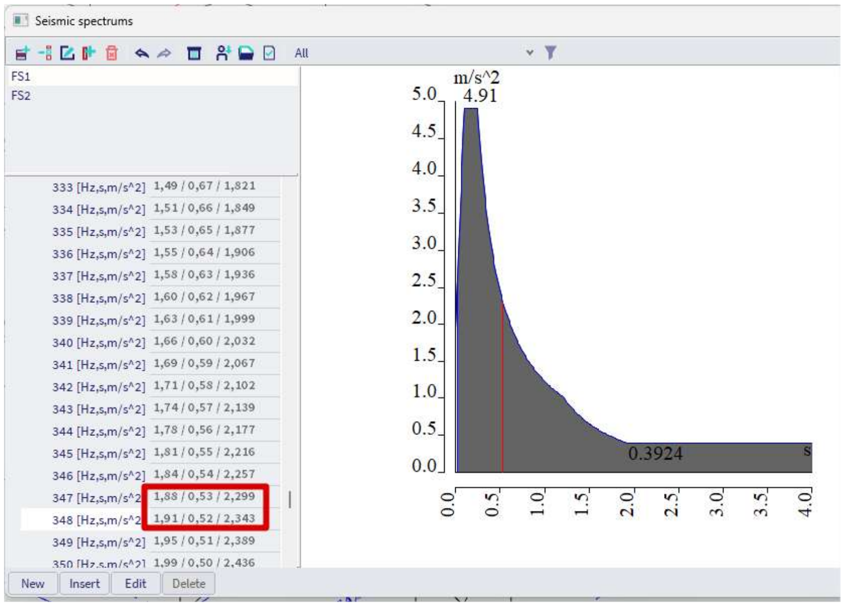
Where: ξ = Damping Ratio expressed in percent.

For a default damping ratio of 5%, η equals unity.
The lower limit of 0,55 for the Damping Coefficient indicates that Damping Ratio's higher than ± 28,06% have no further influence on the seismic spectrum.

For the exact application of η in the formulas of the seismic design spectra, reference is made to Eurocode 8 [7].
In this example, the damping ratio of 4% causes the following Damping Coefficient:

$$\eta = \sqrt{\frac{10}{(5+4)}} = 1{,}0541$$

This indicates that the spectral accelerations will be augmented by 5% due to the fact that less damping is present in the structure.
The spectral acceleration for mode 1 is in this case is 2,327 $\mathrm{m/s^2}$:



The spectral acceleration can thus be multiplied by η:

$$S_{ax,(1)} = 2{,}327\,\mathrm{m/s^2} * 1{,}0541 = 2{,}452\,\mathrm{m/s^2}$$

With these new spectral accelerations, the calculation of the Base Shear, Overturning Moment, … can be repeated.

Manual Calculation

In this paragraph, the application of the CQC method is illustrated for the global response of the Base Shear.

- Mode 1 : $\omega_{(1)} = 11{,}9203\mathrm{rad/s}$ $\quad F_{(1)} = 313{,}89\mathrm{kN}$
- Mode 2 : $\omega_{(2)} = 17{,}6675\mathrm{rad/s}$ $\quad F_{(2)} = 17{,}31\mathrm{kN}$

Using a spreadsheet, the Modal Correlation coefficients $\boldsymbol{\rho_{i,j}}$ are calculated with a damping ratio $\boldsymbol{\xi_{i,j}}$ of 4**%**.

$\omega_{,1}$	11,9203
$\omega_{,19}$	17,6675
r	1,482136
ξ	0,04
$\rho_{1,19}$	0,03846

Mode	1	2
1	1	0, 03846
2	0, 03846	1

$$R_{tot} = \sqrt{\sum_{i=1}^{N}\sum_{j=1}^{N} R_{(i)}.\rho_{i,j}.R_{(j)}}$$

$$R_{tot} = \sqrt{\begin{matrix} 313{,}89kN * 1 * 313{,}89kN \\ +2 * 313{,}89kN * 0{,}03846 * 17{,}31kN \\ +17{,}31kN * 0{,}03846 * 313{,}89kN \\ +17{,}31kN * 1 * 17{,}31kN \end{matrix}}$$

$$R_{tot} = 315{,}03$$

3.4 Seismic combinations

There are different possibilities to create load combinations which include also seismic load cases.

First of all, three load cases are created.
The general format of effects of actions should be:

$$E_d = E\left(G_{k,j}; P; A_{Ed}; \psi_{2,i} Q_{k,i}\right) \qquad j \geq 1\,; i \geq 1$$

The combination of actions in brackets can be expressed as:

$$\sum_{j\geq 1} G_{k,j} \text{"+"} P \text{"+"} A_{Ed} \text{" + "} \sum_{i \geq 1} \psi_{2,i} Q_{k,i}$$

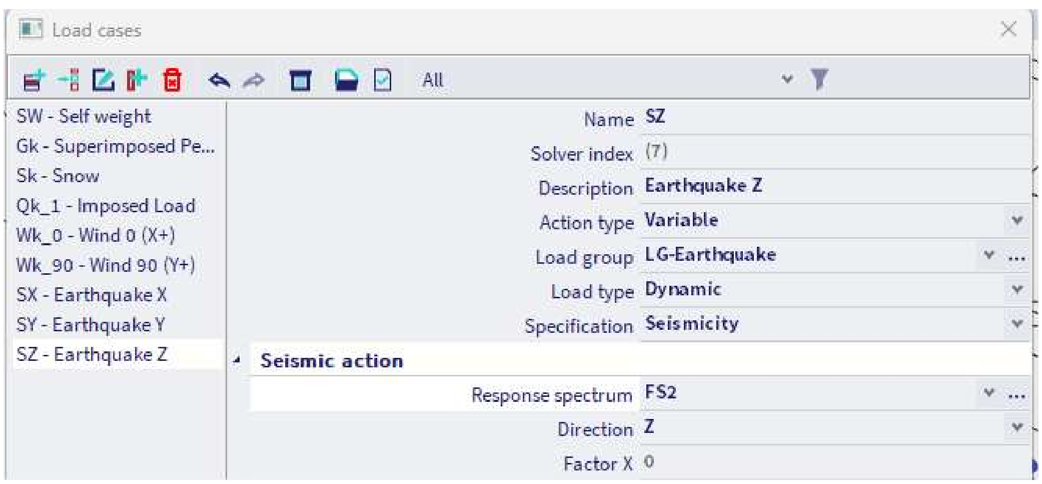
Where E_d:

$$E_{Edx} + 0{,}3.E_{Edy} + 0{,}3.E_{Edz}$$
$$0{,}3.E_{Edx} + E_{Edy} + 0{,}3.E_{Edz}$$
$$0{,}3.E_{Edx} + 0{,}3.E_{Edy} + E_{Edz}$$

Note: the seismic analysis in Z direction is in most cases irrelevant and it can be ignored.

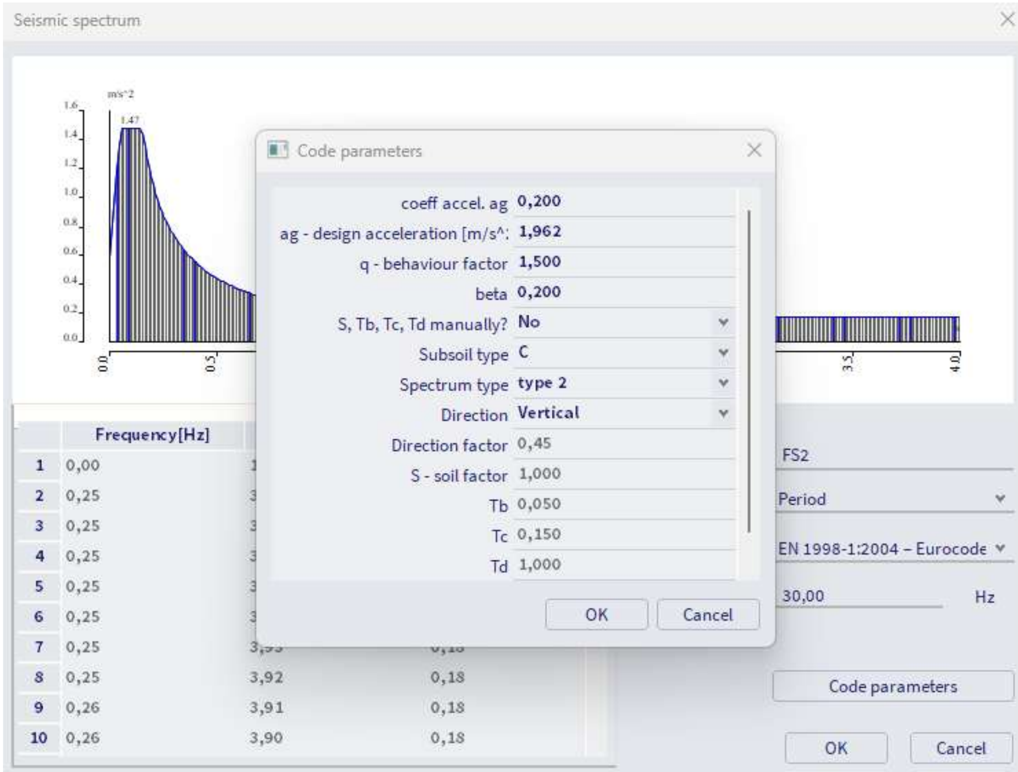
So, there load cases include respectively the seismic spectra in the directions X, Y and Z.

For example:



Please note that a different Eurocode must be generated for the vertical direction. In SCIA Engineer, a load case must be made for component of the earthquake in the X-direction, another for the Y-direction, and another for the Z-direction. Please make sure that the ‘factor’ just underneath the spectrum, « Coef.Z », is not set to ‘zero’, since the accelerations in the seismic spectrum will be multiplied with this value.

The spectrum is Z-direction is different from the horizontal in Xand Y-direction and needs to be created.



Next, we have to assign a type of load group to the seismic load case.

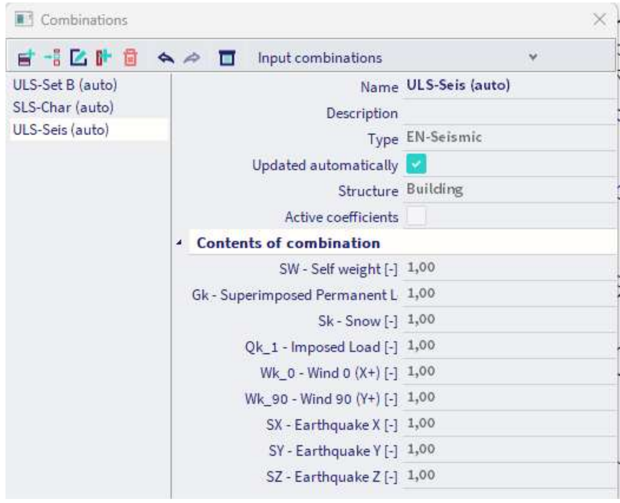
First of all, the relation between load cases in the same group has to be defined. The three seismic spectra have to appear always in the same combination. So, the option 'together' will be chosen here.
Next, the type of load has to be selected: for this, the special type 'seismic' has been implemented.

After the creation of seismic load cases, the combinations can be made. For this purpose, a new type of combinations was implemented: namely the Seismic combination according to the EC-EN.
To use this special type of combination, the seismic load cases must have a load group with properties 'seismic' and 'together' assigned to it. Also no active coefficients can be used.

This combination envelope will automatically look at the seismic load cases with both a positive and a negative coefficient, and will automatically make one of the seismic load cases the primary load case and the others secondary load cases.

If we would not yet take into account that the coefficients can be both positive and negative, then an example would be:

$$E_{Edx} + 0{,}3.E_{Edy} + 0{,}3.E_{Edz}$$
$$0{,}3.E_{Edx} + E_{Edy} + 0{,}3.E_{Edz}$$
$$0{,}3.E_{Edx} + 0{,}3.E_{Edy} + E_{Edz}$$



In the case of the EC-EN, we have to make two sets of combinations, one for the deformations and one for the internal forces. This means that we would have in total six EN-Seismic load cases.

For internal forces, the load cases have to be introduced as described above.
For deformation results, we must create three new load cases, and multiply all displacements results by the behaviour factor q, as described on article 4.3.4 of EN 1998-1-1 :

4.3.4 Displacement calculation

(1)P If linear analysis is performed the displacements induced by the design seismic action shall be calculated on the basis of the elastic deformations of the structural system by means of the following simplified expression:

$$d_s = q_d \, d_e \tag{4.23}$$

where

d_s is the displacement of a point of the structural system induced by the design seismic action;

q_d is the displacement behaviour factor, assumed equal to q unless otherwise specified;

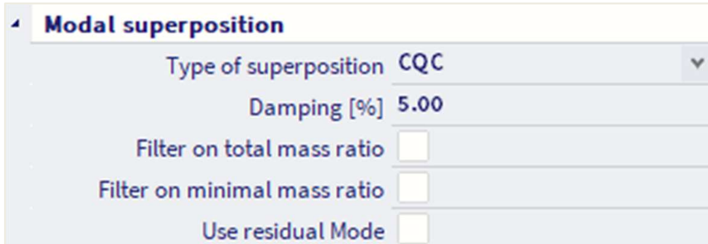
d_e is the displacement of the same point of the structural system, as determined by a linear analysis based on the design response spectrum in accordance with **3.2.2.5.**

3.5 Mass in analysis

As mentioned before, the sum of the effective modal masses for the modes taken into account must amount to at least 90% (EN 1998-1-1 art.4.3.3.3). The user can try to achieve this with the following possibilities:

- Take more natural frequencies into account
- Assign mass more to nodes/connection instead of beams (to avoid local eigenmodes).

The mass which has not been taken into account (for example, if the effective modal mass is 90%, then there is 10% not taken into account), can be treated in two possible different ways in SCIA Engineer:



The used method is set in each seismic load case and is again displayed in the linear calculation protocol. Let's take as example that the effective modal mass in a direction is 90%. Then how can the other 10% be treated?

- If the option « Use residual mode » is not ticked: in this case, the 10% would be ignored. We would only take into account 90% of the mass of the structure to calculate the effects of an earthquake.
- If the option « Use residual mode » is ticked: in this case, a 'fictive' mode corresponding to the combination of all missing modes can be calculated. But since these missing modes are over different natural frequencies, the last found frequency will also be the natural frequency of this mode. In the calculation, the forces in this mode will be calculated in the same way as in the other modes.

A detailed explanation of these modes by using examples can be found in Annex D.

3.6 Modal superposition

The response spectrum method uses a modal superposition of the relevant eigenmodes of the structure. The methods which are used for modal superposition are the ones described at the beginning of the chapter: SRSS or CQC.

These methods have the advantage of very easily providing design values of all results (displacements, internal forces…) but only part of the information is available:

- Min and max values of any result can be determined;
- The actual sign of a result cannot be defined;
- The concomitance of separate results cannot be defined.

The loss of concomitance and sign of results is an issue typically when computing resulting forces in shear walls: it is not possible to compute a resultant from internal forces after modal superposition, as typically all raw results are positive.
Computing resultant forces in one of those shear walls would typically give near-zero moments and extremely overestimated axial forces.

An automatic method can be used since using signed results (described below) is only a workaround to obtain usable resulting forces.
The rigorous method for computation of resultants in the context of the response spectrum method can be summarized as follows:

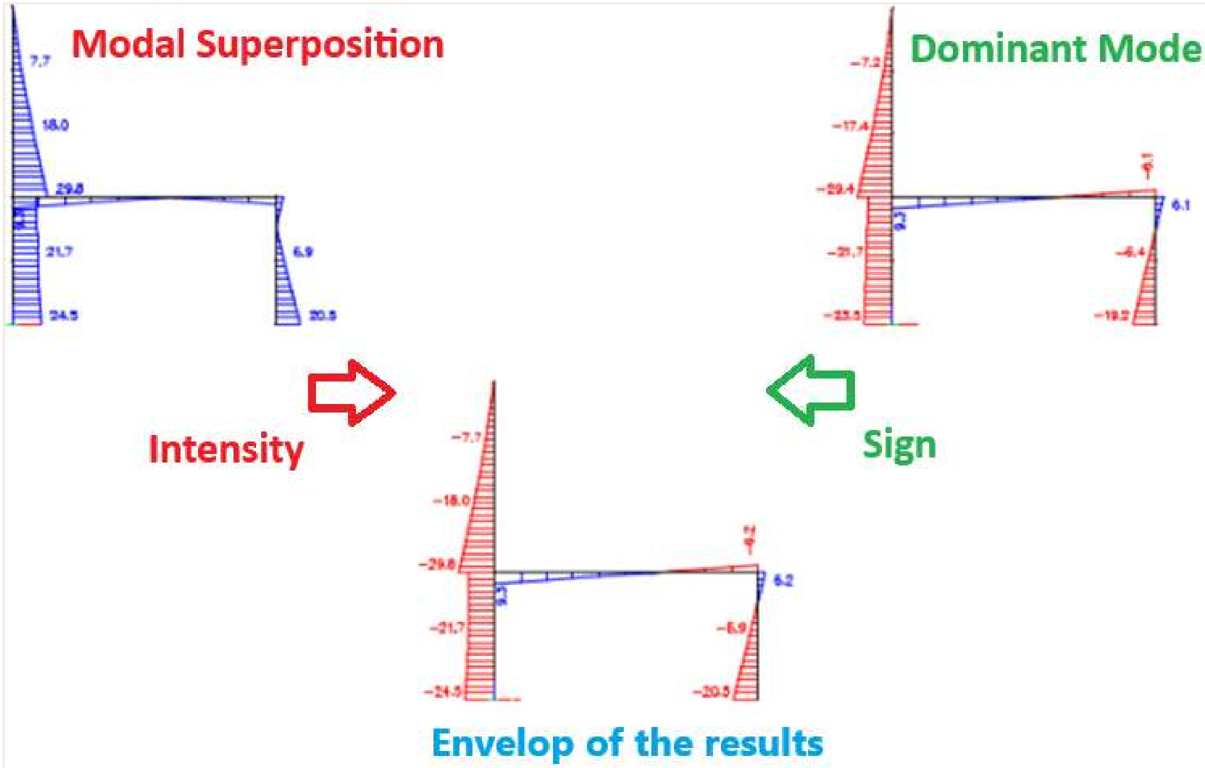
- Compute the local internal forces for each eigenmode;
- Compute the resultant force for each eigenmode separately;
- Apply the modal superposition to the obtained modal resultant values.

When proceeding so, no result signature is necessary to obtain correct values of resulting forces. Moreover there are cases where the method described in the previous paragraph gives overestimated results of most result components and can therefore only be seen as an approximation. The method described here is clearly more accurate.

This option is enabled by default for new projects in SCIA Engineer. For old projects (created before version 2013), you have to open the main menu Tools / Calculation & Mesh / Solver Settings.

To obtain usable values of resulting forces, a possibility is the so-called "signed results" method.
It consists of applying some signature scheme to raw results of the modal superposition. A classical approach uses the sign of the most significant eigenmode.

It is however very important to know that this method will only give good result if there is 1 and only 1 eigenmode of great importance in that respective direction (compared to the other eigenmodes).



Applying this to shear walls, it is possible to “sign” the internal forces, making them suitable for computation of resulting forces.

You can sign results in SCIA Engineer by selecting a signature mode manually or a default mode determined by the program. If the Automatic is chosen, the mode shape with the biggest mass participation is used (sum of direction X, Y and Z).

CHAPTER 4 : REDUCED ANALYSIS MODEL

4.1 Theory

The actual tendency in FE structural analysis is using full 3D modelling of the considered structure. SCIA Engineer obeys that rule as structures are usually modelled in 3D using beam and shell elements, including buildings.

Once a detailed 3D model is ready for statical analysis of a structure, it is natural to use it also for dynamic analysis and, more specifically, for seismic design. A typical issue of full 3D model is, that seismic design regards mostly the global behaviour of the structure whence the full mesh of the structure provides a lot of information about local behaviours. When performing the modal analysis, the full mesh finds all local and global vibration modes, but the local modes are irrelevant for the overall seismic response of the structure. It appears hence logical to use a different, reduced mesh for the dynamic analysis, which ignores these local modes.

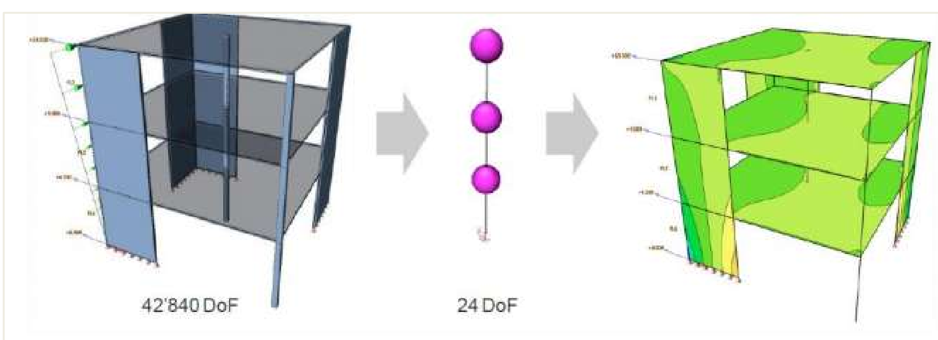
There are well known matrix condensation techniques (Guyan Reduction, also known as static condensation) which allow the user to obtain a reduced system in a very efficient way, but those methods are not well suited for dynamic analysis. An Improved Reduced System (IRS) method has been developed which takes into account not only the stiffness matrix of the system, but also the mass matrix during the condensation process. That method proved to give excellent results for dynamic analysis, with both modal analysis and direct time integration methods.

The algorithm implemented in SCIA Engineer uses the IRS method and consists of 3 steps:

1. The IRS method is used for condensing the mesh of the analysis model.

2. The modal analysis is performed using the reduced mesh, which has typically 1’000 times less degrees of freedom than the original full mesh. This makes the calculation of eigenvalues massively faster on large structures and also avoids unwanted local modes. The latter is particularly interesting for seismic analysis.

3. The results of the reduced system are expanded to the original full mesh, allowing for output of detailed results in the entire structure.



Full 3D storey-based input

Condensed model & dynamic analysis

Expand back to full mesh for result input

The IRS method allows:

1. **Elimination of irrelevant, local bending vibration modes in the slabs**: local modes in all structural elements are implicitly removed, due to the elimination of unwanted degrees of freedom. Of course, adding more reduction nodes would allow for more detailed analysis of local modes, but it is particularly interesting for seismic analysis to keep in the reduced model only the nodes that are strictly necessary to reproduce the typical seismic behaviour of a building. Ultimately, it is up to the user to choose the reduction points in such a way that the wanted eigenmodes are obtained.

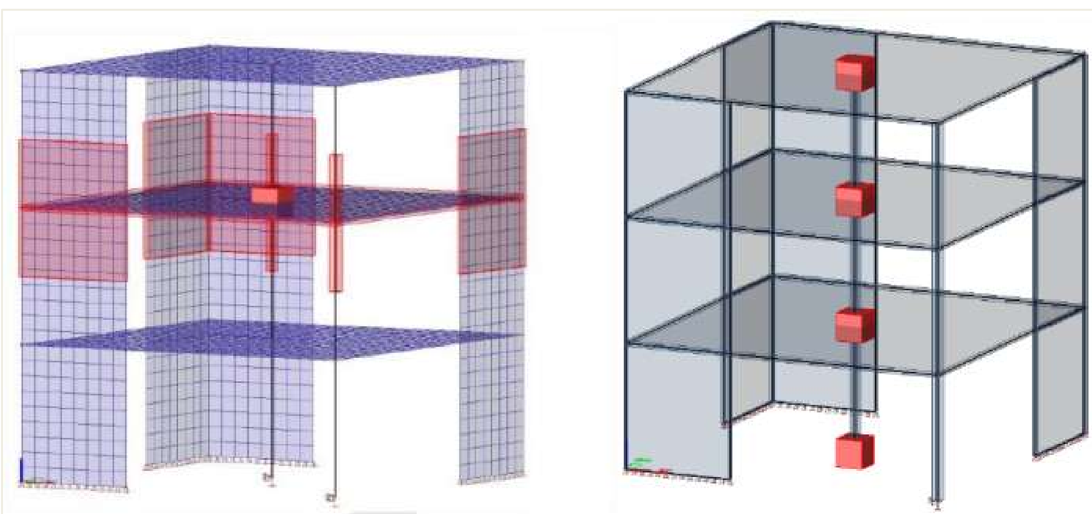
2. **Reduction of computation time**: the computation time is reduced, due to the drastic reduction of the number of degrees of freedom; actually, the reduction is even more important than with diaphragms, as supporting members are also condensed.

3. **Easy handling of mass eccentricities for each deck**: the IRS analysis uses a full mass matrix, which allows for exact implementation of mass eccentricity in each node of the reduced system.

Remark: The elimination of unwanted frame effects from the structural behaviour (considering deck slabs as diaphragms) is not addressed by the IRS analysis in itself, as it does not modify the mechanical behaviour of the structure. However, as unwanted local bending modes are implicitly removed from the reduced system, so-called flexible diaphragms may be easily simulated by significantly reducing the bending stiffness of deck slabs. Not only does that allow obtaining classical diaphragm behaviour by means of a very low bending stiffness, but also intermediate behaviour where the bending stiffness is less drastically reduced and frame effects are therefore reduced, but not completely removed.

The condensed model is obtained from Reduction nodes. R-nodes are placed in each storey, at the specified level, in the middle of the structure (all R-nodes are located on the same vertical axis).

During the analysis, the reduced model is automatically generated from the full mesh of the structure. Each node of the full mesh is mapped to the closest R-node. In a typical building configuration, this means that each R-node will receive the stiffness, loads and masses from the corresponding deck slab, from the top half of the supporting members below the slab and from the bottom half of the supporting members above the slab.



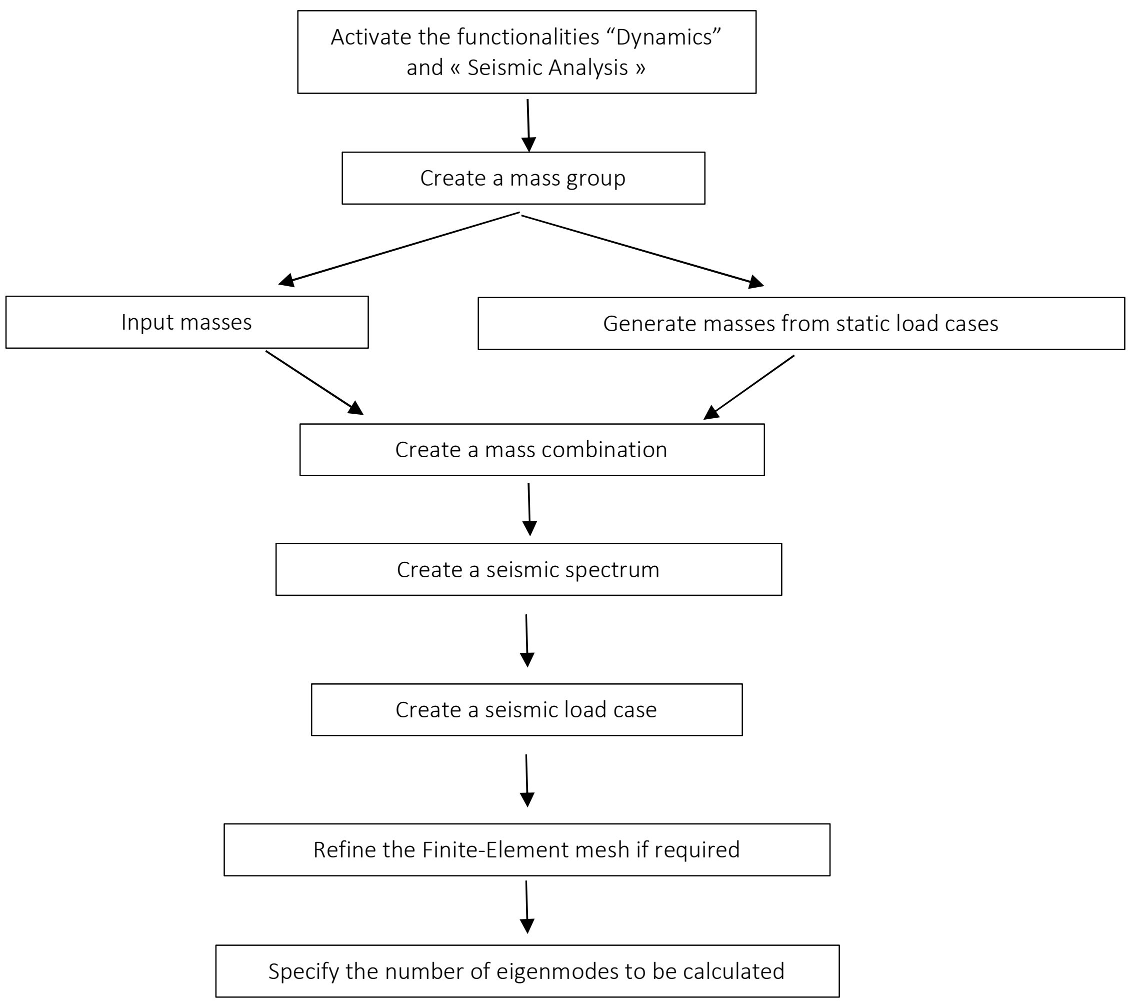
Unlike the classical modal analysis, which typically uses a lumped mass matrix (only diagonal terms are non-zero), the reduced system uses a full mass matrix , with non-zero values out of the diagonal. This means that mass eccentricities can be taken into account easily by the reduced system. The very small size of the reduced system allows using the full mass matrix.

Therefore the reduction points – or so-called R-nodes – that will constitute the reduced model do not need to be located in a particular position, such as the mass centre of each storey. As the structure may have to be calculated several times with various distributions of the masses, the mass centre of each storey is likely to be slightly different depending on the selected mass combination. Thanks to the use of a full mass matrix, the same R-nodes may be used in all cases.

During the analysis, the reduced model is computed automatically from the full mesh. Each node of the full mesh is mapped to the closest R-node of the reduced model.

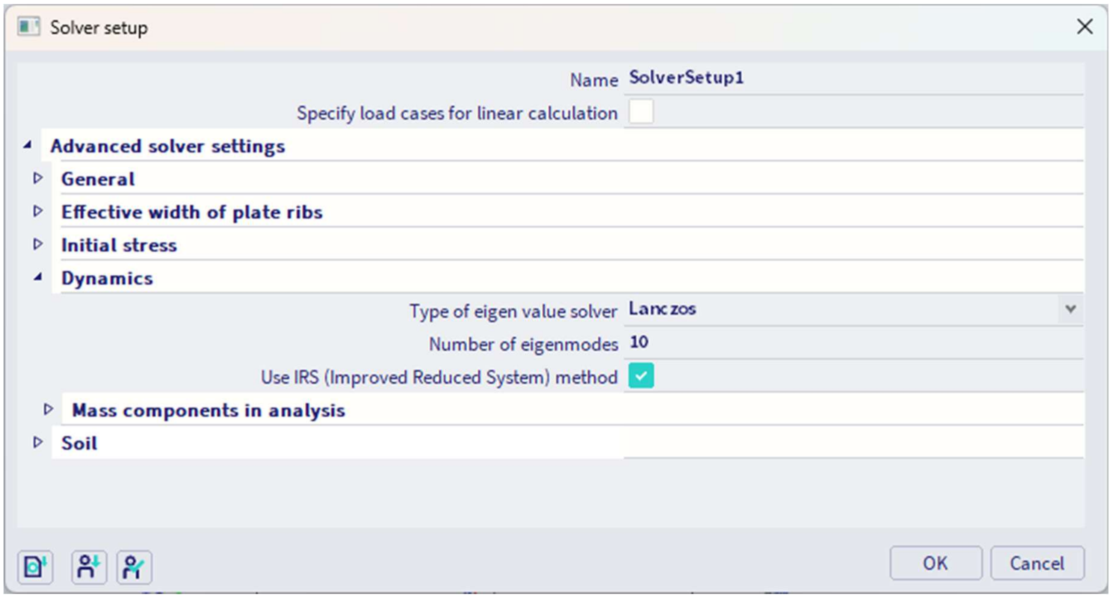
4.2 IRS method in SCIA Engineer

To make an IRS calculation, you first have to perform all the steps as described in detail for seismic calculation in previous chapters. As a reminder, those steps are:

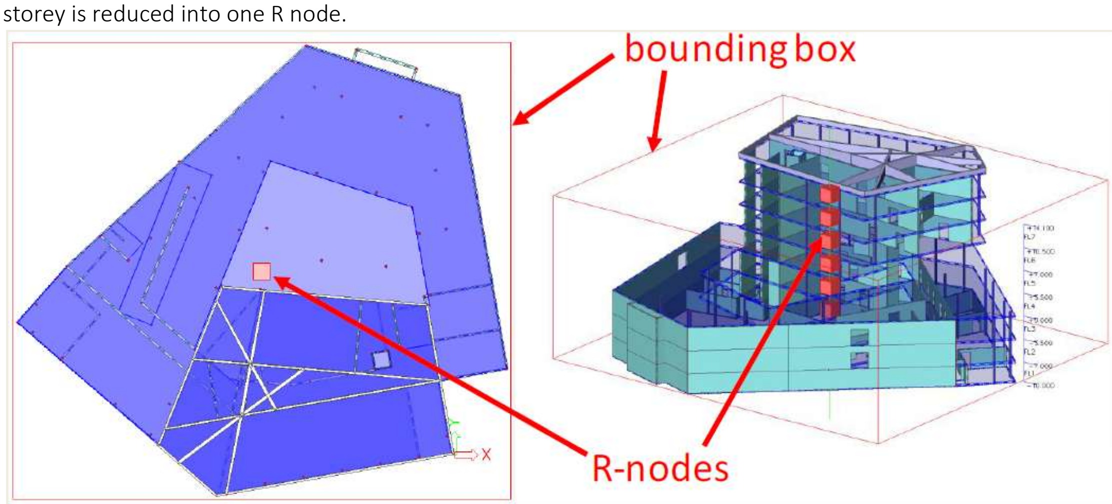


Before doing the linear analysis, the additional steps you have to execute in order to make an IRS calculation are:

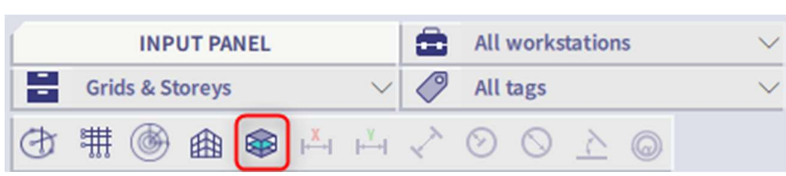
1. You have to enable the reduced model analysis in the project. This can be done via the main menu Tools / Configuration and mesh / Solver Settings:



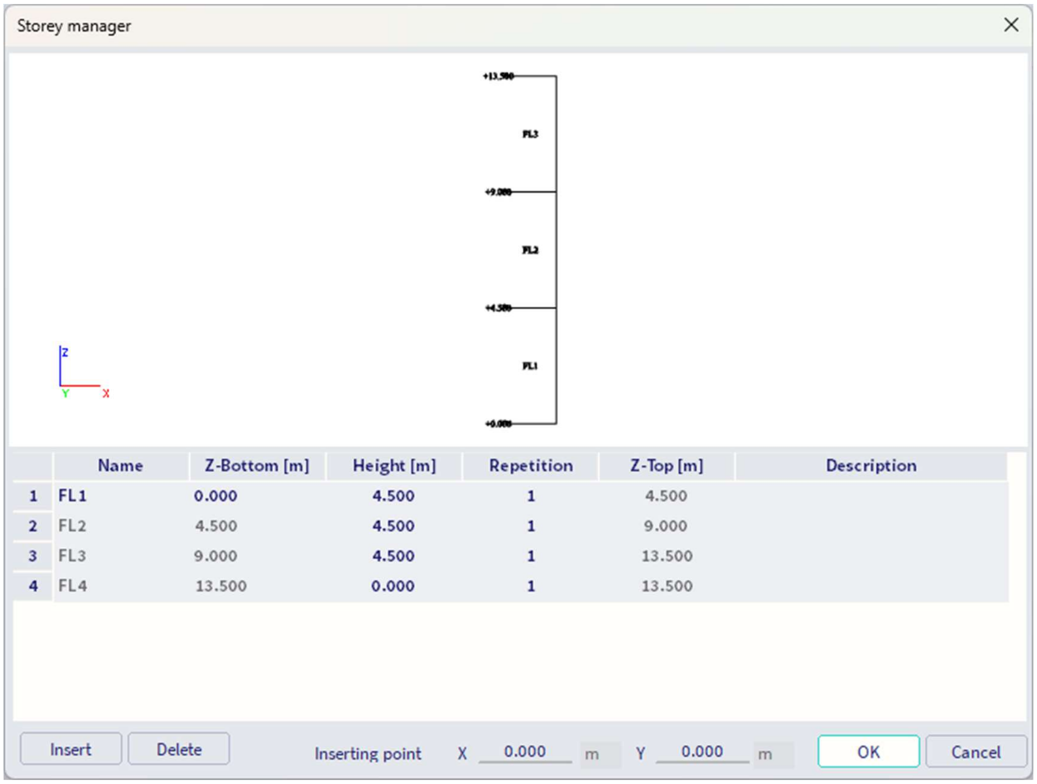
2. Define the building storeys. The Reduction nodes will be calculated from the storey data. In SCIA Engineer, each storey is reduced into one R node.



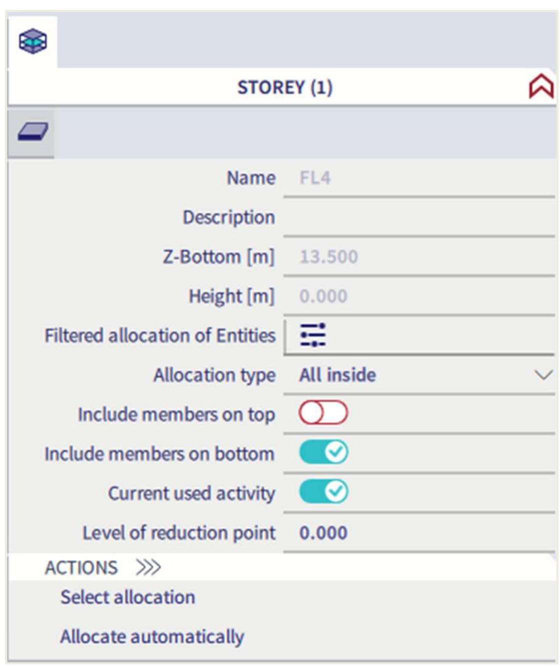
To introduce the building storeys, go to the input panel and in « Line grid and storeys », click on “Storeys”:



The storey manager where you can input the levels opens:



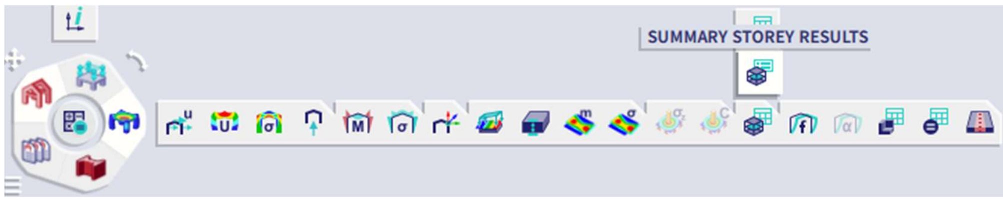
With the default settings, the deck slab of each storey is located at the bottom of the storey, and so is the corresponding R-node. It is recommended to keep it that way. This can be seen from the storey Properties:



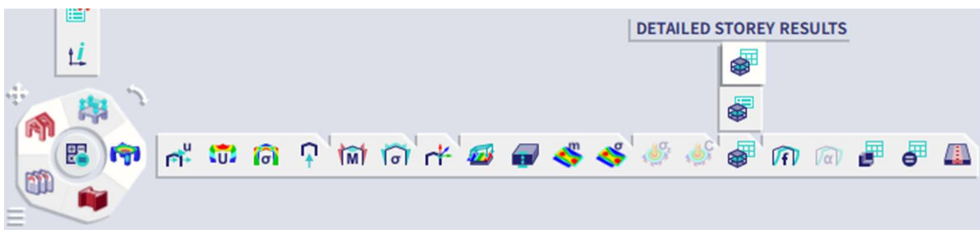
3. Once the linear calculation has been executed, results are available. There are fundamentally two types of results available after an IRS analysis:

- The results of the reduced model are automatically expanded to the original mesh and are accessible through standard output. This will not be detailed here as it is the same as what has been explained in the previous chapters.

- Some dedicated results, coming directly from the reduced model, are available in “Results” workstation, and « Summary Storey Results ». This typically gives information about the masses, displacements and accelerations at each storey in the reduced model.



- Other results can be displayed via the « Results » workstation as « Detailed Storey Results »: this menu can be used to display results from the full mesh analysis. It may be used for results from any linear analysis, with or without dynamic analysis, with or without IRS analysis. It provides results in all supporting members, with easy selection of members per storey. Walls and columns may be represented on the same drawing. Typical provided results are: internal forces, resultants per wall or per storey…

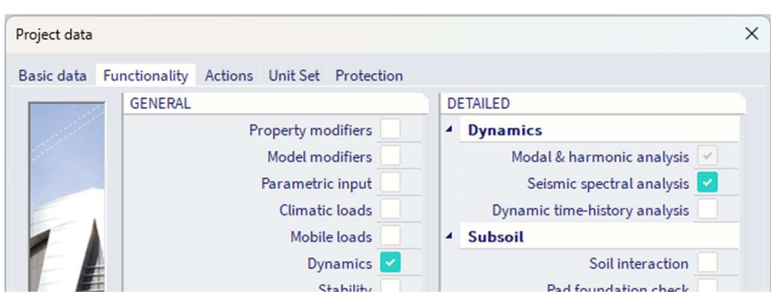


Example 04-1.esa

Open the corresponding project. We are going to apply the principles seen above to this small building.

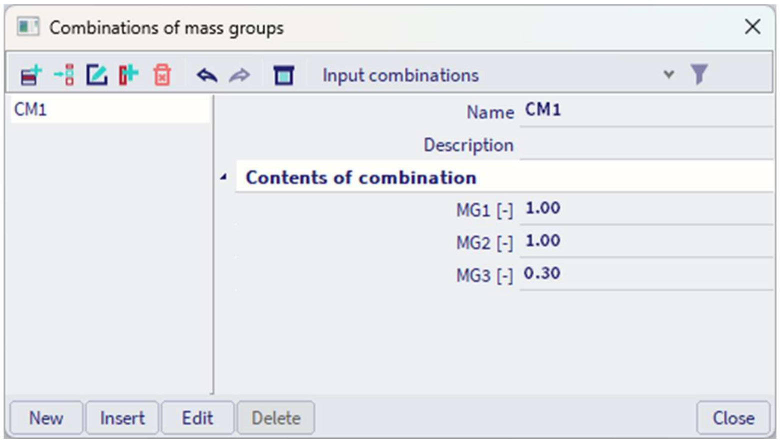
Step 1: set up the seismic model

- Activate “Dynamic” and “Seismic” functionalities from the Project data menu.

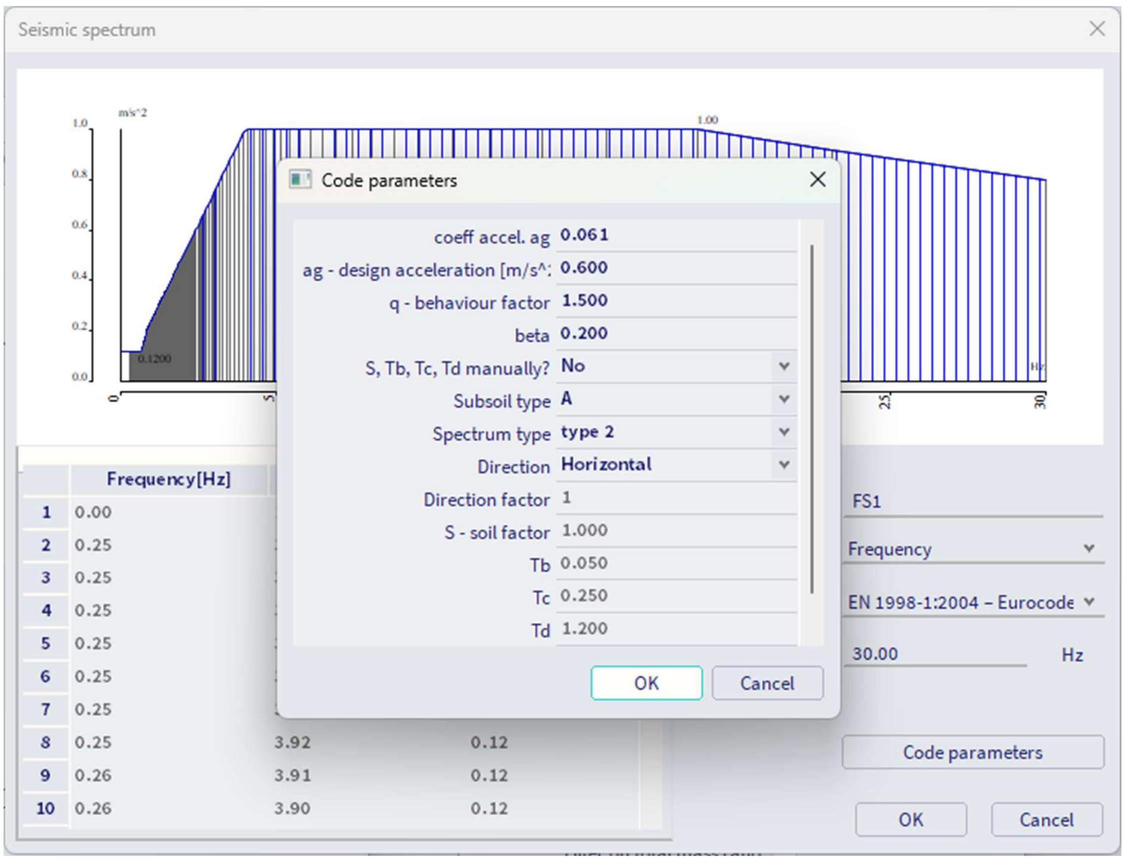


- Create mass groups. For this example, we are going to consider 3 mass groups related to 3 static predefined load cases : self-weight, dead load (DL) and live load (LL).

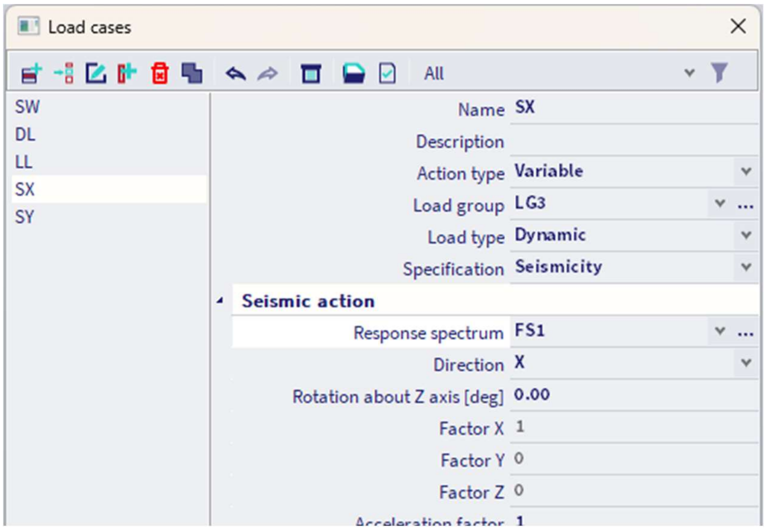
- Create a combination of mass groups

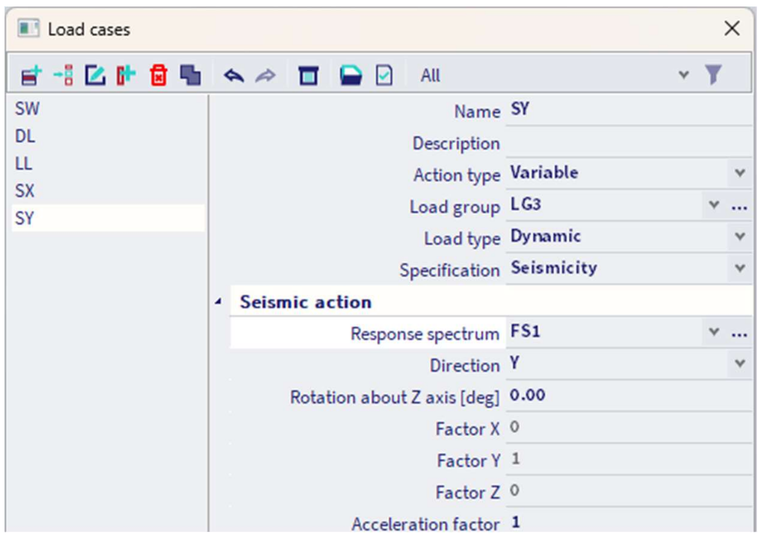


- Define a **seismic spectrum**. Let's consider a seismic spectrum with the following parameters :

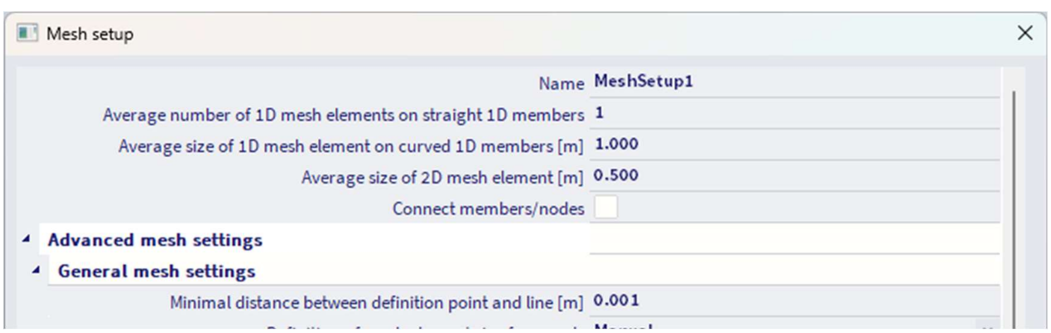


- Create the seismic load cases in X and Y direction in the **"Load cases"** window:





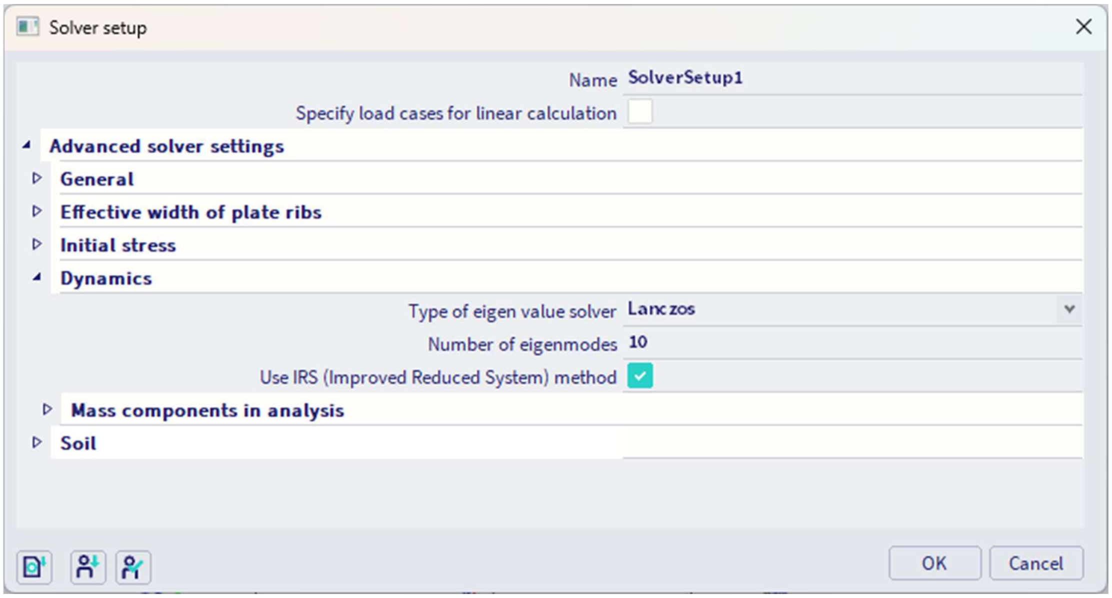
- Refine the mesh. For this example, we set the mesh as follow:



- Choose the number of frequencies which have to be calculated (**Solver setup**). We chose **10 values**.

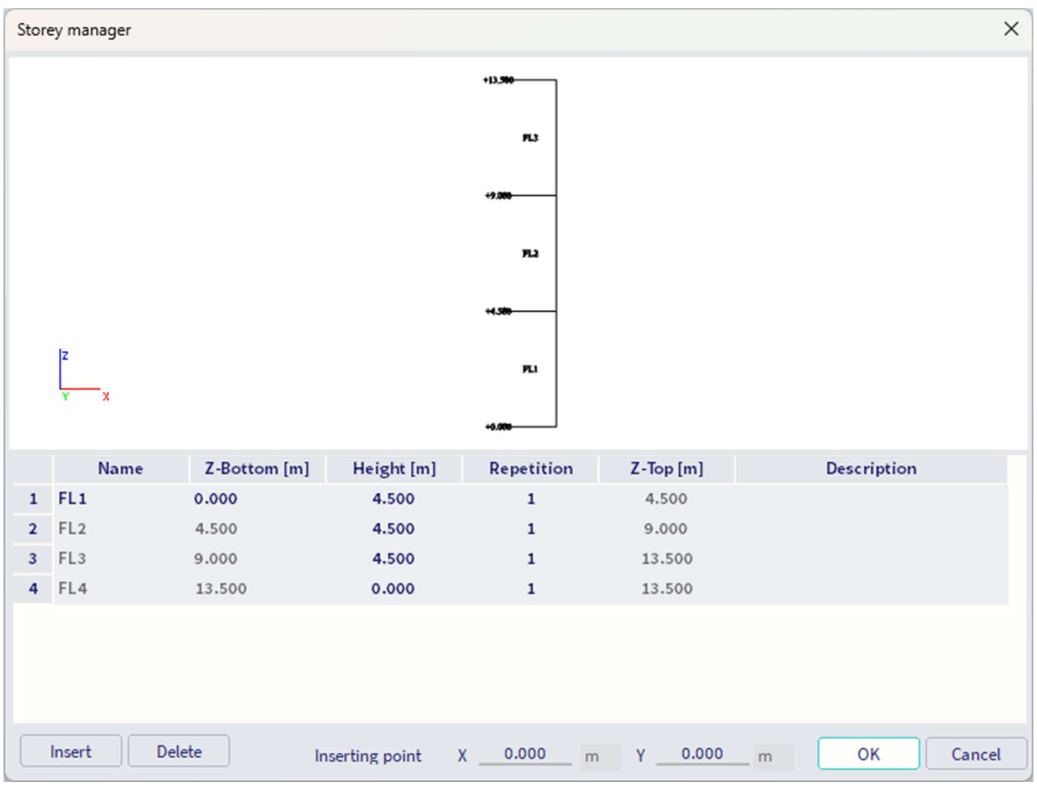
Step 2: activate the option "Use IRS (Improved Reduced Model)"

Activate the option "Use the Improved Reduced Model" from the "Solver Setup".

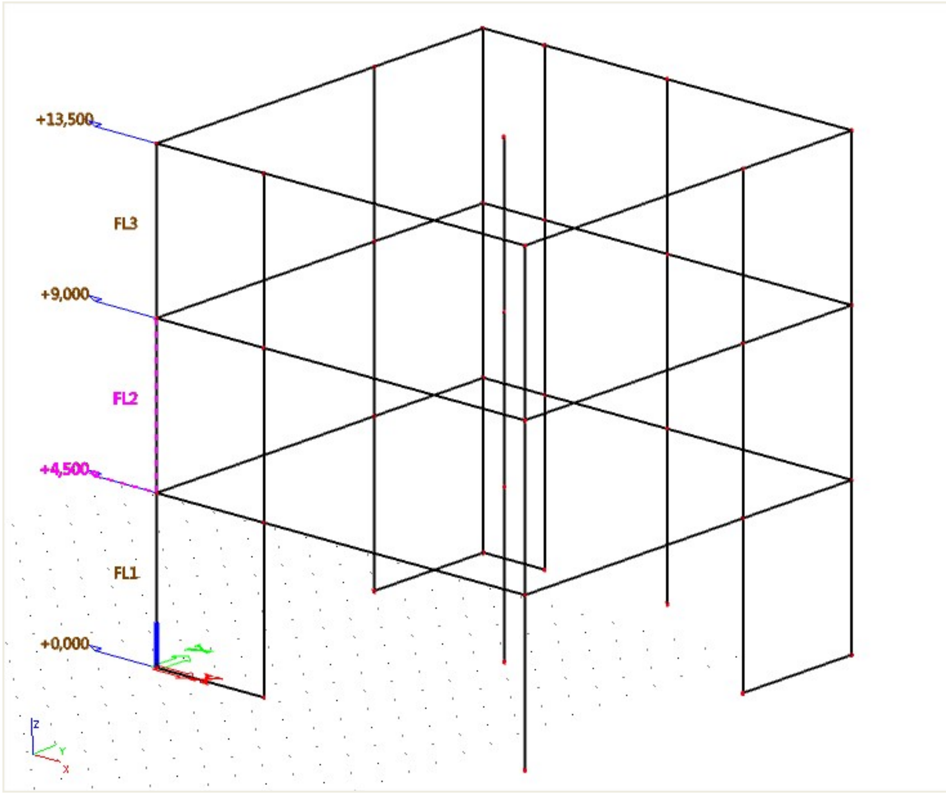


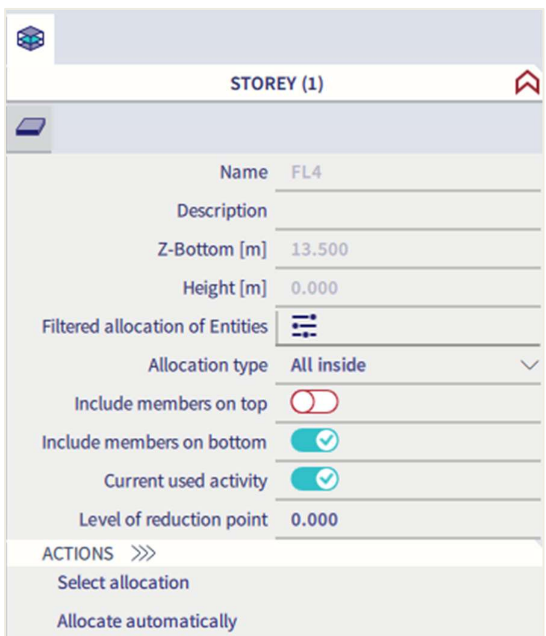
Step 3: define storeys

Define the storeys from the input panel:



The levels are shown graphically. If you select a storey level, you can adapt its properties from the Properties panel:





You can check if the supporting members of the building are properly allocated to storeys using the 'Filtered Allocation of Entities' property.

Optionally, R-nodes may be placed at any level in each storey. The storey property "level of reduction point" allows selecting the exact height of the reduction point for each storey separately. 0 corresponds to the bottom of the storey, 1 to the top of the storey.

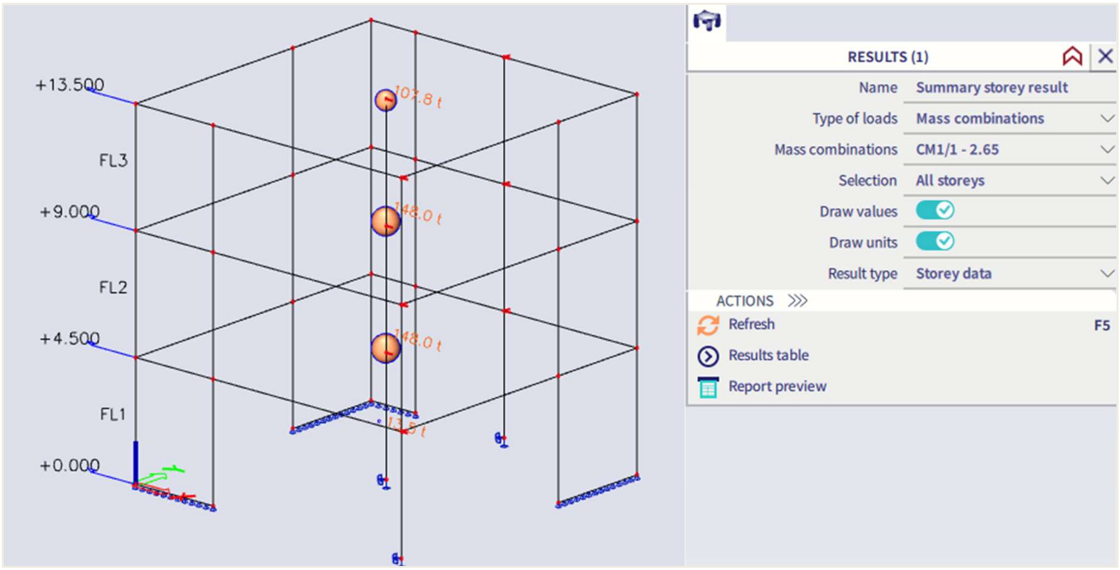
Step 4: perform the linear calculation and check the results

Step 5: summary storey results

There are 3 types of results: storey data, displacements and accelerations.

- Storey data:

  Storey data displays for each storey the total mass and the coordinates of the mass center. It is only available with mass combinations.



Summary storey result

Storey data:
Eigen solution
Selection: All
Mass combinations : CM1/1 - 2.65

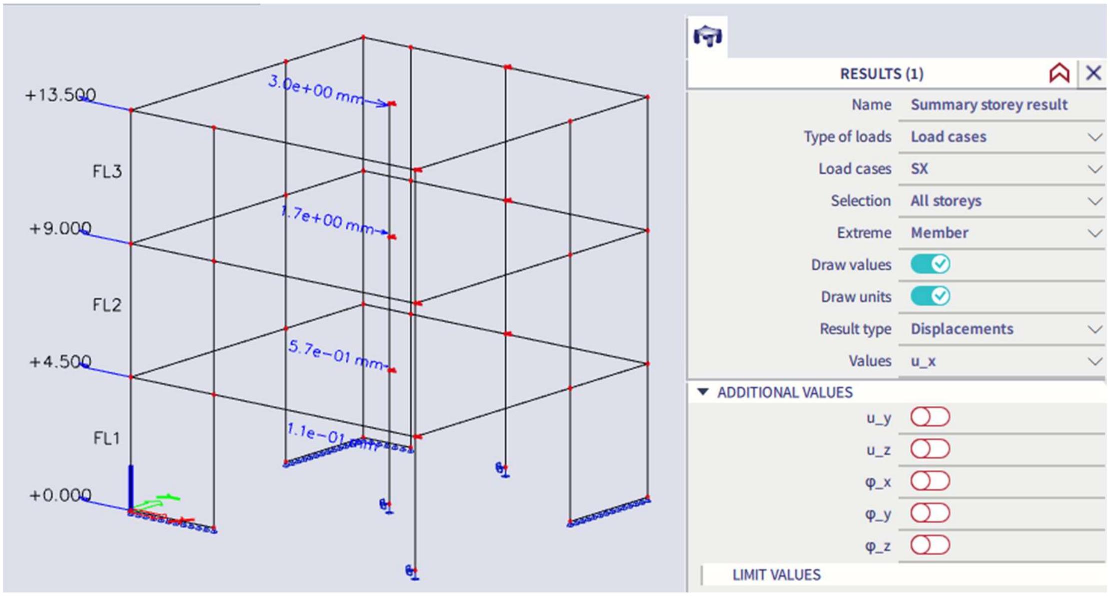
Name	M [t]	XG [m]	YG [m]	ZG [m]
FL1	13.5	4.157	7.704	1.250
FL2	148.0	5.663	6.350	4.500
FL3	148.0	5.663	6.350	9.000
FL4	107.8	5.769	6.240	13.343

- Displacements & accelerations:

  Displacements & Accelerations are available for eigenmodes and seismic load cases. The values of displacement & acceleration components are given at the mass centre of each storey.

  Results for mass combinations are raw, normalized results from modal analysis, without effect of response spectrum.

  Results for seismic load cases are values after modal superposition.



Summary storey result

Storey Displacements:

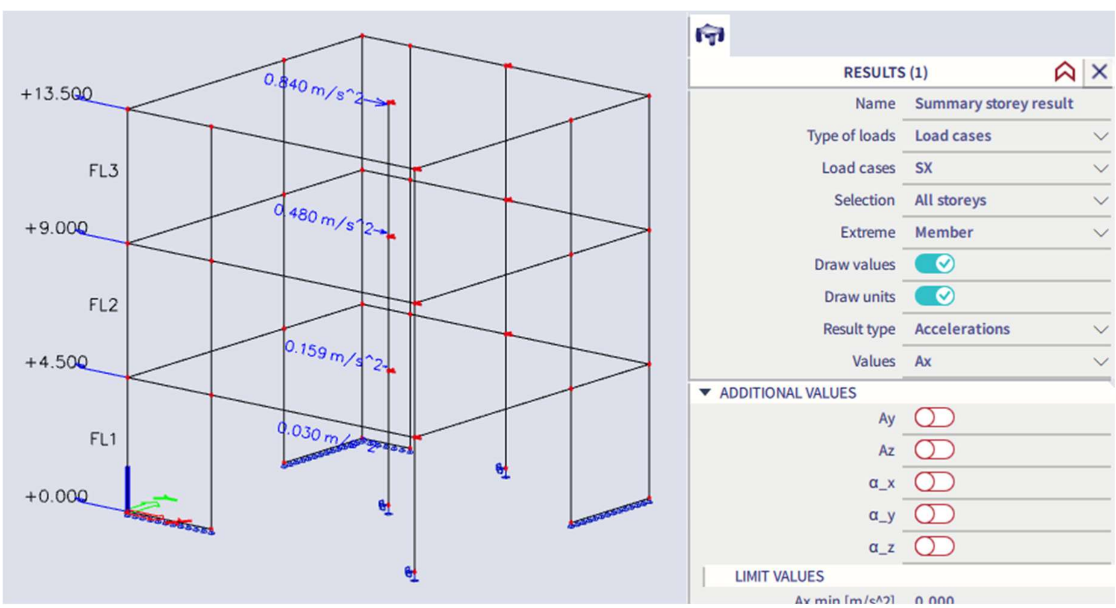
Linear calculation, Extreme: Member

Selection: All

Load cases : SX

Name	u_x [mm]	u_y [mm]	u_z [mm]	φ_x [mrad]	φ_y [mrad]	φ_z [mrad]
FL1	1,1e-01	2,4e-02	6,2e-05	1,3e-03	2,4e-03	2,4e-03
FL2	5,7e-01	1,7e-01	5,2e-02	9,1e-03	4,8e-03	9,0e-03
FL3	1,7e+00	5,2e-01	7,1e-02	1,4e-02	6,8e-03	2,5e-02
FL4	3,0e+00	9,2e-01	9,1e-02	1,5e-02	7,2e-03	4,3e-02

Using the option 'Additional values' in the properties windows you can display more components:



Step 6: detailed storey results

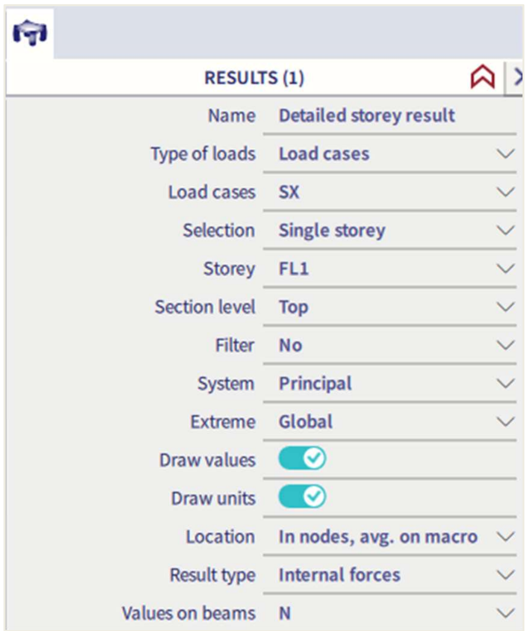
Typical provided results are: internal forces, resultants per wall or per storey…

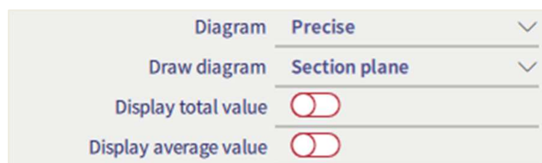
Mainly two types of results are available here:

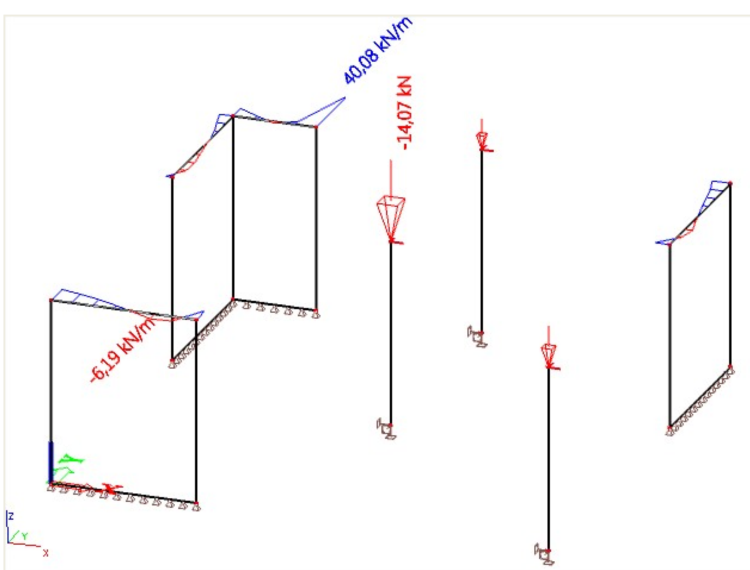
- <u>Internal forces in supporting members</u>

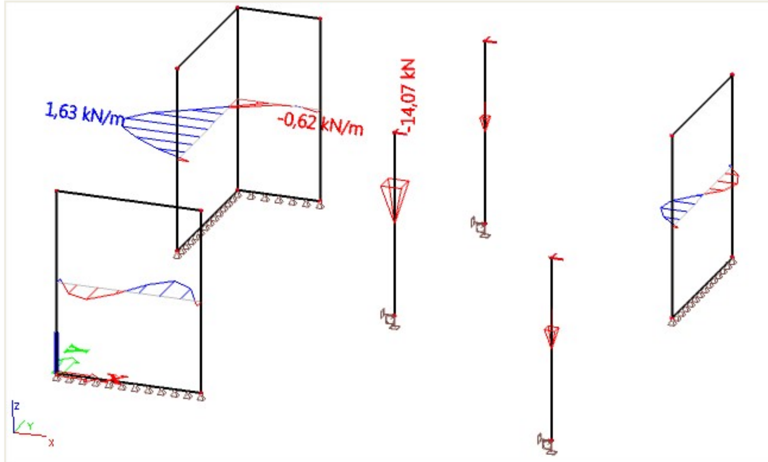
  The result can be displayed on different section levels:

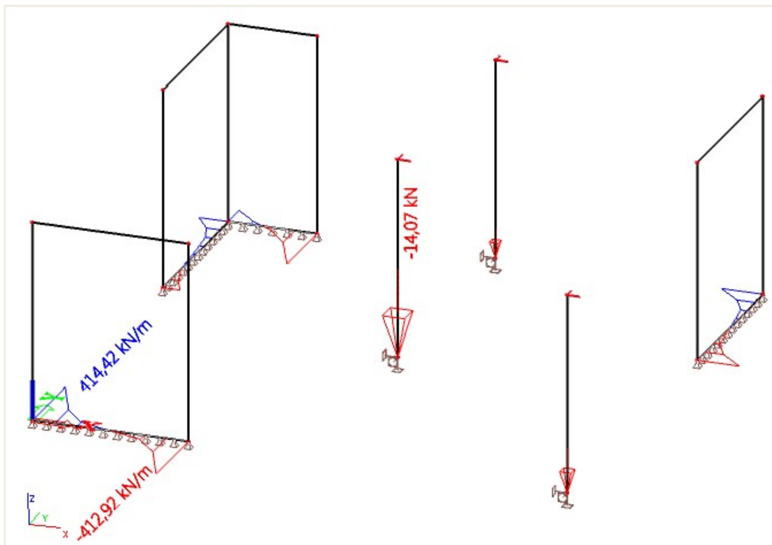
  - Top (section at the top of each storey)
  - Middle (section at mid-height of the each storey)
  - Bottom (section at the bottom of each storey)
  - User defined











Detailed storey result

Linear calculation, Extreme: Global, System: Principal
Selection: FL1
Load cases : SX
Columns:

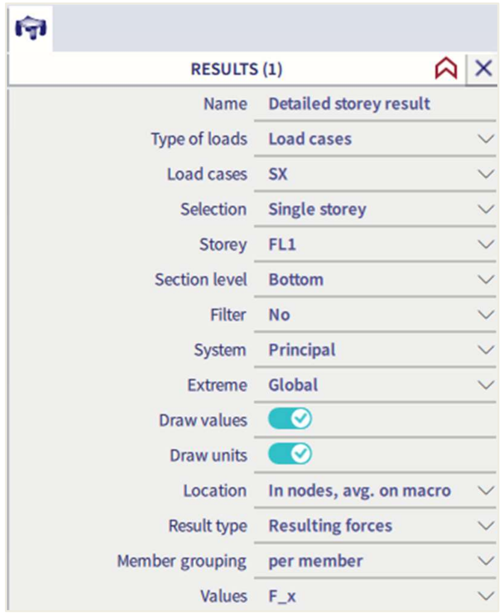
Name	Storey	x [m]	y [m]	z [m]	N [kN]	V_y [kN]	V_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
B22	FL1	6.000	6.000	4.500	**14.02**	**0.26**	**1.05**	**0.03**	**1.94**	**0.43**
B37	FL1	12.000	0.000	4.500	6.87	0.25	**0.56**	**0.04**	**0.54**	0.29
B40	FL1	6.000	12.000	4.500	**4.97**	**0.17**	1.02	0.04	1.75	**0.15**

Walls:

Name	Storey	x [m]	y [m]	z [m]	n_x [kN/m]	n_y [kN/m]	n_{xy} [kN/m]	m_x [kNm/m]	m_y [kNm/m]	m_{xy} [kNm/m]	v_x [kN/m]	v_y [kN/m]
S4	FL1	12.000	10.000	4.500	0.64	**2.18**	28.02	0.08	0.43	0.05	0.19	0.30
S7	FL1	0.000	8.000	4.500	2.87	58.55	**1.94**	0.03	1.59	1.00	0.35	4.92
S15	FL1	1.500	0.000	4.500	3.74	45.62	52.97	**0.02**	0.11	0.22	0.48	0.08
S15	FL1	0.000	0.000	4.500	12.86	**323.53**	7.66	**0.48**	0.64	0.04	1.84	1.88
S15	FL1	2.000	0.000	4.500	0.92	46.64	**53.10**	0.03	**0.09**	0.26	0.50	0.14
S4	FL1	12.000	11.000	4.500	4.32	72.74	20.59	0.09	0.41	**0.02**	0.27	0.23
S7	FL1	0.000	8.500	4.500	3.04	45.38	3.22	0.05	0.14	0.66	**0.03**	1.58
S15	FL1	3.500	0.000	4.500	7.96	301.84	7.28	0.22	1.90	0.90	**2.20**	5.60
S7	FL1	0.000	10.500	4.500	**0.50**	64.10	10.38	0.07	0.53	0.37	0.10	**0.08**
S10	FL1	2.000	12.000	4.500	**32.69**	232.94	23.41	0.03	**3.54**	**1.61**	1.16	**11.18**

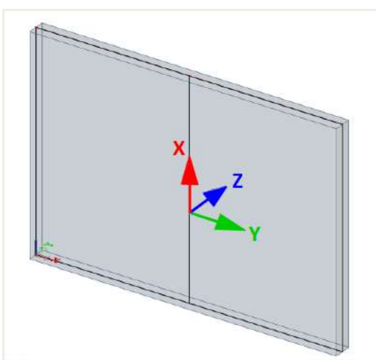
- <u>Resulting forces (by member)</u>

  Location = by member: compute the resulting forces are computed for each wall member separately.



Resulting forces in 1D members (columns) are identical to internal forces in 1D members.

Resulting forces in 2D members (walls) compute the resultant at the centre of each wall, according to a dedicated local coordinate system, regardless of the System output setting. The coordinate system that is used is the same as the LCS of a vertical rib placed in the middle of the wall. It is also the same coordinate system that is used for integration strips.

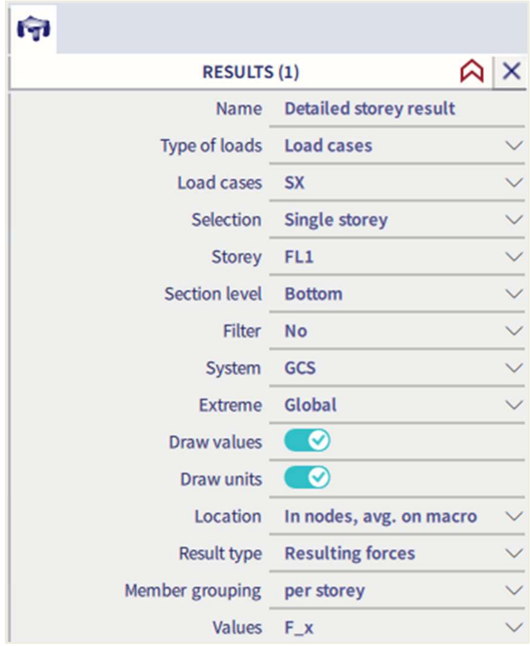


The local X axis is vertical, upwards.

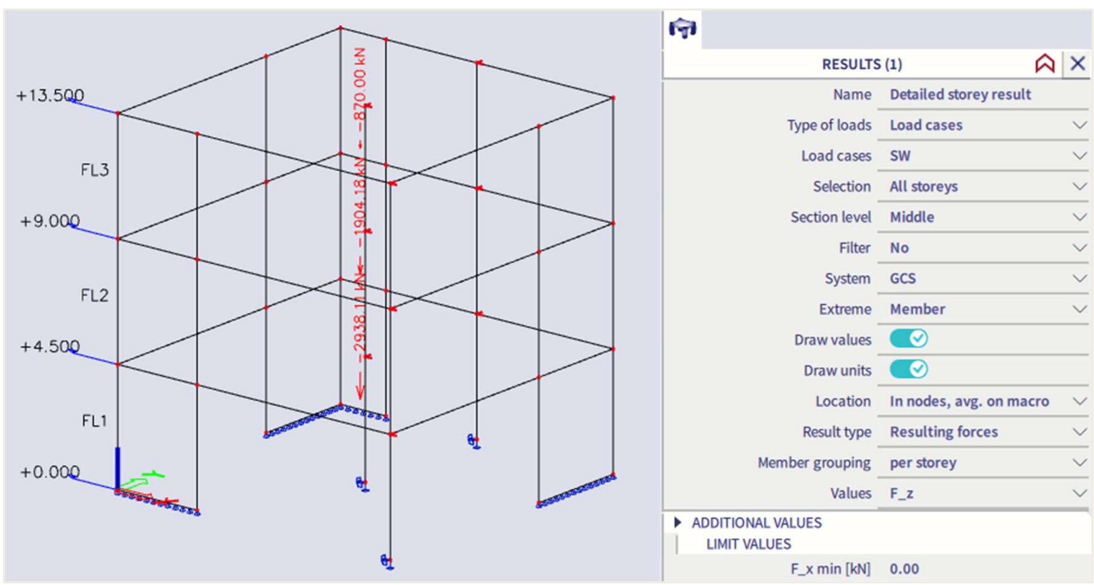
The local Z axis is identical to the Z LCS of the wall.

In this way, resulting forces in walls can be easily displayed together, consistently with internal forces in columns on a single drawing.

- Resulting forces (by storey)
  Location = by storey: the resulting forces are computed for each storey, considering all the supporting members at once; 1D (columns) and 2D members (walls) are taken into account together



Total vertical forces in all storeys:



Detailed storey result

Linear calculation, Extreme: Member, System: GCS
Selection: All
Load cases : SW

Resulting forces per storey

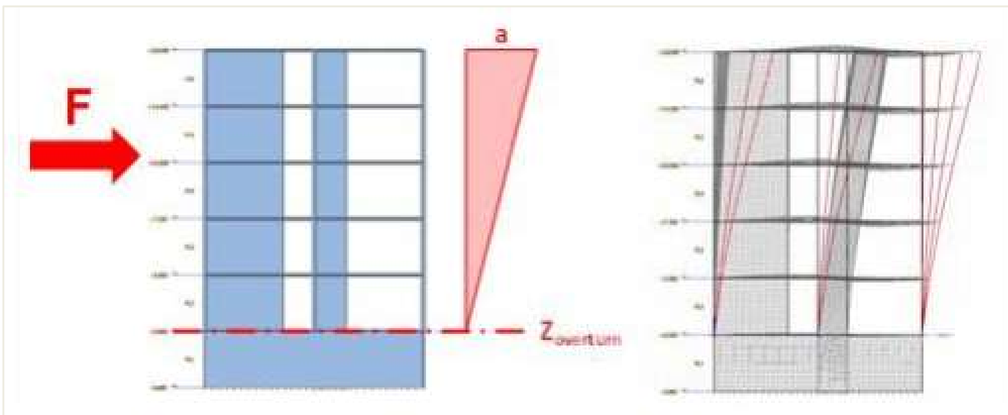
Name	Storey	x [m]	y [m]	z [m]	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
FL1		4.507	7.549	2.250	0.41	0.92	-2938.11	3281.44	3163.40	5.34
FL2		4.507	7.549	6.750	-1.00	0.65	-1904.18	2187.60	2109.18	-0.53
FL3		4.507	7.549	11.250	-1.18	0.69	-870.00	1093.34	1054.57	-1.19

4.3 Accidental eccentricity (accidental torsion)

The accidental eccentricity accounts for inaccuracies in the distribution of masses in the structure. Design codes usually take it into account as an additional eccentricity that is defined as a fraction of the size of the structure.

In the Eurocode 8, the accidental eccentricity for a given floor is defined as 5% of the width of the floor perpendicularly to the direction of the acting seismic action.

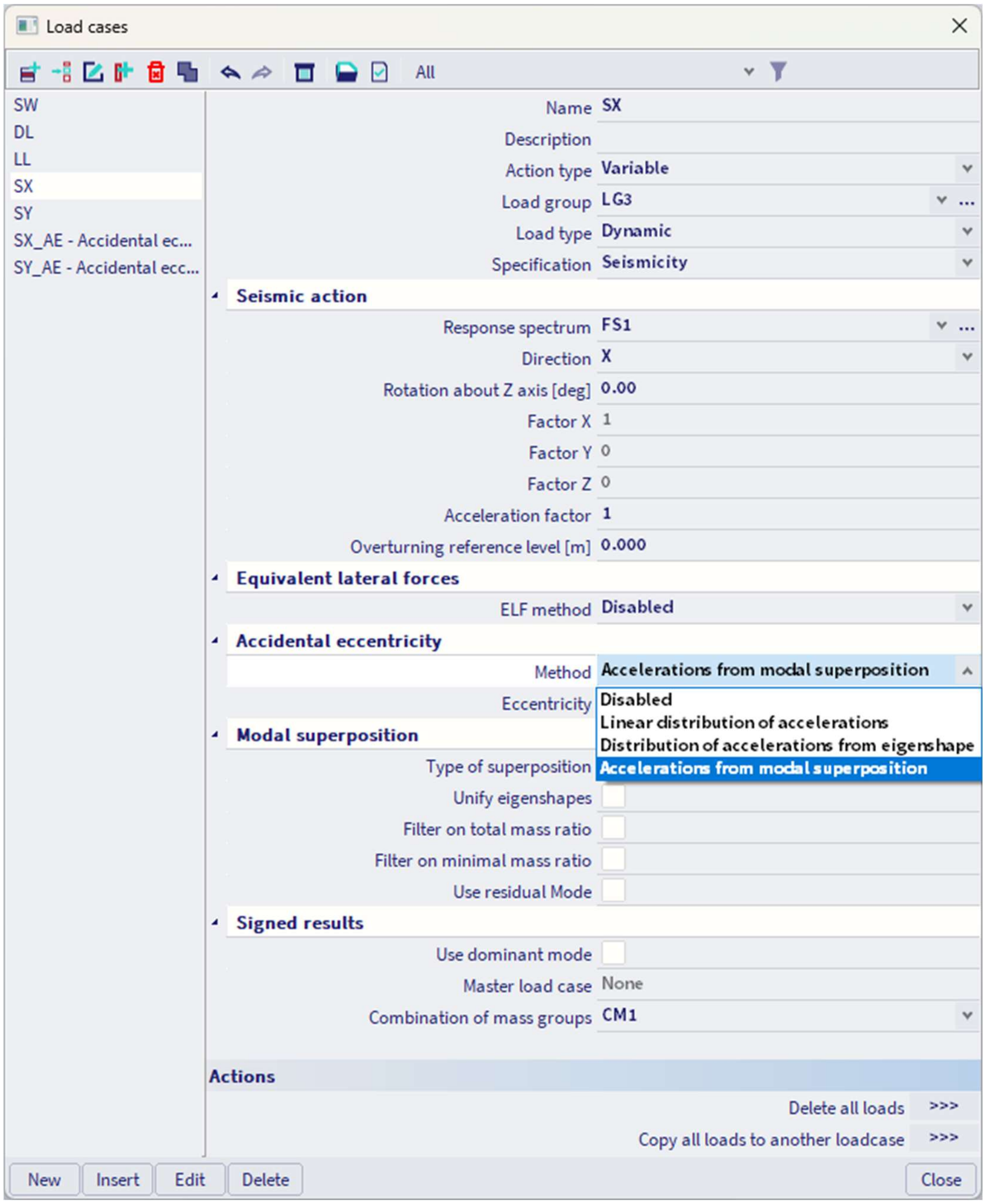
In SCIA Engineer, using the IRS condensed model allows introducing accidental eccentricity easily, since the condensed model uses only one R-node per storey. The accidental eccentricity may be taken into account either as real mass eccentricity or as additional torsion actions (simplified method according to the design codes).
However, SCIA Engineer uses the simplified method using additional torsion moment.
Accidental eccentricity is added through static loading (acc. EN 1998-1 4.3.3.3.3)



Example 04-2.esa

In SCIA Engineer, the accidental torsion can be accounted for in a seismic project using the **IRS method**.

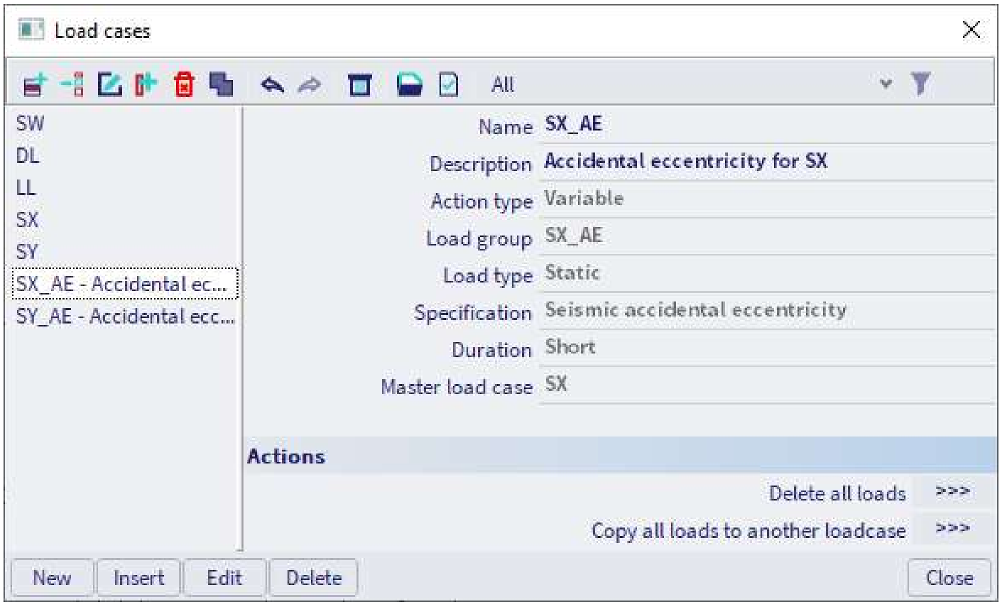
Open the **Load cases** window and select one type of Accidental eccentricity:

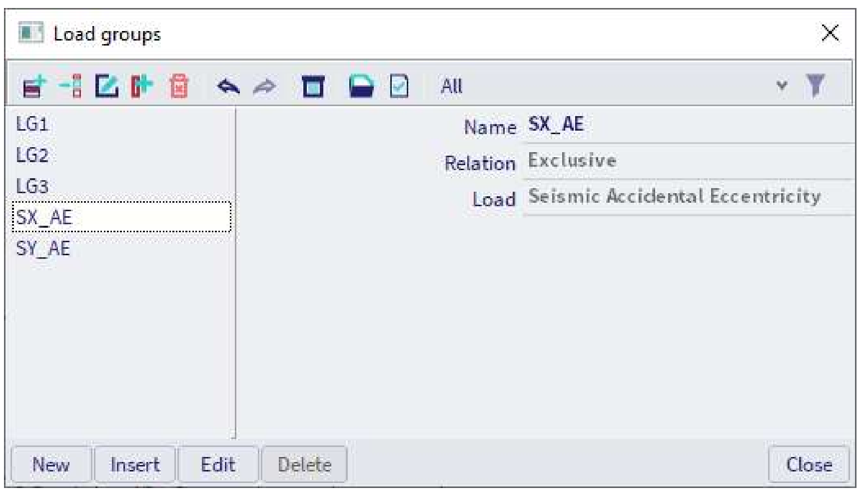


The following methods are available for calculation of AE moments:

- Linear distribution of accelerations (EN 1998-1 4.3.3.3.3 and formula (4.11))
- Distribution of accel. from eigenshape (EN 1998-1 4.3.3.3.3 and formula (4.10))
- Accelerations from modal superposition

Once the accidental eccentricity is selected, a new AE load case and also a new load group are automatically created:





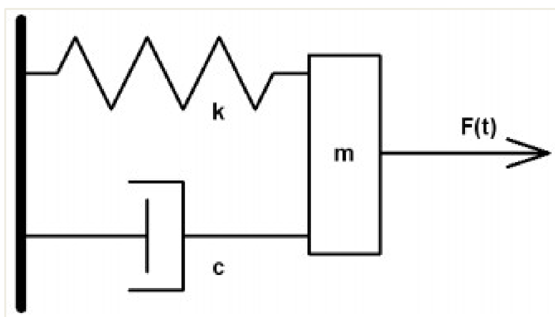
CHAPTER 5 : FORCED VIBRATIONS : HARMONIC LOAD

In this chapter, the forced vibration calculation is examined. More specifically, the structure will now be loaded with an external harmonic load, which will cause the structure to vibrate.
A forced vibration calculation can be required to check the response of a building near a railroad or major traffic lane, to check vibrations due to machinery, to verify structural integrity of a floor loaded by an aerobics class,…
As in the previous chapter, first the theory will be discussed. The theory will then be illustrated by examples, which will again be verified by manual calculations.

5.1 Theory

To understand what is going on during the dynamic analysis of a complex structure with frames or finite elements, the forced vibration of a SDOF (Single Degree Of Freedom) system is regarded in detail. A complete overview can be found in reference [1].

Consider the following system:



A body of mass **m** can move in one direction. A spring of constant stiffness **k**, which is fixed at one end, is attached at the other end to the body. The mass is also subjected to damping with a damping capacity **c**. An external time dependant force **F(t)** is applied to the mass.

The equation of motion can be written as:

$$m.\ddot{y}(t) + c.\dot{y}(t) + k.y(t) = F(t) \tag{3.1}$$

When the acting force on this system is a harmonic load, equation (3.1) can be rewritten as follows:

$$m.\ddot{y}(t) + c.\dot{y}(t) + k.y(t) = F.\sin(\nu.t) \tag{3.2}$$

With:

F: amplitude of the harmonic load
ν: circular frequency of the harmonic load

A solution to this equation is the following:

$$y(t) = e^{-\xi\omega t}.[A.\cos(\omega_D t) + B.\sin(\omega_D t)] + Y_S.\frac{\sin(\nu.t - \theta)}{\sqrt{(1-r^2)^2 + (2r\xi)^2}} \tag{3.3}$$

Where:

Y_S: the static deflection

$$Y_S = \frac{F}{k} \tag{3.4}$$

ξ: the damping ratio

$$\xi = \frac{c}{2.m.\omega} \tag{3.5}$$

ω_D: the damped circular frequency

$$\omega_D = \omega.\sqrt{1-\xi^2} \tag{3.6}$$

tan(θ):

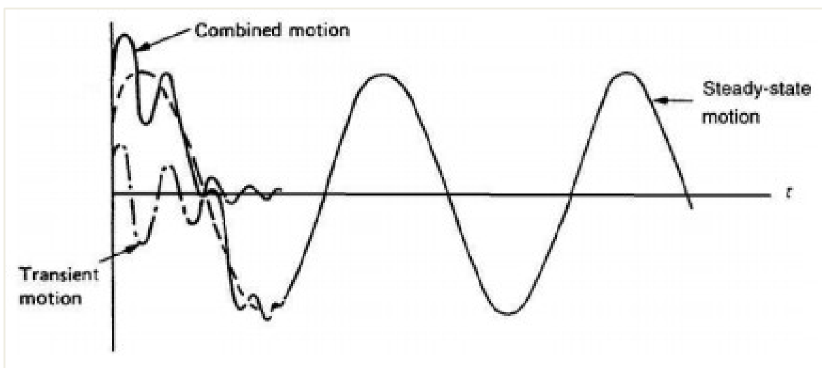
$$\tan(\theta) = \frac{2.\xi.r}{1-r^2} \tag{3.7}$$

r: the frequency ratio

$$r = \frac{\nu}{\omega} \tag{3.8}$$

The angle θ signifies that the displacement vector lags the force vector, that is, the motion occurs after the application of the force. A and B are constants which are determined from the initial displacement and velocity.

The first term of equation (3.3) is called the *Transient motion*. The second term is called the *Steady-state motion*. Both terms are illustrated on the following figure:



The amplitude of the transient response decreases exponentially ($e^{-\xi\omega t}$). Therefore, in most practical applications, this term is neglected and the total response *y(t)* can be considered as equal to the steady-state response (after a few periods of the applied load).

Equation (3.3) can then be written in a more convenient form:

$$\frac{Y}{Y_S} = \frac{1}{\sqrt{(1-r^2)^2 + (2r\xi)^2}} \tag{3.9}$$

(Y/Y_S) is known as the **Dynamic Magnification factor**, because Y_S is the static deflection of the system under a steady force F and Y is the dynamic amplitude.

The importance of mechanical vibration arises mainly from the large values of (Y/Y_S) experienced in practice when the frequency ratio **r** has a value near unity: this means that a small harmonic force can produce a large amplitude of vibration. This phenomenon is known as *resonance*. In this case, the dynamic amplitude does not reach an infinite value but a limiting value:

$$Y_S/2\xi$$

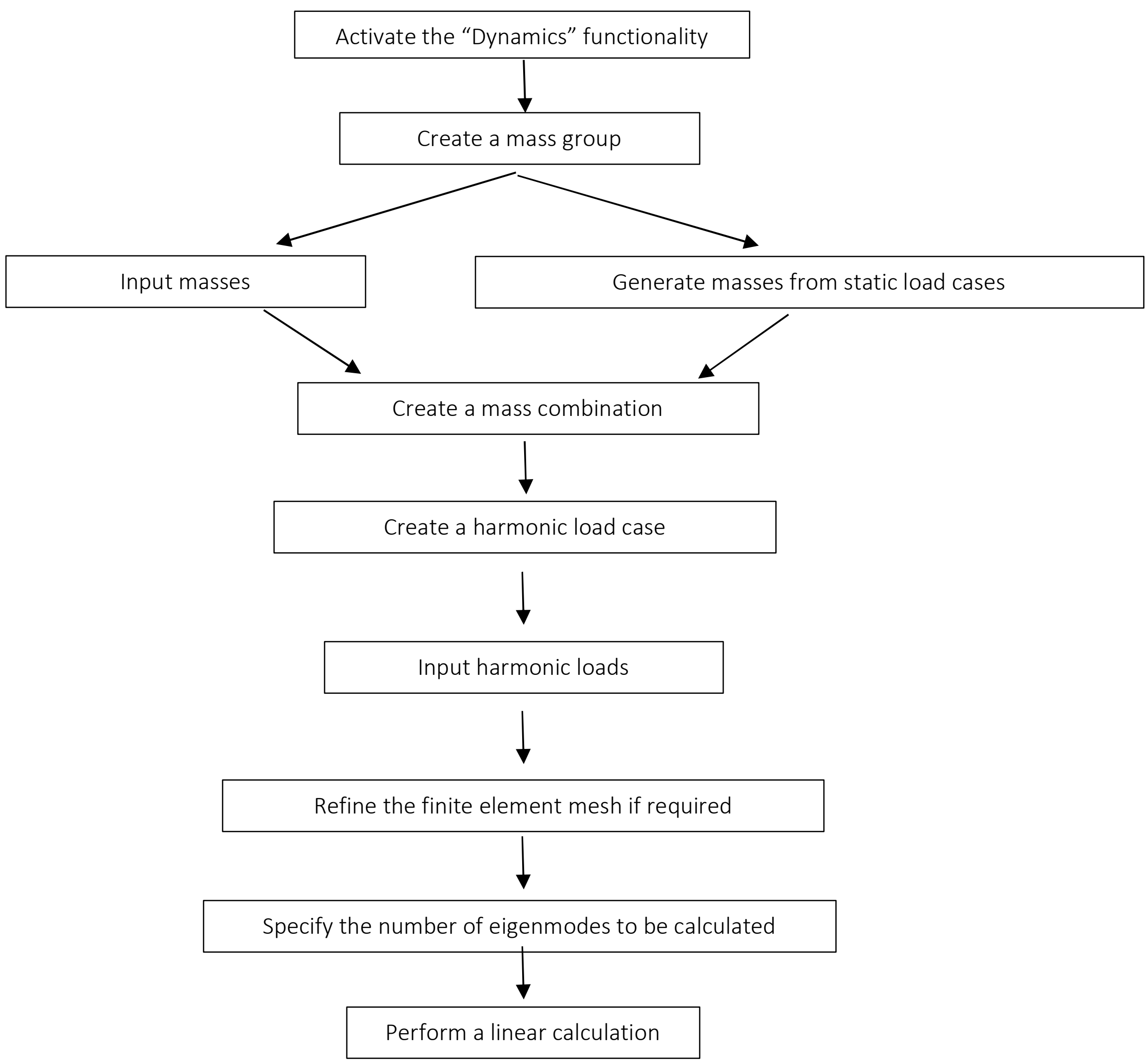
5.2 Harmonic load in SCIA Engineer

In SCIA Engineer, a Harmonic Load can be inputted after creating a Combination of Mass Groups. This implies that the steps used to perform a Free Vibration calculation still apply here and are expanded by the properties of the Harmonic Load.
Conform the theory, a Harmonic Load is defined by a forcing frequency and an amplitude. To specify the damping ratio of the structure, the damping ratio can be inputted. Note that the damping ratio and the logarithmic decrement are looked upon in more detail in chapter « Damping ».

Harmonic Loads in SCIA Engineer are always defined as nodal forces i.e. a nodal load or a nodal moment. More than one node of the structure can be loaded in a load case, but the frequency of all solicitations is equal to the forcing frequency specified for that load case.

As specified in the theory, the static results are multiplied by the dynamic magnification factor. The dynamic calculation is thus transformed to an equivalent static calculation. Therefore, a Linear Calculation needs to be executed. During this calculation, the Free Vibration Calculation will also be performed since this data is needed for the result of the Harmonic Load.

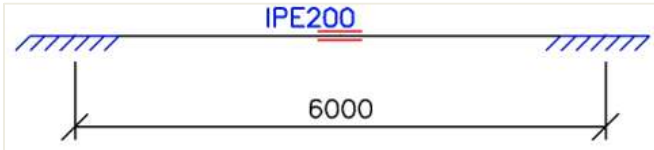
The following diagram shows the required steps to perform a Forced Vibration calculation:



This diagram is illustrated in the following examples.

Example 05-1.esa

In this example, a beam on two clamped supports is modelled. The beam has a cross-section type IPE200, a length of 6 m and is manufactured in S 235 according to EC-EN. A node has been added to the middle of the beam, in which a mass of 200 kg will be inputted.



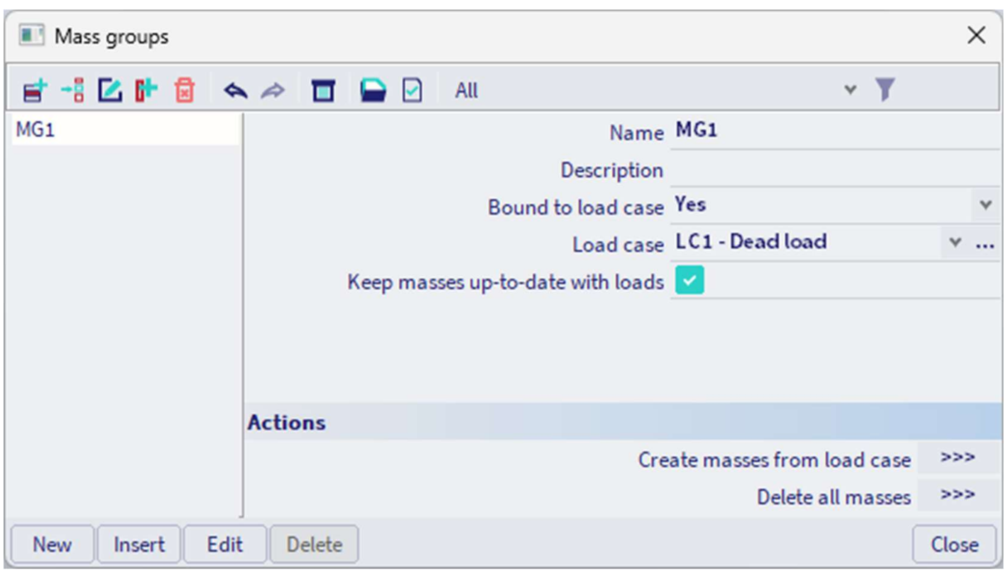
One static load case is created: the self-weight of the beam. However, in order not to take the self-weight into account for the dynamic calculation, the volumetric mass of S 235 can be set to 1 kg/m³ in the Material Library. This will render it easier to check the results through a manual calculation.
The mass of 200 kg is vibrating with a frequency of 5 Hz. The damping ratio of the system is taken as 5%.

Step 1: functionality

The first step in the Dynamic calculation is to activate the functionality Dynamics on the Functionality tab in the Project Data.

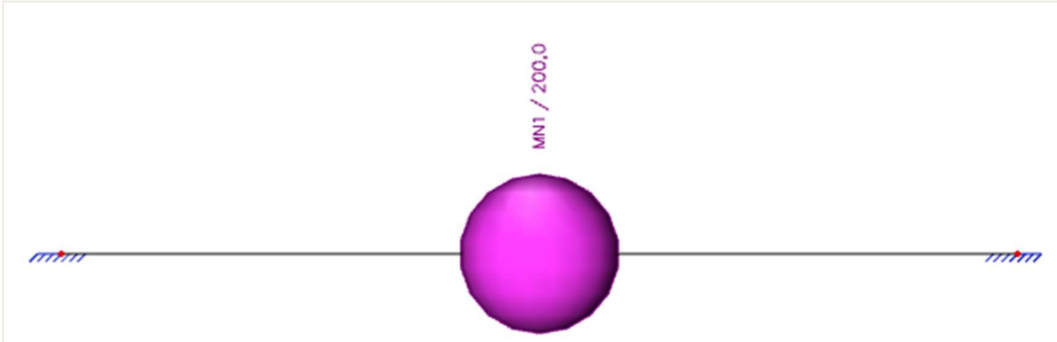
Step 2: mass group

The second step is to create a Mass Group



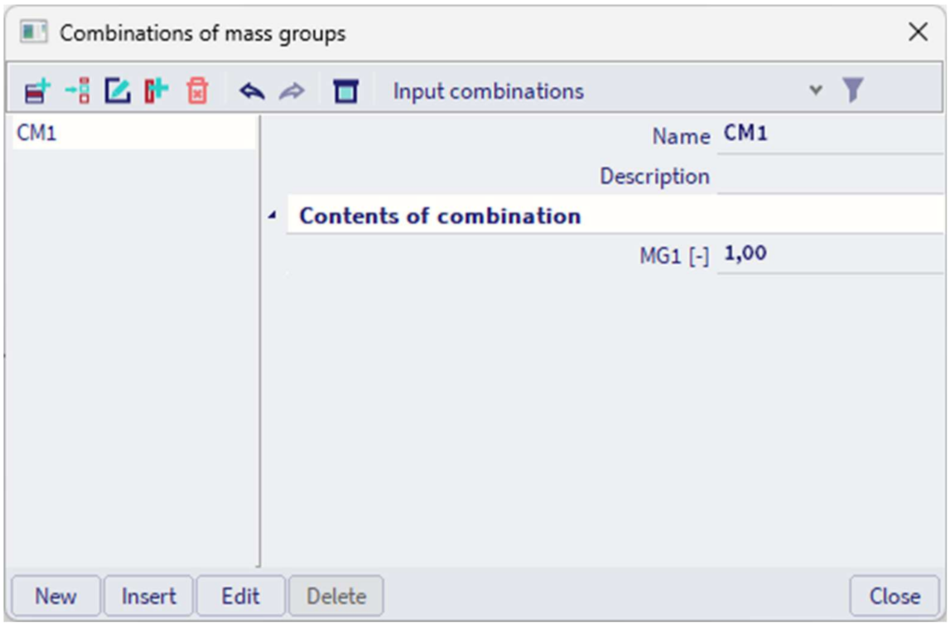
Step 3: masses

After the Mass Group has been created; the mass of 200 kg can be inputted in the middle of the beam.



Step 4: mass matrix

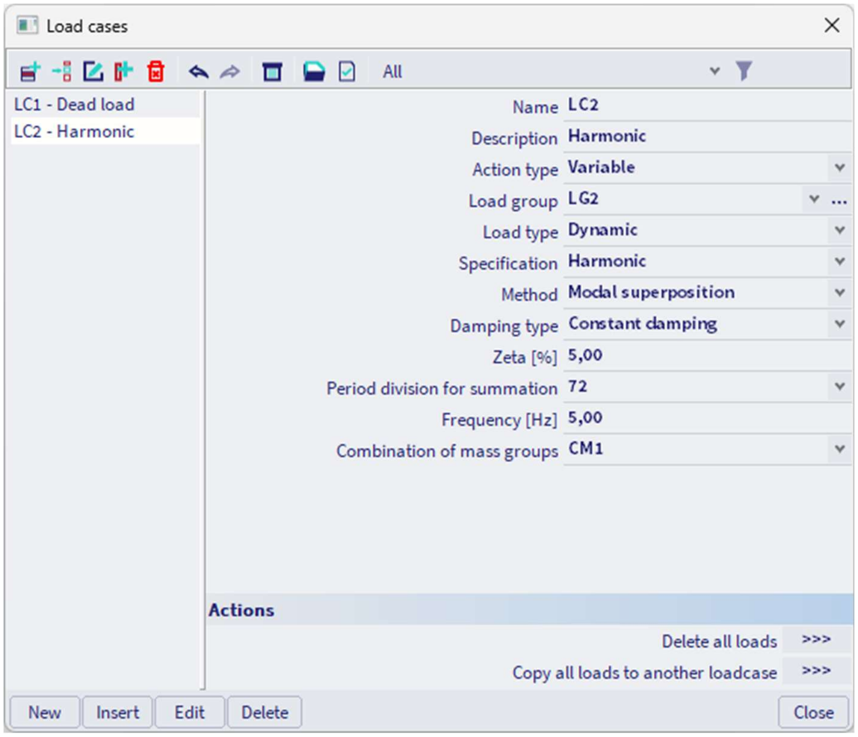
Next, the Mass Group is put within a Combination of Mass Groups, which can be used for defining the harmonic load.



Step 5: harmonic load case

After creating a Combination of Mass Groups, an harmonic load case can be defined through Load cases, Combinations > Load Cases.

The Action type is defined on Variable, the Load type is Dynamic.
On "Specification", the type of load case « Earthquake » is defined by default. But in this case, it is an Harmonic load case. The excitation frequency of the harmonic load is 5Hz.

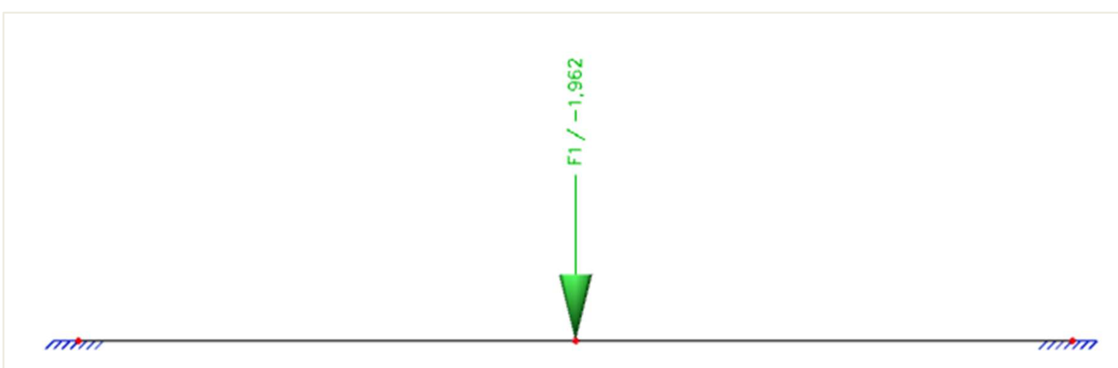


The damping ratio is constant and is equal to 5%.

The last option, Combination of mass groups, shows which mass combination (mass matrix) will be used for the calculation of the harmonic load case.

Step 6: introduction of a point load

The parameters of the load case have been defined, what is left is inputting the amplitude of the load. The mass was 200 kg.
This corresponds to a load of 1,962 kN using 9,81 m/s² for the acceleration of gravity.
This load can be inputted through the input panel « Point load on node »:

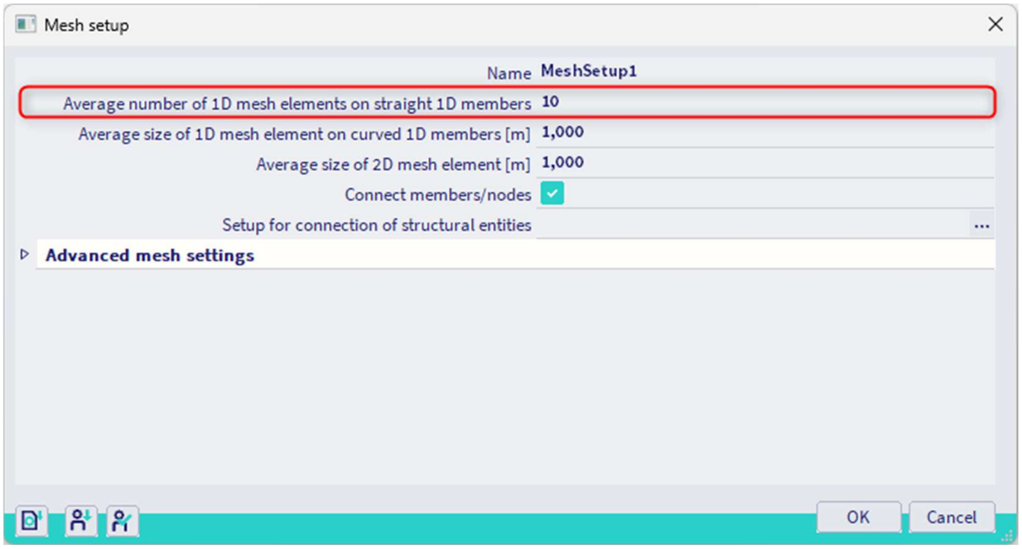


NB:

As specified in the theory, more than one harmonic load can be inputted in the same harmonic load case however the harmonic parameters like damping and forcing frequency are defined on the level of the load case. This implies that, for example, when several harmonic loads are vibrating with different frequencies, different load cases have to be created.

Step 7: mesh setup

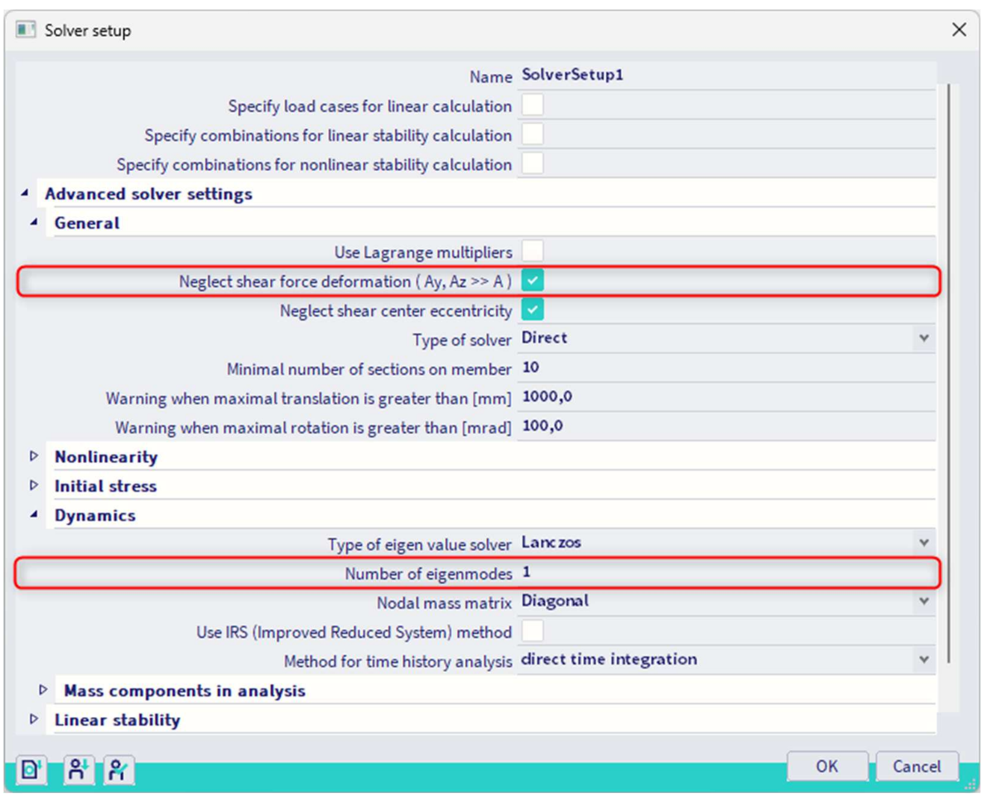
To obtain precise results for the dynamic calculation, the Finite Element Mesh is refined.
This can be done through the main menu Tools / Calculation & Mesh / Mesh settings.



The Average number of tiles of 1D element is set to 10.

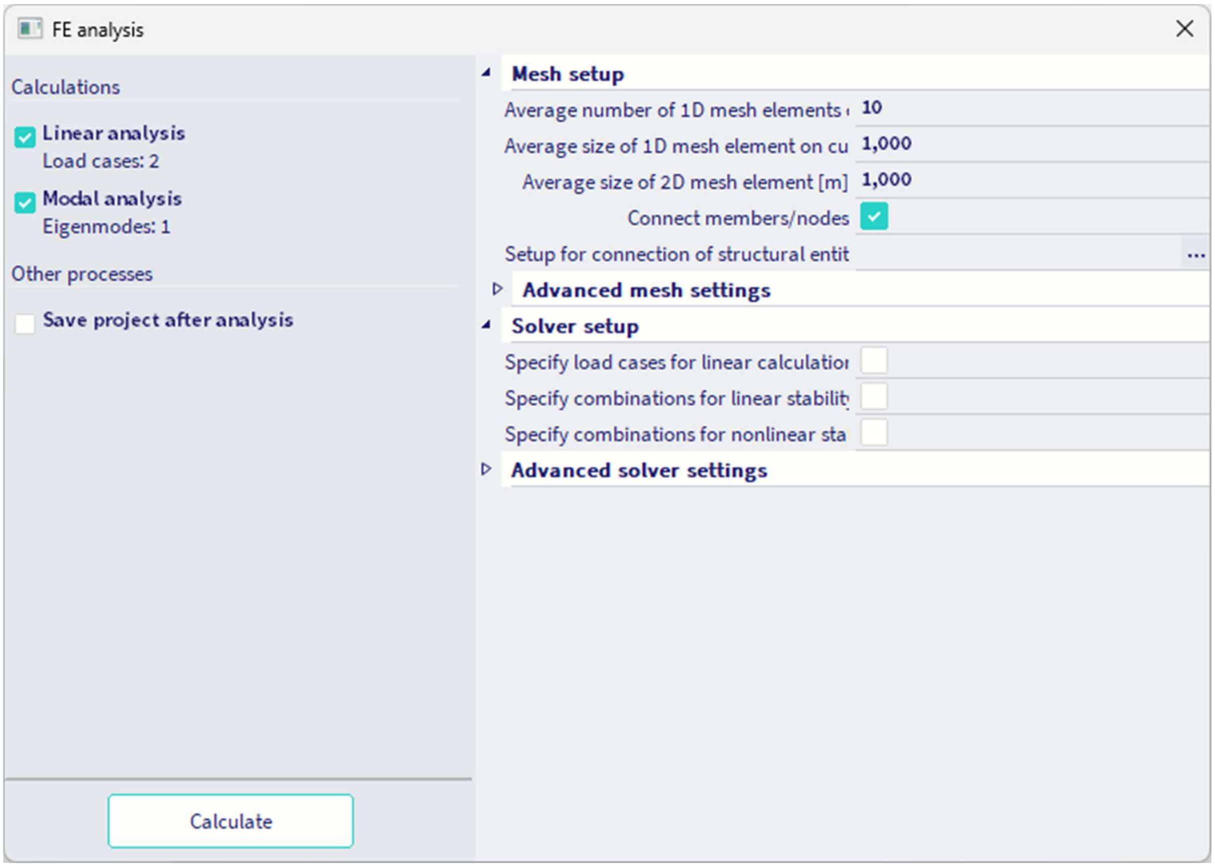
Step 8: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. For this example, only one eigenmode is required so in Calculation & Mesh / Solver Settings the number of frequencies is set to 1.
To compare the results with a manual calculation, the shear force deformation is neglected.



Step 9: modal analysis

All steps have been executed so the Linear calculation and modal analysis can be started through Calculation, mesh > Calculation.

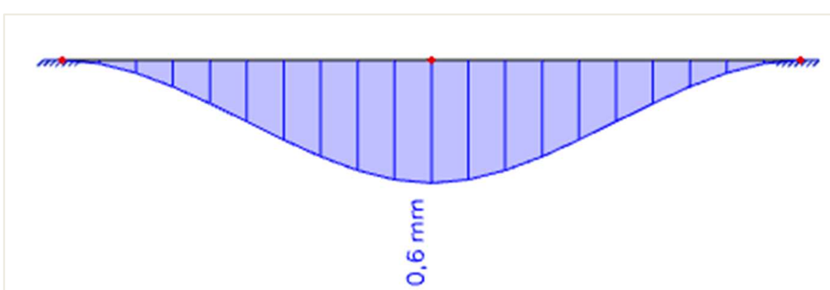


This gives the following results:

Eigen frequencies

N	f [Hz]	ω [1/s]	ω² [1/s²]	T [s]
Mass combination : CM1				
1	21,43	134,66	18132,59	0,05

The deformation for the harmonic load shows the following:



1D deformations

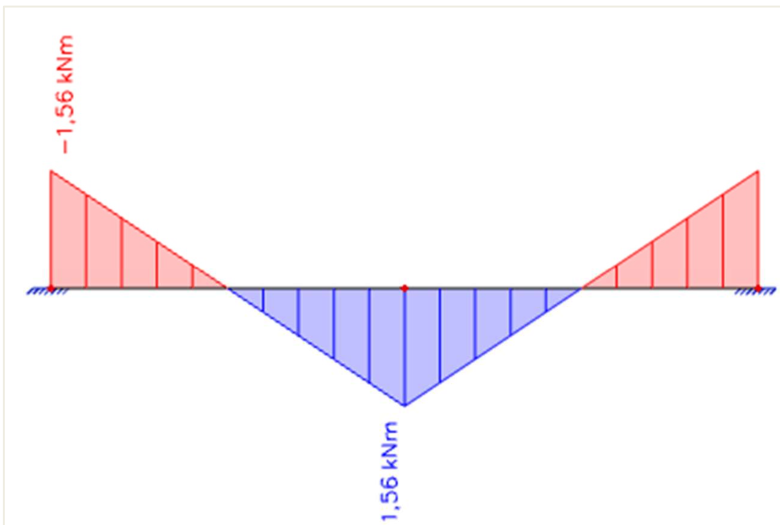
Linear calculation
Load case: LC2
Coordinate system: Global
Extreme 1D: Global
Selection: All

Deformations

Name	dx [m]	Case	u_x [mm]	u_z [mm]	φ_y [mrad]	U_{total} [mm]
B1	0,000	LC2	**0,0**	**0,0**	0,0	0,0
B1	3,000-	LC2	0,0	**-0,6**	0,0	**0,6**
B1	4,500-	LC2	0,0	-0,3	**-0,3**	0,3
B1	1,500-	LC2	**0,0**	-0,3	**0,3**	0,3

It is however very important to keep in mind that this is a vibration: half a period later the deformation is to the upper side of the beam instead of the lower side.

The moment diagram for the harmonic load would give the next diagram:



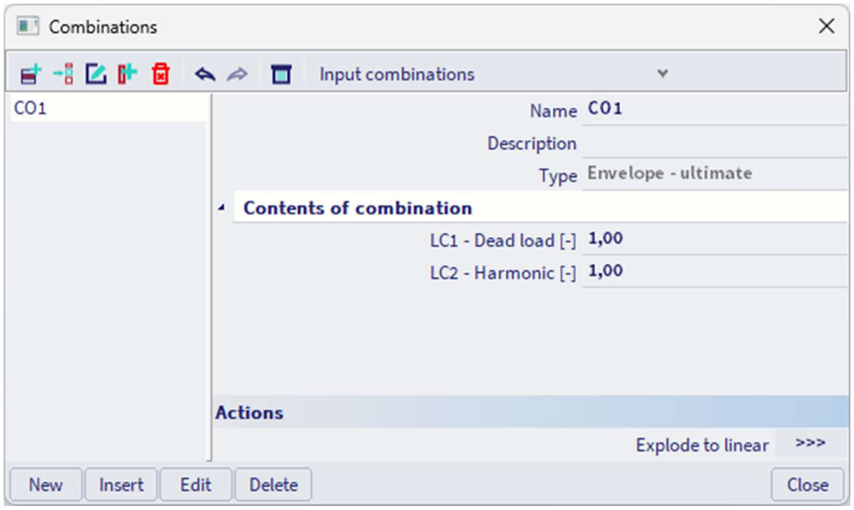
1D internal forces

Linear calculation
Load case: LC2
Coordinate system: Member
Extreme 1D: Global
Selection: All

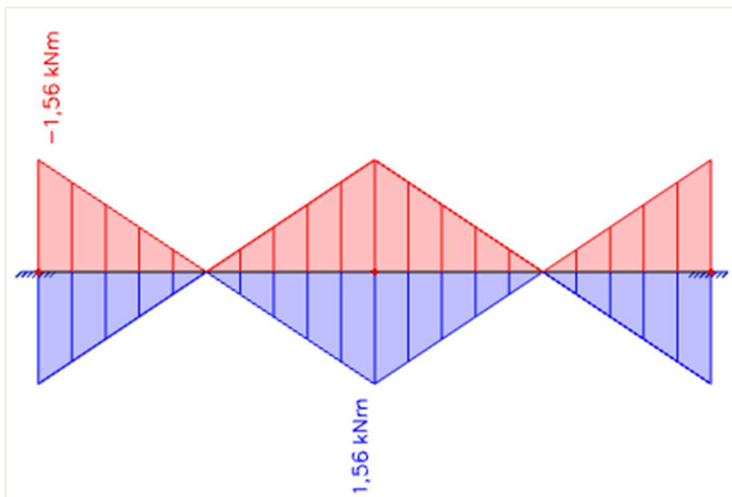
Name	dx [m]	Case	N [kN]	V_z [kN]	M_y [kNm]
B1	5,400+	LC2	0,00	**-1,04**	-0,93
B1	0,000	LC2	0,00	**1,04**	**-1,56**
B1	3,000-	LC2	**0,00**	1,04	**1,56**

This diagram is completely analogous to the moment diagram which one finds for a simple point load.
However, when performing dynamic calculations, one must always take into account both directions of the loading since the load vibrates in both directions.
In SCIA Engineer, this double sided deformation can easily be checked by creating combinations of type code or envelope. In these combinations, the dynamic load cases will be accounted for with both a positive and a negative combination coefficient and thus both sides of the vibration amplitude are taken into account.

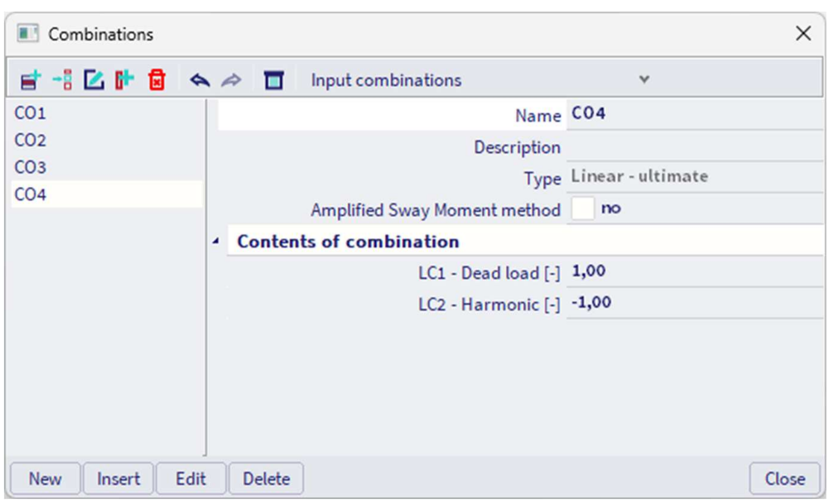
In this example, a combination of type Envelope - ultimate is created which contains only the harmonic load case.



The moment diagram for this combination shows the following:



The vibration effect is correctly taken into account: both sides of the vibration are visible. This is also shown in the **Combination Key** of the **Document**; which shows the two generated Linear combinations from the Envelope combination (Local Extremes):



Manual calculation

In order to check the results of SCIA Engineer, a manual calculation is performed.
First, the calculated eigen frequency is checked using formula (2.3).
Using default engineering tables [11], the maximum static deformation of a beam with length L, clamped at both sides and loaded with a load F in the middle is given as:

$$\delta_{max} = \frac{FL^3}{129EI} \tag{3.12}$$

Where:

F = 1,962 kN = 1962 N
L = 6 m = 6000 mm
E = 210000 N/mm²
I = 19430000 mm^4

So:

$$\delta_{max} = \frac{(1962N) * (6000mm)^3}{129 * 210000\ {}^{N}/_{mm^2} * 19430000mm^4} = 0{,}54095mm$$

The k rigidity of this system can then be calculated:

$$k = \frac{F}{\delta_{max}} = \frac{1962N}{0{,}54095mm} = 3626{,}93\ {}^{N}/_{mm} = 3626933{,}33\ {}^{N}/_{m}$$

Applying formula (2.3):

$$\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{3626933{,}33\ {}^{N}/_{m}}{200kg}} = 134{,}67\ {}^{rad}/_{s}$$

So:

$$f = \frac{\omega}{2\pi} = \mathbf{21{,}43Hz}$$

This result corresponds exactly to the result calculated by SCIA Engineer.

Now the eigen frequency is known, the results of the harmonic load can be verified.
The harmonic load had a forcing frequency of 5 Hz, which corresponds to a circular frequency of 31,416 rad/s.

Applying formula (3.8) the frequency ratio can be calculated:

$$r = \frac{\nu}{\omega} = \frac{31{,}416\ {}^{rad}/_{s}}{134{,}67\ {}^{rad}/_{s}} = 0{,}233289$$

The frequency ratio can then be used in formula (3.9) to calculate the Dynamic Magnification Factor:

$$\frac{Y}{Y_S} = \frac{1}{\sqrt{(1 - r^2)^2 + (2r\xi)^2}} = \frac{1}{\sqrt{(1 - 0{,}233289^2)^2 + (2 * 0{,}233289 * 0{,}05)^2}} = \mathbf{1{,}0572}$$

This implies that the static results need to be multiplied by 1,0572 to obtain the dynamic results.
The static deformation was calculated as $\delta_{max} = 0{,}54095$ mm.

The dynamic deformation is equal to 1,0572 * 0,54095mm = 0,5719 mm.

This result corresponds exactly to the result calculated by SCIA Engineer.

In the same way the moment in the middle of the beam can be calculated.
Using default engineering tables [11], the maximum static moment in the middle of a beam with length L, clamped at both sides and loaded with a load F in the middle is given as:

$$M = \frac{FL}{8} = \frac{1{,}962kN * 6m}{8} = 1{,}4715kNm$$

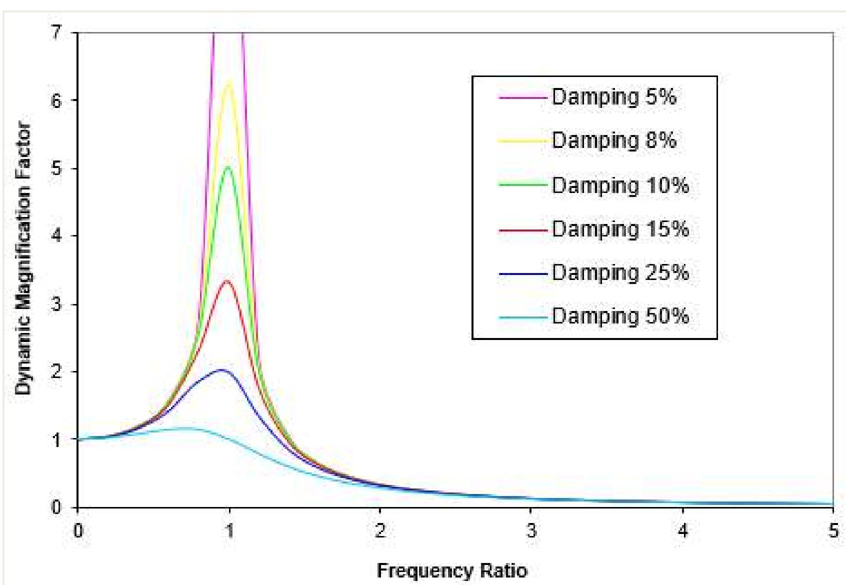
The dynamic moment is equal to 1,0572 * 1,4715kNm = 1,556kNm
This result corresponds exactly to the result calculated by SCIA Engineer.

5.3 Resonance

As specified in the theory, resonance occurs when the frequency ratio **r** has a value near unity. In this case, large values for the Dynamic Amplification factor are obtained.
To illustrate this, the calculation of the Dynamic Amplification Factor is repeated for different frequency ratios and different damping percentages. The results are given in the following table:

Frequency Ratio	Forcing Frequence [Hz]	Mag. factor Damping 5%	Mag. factor Damping 8%	Mag. factor Damping 10%	Mag. factor Damping 15%	Mag. factor Damping 25%	Mag. factor Damping 50%
0,0	0,00	1,0000	1,0000	1,0000	1,0000	1,0000	1,0000
0,2	4,29	1,0414	1,0411	1,0408	1,0396	1,0361	1,0198
0,4	8,57	1,1891	1,1870	1,1851	1,1785	1,1581	1,0748
0,6	12,86	1,5557	1,5452	1,5357	1,5041	1,4148	1,1399
0,8	17,15	2,7116	2,6173	2,5384	2,3113	1,8582	1,1399
1,0	21,43	10,0000	6,2500	5,0000	3,3333	2,0000	1,0000
1,2	25,72	2,1926	2,0830	1,9952	1,7590	1,3440	0,7824
1,4	30,01	1,0308	1,0144	1,0000	0,9543	0,8417	0,5891
1,6	34,29	0,6377	0,6326	0,6280	0,6127	0,5704	0,4475
1,8	38,58	0,4450	0,4428	0,4408	0,4340	0,4142	0,3480
2,0	42,87	0,3326	0,3315	0,3304	0,3269	0,3162	0,2774
2,2	47,15	0,2600	0,2593	0,2587	0,2567	0,2503	0,2260
2,4	51,44	0,2098	0,2094	0,2090	0,2077	0,2037	0,1876
2,6	55,73	0,1734	0,1732	0,1729	0,1720	0,1694	0,1582
2,8	60,01	0,1461	0,1459	0,1457	0,1451	0,1432	0,1353
3,0	64,30	0,1249	0,1248	0,1246	0,1242	0,1229	0,1170
3,2	68,59	0,1082	0,1081	0,1080	0,1076	0,1066	0,1023
3,4	72,87	0,0946	0,0946	0,0945	0,0943	0,0935	0,0901
3,6	77,16	0,0836	0,0835	0,0835	0,0833	0,0827	0,0801
3,8	81,45	0,0744	0,0743	0,0743	0,0741	0,0737	0,0716
4,0	85,73	0,0666	0,0666	0,0666	0,0665	0,0661	0,0644
4,2	90,02	0,0601	0,0600	0,0600	0,0599	0,0596	0,0583
4,4	94,31	0,0545	0,0544	0,0544	0,0543	0,0541	0,0530
4,6	98,59	0,0496	0,0496	0,0496	0,0495	0,0493	0,0484
4,8	102,88	0,0454	0,0453	0,0453	0,0453	0,0451	0,0443
5,0	107,17	0,0417	0,0416	0,0416	0,0416	0,0414	0,0408

In order to draw conclusions, the numerical results are plotted graphically:



Amplitude – frequency response

First of all, the resonance phenomenon is clearly visible. When the frequency ratio equals unity, the Dynamic Magnification factor becomes very large indicating that a small harmonic load can produce a large amplitude of vibration. Second, the influence of the damping ratio on the system response in resonance is significant. With a damping ratio of **5%**, the magnification factor is about **10**; with a damping ratio of **50%**, the magnification factor is reduced to **1**.

In general, the following can be concluded from this graphic [1]:
The system response at low frequencies is **stiffness-dependent**. In the region of resonance, the response is **damping-dependent** and at high frequencies, the response is governed by the system mass: **mass-dependent**.
It is important to keep this in mind when attempting to reduce the vibration of a structure. For example, the application of increased damping will have little effect if the excitation and response frequencies are in a region well away from resonance, such as that controlled by the mass of the structure.

The effect of resonance can also be illustrated in SCIA Engineer.
In the project "Harmonic_Load_1", the excitation frequency is 5 Hz. The eigenfrequency is 21.43 Hz. So this is not in the resonance area.

To see the response in function of the frequency, we can create several load cases with other excitation frequency. You can easily do this by copying the existing load case and changing the excitation frequency. This is shown in the next example.

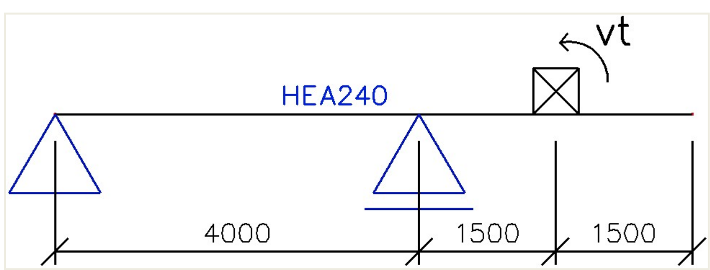
Example 05-2.esa

Another common application of a harmonic load is a structure loaded with a plunger system or a motor. Both the reciprocating effect of the plunger and the rotating unbalance of the motor produce an exciting force of the inertia type of the system.
For an unbalanced body of mass **m**, at an effective radius **e**, rotating at an angular speed **ν**, the exciting force **F** can be written as [1]:

$$F = m_r \cdot e \cdot \nu^2 \tag{3.13}$$

This is illustrated in following example.

An electric motor with a mass of **500 kg** is mounted on a simply supported beam with overhang. The beam has a cross-section type **HE240A** and is manufactured in **S 235** according to **EC-EN**. The beam has a length of **4 m** and the overhang is **3 m**.
The motor has an unbalance of **0,6 kgm**. The damping ratio of the system is taken as **10%**.



The motor can operate at speeds of **800**, **1000** and **1200 rpm**. For each of these speeds, the amplitude of forced vibration needs to be calculated to check, for example, if the vibrations induced by the motor are acceptable.

One static load case is created: the **self-weight** of the beam. However, in order not to take the self-weight into account for the dynamic calculation, the volumetric mass of **S235** can be set to **1 kg/m³** in the **Material** Library. This will render it easier to check the results through a manual calculation.

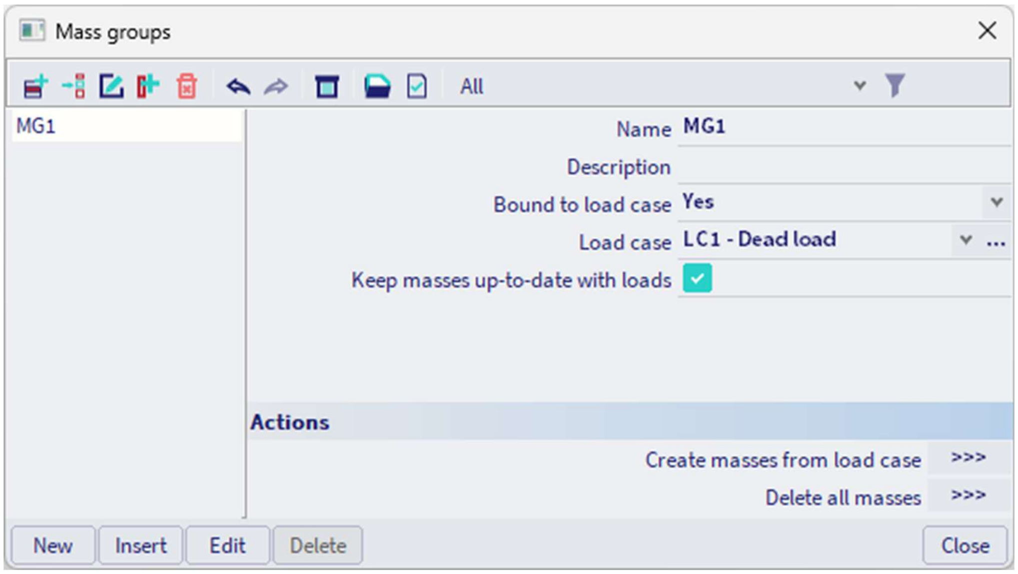
A node has been added to the middle of the overhang to specify the location of the motor.

Step 1: functionality

The first step in the Dynamic calculation is to activate the functionality Dynamics on the Functionality tab in the Project Data.

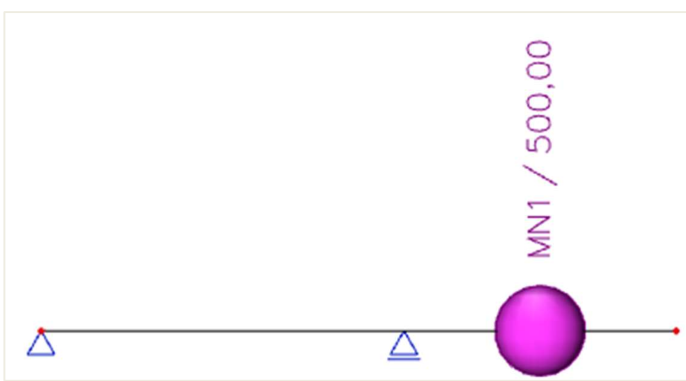
Step 2: mass group

The second step is to create a Mass Group.



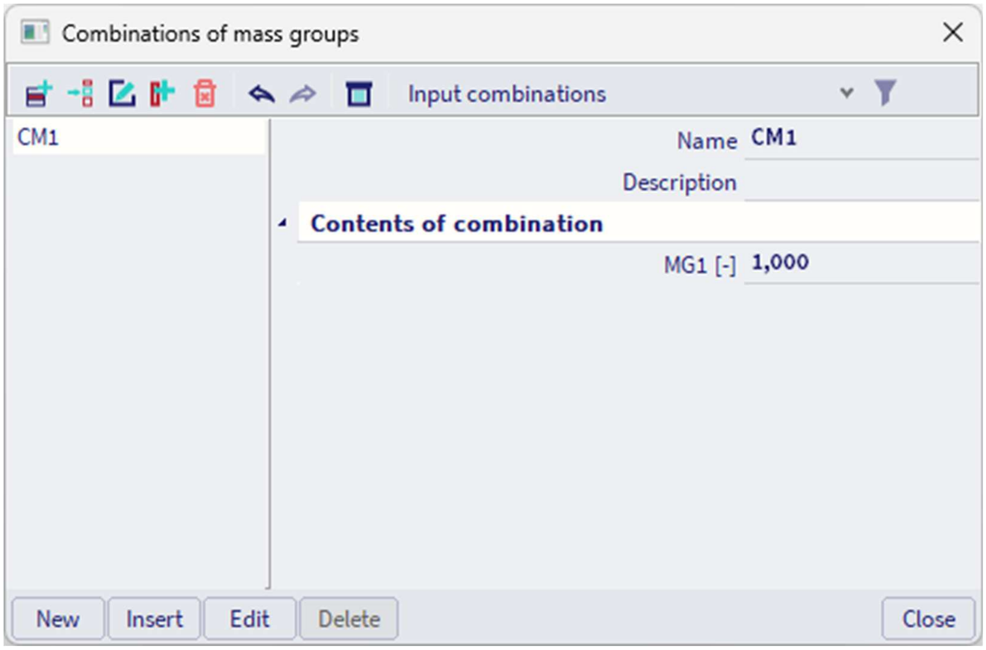
Step 3: masses

After the Mass Group has been created; the 500 kg mass of the motor can be inputted in the middle of the overhang:



Step 4: mass matrix

Next, the Mass Group is put within a Combination of Mass Groups, which can be used for defining the harmonic loads at the different speeds:



Step 5: load cases definition

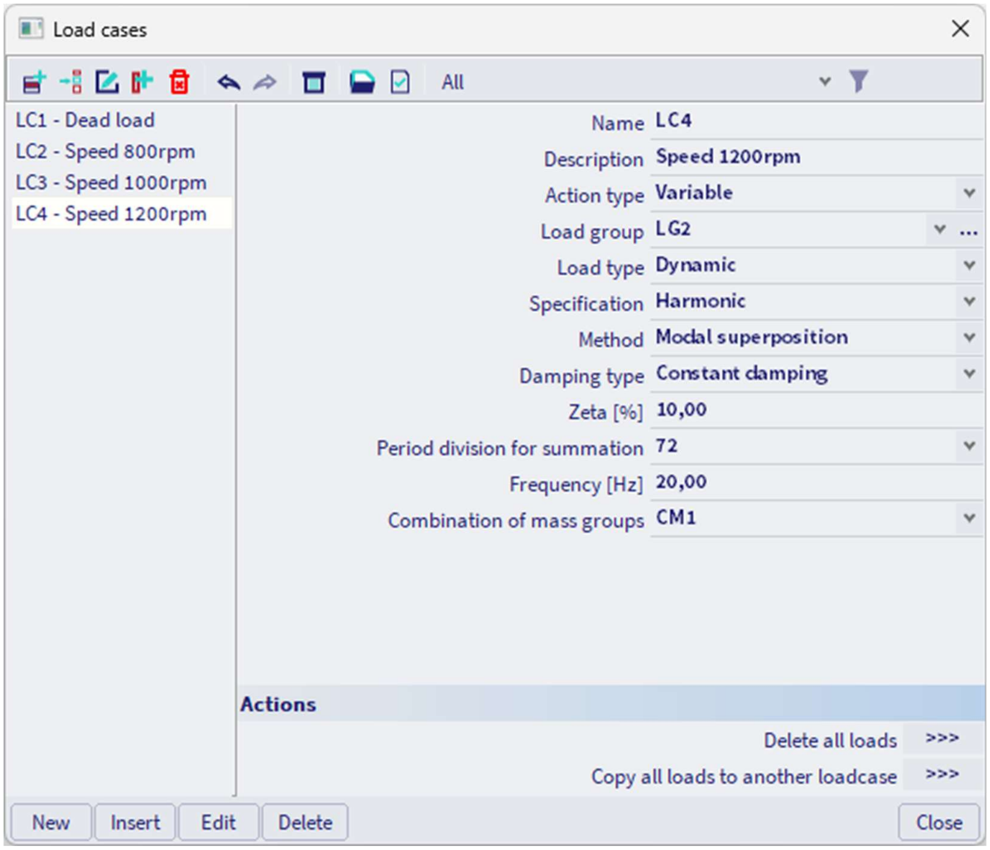
After creating the Mass Combination, three harmonic load cases can be defined, one for each speed. Each load case uses the same Mass Combination and has the same damping specifications.
The damping ratio is constant and equal to 10%.

The forcing frequency is different for each load case and can be calculated from the given speeds:

$$\nu_{800} = 800\text{rpm} * \frac{2\pi\text{rad}}{1\text{rev}} * \frac{1\text{min}}{60\text{s}} = 83{,}78\,{}^{\text{rad}}/_{\text{s}} \qquad => \ f_{800} = 13{,}33\text{Hz}$$

$$\nu_{1000} = 1000\text{rpm} * \frac{2\pi\text{rad}}{1\text{rev}} * \frac{1\text{min}}{60\text{s}} = 104{,}72\,{}^{\text{rad}}/_{\text{s}} \qquad => \ f_{1000} = 16{,}67\text{Hz}$$

$$\nu_{1200} = 1200\text{rpm} * \frac{2\pi\text{rad}}{1\text{rev}} * \frac{1\text{min}}{60\text{s}} = 125{,}66\,{}^{\text{rad}}/_{\text{s}} \qquad => \ f_{1200} = 20{,}00\text{Hz}$$



Step 6: harmonic forces

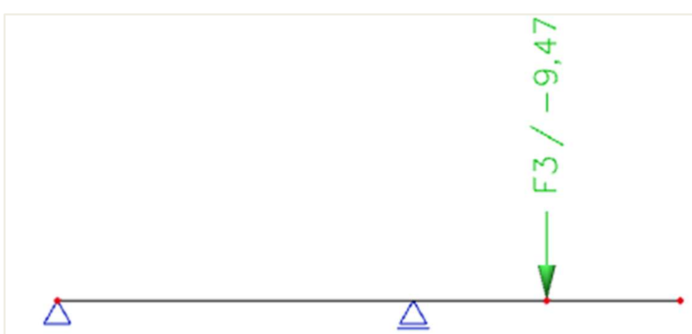
The parameters of the harmonic loads have been defined. What is left is inputting the amplitude of the three exciting forces.
Using formula (3.13) these forces can be calculated from the forcing circular frequency and the mass unbalance.

$$F_{800} = m_r.e.v_{800}^2 = 0{,}6\text{kgm} * \left(83{,}78\, {}^{\text{rad}}/_{\text{s}}\right)^2 = 4211{,}03\text{N} = 4{,}21\text{kN}$$

$$F_{1000} = m_r.e.v_{1000}^2 = 0{,}6\text{kgm} * \left(104{,}72\, {}^{\text{rad}}/_{\text{s}}\right)^2 = 6579{,}74\text{N} = 6{,}58\text{kN}$$

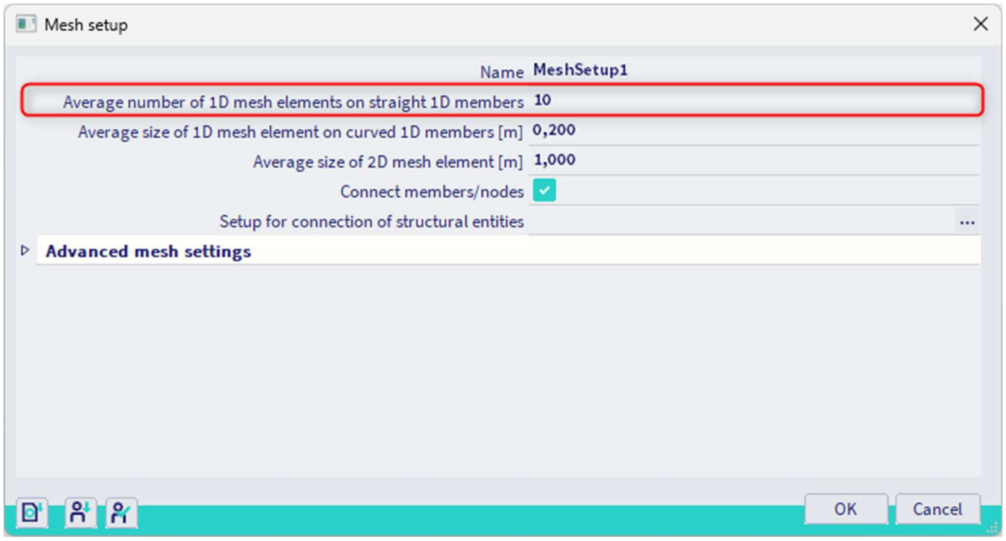
$$F_{1200} = m_r.e.v_{1200}^2 = 0{,}6\text{kgm} * \left(125{,}66\, {}^{\text{rad}}/_{\text{s}}\right)^2 = 9474{,}82\text{N} = 9{,}47\text{kN}$$

The loads are inputted through Load > Point Force > In Node:



Step 7: mesh setup

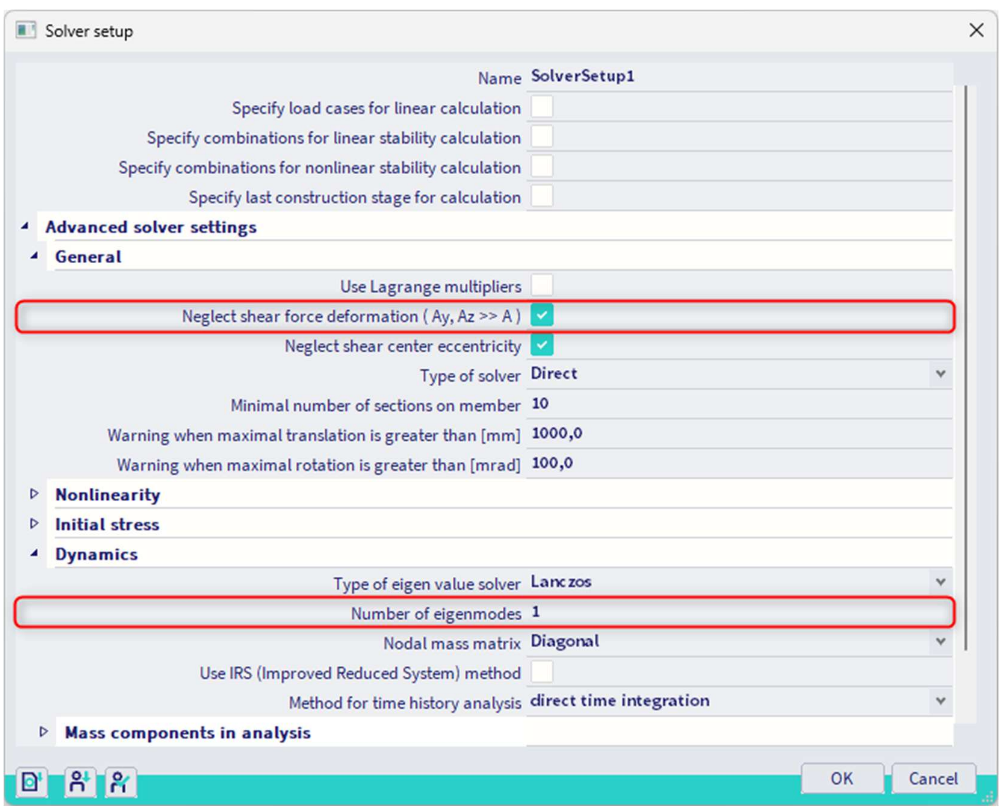
To obtain precise results for the dynamic calculation, the Finite Element Mesh is refined.
This can be done through Calculation & Mesh / Mesh Settings.



The Average number of tiles of 1D element is set to 10.

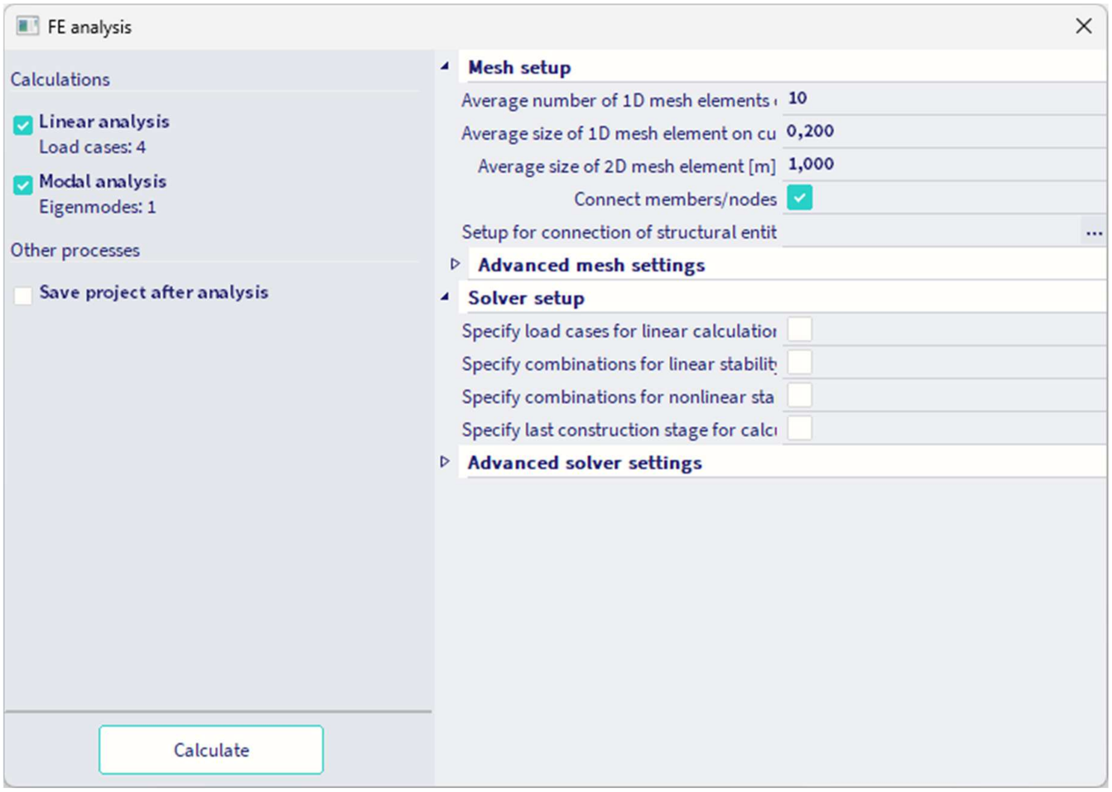
Step 8: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. For this example, only one eigenmode is required so in Calculation & Mesh / Solver Settings the number of frequencies is set to 1.
To compare the results with a manual calculation, the shear force deformation is neglected.



Step 9: modal analysis

All steps have been executed so the Linear calculation and modal analysis can be started through Calculation, mesh > Calculation.



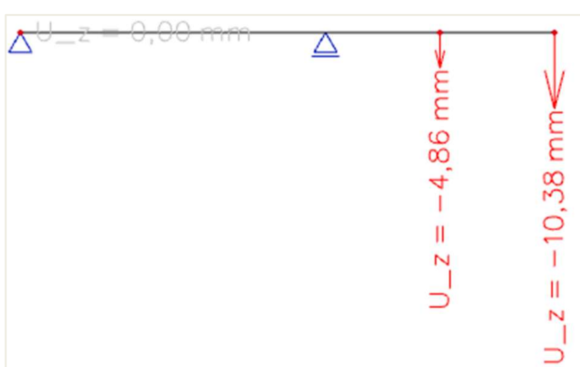
This gives the following results:

Eigen frequencies

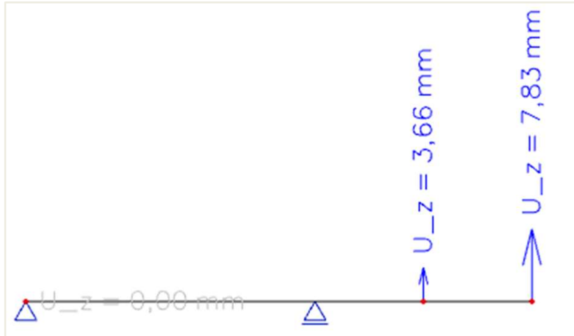
N	f [Hz]	ω [1/s]	ω² [1/s²]	T [s]
Mass combination : CM1				
1	14,15	88,88	7899,96	0,07

The nodal deformations for the harmonic loads at the location of the motor are the following:

- At 800 rpm:



- At 1000 rpm:



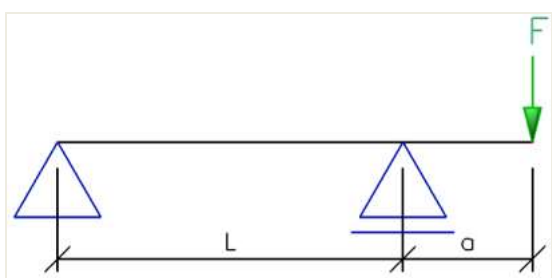
- At 1200 rpm:

As stated in the previous example, it is important to keep in mind that the signs are not relevant since a vibration occurs on both sides of the equilibrium position.

Manual calculation

In order to check the results of SCIA Engineer, a manual calculation is performed [15].
First, the calculated eigen frequency is checked using formula (2.3)
Using default engineering tables [11], the maximum static deformation of a simply supported beam with length L, an overhang with length a and loaded with a load F at the end of the overhang is given as:

$$\delta_{\max} = \frac{Fa^2(L+a)}{3EI} \tag{3.14}$$



The rigidity k of this system can then be calculated:

$$k = \frac{F}{\delta_{\max}} = \frac{3EI}{a^2(L+a)}$$

Where:

L = 4 m = 4000 mm
a = 1,5 m = 1500 mm
E = 210000 N/mm²
I = 77600000 mm⁴

So:

$$k = \frac{3 * \left(210000 \,{}^{N}\!/_{mm^2}\right) * (77600000mm^4)}{(1500mm)^2 * (4000mm + 1500mm)} = 3950{,}55 \,{}^{N}\!/_{mm} = 3950545{,}45 \,{}^{N}\!/_{m}$$

Applying formula (2.3):

$$\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{3950545{,}45 \,{}^{N}\!/_{m}}{500kg}} = 88{,}89 \,{}^{rad}\!/_{s}$$

So:

$$f = \frac{\omega}{2\pi} = \mathbf{14{,}15Hz}$$

This result corresponds exactly to the result calculated by SCIA Engineer.

Applying formula (3.8) the frequency ratios can be calculated for each motor speed:

$$r_{800} = \frac{\nu_{800}}{\omega} = \frac{83{,}78 \,{}^{rad}\!/_{s}}{88{,}89 \,{}^{rad}\!/_{s}} = 0{,}9425$$

$$r_{1000} = \frac{\nu_{1000}}{\omega} = \frac{104{,}72 \,{}^{rad}\!/_{s}}{88{,}89 \,{}^{rad}\!/_{s}} = 1{,}1781$$

$$r_{1200} = \frac{\nu_{1200}}{\omega} = \frac{125{,}66 \,{}^{rad}\!/_{s}}{88{,}89 \,{}^{rad}\!/_{s}} = 1{,}4137$$

The frequency ratios can then be used in formula (3.9) to calculate the Dynamic Magnification Factors. When also applying formula (3.4) the Dynamic Amplitude can be calculated for each speed:

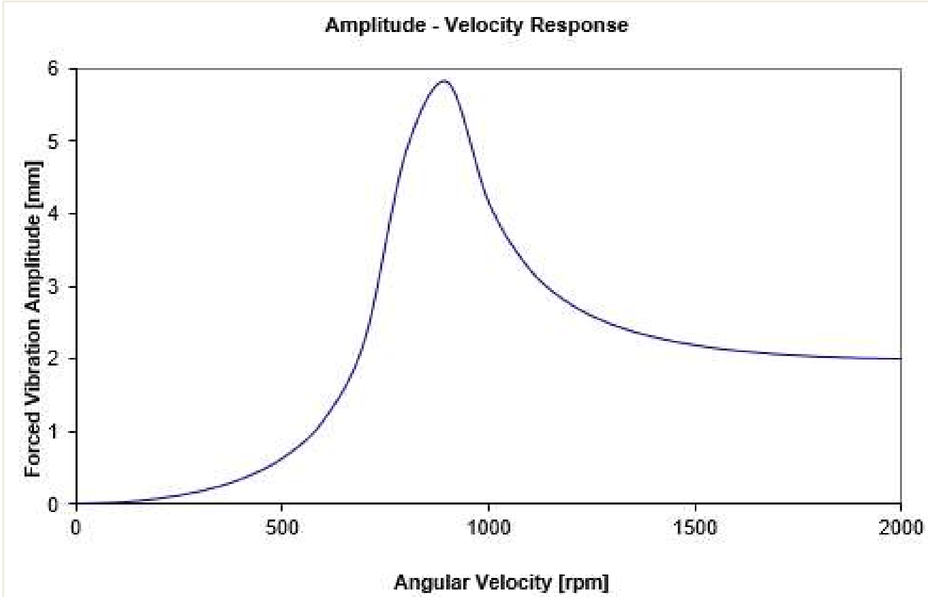
$$Y_{800} = \frac{{}^{F_{800}}\!/_{k}}{\sqrt{(1 - r_{800}^2)^2 + (2r_{800}\xi)^2}} = \frac{{}^{4211{,}03N}\!/_{3950545{,}45 \,{}^{N}\!/_{m}}}{\sqrt{(1 - 0{,}9425^2)^2 + (2 * 0{,}9425 * 0{,}10)^2}} = 4{,}86mm$$

$$Y_{1000} = \frac{{}^{F_{1000}}\!/_{k}}{\sqrt{(1 - r_{1000}^2)^2 + (2r_{1000}\xi)^2}} = \frac{{}^{6579{,}74N}\!/_{3950545{,}45 \,{}^{N}\!/_{m}}}{\sqrt{(1 - 1{,}1781^2)^2 + (2 * 1{,}1781 * 0{,}10)^2}} = 3{,}67mm$$

$$Y_{1200} = \frac{{}^{F_{1200}}\!/_{k}}{\sqrt{(1 - r_{1200}^2)^2 + (2r_{1000}\xi)^2}} = \frac{{}^{9474{,}82N}\!/_{3950545{,}45 \,{}^{N}\!/_{m}}}{\sqrt{(1 - 1{,}4137^2)^2 + (2 * 1{,}4137 * 0{,}10)^2}} = 2{,}31mm$$

These results correspond exactly to the results calculated by SCIA Engineer.

In the same way as in the previous example, the calculation can be repeated for several angular velocities. The result is shown graphically on the following figure:



Amplitude – velocity response

NB:

The main feature to notice is the decrease in vibration amplitude when the forcing frequency increases due to moving away from resonance [15].

CHAPITRE 6 : REFERENCES

[1] Beards F., Structural Vibration: Analysis and Damping, Arnold, London, 1996.

[2] H. Buchholdt, Structural Dynamics for Engineers, Thomas Telford Publications, 1997.

[3] Chopra A., Dynamics of Structures: Theory and Applications to Earthquake Engineering, Prentice Hall, New Jersey, 2001.

[4] Eurocode 1: Basis of design and actions on structures, Part 2-4: Actions on structures - Wind actions, ENV 1991-2-4, 1995.

[5] Eurocode 3: Design of Steel Structures, Part 1-9: Fatigue Strength of Steel Structures, ENV 1993-1-9: 2002.

[6] Eurocode 8: Design of structures for earthquake resistance, Part 1-1: General rules, seismic actions and rules for buildings, ENV 1998-1-1, 1994.

[7] Eurocode 8: Design of structures for earthquake resistance, Part 1: General rules, seismic actions and rules for buildings, EN 1998-1, 2004.

[8] Eurocode 8: Design provisions for earthquake resistance of structures, Part 6: Towers, masts and chimneys, Annex B, ENV 1998-6, 2003.

[9] Eurocode 8: National Application Document for Belgium, NBN-ENV 1998-1-1, 2002 NAD.

[10] Murray M., Allen D., Ungar E., Floor Vibrations due to Human Activity, AISC, 2003.

[11] Leijendeckers P.P.H., Fortuin J.B., Van Herwijnen F., Leegwater H., Polytechnisch zakboekje (Polytechnic pocket notebook), 48e druk, Koninklijke PBNA, 1998 (in Dutch).

[12] Ghali A., Neville A.M., Structural Analysis: A unified classical and matrix approach, Chapman and Hall, London, 1990.

[13] EPFL-ENAC-SGC, Génie Parasismique, Quotient de Rayleigh (Rayleigh Quotient), Suisse, 2004 (in French).

[14] Xiang Y., Wei G.W., Exact Solutions for Vibration of Multi-span Rectangular Mindlin Plates, Journal of Vibration and Acoustics Vol.125, ASME, 2002.

[15] Whalen T.M., Course CE573: Structural Dynamics, Problem 3.10, Purdue University, Indiana, 2004.

[16] SCIA Group NV, ESA-Prima Win: Dynamics – Theoretical Background, Herk-de-Stad, 2002.

[17] U.S. Army Corps of Engineers, Engineering and Design - Seimic Design Provisions for Roller Compacted Concrete Dams, EP 1110-2-12, 1995.

[18] Wilson E., Three-Dimensional Static and Dynamic Analysis of Structures, CSI, Berkley, 2002.

[19] ASCE4-98, Seismic Analysis of Safety-Related Nuclear Structures and Commentary, 2000.

[20] Chowdhury I., Dasgupta S., Computation of Rayleigh Damping Coefficients for Large Systems, EJGE, 2003.

[21] NF P 06-013, PS92: Les Règles de la construction Parasismique (Rules for Seismic Structures), 1995 (in French).

[22] Vandepitte D., Berekening van Constructies (Calculation of Structures), Story-Scientia, Gent, 1979 (in Dutch). www.berekeningvanconstructies.be

[23] Transvik P., Alpsten G., Dynamic Behaviour under Wind Loading of a 90m Steel Chimney, Report S-01041, 2002.

[24] Czech Standard CSN 73 0035: Actions on Structures, 1986 (in Czech).

[25] Vandewalle J., Cools R., Lineaire Algebra en Analytische Meetkunde (Linear Algebra and Analytical Geometry), Garant, Leuven, 1998 (in Dutch).

[26] Toratti T., Seismic Design of Timber Structures, Tekes, 2001.

[27] Billah Y., Scanlan R., Resonance, Tacoma Narrows bridge failure, and undergraduate physics textbooks, Am. J. Phys. 59 (2), American Association of Physics Teachers, 1991.

[28] Lee I., AE514: Finite Element Methods in Structural Dynamics, Aero elasticity and Structural Dynamics Lab., Dept. of Aerospace Engineering, Kaist, 2001.

[29] Vladimír T., Reinhard M., Local magnitude, surface wave magnitude and seismic energy, Studia Geophysica et Geodaetica, 1991.

[30] Nederlands Normalisatie-instituut, Beoordeling van de constructieve veiligheid van een gebouw bij nieuwbouw, verbouw en afkeuren – Grondslagen voor aardbevinstbelastingen: geïnduceerde grondslagen.

CHAPTER 7 : ANNEX A : EARTHQUAKE MAGNITUDE

To assess the magnitude of earthquakes, a scale to describe the energy released during an earthquake was developed by Richter in the 1930s. This is named the Richter scale and it is the most common scale used today to describe earthquakes [26].

The magnitude of an earthquake on the Richter scale is determined by a so-called Wood-Anderson seismograph maximum amplitude, where M = log(a), and a is the maximum amplitude $[\mu m]$ at a 100 km distance from the epicentre.

The seismic action on buildings cannot be described by the Richter scale magnitude and this may not be used in the design. However, Housner in 1970 developed empirical relationships between the magnitude, the duration and the peak ground acceleration to be used in design:

Magnitude on the Richter scale	Peak ground acceleration (% g)	Duration (s)
5,0	9	2
5,5	15	6
6,0	22	12
6,5	29	18
7,0	37	24
7,5	45	30
8,0	50	34
8,5	50	37

CHAPTER 8 : ANNEX B : NUMERICAL DAMPING VALUES

In this annex, some numerical values for structural damping are given.

8.1 EC8 – Part 6

EC8 part 6 (ENV 1998-6:2003 Annex B) suggest the following values for the damping ratio:

Structural material	Damping ratio ξ
Steel elements	1% - 4%
Concrete elements	2% - 7%
Ceramic cladding	1,5% - 5%
Brickwork lining	3% - 10%

8.2 EC1 – Part 2-4

Other values for damping are suggested by EC1 – part 2-4 (ENV 1991-2-4: 1995 Annex C).
The fundamental logarithmic decrement d is given by:

$$d = d_s + d_a + d_d$$

Where:

- d_s: fundamental structural damping
- d_a: fundamental aero dynamical damping
- d_d: fundamental damping due to special devices

The structural damping is given by:

$$d_s = a_1 . n_1 + b_1$$

$$d_s \geq \delta_{min}$$

Where:

- η_1: fundamental flexural frequency.
- a_1, b_1, δ_{min}: parameters given in the following table for different structural types.

Structural type	a_1	b_1	δ_{min}
Reinforced concrete buildings	0,045	0,030	0,080
Steel buildings	0,045	0	0,050
Mixed structures : concrete + steel	0,080	0	0,080
Reinforced concrete towers	0,050	0	0,025

Lattice steel towers		0	0,030	0
Reinforced concrete chimneys		0,075	0	0,030
Prestressed steel cable		0	0,010	0
Unlined welded steel stacks		0	0,015	0
Steel stack with one liner or thermal insulation		0	0,025	0
Steel stack with two or more liners		0	0,030	0
Steel with brick liner		0	0,070	0
Coupled stacks without liner		0	0,015	0
Guyed steel stack without liner		0	0,040	0
Steel bridges	Welded	0	0,020	0
	High resistance bolts	0	0,030	0
	Ordinary bolts	0	0,050	0
Concrete bridges	Prestressed without cracks	0	0,040	0
	With cracks	0	0,100	0
Bridge cables	Parallel cables	0	0,006	0
	Spiral cables	0	0,020	0

For example, for a steel building with first frequency of 3Hz, the logarithmic decrement is:

$$0{,}045*3 + 0 = 0{,}135 \quad (> 0{,}05)$$

8.3 Reference [22]

Other values for the logarithmic decrement are suggested by the reference [22]:

Structural material	Logarithmic decrement
Steel (welded)	0,025
Reinforced or prestressed concrete	0,056
Brickwork	0,25
Wood	0,13

In this reference, preliminary formulas can also be found for aerodynamic damping and damping caused by the foundation.

CHAPTER 9 : ANNEX C : MANUAL CALCULATIONS SPECTRAL ANALYSIS

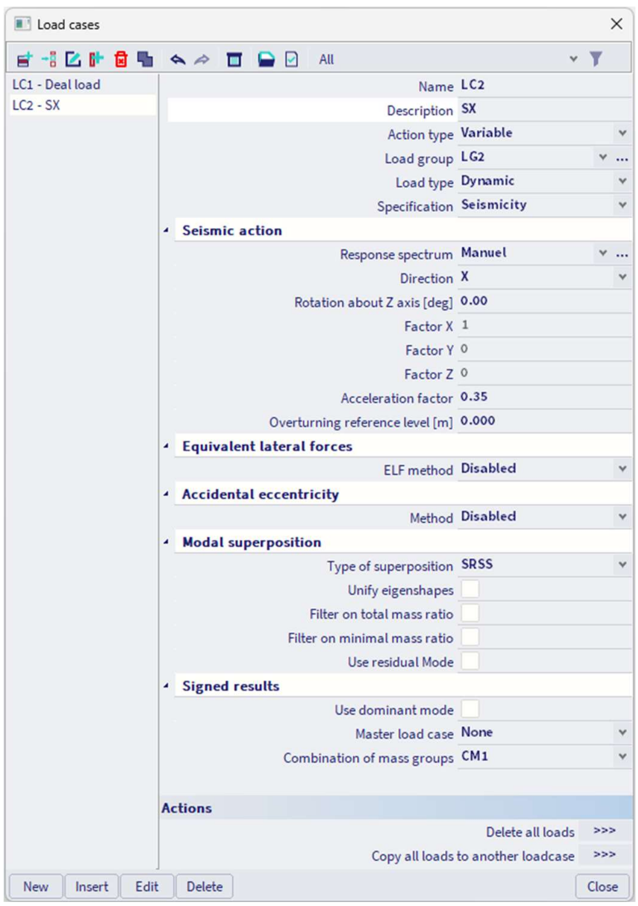
9.1 Spectral analysis of 3-2 example (example C-1)

In this paragraph, the seismic results of SCIA Engineer are calculated manually to give a clear understanding of the applied formulas. All formulas can be found in the paragraph “Calculation Protocol” of this chapter.

The reference project is not completely the same as the one described in example 3-2. The differences will be shown first before starting the manual calculation.

9.1.1 Seismic load case

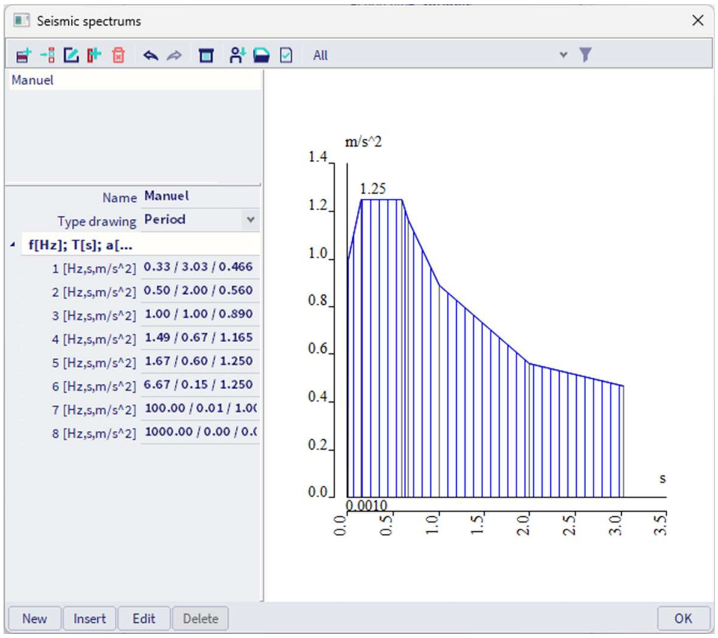
The properties which have been used in the seismic load case can be seen here:



A different acceleration factor has been used. This reduces the accelerations given by the spectrum.

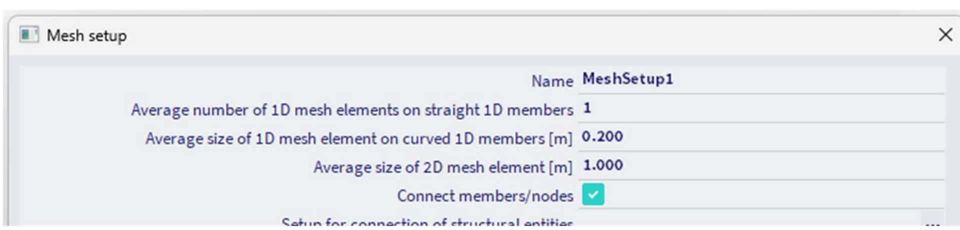
9.1.2 Spectrum

A manual seismic spectrum is used:

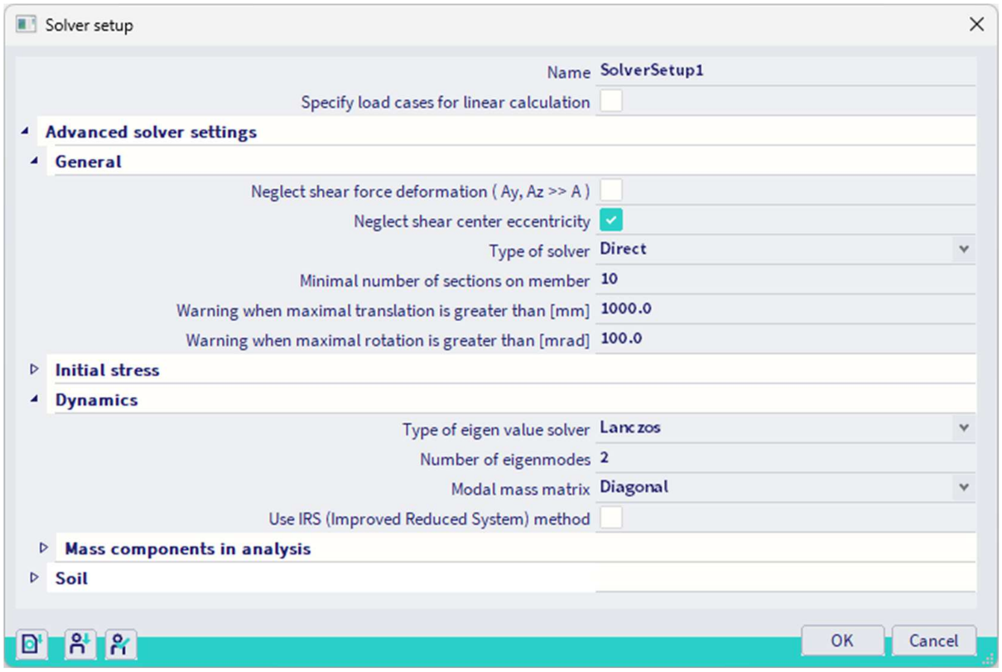


9.1.3 Finite element mesh and solver setup

The finite element mesh has not been refined:



The solver also has not been changed to neglect shear deformations.



9.2 Manual calculation of 3-2 example (example C-1)

9.2.1 Verification of modal participation factors

First, the **Modal Participation Factors** of the **Eigen Frequency Calculation Protocol** are verified.

As shown in the Deformation of Nodes, the normalized modal shapes for both modes were the following:

Displacement of nodes

Modal shapes are normalized, so that the generalized
Mass combination: CM1/1 - 0.53
Extreme: Node
Selection: All

Name	Case	Ux [mm]	Uz [mm]
N1	CM1/1 - 0.53	0.0	0.0
N2	CM1/1 - 0.53	-6.1	0.0
N3	CM1/1 - 0.53	-20.8	0.0
N4	CM1/1 - 0.53	-39.1	0.0

Displacement of nodes

Modal shapes are normalized, so that the generalized
Mass combination: CM1/2 - 3.43
Extreme: Node
Selection: All

Name	Case	Ux [mm]	Uz [mm]
N1	CM1/2 - 3.43	0.0	0.0
N2	CM1/2 - 3.43	25.7	0.0
N3	CM1/2 - 3.43	30.5	0.0
N4	CM1/2 - 3.43	-20.2	0.0

$$\{\varphi_1\} = \begin{Bmatrix} 0{,}039111 \\ 0{,}020803 \\ 0{,}006128 \\ 0 \end{Bmatrix} \qquad \{\varphi_2\} = \begin{Bmatrix} -0{,}020233 \\ 0{,}030451 \\ 0{,}025755 \\ 0 \end{Bmatrix}$$

Participation factor:

$$\gamma_{k,(j)} = \{\phi_k\}^T * \{m\}$$

$$\gamma_{x,(1)} = 0{,}039111 * 500 + 0{,}0200803 * 500 + 0{,}006128 * 500 = 33{,}021$$

$$\gamma_{x,(2)} = -0{,}020233 * 500 + 0{,}030451 * 500 + 0{,}025755 * 500 = 17{,}984$$

Effective mass:

$$M_{k,ef,(j)} = \gamma_{k,(j)}^2$$

$$M_{x,ef,(1)} = (33{,}021)^2 = 1090{,}39$$
$$M_{x,ef,(2)} = (17{,}984)^2 = 323{,}42$$

Participation mass ratio:

$$L_{k,(j)} = \frac{M_{k,ef,(j)}}{M_{k,tot}}$$

$$L_{x,(1)} = \frac{1090{,}39}{500 + 500 + 500} = 0{,}7269$$

$$L_{k,(j)} = \frac{323{,}42}{500 + 500 + 500} = 0{,}2156$$

These results correspond to the results obtained by SCIA Engineer.
They can be found in SCIA Engineer in the Calculation protocol (Eigen frequency):

Relative modal masses

Mode	ıega [rad,	Period [s]	Freq. [Hz]	Γ_{xi}	Γ_{yi}	Γ_{zi}	W_{xi}/W_{xtot}	W_{yi}/W_{ytot}	W_{zi}/W_{ztot}	W_{xi_R}/W_{xtot}	W_{yi_R}/W_{ytot}	W_{zi_R}/W_{ztot}
1	3.30085	1.90	0.53	-33.0211	0.0000	0.0000	0.7269	0.0000	0.0000	0.0000	0.2720	0.0000
2	21.5274	0.29	3.43	17.9865	0.0000	0.0000	0.2157	0.0000	0.0000	0.0000	0.5286	0.0000
							0.9426	0.0000	0.0000	0.0000	0.8006	0.0000

9.2.2 Details of the seismic calculation

Next, the details of the seismic calculation found in the Calculation Protocol for the Linear Calculation are verified:

Mode	Freq. [Hz]	Damp ratio	Damp coef.	Wi/Wtot [-]	Sax [m/s²]	Say [m/s²]	Saz [m/s²]	G(j) [-]	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.53	0.05	1	0.73	0.207	0.000	0.000	-0.63	0.23	0.00	0.00	-2.26
2	3.43	0.05	1	0.22	0.438	0.000	0.000	0.02	0.14	0.00	0.00	-0.41
Level=	0.00			0.94					0.27	0.00	0.00	2.29

The spectral acceleration S_{ax} for both modes is calculated using the defined seismic spectrum.

The spectrum for ground type B with a behaviour factor q = 2 gives the following values for $S_d(T)/\alpha$:

	Frequency[Hz]	Period[s]	Acceleration[m/s^2]
1	0.33	3.03	0.47
2	0.50	2.00	0.56
3	1.00	1.00	0.89
4	1.49	0.67	1.17
5	1.67	0.60	1.25
6	6.67	0.15	1.25
7	100.00	0.01	1.00
8	1000.00	0.00	0.00
*	0.00	0.00	0.00

The first mode has a period T_1 of 1,9036 s => $S_d(T_1)/\alpha$ = 0,5918 m/s²
The second mode has a period T_2 of 0,2920 s => $S_d(T_2)/\alpha$ = 1,25 m/s²

In this example, the coefficient of acceleration α was 0,35

=> $S_{ax,(1)}$ = 0,5918m/s² * 0,35 = 0,2071 m/s²
=> $S_{ax,(2)}$ = 1,25m/s² * 0,35 = 0,4375 m/s²

These results correspond to the results obtained by SCIA Engineer. The small deviation is due to the fact that SCIA Engineer uses more decimals. In the further analysis, the spectral accelerations of SCIA Engineer are used.

Mode coefficient:

$$G_{k,(j)} = \frac{S_{a,k,(j)} * \gamma_{k,(j)}}{\omega_{(j)}^2}$$

$$G_{x,(1)} = \frac{0,2019 * 33,021}{(3,3007)^2} = 0,6119$$

$$G_{x,(2)} = \frac{0,4380 * 17,984}{(21,5192)^2} = 0,0170$$

These results correspond to the results obtained by SCIA Engineer.

The necessary intermediate results are calculated so the response of each mode can now be obtained.
First, for each mode, the lateral force in each node can be calculated. These lateral forces can then be used to calculate the base shear and overturning moment.

Mode 1:

Lateral force in node i:

$$F_{i,k,(j)} = m_{i,k,(j)} * S_{a,k,(j)} * \gamma_{k,(j)} * \phi_{i,k,(j)}$$

$$F_{4,x,(1)} = 500\text{kg} * \frac{0{,}2019\text{m}}{\text{s}^2} * 33{,}021 * 0{,}039111 = 130{,}38\text{N}$$
$$F_{3,x,(1)} = 500\text{kg} * \frac{0{,}2019\text{m}}{\text{s}^2} * 33{,}021 * 0{,}020803 = 69{,}35\text{N}$$
$$F_{2,x,(1)} = 500\text{kg} * \frac{0{,}2019\text{m}}{\text{s}^2} * 33{,}021 * 0{,}006128 = 20{,}43\text{N}$$
$$F_{1,x,(1)} = 0\text{N}$$

Base shear force:

$$F_{k,(j)} = \sum_i F_{i,k,(j)}$$

$$F_{x,(1)} = 130{,}38\text{N} + 69{,}35\text{N} + 20{,}43\text{N} = 220{,}129\text{N} = 0{,}2201\text{kN}$$

Overturning moment in node i:

$$M_{i,k,(j)} = F_{i,k,(j)} * z_i$$

$$M_{4,y,(1)} = -130{,}38\text{N} * 12\text{m} = -1564{,}50\text{Nm}$$
$$M_{3,y,(1)} = -69{,}35\text{N} * 8\text{m} = -554{,}77\text{Nm}$$
$$M_{2,y,(1)} = -20{,}43\text{N} * 4\text{m} = -81{,}71\text{Nm}$$
$$M_{1,y,(1)} = 0\text{N.m}$$

Overturning moment:

$$M_{k,(j)} = \sum_i M_{i,k,(j)}$$

$$M_{y,(1)} = -1564{,}50\text{Nm} - 554{,}77\text{Nm} - 81{,}71\text{Nm} = -2200{,}89\text{Nm} = -2{,}2009\text{kNm}$$

NB:

In this mode, all lateral forces in the nodes are oriented the same way. The lateral loads in the nodes are in this case oriented in the negative X-direction so the Base Shear Force is oriented in the positive X-direction. The lateral loads in the nodes thus produce a negative Overturning Moment around the Y-axis. An example of this principle can be found in reference [26].

However, as stated in the previous chapters, the signs ***have no absolute importance*** *since vibration amplitudes always occur on both sides of the equilibrium position.*

Mode 2:

Lateral force in node i:

$$F_{i,k,(j)} = m_{i,k,(j)} * S_{a,k,(j)} * \gamma_{k,(j)} * \phi_{i,k,(j)}$$

$$F_{4,x,(2)} = 500\text{kg} * \frac{0{,}4380\text{m}}{\text{s}^2} * 17{,}984 * -0{,}020233 = -79{,}69\text{N}$$
$$F_{3,x,(2)} = 500\text{kg} * \frac{0{,}4380\text{m}}{\text{s}^2} * 17{,}984 * 0{,}030451 = 119{,}93\text{N}$$
$$F_{2,x,(2)} = 500\text{kg} * \frac{0{,}4380\text{m}}{\text{s}^2} * 17{,}984 * 0{,}025755 = 101{,}44\text{N}$$
$$F_{1,x,(2)} = 0\text{N}$$

Base shear force:

$$F_{k,(j)} = \sum_i F_{i,k,(j)}$$

$$F_{x,(2)} = -79{,}69\text{N} + 119{,}93\text{N} + 101{,}44\text{N} = 141{,}68\text{N} = 0{,}1417\text{kN}$$

Overturning moment in node i:

$$M_{i,k,(j)} = F_{i,k,(j)} * z_i$$

$$M_{4,y,(2)} = -79{,}69\text{N} * 12\text{m} = -956{,}25\text{Nm}$$
$$M_{3,y,(2)} = -119{,}93\text{N} * 8\text{m} = -959{,}45\text{Nm}$$
$$M_{2,y,(2)} = -101{,}44\text{N} * 4\text{m} = -405{,}74\text{Nm}$$
$$M_{1,y,(2)} = 0\text{N.m}$$

Overturning moment:

$$M_{k,(j)} = \sum_i M_{i,k,(j)}$$

$$M_{y,(2)} = 956{,}25\text{Nm} - 959{,}45\text{Nm} - 405{,}74\text{Nm} = -408{,}94\text{Nm} = -0{,}4089\text{kNm}$$

To obtain the global response, the modal responses need to be combined. In this example the **SRSS-method** was used:

$$F_x = \sqrt{\left(F_{x,(1)}\right)^2 + \left(F_{x,(2)}\right)^2} = \sqrt{(0{,}2201\text{kN})^2 + (0{,}1417\text{kN})^2} = 0{,}2618\text{kN}$$

$$M_y = \sqrt{\left(M_{y,(1)}\right)^2 + \left(M_{y,(2)}\right)^2} = \sqrt{(-2{,}2009\text{kN})^2 + (-0{,}4089\text{kN})^2} = 2{,}238\text{kN.m}$$

These results correspond almost exactly to the results obtained by SCIA Engineer. We will show them again:

Mode	Freq. [Hz]	Damp ratio	Damp coef.	Wi/Wtot [-]	Sax [m/s²]	Say [m/s²]	Saz [m/s²]	G(j) [-]	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.53	0.05	1	0.73	0.207	0.000	0.000	-0.63	0.23	0.00	0.00	-2.26
2	3.43	0.05	1	0.22	0.438	0.000	0.000	0.02	0.14	0.00	0.00	-0.41
Level=	0.00			0.94					0.27	0.00	0.00	2.29

As specified in the theory, these same principles can now be used to calculate the displacements and accelerations for each mode. These modal responses can then be combined again to obtain the global displacements and accelerations of the structure.

Mode 1:

Displacement in node i:

$$u_{i,k,(j)} = G_{k,(j)}.\phi_{i,k,(j)}$$

$$u_{4,x,(1)} = 0{,}6119 * 0{,}039111 = 0{,}02393\text{m} = 23{,}93\text{mm}$$
$$u_{3,x,(1)} = 0{,}6119 * 0{,}020803 = 0{,}01273\text{m} = 12{,}73\text{mm}$$
$$u_{2,x,(1)} = 0{,}6119 * 0{,}006128 = 0{,}00375\text{m} = 3{,}75\text{mm}$$
$$u_{1,x,(1)} = 0\text{mm}$$

Acceleration in node i:

$$\ddot{u}_{i,k,(j)} = \omega_{(j)}^2.G_{k,(j)}.\phi_{i,k,(j)}$$

$$a_{4,x,(1)} = 3{,}3007^2 * 0{,}6119 * 0{,}039111 = 0{,}26073\text{m/s}^2 = 260{,}73\text{mm/s}^2$$
$$a_{3,x,(1)} = 3{,}3007^2 * 0{,}6119 * 0{,}020803 = 0{,}13868\text{m/s}^2 = 138{,}68\text{mm/s}^2$$
$$a_{2,x,(1)} = 3{,}3007^2 * 0{,}6119 * 0{,}006128 = 0{,}04085\text{m/s}^2 = 40{,}85\text{mm/s}^2$$
$$a_{1,x,(1)} = 0\text{mm/s}^2$$

Mode 2:

Displacement in node i:

$$u_{i,k,(j)} = G_{k,(j)}.\phi_{i,k,(j)}$$

$$u_{4,x,(2)} = 0{,}0170 * (-0{,}020233) = -0{,}00034\text{m} = -0{,}34\text{mm}$$
$$u_{3,x,(2)} = 0{,}0170 * 0{,}030451 = 0{,}00052\text{m} = 0{,}52\text{mm}$$
$$u_{2,x,(2)} = 0{,}0170 * 0{,}025755 = 0{,}00044\text{m} = 0{,}44\text{mm}$$
$$u_{1,x,(2)} = 0\text{mm}$$

Acceleration in node i:

$$\ddot{u}_{i,k,(j)} = \omega_{(j)}^2.G_{k,(j)}.\phi_{i,k,(j)}$$

$$a_{4,x,(2)} = 21{,}5192^2 * 0{,}0170 * (-0{,}020233) = -\,0{,}15928\text{m/s}^2 = -159{,}28\text{mm/s}^2$$
$$a_{3,x,(2)} = 21{,}5192^2 * 0{,}0170 * 0{,}030451 = 0{,}23972\text{m/s}^2 = 239{,}72\text{mm/s}^2$$
$$a_{2,x,(2)} = 21{,}5192^2 * 0{,}0170 * 0{,}025755 = 0{,}20275\text{m/s}^2 = 202{,}75\text{mm/s}^2$$
$$a_{1,x,(2)} = 0\text{mm/s}^2$$

To obtain the global response, the modal responses need to be combined. In this example the SRSS-method was used.

Displacements:

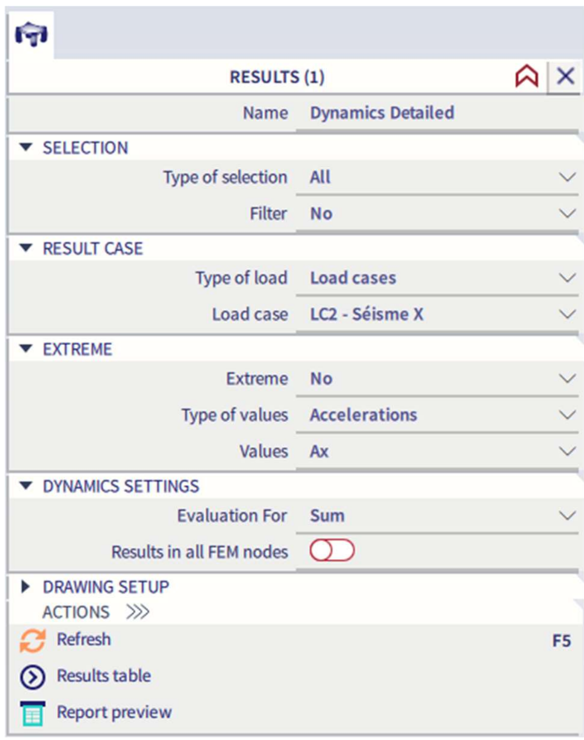
$$u_{4,x} = \sqrt{\left(u_{4,x,(1)}\right)^2 + \left(u_{4,x,(2)}\right)^2} = \sqrt{(23{,}93)^2 + (-0{,}34)^2} = 23{,}93\text{mm}$$
$$u_{3,x} = \sqrt{\left(u_{3,x,(1)}\right)^2 + \left(u_{3,x,(2)}\right)^2} = \sqrt{(12{,}73)^2 + (0{,}52)^2} = 12{,}74\text{mm}$$
$$u_{2,x} = \sqrt{\left(u_{2,x,(1)}\right)^2 + \left(u_{2,x,(2)}\right)^2} = \sqrt{(3{,}75)^2 + (0{,}44)^2} = 3{,}78\text{mm}$$
$$u_{1,x} = 0$$

Accelerations:

$$a_{4,x} = \sqrt{\left(a_{4,x,(1)}\right)^2 + \left(a_{4,x,(2)}\right)^2} = \sqrt{(260{,}73)^2 + (-159{,}28)^2} = 305{,}53\text{mm/s}^2$$
$$a_{3,x} = \sqrt{\left(a_{3,x,(1)}\right)^2 + \left(a_{3,x,(2)}\right)^2} = \sqrt{(138{,}68)^2 + (239{,}72)^2} = 276{,}94\text{mm/s}^2$$
$$a_{2,x} = \sqrt{\left(a_{2,x,(1)}\right)^2 + \left(a_{2,x,(2)}\right)^2} = \sqrt{(40{,}85)^2 + (202{,}75)^2} = 206{,}82\text{mm/s}^2$$
$$a_{1,x} = 0\text{mm/s}^2$$

The main menu Results / Dynamics / Seismic Detailed, or the below icon, was designed to view these modal displacements and accelerations.

In the Properties Window, the options for viewing the modal results can be set:



- In the field « Load Cases », a seismic load case or a mass combination can be selected.

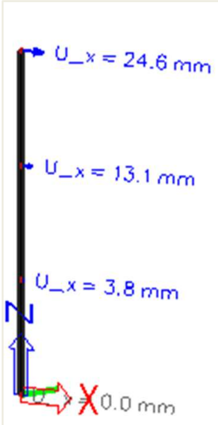
- The filed « Modal results » allow choosing between the displacements or accelerations.

- « Evaluation for » is used to specify which results need to be shown: the results for a specific Eigenmode, the results for All Eigenmodes or the global, Summarized results.

The results for each mode and the summarized results are shown on the next pages for both the displacement and the accelerations.

Displacements:

Mode 1:

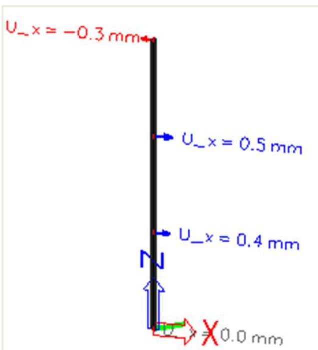


Dynamics Detailed

Linear calculation
Load case: LC2
Evaluation For: CM1/1 - 0.53
Selection: All
Displacements

Name	Case	U_x [mm]	U_z [mm]	Φ_y [mrad]	U_{total} [mm]
N1	LC2	0.0	0.0	0.0	0.0
N2	LC2	3.8	0.0	1.8	3.8
N3	LC2	13.1	0.0	2.7	13.1
N4	LC2	24.6	0.0	3.0	24.6

Mode 2:

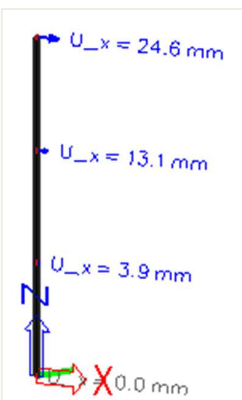


Dynamics Detailed

Linear calculation
Load case: LC2
Evaluation For: CM1/2 - 3.43
Selection: All
Displacements

Name	Case	U_x [mm]	U_z [mm]	Φ_y [mrad]	U_{total} [mm]
N1	LC2	0.0	0.0	0.0	0.0
N2	LC2	0.4	0.0	0.1	0.4
N3	LC2	0.5	0.0	-0.1	0.5
N4	LC2	-0.3	0.0	-0.3	0.3

Summarized:



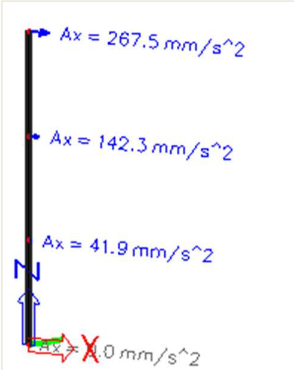
Dynamics Detailed

Linear calculation
Load case: LC2
Evaluation For: Sum
Selection: All
Displacements

Name	Case	U_x [mm]	U_z [mm]	Φ_y [mrad]	U_{total} [mm]
N1	LC2	0.0	0.0	0.0	0.0
N2	LC2	3.9	0.0	1.8	3.9
N3	LC2	13.1	0.0	2.7	13.1
N4	LC2	24.6	0.0	3.0	24.6

Accelerations:

Mode 1:

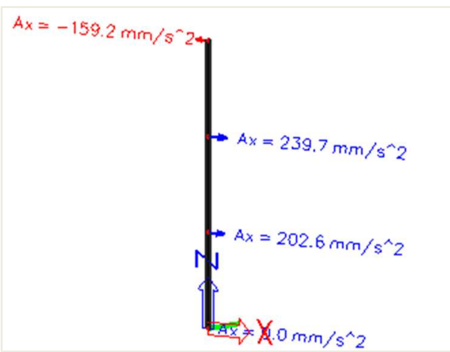


Dynamics Detailed

Linear calculation
Load case: LC2
Evaluation For: CM1/1 - 0.53
Selection: All
Accelerations

Name	Case	Ax [mm/s²]	Az [mm/s²]	α_y [mrad/s²]	A_{total} [mm/s²]
N1	LC2	0.0	0.0	0.0	0.0
N2	LC2	41.9	0.0	19.3	41.9
N3	LC2	142.3	0.0	29.4	142.3
N4	LC2	267.5	0.0	32.2	267.5

Mode 2:

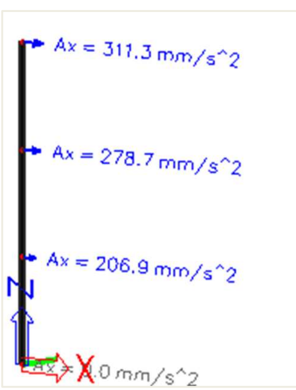


Dynamics Detailed

Linear calculation
Load case: LC2
Evaluation For: CM1/2 - 3.43
Selection: All
Accelerations

Name	Case	Ax [mm/s²]	Az [mm/s²]	α_y [mrad/s²]	A_{total} [mm/s²]
N1	LC2	0.0	0.0	0.0	0.0
N2	LC2	202.6	0.0	57.0	202.6
N3	LC2	239.7	0.0	-51.1	239.7
N4	LC2	-159.2	0.0	-123.4	159.2

Summarized:



Dynamics Detailed

Linear calculation
Load case: LC2
Evaluation For: Sum
Selection: All
Accelerations

Name	Case	Ax [mm/s²]	Az [mm/s²]	α_y [mrad/s²]	A_{total} [mm/s²]
N1	LC2	0.0	0.0	0.0	0.0
N2	LC2	206.9	0.0	60.2	206.9
N3	LC2	278.7	0.0	58.9	278.7
N4	LC2	311.3	0.0	127.6	311.3

When comparing the results of the manual calculation and those obtained by SCIA Engineer, it is clear that both calculations correspond.

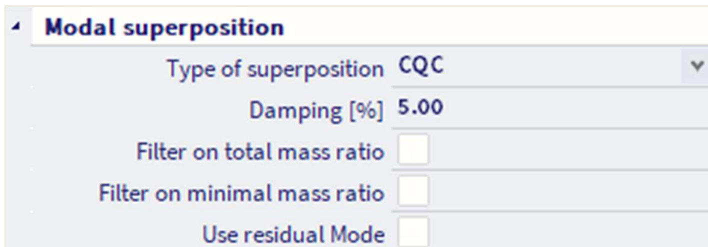
As specified in the theory, when using the CQC-method, a damping spectrum needs to be defined. To illustrate this, the above example is calculated again, but now using the CQC-method for the modal combination.

CHAPTER 10 : ANNEX D : MISSING MASS IN NODES

As mentioned before, the sum of the effective modal masses for the modes taken into account must amount to at least 90% (EN 1998-1-1 art.4.3.3.3). The user can try to achieve this with the following possibilities:

- Take more natural frequencies into account
- Assign mass more to nodes/connection instead of beams (to avoid local eigenmodes).

The mass which has not been taken into account (for example, if the effective modal mass is 90%, then there is 10% not taken into account), can be treated in 3 possible different ways in SCIA Engineer:



The used method is set in each seismic load case and is again displayed in the linear calculation protocol. Let's take as example that the effective modal mass in a direction is 90%. Then how can the other 10% be treated?

- If the option « Use residual mode » is not ticked: in this case, the 10% would be ignored. We would only take into account 90% of the mass of the structure to calculate the effects of an earthquake.
- If the option « Use residual mode » is ticked: in this case, a 'fictive' mode corresponding to the combination of all missing modes can be calculated. But since these missing modes are over different natural frequencies, the last found frequency will also be the natural frequency of this mode. In the calculation, the forces in this mode will be calculated in the same way as in the other modes.

In the following examples the differences are explained in detail.

In these projects the following general principle is used:
First of all, a seismic spectrum is introduced. For this spectrum the modal displacements are calculated for each mode, in this case there are 2 modes. Afterwards, the displacements are transformed in real load cases. For these 2 load cases the results of the internal forces and reactions can be asked. According to the specific analysis method, the results are summed. On that way, one can compare these results with the output of the internal forces of the seismic load case. This will be done with the following three types of 'mass in analysis'.

10.1 Spectral analysis example without « residual mode »

Example D-1.esa: spectral analysis without residual mode

If the option 'Residual mass' is not ticked, the standard calculation is used. In this case, the participation mass from all modes is taken into account and the user has to consider the 90% rule of the Eurocode. In other words, using this method it's important that the total amount of the masses in X, Y and Z are sufficient.

In the example, a structure (3mx6m), made of beams and columns with rectangular cross-sections (beams cross-section 15*30 except B4 which is 20*60; columns cross-section 15x15 except B3 which is 20*60), is subjected to dynamic forces. The members are manufactured in C25/30 according to EC-EN. The height of each column is 5m.

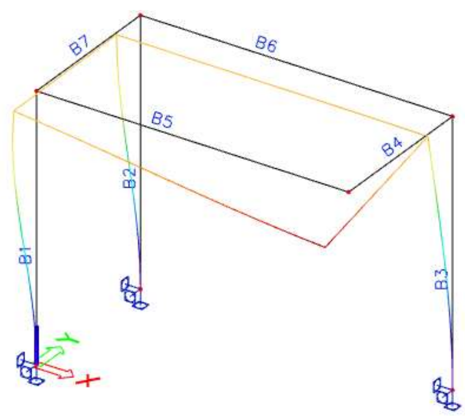
Next, a seismic load case is introduced. The seismic spectrum acts in 3 directions. An acceleration of 2 m/s² is given in function of the frequency.
The evaluation method SRSS is used together without the option 'Residual mass'.

The eigen frequency analysis gives the following output:

Eigen frequencies

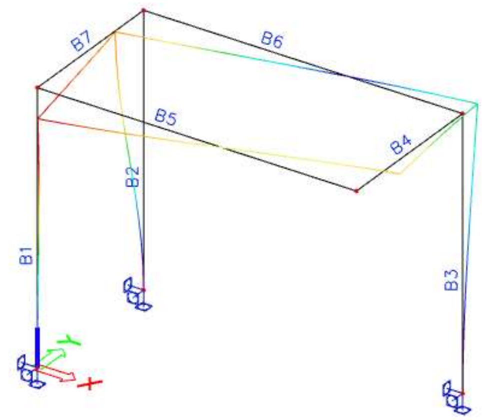
N	f [Hz]	ω [1/s]	ω² [1/s²]	T [s]
Mass combination : CM1				
1	2.05	12.90	166.40	0.49
2	2.39	15.03	225.81	0.42

Deformation for mass combination CM1/1-1,64:



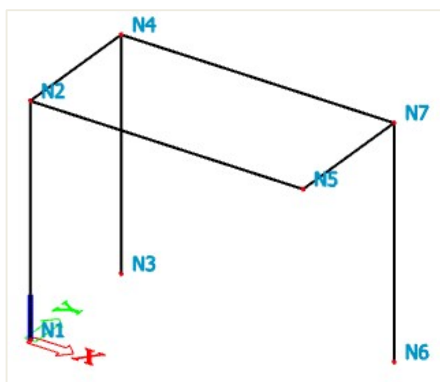
Name	Case	Ux [mm]	Uy [mm]	Uz [mm]	Φx [mrad]	Φy [mrad]	Φz [mrad]	Utotal [mm]
N1	CM1/1 - 2.05	0.00	0.00	0.00	0.00	0.00	0.00	0.00
N2	CM1/1 - 2.05	1.27	-15.15	-0.04	0.34	2.02	0.07	15.21
N3	CM1/1 - 2.05	0.00	0.00	0.00	0.00	0.00	0.00	0.00
N4	CM1/1 - 2.05	0.78	-15.15	0.04	0.34	0.15	0.08	15.17
N5	CM1/1 - 2.05	1.27	-15.64	-12.89	4.27	1.35	0.15	20.31
N6	CM1/1 - 2.05	0.00	0.00	0.00	0.00	0.00	0.00	0.00
N7	CM1/1 - 2.05	0.78	-15.64	0.00	4.32	0.27	0.12	15.65

Deformation for mass combination CM1/2-1,90:



Name	Case	Ux [mm]	Uy [mm]	Uz [mm]	Φx [mrad]	Φy [mrad]	Φz [mrad]	Utotal [mm]
N1	CM1/2 - 2.39	0.00	0.00	0.00	0.00	0.00	0.00	0.00
N2	CM1/2 - 2.39	13.37	-19.69	-0.01	0.24	-0.61	4.27	23.80
N3	CM1/2 - 2.39	0.00	0.00	0.00	0.00	0.00	0.00	0.00
N4	CM1/2 - 2.39	0.43	-19.69	0.02	0.24	-0.08	4.55	19.69
N5	CM1/2 - 2.39	13.37	9.32	7.12	-2.35	-0.80	4.73	17.78
N6	CM1/2 - 2.39	0.00	0.00	0.00	0.00	0.00	0.00	0.00
N7	CM1/2 - 2.39	0.43	9.32	0.00	-2.40	0.07	3.55	9.33

The masses of the participating nodes (N2, N4, N5 and N7) are needed. The mass is attributed to the end nodes of each member.



Calculation of mass X for N2:

$$\text{Mass X} = 2500\text{kg/m}^3 * [(2{,}5 * 0{,}15 * 0{,}15) + (3 * 0{,}3 * 0{,}15) + (1{,}5 * 0{,}3 * 0{,}15)]$$
$$= 646{,}875 \text{ kg}$$

The total mass matrix is:

Node	Mass x (kg)	Mass y (kg)	Mass z (kg)
N2	646,875	646,875	646,875
N4	646,875	646,875	646,875
N5	787,5	787,5	787,5
N7	1537,5	1537,5	1537,5
Total	3618,75	3618,75	3618,75

The modal participation factor is calculated as:

$$\gamma_{k(j)} = \{\phi_k\}^T | \gamma_{k(j)} = \{\phi_k\}^T \{m\}$$

Calculation of γ_x for mode 1:

$$\{\phi_{x,(1)}\} = \begin{Bmatrix} 0 \\ -0{,}001267 \\ 0 \\ -0{,}000778 \\ -0{,}001265 \\ 0 \\ -0{,}000777 \end{Bmatrix} \quad \text{and} \quad \{m\} = \begin{Bmatrix} 0 \\ 646{,}875 \\ 0 \\ 646{,}875 \\ 787{,}5 \\ 0 \\ 1537{,}5 \end{Bmatrix}$$

So:

$$\gamma_{x,(1)} = -0{,}001267 * 647 - 0{,}000778 * 647 - 0{,}001265 * 788 - 0{,}000777 * 1538$$
$$\gamma_{x,(1)} = -3{,}514$$

The participation factor matrix is:

(j) Units	γ_x ($kg^{1/2}$)	γ_y ($kg^{1/2}$)	γ_z ($kg^{1/2}$)
1	-3,514	55,959	10,158
2	20,115	-3,812	5,614

Out of this matrix the **effective masses** can be calculated:

$$M_{ef,k,(j)} = \gamma^2_{k,(j)}$$

Calculation of M_{ef} for mode 1 in direction x:

$$M_{ef,x,(1)} = -3{,}514^2 = 12{,}346$$

(j) Units	$M_{ef,x}$ (kg)	$M_{ef,y}$ (kg)	$M_{ef,z}$ (kg)
1	12,346	3131,374	103,182
2	404,603	14,533	31,517

The formula for the **participation mass ratio** is as follows:

$$L_{k,(j)} = \frac{M_{ef,k,(j)}}{M_{tot,k}}$$

$$L_{k,(1)} = \frac{12{,}346}{3618{,}75} = 0{,}0034$$

(j) Units	L_x (-)	L_y (-)	L_z (-)
1	0,0034	0,8653	0,0285
2	0,1118	0,0040	0,0087

The **acceleration response spectrum S** has the constant value of 2m/s² :

(j) Units	S_x (m/s²)	S_y (m/s²)	S_z (m/s²)
1	2	2	2
2	2	2	2

Calculation of **mode coefficient** in each direction:

$$G_{k,(j)} = \frac{S_{a,k,(j)} * \gamma_{k,(j)}}{\omega^2_{(j)}}$$

For example for direction x and mode 1:

$$\mathrm{G}_{\mathrm{x},(1)} = \frac{2 * -3{,}514}{166{,}4} = -0{,}042$$

(j) Units	G_x (m*kg$^{1/2}$)	G_y (m*kg$^{1/2}$)	G_z (m*kg$^{1/2}$)	G (m*kg$^{1/2}$)
1	-0,042	0,673	0,122	0,7524
2	0,178	-0,034	0,050	0,1941

Now, the lateral forces can be calculated in each node:

$$\mathrm{F}_{\mathrm{i,k,(j)}} = \mathrm{m}_{\mathrm{i,k,(j)}} * \ddot{\mathrm{u}}_{\mathrm{i,k,(j)}} = \mathrm{m}_{\mathrm{i,k,(j)}} * \mathrm{G}_{(\mathrm{j})} * \phi_{\mathrm{i,k,(j)}} * \omega_{(j)}^2$$

As example, this is calculated for node 2 in direction X:

$$\mathrm{F}_{\mathrm{N2,x,(1)}} = 646{,}875 * 0{,}7524 * (-0{,}001267) * 166{,}4 = -102{,}6\mathrm{N}$$

Mode1

Node	F_x (1) (N)	F_y (1) (N)	F_z (1) (N)
N2	-102,6	1227,3	3,4
N4	-63,0	1227,3	-2,9
N5	-124,7	1541,8	1271,1
N7	-149,6	3010,0	0,2
Total	-439,9	7006,3	1271,8

Mode2

Node	F_x (2) (N)	F_y (2) (N)	F_z (2) (N)
N2	379,1	-558,3	0,3
N4	12,2	-558,3	0,6
N5	461,4	321,6	245,8
N7	29,0	627,8	-0,1
Total	881,7	-167,1	246,1

The shear forces in direction X, Y and Z:

$$\mathrm{F}_{\mathrm{k,(j)}} = \sum_i F_{i,k,(j)} l$$

For mode 1 in direction x:

$$\mathrm{F}_{\mathrm{x},(1)} = \frac{-439{,}9}{1000} = -0{,}4399\mathrm{kN}$$

(j) Units	F_x (kN)	F_y (kN)	F_z (kN)
1	-0,4399	7,0063	1,2718
2	0,8817	-0,1671	0,2461
Total	0,99	7,01	1,30

The overturning moment in each node for each direction is:

$$\mathrm{M}_{\mathrm{i,k,(j)}} = \mathrm{F}_{\mathrm{i,k,(j)}} * \mathrm{z_i}$$

$$\mathrm{M}_{\mathrm{N2,x,(1)}} = \mathrm{F}_{\mathrm{N2,y,(1)}} * (\text{height} - \text{overturning height})$$
$$\mathrm{M}_{\mathrm{N2,x,(1)}} = 1227{,}3\mathrm{N} * (5\mathrm{m} - 0\mathrm{m})$$
$$\mathrm{M}_{\mathrm{N2,x,(1)}} = -6136{,}4\mathrm{N.m}$$

The other values are:

Mode1

Node	M_x (1) (N.m)	M_y (1) (N.m)
N2	-6136,4	513,1
N4	-6136,4	315,1
N5	-7709,0	623,6
N7	-15049,9	747,9

Mode2

Node	M_x (2) (N.m)	M_y (2) (N.m)
N2	2791,4	-1895,4
N4	2791,4	-60,8
N5	-1608,1	-2307,1
N7	-3139,2	-145,2

The sum of the moments for each node gives the **overturning moment in base**:

(j) Units	$\mathrm{M_x}$ (kN)	$\mathrm{M_y}$ (kN)
1	-35,0317	2,1997
2	0,8355	-4,4085
Total	35,04	4,93

The moments for each separate mode are combined with the SRSS-method.

Calculation of the **modal displacement**:

$$\mathrm{u_{i,k,(j)}} = \mathrm{G_{(j)}} * \phi_{\mathrm{k,(j)}}$$

For instance for node 2 in direction X and first mode:

$$\{\mathrm{G_{(1)}}\} = \{0{,}7524\} \qquad \text{and} \qquad \{\phi_{\mathrm{N2,x,(1)}}\} = \{-0{,}001267\}$$

So:

$$\mathrm{u_{N2,x,(1)}} = (0{,}7524 * -0{,}001267) * 1000 = -0{,}95\mathrm{mm}$$

Other values are:

Mode1

Node	u_x (mm)	u_y (mm)	u_z (mm)
N2	-0,95	11,40	0,03
N4	-0,59	11,40	-0,03
N5	-0,95	11,77	9,70
N7	-0,58	11,77	0,00

Mode2

Node	u_x (mm)	u_y (mm)	u_z (mm)
N2	2,60	-3,82	0,00
N4	0,08	-3,82	0,00
N5	2,59	1,81	1,38
N7	0,08	1,81	0,00

Total

Node	u_x (mm)	u_y (mm)	u_z (mm)
N2	2,76	12,03	0,03

N4	0,59	12,03	0,03
N5	2,76	11,90	9,80
N7	0,59	11,90	0,00

Calculation of the **modal acceleration**:

$$\ddot{u}_{i,k,(j)} = \omega_{(j)}^{2} * G_{(j)} * \phi_{k,(j)}$$

For instance for node 2 in direction X and first mode:

$$\ddot{u}_{N2,x,(1)} = -0{,}95 * 166{,}4 = -158{,}6\text{mm/s}^2$$

Mode1

Node	a_x (mm/s²)	a_y (mm/s²)	a_z (mm/s²)
N2	-158,6	1897,2	5,3
N4	-97,4	1897,2	-4,5
N5	-158,4	1957,8	1645,2
N7	-97,3	1957,7	0,1

Mode2

Node	a_x (mm/s²)	a_y (mm/s²)	a_z (mm/s²)
N2	586,0	-863,0	-0,4
N4	18,8	-863,0	1,0
N5	585,9	408,4	312,1
N7	18,9	408,4	0,0

Total

Node	a_x (mm/s²)	a_y (mm/s²)	a_z (mm/s²)
N2	607,1	2084,3	5,3
N4	99,2	2084,3	4,6
N5	606,9	2000,0	1644,1
N7	99,1	1999,9	0,1

Next, the displacements are inputted on the structure by means of a load case:

INPUT TABLE   Loads   Type here

	Name	Direction	Reference	Value ...	Suppo...	Load case
1	TRS1	X	Relative	2.60	Sn2	LC4
2	TRS2	X	Relative	2.60	Sn5	LC4
3	TRS3	X	Relative	0.08	Sn4	LC4
4	TRS4	X	Relative	0.08	Sn7	LC4
5	TRS5	Y	Relative	-3.82	Sn2	LC4
6	TRS6	Y	Relative	-3.82	Sn4	LC4
7	TRS7	Y	Relative	1.81	Sn5	LC4
8	TRS8	Y	Relative	1.81	Sn7	LC4
9	TRS9	Z	Relative	1.38	Sn5	LC4
10	TRS10	X	Relative	-0.95	Sn2	LC3
11	TRS11	X	Relative	-0.95	Sn5	LC3
12	TRS12	X	Relative	-0.59	Sn4	LC3
13	TRS13	X	Relative	-0.59	Sn7	LC3
14	TRS14	Y	Relative	11.40	Sn2	LC3
15	TRS15	Y	Relative	11.40	Sn4	LC3
16	TRS16	Y	Relative	11.77	Sn5	LC3
17	TRS17	Y	Relative	11.77	Sn7	LC3
18	TRS18	Z	Relative	9.70	Sn5	LC3
*						

For these load cases the following internal forces are computed:

Name	dx [m]	Case	N [kN]	V_y [kN]	V_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
B1	0.000	LC3	4.38	-1.35	0.36	0.01	-0.49	3.44
B1	5.000	LC3	4.38	-1.35	0.36	0.01	1.29	-3.30
B2	0.000	LC3	-3.73	-1.35	-0.04	0.01	0.12	3.44
B2	5.000	LC3	-3.73	-1.35	-0.04	0.01	-0.06	-3.30
B3	0.000	LC3	0.62	-4.32	-0.78	0.28	6.54	18.85
B3	5.000	LC3	0.62	-4.32	-0.78	0.28	2.65	-2.73
B4	0.000	LC3	0.00	0.35	0.08	-4.43	-1.48	-0.40
B4	3.000	LC3	0.00	0.35	0.08	-4.43	-1.24	0.63
B5	0.000	LC3	0.00	-0.12	-1.19	1.48	2.69	0.35
B5	6.000	LC3	0.00	-0.12	-1.19	1.48	-4.43	-0.40
B6	0.000	LC3	0.00	-0.12	0.54	1.49	-1.46	0.33
B6	6.000	LC3	0.00	-0.12	0.54	1.49	1.78	-0.36
B7	3.000	LC3	0.00	0.23	-3.19	-1.40	-4.80	0.34
B7	0.000	LC3	0.00	0.23	-3.19	-1.40	4.78	-0.35

Name	dx [m]	Case	N [kN]	V_y [kN]	V_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
B1	0.000	LC4	-0.27	0.46	0.36	-0.15	-0.88	-1.17
B1	5.000	LC4	-0.27	0.46	0.36	-0.15	0.94	1.15
B2	0.000	LC4	0.59	0.46	0.01	-0.16	-0.03	-1.17
B2	5.000	LC4	0.59	0.46	0.01	-0.16	0.04	1.15
B3	0.000	LC4	-0.08	-0.77	0.49	-2.26	-1.52	3.07
B3	5.000	LC4	-0.08	-0.77	0.49	-2.26	0.94	-0.76
B4	0.000	LC4	0.00	-0.55	-0.08	-0.91	-0.25	-0.12
B4	3.000	LC4	0.00	-0.55	-0.08	-0.91	-0.50	-1.78
B5	0.000	LC4	0.00	-0.05	-0.33	0.25	1.04	0.20
B5	6.000	LC4	0.00	-0.05	-0.33	0.25	-0.91	-0.12
B6	0.000	LC4	0.00	-0.13	0.01	0.26	-0.06	0.31
B6	6.000	LC4	0.00	-0.13	0.01	0.26	-0.03	-0.48
B7	0.000	LC4	0.00	0.06	0.60	-0.10	-0.90	-0.05
B7	3.000	LC4	0.00	0.06	0.60	-0.10	0.89	0.14

According to the SRSS-method the following formula is used:

$$R_{tot} = \sqrt{\sum_{j=1}^{N} R_{(j)}^2}$$

Take for instance the normal force in member B1:

$$N_{tot} = \sqrt{(4{,}38\text{kN})^2 + (-0{,}27\text{kN})^2} = 4{,}39\text{kN}$$

Nom	dx [m]	Cas	N [kN]	V_y [kN]	V_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
B1	0.000	LC2	4.39	1.43	0.51	0.15	1.00	3.63
B1	1.818	LC2	4.39	1.43	0.51	0.15	0.27	1.04
B1	5.000	LC2	4.39	1.43	0.51	0.15	1.59	3.50
B1	2.500+	LC2	4.39	1.43	0.51	0.15	0.40	0.07
B2	0.000	LC2	3.78	1.43	0.04	0.16	0.13	3.63
B2	3.182	LC2	3.78	1.43	0.04	0.16	0.01	0.90
B2	2.500+	LC2	3.78	1.43	0.04	0.16	0.03	0.07
B3	0.000	LC2	0.63	4.38	0.91	2.27	6.65	19.08
B3	5.000	LC2	0.63	4.38	0.91	2.27	2.83	2.82
B3	4.545	LC2	0.63	4.38	0.91	2.27	3.09	0.86
B4	3.000	LC2	1.44	0.65	0.12	4.53	1.32	1.88
B4	0.000	LC2	1.44	0.65	0.12	4.53	1.50	0.42
B5	2.308	LC2	0.24	0.14	1.23	1.50	0.29	0.10
B5	3.000-	LC2	0.24	0.14	1.23	1.50	0.87	0.05
B5	6.000	LC2	0.24	0.14	1.23	1.50	4.53	0.42
B6	2.769	LC2	0.27	0.18	0.54	1.51	0.06	0.06
B6	6.000	LC2	0.27	0.18	0.54	1.51	1.77	0.60
B7	1.500-	LC2	0.04	0.24	3.25	1.40	0.01	0.05
B7	3.000	LC2	0.04	0.24	3.25	1.40	4.88	0.38

These values correspond with the internal forces for the seismic load case in the project.

The same can be done for the reactions:

Name	Case	R_x [kN]	R_y [kN]	R_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
Sn1/N1	LC3	-0.36	-1.35	-4.38	3.44	-0.49	0.01
Sn6/N6	LC3	0.78	-4.32	-0.62	18.85	6.54	0.28
Sn3/N3	LC3	0.04	-1.35	3.73	3.44	0.12	0.01

Name	Case	R_x [kN]	R_y [kN]	R_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
Sn1/N1	LC4	-0.36	0.46	0.27	-1.17	-0.88	-0.15
Sn6/N6	LC4	-0.49	-0.77	0.08	3.07	-1.52	-2.26
Sn3/N3	LC4	-0.01	0.46	-0.59	-1.17	-0.03	-0.16

Calculation of the reaction for N1:

$$R_x = \sqrt{(-0{,}36\text{kN})^2 + (-0{,}36\text{kN})^2} = 0{,}51\text{kN}$$

Nom	Cas	R_x [kN]	R_y [kN]	R_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
Sn1/N1	LC2	**0.51**	**1.43**	**4.39**	**3.63**	**1.00**	**0.15**
Sn2/N6	LC2	**0.91**	**4.38**	**0.63**	**19.08**	**6.65**	**2.27**
Sn3/N3	LC2	**0.04**	**1.43**	**3.78**	**3.63**	**0.13**	**0.16**

After verifying the results for the seismic load case, we can conclude that these values of the manual calculation correspond to the calculated values by SCIA Engineer.

10.2 Spectral analysis example with « residual mode »

Example D-2.esa: spectral analysis with residual mode

If there is too less mass taken into account with the standard method, more mass will be added to satisfy the prescriptions of the EC.
The aim of this method is to evaluate the missing mass as an extra mode which is computed as an equivalent static load case. The static load case represents the weight of the missing mass under the cut-off acceleration. Afterwards it's summed depending the selected rule SRSS, CQC, MAX.

This missing mass is taken in the seismic analysis as an extra mode which represents the weight of the missing mass. The modal result of this mode is computed by a static equivalent load case.

The effective masses are calculated for each separate node. In the other method, the effective mass was determined for each direction in each mode. Now, this parameter will be calculated for each different node in direction X,Y and Z for each mode. Later, this missing mass will be taken into account by means of an extra load case.

Effective mass in node:

$$M_{\text{eff,k,(j),i}} = \frac{M_{k,i} * \varphi_{k,(j),i} * M_{\text{eff,k,(j)}}}{1000 * \gamma_{k,j}}$$

Calculation of the effective mass in direction X for mode 1 and N2:

$$M_{\text{eff,N2,x,(1)}} = \frac{646{,}875\text{kg} * (-12{,}67) * 12{,}346}{10000 * (-3{,}514)} = 2{,}9$$

Effective mass in nodes (k direction, j mode):

Mode 1

Node	M_x (kg)	M_y (kg)	M_z (kg)
N2	2,9	548,5	0,3
N4	1,8	548,5	-0,2
N5	3,5	689,1	103,1
N7	4,2	1345,3	0
Total	12,346	3131,374	103,182

Mode 2

Node	M_x (kg)	M_y (kg)	M_z (kg)
N2	174,0	48,6	0
N4	5,6	48,6	0,1
N5	211,7	-28,0	31,5
N7	13,3	-54,6	0
Total	404,60	14,53	31,52

Mode 1 & 2

Node	M_x (kg)	M_y (kg)	M_z (kg)
N2	176,8	597,1	0,2
N4	7,4	597,1	-0,2
N5	215,2	661,1	134,6
N7	17,5	1290,7	0
Total	416,9494	3145,9065	134,6994

The **missing mass** is the difference between the total mass for each node minus the effective mass:

$$M_{missing,N2x} = 646{,}9 - 176{,}8 = 470\text{kg}$$

Node	M_x (kg)	M_y (kg)	M_z (kg)
N2	470,0	49,8	646,6
N4	639,5	49,8	647,0
N5	572,3	126,4	652,9
N7	1520,0	246,8	1537,5

Out of these missing masses, **load cases** are generated. This by the formula:

$$\text{Load case}_{i,k} = M_{missing,i,k} * S_{k,cutoff}$$

Node	F_x (kN)	F_y (kN)	F_z (kN)
N2	0,940	0,100	1,293
N4	1,279	0,100	1,294
N5	1,145	0,253	1,306
N7	3,040	0,494	3,075
Total	6,4036	0,9457	6,9681

NB: The cut-off acceleration is the acceleration of the cut-off frequency, this the last calculated frequency.

Calculation of the **mode coefficient**:

$$G_{k,(j)} = \frac{S_{a,k,(j)} * \gamma_{k,(j)}}{\omega_{(j)}}$$

$$G_{x,(1)} = \frac{\frac{2m}{s^2} * \left(-3{,}514kg^{\frac{1}{2}}\right)}{166{,}4/s^2} = -0{,}042m.kg^{1/2}$$

(j) Units	G_x ($m*kg^{1/2}$)	G_y ($m*kg^{1/2}$)	G_z ($m*kg^{1/2}$)	G ($m*kg^{1/2}$)
1	-0,042	0,673	0,122	0,752
2	0,178	-0,034	0,050	0,194

Calculation of the lateral forces:

$$F_{i,k,(j)} = m_{i,k,(j)} * \ddot{u}_{i,k,(j)} = m_{i,k,(j)} * G_{(j)} * \phi_{i,k,(j)} * \omega_{(j)}^2$$

$$F_{1,x,(1)} = \frac{646{,}9kg * 0{,}75m.kg^{\frac{1}{2}} * (-12{,}67mm) * 166/s^2}{10000} = -102{,}6N$$

Mode 1

Node	F_x(1) (N)	F_y(1) (N)	F_z(1) (N)
N2	-102,6	1227,3	3,4
N4	-63,0	1227,3	-2,9
N5	-124,7	1541,8	1271,1
N7	-149,6	3010,0	0,2
Total	-439,9	7006,3	1271,8

Mode 2

Node	F_x(2) (N)	F_y(2) (N)	F_z(2) (N)
N2	379,1	-558,3	-0,3
N4	12,2	-558,3	0,6
N5	461,4	321,6	245,8
N7	29,0	627,8	-0,1
Total	881,7	-167,1	246,1

Calculation of the shear force in base:

$$F_{k,(j)} = \sum_i F_{i,k,(j)} l$$

(j) units	F_x (kN)	F_y (kN)	F_z (kN)
1	-0,4399	7,0063	1,2718
2	0,8817	-0,1671	0,2461
R	6,4036	0,9457	6,9681
Total	6,5	7,1	7,1

The overturning moment in each node is calculated as follows:

$$M_{i,k,(j)} = F_{i,k,(j)} * z_i$$

The height z_i is equal to the height of the concerning node minus the overturning height. In this case, the overturning height is equal to zero.

Mode 1

Node	M_x(1) (N.m)	M_y(1) (N.m)
N2	-6136,4	513,1
N4	-6136,4	315,1
N5	-7709,0	623,6
N7	-15049,9	747,9

Mode 2

Node	$M_{x}(2)$ (N.m)	$M_{y}(2)$ (N.m)
N2	2791,4	-1895,4
N4	2791,4	-60,8
N5	-1608,1	-2307,1
N7	-3139,2	-145,2

In this case, an extra overturning moment is calculated for the residual load case:

$$\mathrm{M_{N2,y,(1)}} = 0{,}94\mathrm{kN} * (5\mathrm{m} - 0\mathrm{m}) = -4{,}7\mathrm{kN.m}$$

Mode R

Node	$M_{x}(R)$ (kN.m)	$M_{y}(R)$ (kN.m)
N2	0	-4,7
N4	0	-6,4
N5	0	-5,7
N7	0	-15,2

The letter R stands for the residual mode.

For each mode the **sum of the overturning moments** are taken, afterwards the results are combined with the SRSS method:

(j) units	M_{x} (kN)	M_{y} (kN)
1	-35,0317	2,1997
2	0,8355	-4,4085
R	0,0000	-32,0180
Total	35,0	32,4

Calculation of the **modal displacement**:

$$\mathrm{u_{i,k,(j)}} = \mathrm{G_{(j)}} * \phi_{\mathrm{k,(j)}}$$

Mode 1

Node	u_x (mm)	u_y (mm)	u_z (mm)
N2	-0,95	11,40	0,03
N4	-0,59	11,40	-0,03
N5	-0,95	11,77	9,70
N7	-0,58	11,77	0,00

Mode 2

Node	u_x (mm)	u_y (mm)	u_z (mm)
N2	2,60	-3,82	0,00
N4	0,08	-3,82	0,00
N5	2,59	1,81	1,38
N7	0,08	1,81	0,00

To calculate the deformations for mode R, the load cases - generated out of the missing masses - are inputted as real load cases on the nodes of the structure. This gives the following table:

2. Deformation of nodes

Linear calculation, Extreme : Node
Selection : All
Load cases : LC3

Node	Case	Ux [mm]	Uy [mm]	Uz [mm]
N1	LC3	0	0	0
N2	LC3	4,14	4,91	0,03
N3	LC3	0	0	0
N4	LC3	1,46	4,91	0
N5	LC3	4,14	8,25	6,74
N6	LC3	0	0	0
N7	LC3	1,45	8,25	0

The deformations for each mode (namely mode 1, mode 2 and mode R) are combined with the SRSS-formula:

Total

Node	u_x (mm)	u_y (mm)	u_z (mm)
N2	4,98	12,99	0,04
N4	1,58	12,99	0,03
N5	4,98	14,48	11,89
N7	1,57	14,48	0,00

The same for the **modal acceleration**:

$$\ddot{u}_{i,k,(j)} = \omega_{(j)}^2 * G_{(j)} * \phi_{k,(j)}$$

Mode 1

Node	a_x (mm/s²)	a_y (mm/s²)	a_z (mm/s²)
N2	-158,6	1897,2	5,3
N4	-97,4	1897,2	-4,5
N5	-158,4	1957,8	1614,2
N7	-97,3	1957,7	0,1

Mode 2

Node	a_x (mm/s²)	a_y (mm/s²)	a_z (mm/s²)
N2	586,0	-863,0	-0,4
N4	18,8	-863,0	1,0
N5	585,9	408,4	312,1
N7	18,9	408,4	0,0

For the mode R, the constant value of 2000mm/s² is used:

Mode R

Node	a_x (mm/s²)	a_y (mm/s²)	a_z (mm/s²)
N2	2000,0	2000,0	2000,0
N4	2000,0	2000,0	2000,0

N5	2000,0	2000,0	2000,0
N7	2000,0	2000,0	2000,0

This gives through the SRSS-method:

Total

Node	a_x (mm/s²)	a_y (mm/s²)	a_z (mm/s²)
N2	2090,1	2888,7	2000,0
N4	2002,5	2888,7	2000,0
N5	2090,1	2828,4	2589,0
N7	2002,5	2828,3	2000,0

In the same way as for the 'missing mass method' the calculated deformations are put on the structure as real load cases. This gives the following internal forces:

Mode 1:

Linear calculation, Extreme : Member, System : Principal Selection : All Load cases : LC3								
Member	**Case**	**dx [m]**	**N [kN]**	**Vy [kN]**	**Vz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
B1	LC3	0	**4,38**	**-1,35**	**0,36**	**0,01**	**-0,49**	**3,44**
B1	LC3	5	4,38	-1,35	0,36	0,01	**1,29**	**-3,3**
B2	LC3	0	**-3,73**	**-1,35**	**-0,04**	**0,01**	**0,12**	**3,44**
B2	LC3	5	-3,73	-1,35	-0,04	0,01	**-0,06**	**-3,3**
B3	LC3	0	**0,62**	**-4,32**	**-0,78**	**0,28**	**6,54**	**18,85**
B3	LC3	5	0,62	-4,32	-0,78	0,28	**2,65**	**-2,73**
B4	LC3	0	**0**	**0,35**	**0,08**	**-4,43**	**-1,48**	**-0,4**
B4	LC3	3	0	0,35	0,08	-4,43	**-1,24**	**0,63**
B5	LC3	0	**0**	**-0,12**	**-1,19**	**1,48**	**2,69**	**0,35**
B5	LC3	6	0	-0,12	-1,19	1,48	**-4,43**	**-0,4**
B6	LC3	0	**0**	**-0,12**	**0,54**	**1,49**	**-1,46**	**0,33**
B6	LC3	6	0	-0,12	0,54	1,49	**1,78**	**-0,36**
B7	LC3	0	**0**	**0,23**	**-3,19**	**-1,4**	**4,78**	**-0,35**
B7	LC3	3	0	0,23	-3,19	-1,4	**-4,8**	**0,34**

Mode 2:

Linear calculation, Extreme : Member, System : Principal Selection : All Load cases : LC4								
Member	**Case**	**dx [m]**	**N [kN]**	**Vy [kN]**	**Vz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
B1	LC4	0	**-0,27**	**0,46**	**0,36**	**-0,15**	**-0,88**	**-1,17**
B1	LC4	5	-0,27	0,46	0,36	-0,15	**0,94**	**1,15**
B2	LC4	0	**0,59**	**0,46**	**0,01**	**-0,16**	**-0,03**	**-1,17**
B2	LC4	5	0,59	0,46	0,01	-0,16	**0,04**	**1,15**
B3	LC4	0	**-0,08**	**-0,77**	**0,49**	**-2,26**	**-1,52**	**3,07**
B3	LC4	5	-0,08	-0,77	0,49	-2,26	**0,94**	**-0,76**
B4	LC4	0	**0**	**-0,55**	**-0,08**	**-0,91**	**-0,25**	**-0,12**
B4	LC4	3	0	-0,55	-0,08	-0,91	**-0,5**	**-1,78**
B5	LC4	0	**0**	**-0,05**	**-0,33**	**0,25**	**1,04**	**0,2**
B5	LC4	6	0	-0,05	-0,33	0,25	**-0,91**	**-0,12**
B6	LC4	0	**0**	**-0,13**	**0,01**	**0,26**	**-0,06**	**0,31**
B6	LC4	6	0	-0,13	0,01	0,26	**-0,03**	**-0,48**
B7	LC4	0	**0**	**0,06**	**0,6**	**-0,1**	**-0,9**	**-0,05**
B7	LC4	3	0	0,06	0,6	-0,1	**0,89**	**0,14**

Mode R:

Linear calculation, Extreme : No, System : Principal Selection : All Load cases : LC3								
Member	**Case**	**dx [m]**	**N [kN]**	**Vy [kN]**	**Vz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
B1	LC3	0	4,232	-0,575	0,824	-0,134	-1,804	1,47
B1	LC3	5	4,232	-0,575	0,824	-0,134	2,314	-1,402
B2	LC3	0	-0,061	-0,574	0,244	-0,144	-0,559	1,47
B2	LC3	5	-0,061	-0,574	0,244	-0,144	0,661	-1,401
B3	LC3	0	2,798	-3,078	5,79	-2,301	-22,431	13,299
B3	LC3	5	2,798	-3,078	5,79	-2,301	6,521	-2,093
B4	LC3	0	-1,406	-1,192	0,005	-4,714	-1,048	0,818
B4	LC3	3	-1,406	-1,192	0,005	-4,714	-1,033	-2,759
B5	LC3	0	0,159	0,242	-1,301	1,048	3,092	-0,632
B5	LC3	6	0,159	0,242	-1,301	1,048	-4,714	0,818
B6	LC3	0	-1,527	0,164	-0,282	1,061	-0,117	-0,525
B6	LC3	6	-1,527	0,164	-0,282	1,061	-1,807	0,457
B7	LC3	0	0,039	-0,478	-1,637	-0,778	2,45	0,766
B7	LC3	3	0,039	-0,478	-1,637	-0,778	-2,461	-0,669

Combination via SRSS method gives:

Member	**Case**	**dx [m]**	**N [kN]**	**Vy [kN]**	**Vz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
B1	LC2	0,00	6,10	1,54	0,97	0,20	2,07	3,92
B1	LC2	5,00	6,10	1,54	0,97	0,20	2,81	3,77
B2	LC2	0,00	3,78	1,54	0,25	0,22	0,57	3,92
B2	LC2	5,00	3,78	1,54	0,25	0,22	0,66	3,77
B3	LC2	0,00	2,87	5,36	5,86	3,24	23,41	23,27
B3	LC2	5,00	2,87	5,36	5,86	3,24	7,10	3,52
B4	LC2	0,00	1,41	1,36	0,11	6,53	1,83	0,92
B4	LC2	3,00	1,41	1,36	0,11	6,53	1,69	3,34
B5	LC2	0,00	0,16	0,27	1,79	1,83	4,23	0,75
B5	LC2	6,00	0,16	0,27	1,79	1,83	6,53	0,92
B6	LC2	0,00	1,53	0,24	0,61	1,85	1,47	0,69
B6	LC2	6,00	1,53	0,24	0,61	1,85	2,54	0,75
B7	LC2	0,00	0,04	0,53	3,64	1,60	5,45	0,84
B7	LC2	3,00	0,04	0,53	3,64	1,60	5,47	0,76

The reactions are:

Mode 1

Linear calculation, Extreme : Node Selection : All Load cases : LC3							
Support	**Case**	**Rx [kN]**	**Ry [kN]**	**Rz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
Sn1/N1	LC3	**-0,36**	**-1,35**	**-4,38**	**3,44**	**-0,49**	**0,01**
Sn2/N6	LC3	**0,78**	**-4,32**	**-0,62**	**18,85**	**6,54**	**0,28**
Sn3/N3	LC3	**0,04**	**-1,35**	**3,73**	**3,44**	**0,12**	**0,01**

Mode 2:

Linear calculation, Extreme : Node Selection : All Load cases : LC4							
Support	**Case**	**Rx [kN]**	**Ry [kN]**	**Rz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
Sn1/N1	LC4	**-0,36**	**0,46**	**0,27**	**-1,17**	**-0,88**	**-0,15**
Sn2/N6	LC4	**-0,49**	**-0,77**	**0,08**	**3,07**	**-1,52**	**-2,26**
Sn3/N3	LC4	**-0,01**	**0,46**	**-0,59**	**-1,17**	**-0,03**	**-0,16**

Mode R:

Linear calculation, Extreme : Node Selection : All Load cases : LC3							
Support	**Case**	**Rx [kN]**	**Ry [kN]**	**Rz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
Sn1/N1	LC3	**-0,82**	**-0,575**	**-4,232**	**1,47**	**-1,804**	**-0,134**
Sn2/N6	LC3	**-5,79**	**-3,078**	**-2,798**	**13,299**	**-22,431**	**-2,301**
Sn3/N3	LC3	**-0,24**	**-0,574**	**0,061**	**1,47**	**-0,559**	**-0,144**

SRSS:

Support	**Case**	**Rx [kN]**	**Ry [kN]**	**Rz [kN]**	**Mx [kNm]**	**My [kNm]**	**Mz [kNm]**
Sn1/N1	LC2	0,97	1,54	6,10	3,92	2,07	0,20
Sn2/N6	LC2	5,86	5,36	2,87	23,27	23,41	3,24
Sn3/N3	LC2	0,25	1,54	3,78	3,92	0,57	0,22

NB:

In case of CQC, we don't assume any correlation with the other modes (i.e. absolute value is added)
The cut-off frequency is the frequency of the latest modes in the analysis. It is the responsibility of the user to select the correct number of modes. This can be done in the Solver Setup.

CHAPTER 12 : DAMPING

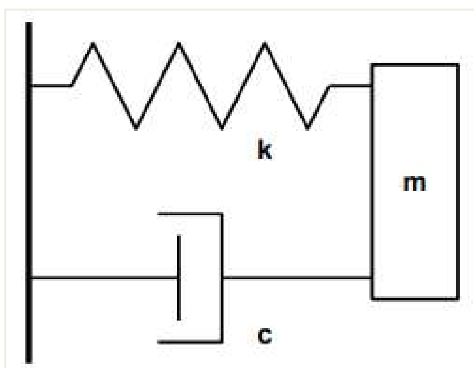
In the previous chapters, the influence of damping on the dynamic response of a structure was shown. Especially in the vicinity of resonance the effect of damping was significant.
In this chapter, damping will be looked upon in more detail. First the theory will be explained after which the input of non-uniform damping in SCIA Engineer is regarded.
By means of the examples of the previous chapter, the influence of damping on the seismic response is illustrated. The chapter is finished with a 3D structure, which takes into account material damping of the different elements.

12.1 Theory

Damping can have different causes. The component that is always present is structural damping. Structural damping is caused by hysteresis in the material: the transfer of small amounts of energy into warmth for each vibration cycle possibly increased by friction between internal parts.
Other causes can be the foundation soil of the building and aerodynamic damping due to the diversion of energy by the air [22]. In many cases, damping is increased by adding artificial dampers to the structure.

In the same way as for the previous chapters, first the theory is examined. A complete overview can be found in reference [1].

Consider the following damped free-vibrating system:



A body mass **m** can move in one direction. A spring of constant stiffness **k**, which is fixed at one end, is attached at the other end to the body. The mass is also subjected to damping with a damping capacity **c**.

The **equation of motion**, using matrix notations can be written as:

$$M.\ddot{x}(t) + C.\dot{x}(t) + K.x(t) = 0 \quad (5.1)$$

A **possible solution** to this equation is one of the type:

$$x = A.e^{st} \tag{5.2}$$

Substituting (5.2) in (5.1) gives:

$$M.s^2.A.e^{st} + C.s.A.e^{st} + K.A.e^{st} = 0 \tag{5.3}$$

This equation can be rewritten as:

$$s^2 + 2.n.s + \omega_n^2 = 0 \tag{5.4}$$

With:

$$n = \frac{C}{2M} \tag{5.5}$$

$$\omega_n = \sqrt{\frac{K}{M}} \tag{5.6}$$

The possible solutions for equation (5.4) are:

$$s = -n \pm \sqrt{n^2 - \omega_n^2} \tag{5.7}$$

It is clear that the response of the system depends on the numerical value of the radical. Therefore the following three possibilities need to be examined:

$$n = \omega_n$$

$$n < \omega_n$$

$$n > \omega_n \tag{5.8}$$

These can be rewritten as:

$$C = 2.\sqrt{K.M}$$

$$C < 2.\sqrt{K.M}$$

$$C > 2.\sqrt{K.M} \tag{5.9}$$

The condition $C = 2.\sqrt{K.M} = C_c$ is called *critical damping*. In this case, the displaced body is restored to equilibrium in the shortest possible time, without oscillation.

The ratio ξ is called the **damping ratio** or the **relative damping**:

$$\xi = \frac{C}{C_c}$$

Therefore, when assuming $n = \xi.\omega_n$, equation (5.5) can be written as:

$$C = 2.\xi.\omega_n.M \tag{5.10}$$

The three possibilities of equation (5.8) can be rewritten as:

$$\xi = 1$$

$$\xi < 1$$

$$\xi > 1$$

(5.11)

When looking at the conditions $\xi = 1$ and $\xi > 1$, it can be shown that there is no harmonic solution.
Only the condition $\xi < 1$ gives a harmonic solution.

Introducing the damped circular frequency:

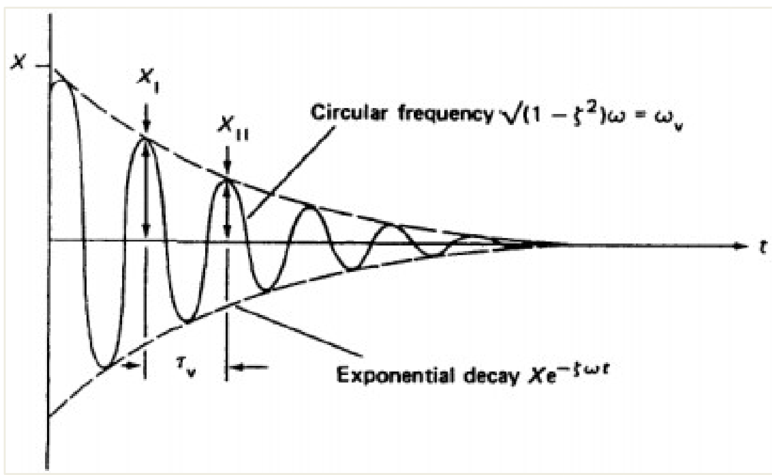
$$\omega_D = \omega_n . \sqrt{1 - \xi^2}$$

the solution to equation (5.1) can be written as:

$$x = e^{-\xi \omega_n t} . \{A . \cos(\omega_D t) + B . \sin(\omega_D t)\}$$

(5.12)

In a previous chapter, this vibration equation was illustrated by the following figure:



A convenient way to determine the damping in a system was shown to be the logarithmic decrement Λ, which is the natural logarithm of the ratio of any two successive amplitudes in the same direction.

$$\Lambda = \ln \frac{X_1}{X_{11}} = \frac{2\pi\xi}{\sqrt{1 - \xi^2}}$$

(5.13)

NB:

As shown above, the circular frequency is reduced by the damping action to obtain the damped circular frequency. However, in many systems this reduction is likely to be small because very small values of ξ are common; for example, in most engineering structures ξ is rarely greater than 0,02. Even if ξ = 0,2; ω_D = 0,98ω_n.

Annex B gives some references for numerical values of the damping ratio.

12.2 Damping in SCIA Engineer

In SCIA Engineer, damping can be specified on 1D elements, 2D elements and supports. The damping of each of these elements (or substructures) will be used to calculate a modal damping ratio for the whole structure for each Eigenmode. In the literature this is described as Composite Damping.

Composite damping is used in partly bolted, partly welded steel constructions, mixed steel-concrete structures, constructions on subsoil, ...
For structural systems that consist of substructures with different damping properties, the composite damping matrix C can be obtained by an appropriate superposition of damping matrices for the individual substructures C_i:

$$C = \sum_{i=1}^{N} C_i \tag{5.14}$$

With:

C_i: the damping matrix for the i^{th} substructure in the global coordinate system.
N: the number of substructures being assembled.

- *Proportional Damping (Rayleigh Damping)*

A way of describing the damping is assuming that the damping matrix is formed by a linear combination of the mass and stiffness matrices.

$$C_i = \alpha_i . M_i + \beta_i . K_i \tag{5.15}$$

With:

α_i and β_i: proportional damping for the i^{th} part of the structure.
M_i: mass matrix for the i^{th} part of the structure in the global coordinate system.
K_i: stiffness matrix for the i^{th} part of the structure in the global coordinate system.

Formulas for these proportional damping coefficients can be found in reference [19].
Examples can be found in reference [20].

- *Stiffness-Weighted Damping*

For structures or structural systems that consist of major substructures or components with different damping characteristics, composite modal damping values can be calculated using the elastic energy of the structure [8], [21]:

$$\xi_j = \frac{\sum_{i=1}^{N} \xi_j . E_i}{E} \tag{5.16}$$

With:

ξ_j: damping ratio of the considered eigenmode.
E: elastic energy of the structure, associated with the modal displacement of the considered eigenmode.
N: number of all substructures.
ξ_i: damping ratio for the $i^{ème}$ substructure.
E_i: elastic energy for the $i^{ème}$ substructure, associated with the modal displacement of the considered eigenmode.

Equation (5.16) can be rewritten in the following way [19]:

$$\xi_j = \frac{\Phi_j^T . \left[\sum_{i=1}^{N} [\xi K]_i\right] . \Phi_j}{\omega_j^2} \tag{5.17}$$

With:
$[\xi K]_i$: stiffness matrix for the i^{th} substructure in the global coordinate system, scaled by the modal damping ratio of the i^{th} substructure.

NB:

This formula may be used as long as the resulting damping values are less than ***20%*** *of critical. If values in excess of 20% are computed, further justification is required.*

As specified, in SCIA Engineer on each element a damping ratio can be inputted. For this ratio, also the damping of the material can be used from which the element is manufactured.

When no damping ratio is inputted on an element, a default value will be used since all elements need a damping ratio before the above formulas can be applied. The input of this default will be shown in the examples.
Analogous to the input of other objects in SCIA Engineer, **Damping** on elements will be grouped in a **Damping Group**. In turn, this Group can be assigned to a **Combination of Mass Groups**.

- *Support damping*

Additional to the damping of 1D and 2D elements, SCIA Engineer allows the input of a damper on a flexible nodal support. The modal damping ratio ξ_j is calculated by the following formula:

$$\xi_j = \alpha . \frac{\Phi_{s,j}^{T} . [\sum_s C_s] . \Phi_{s,j}}{4 . \omega_j} \tag{5.18}$$

With:

ω_j: the circular frequency of mode j
$\Phi_{s,j}$: the modal displacement in support node s for mode j
C_s: the damping constant for the support
α: a user defined parameter (> 0)

The total modal damping ratio can then be calculated as the summation of equations (5.17) and (5.18).

As specified, on all 1D and 2D elements a damping ratio has to be defined. This is not the case with supports, not every support needs to have a damping value.

The following diagram shows how non-proportional damping is inputted in SCIA Engineer:

Activate the functionalities:
- "Dynamics"
- « Non-proportional damping »

↓

Create a damping group

↓

Input dampers

↓

Assign the damping group to a mass combination

↓

Proceed with the steps of the previous chapters.

The use of dampers and the calculation of the composite damping ratio will be illustrated in the following examples.

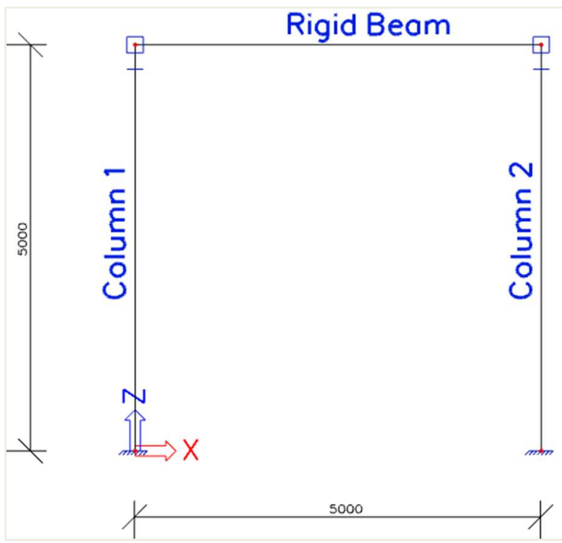
NB: The damping functionality is only available on 32-bit version of SCIA Engineer.

Example 11-1.esa

In this example, the principle of stiffness-weighted damping is illustrated.

A concrete frame is modelled in which the beam is assumed to be rigid. In this case, only the columns take part in the horizontal stiffness of the frame.

The left column has a **Rectangular 500 x 500** section, the right column a **Rectangular 350 x 350** section. The column bases are modelled as rigid. To model the rigid beam, a **Rectangular 500000 x 500000** section is used. To make sure this beam acts as rigid, in the nodes between the columns and the beam, supports are inputted which have a fixed **Translation Z** and **Rotation Ry**. The height of the columns and the length of the beam are taken as **5m**. All elements are manufactured in **C30/37** according to **EC-EN**.



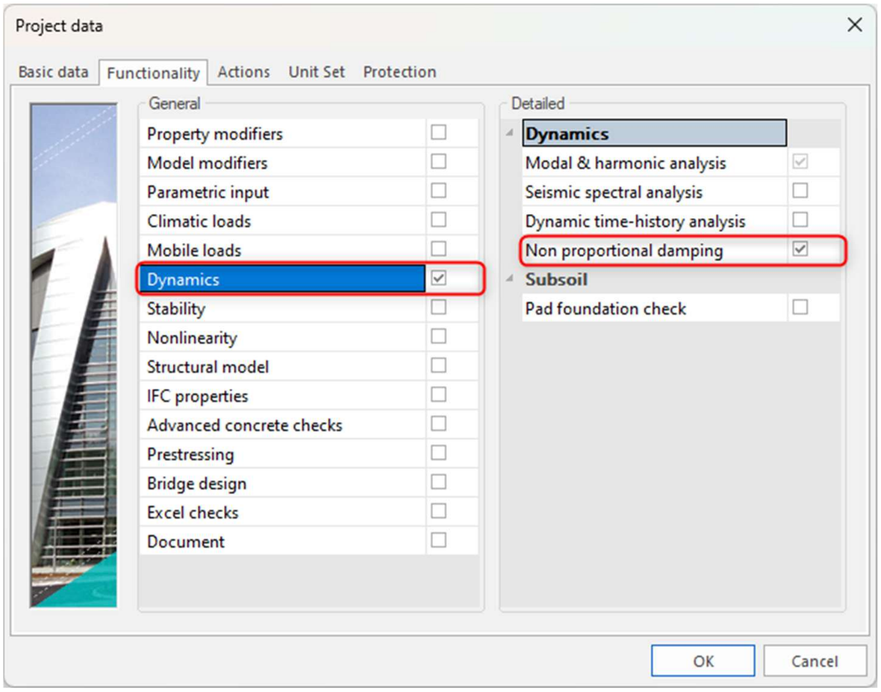
The beam is loaded by a line mass of **500 kg/m**. The left column has a damping ratio of **12%**, the right column a damping ratio of **3%**.

One static load case is created: the **self-weight** of the beam. However, in order not to take the self-weight into account for the dynamic calculation, the volumetric mass of **C30/37** can be set to **1e-10 kg/m³** in the **Material** Library. This low value is chosen to avoid any influence by the rigid beam.

The steps of the Free Vibration calculation are followed and extended with the input of damping.

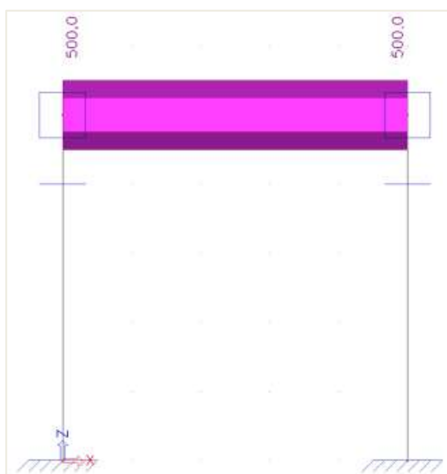
Step 1: functionality

The first step in the Dynamic calculation is to activate the functionalities **Dynamics** and **Non-Proportional Damping** on the **Functionality** tab in the **Project Data**.



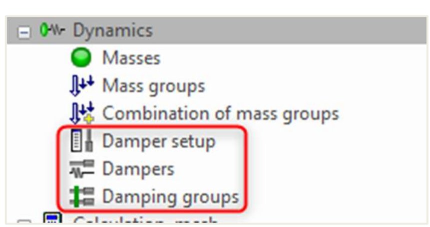
Step 2: mass group and masses

A Mass Group is created after which the line mass of 500 kg/m can be inputted on the rigid beam.

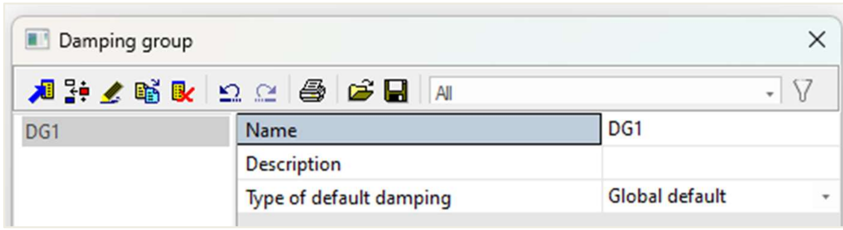


Step 3: damping

Before creating a Combination of Mass Groups, the dampers are inputted.



First of all, a Damping Group is created.

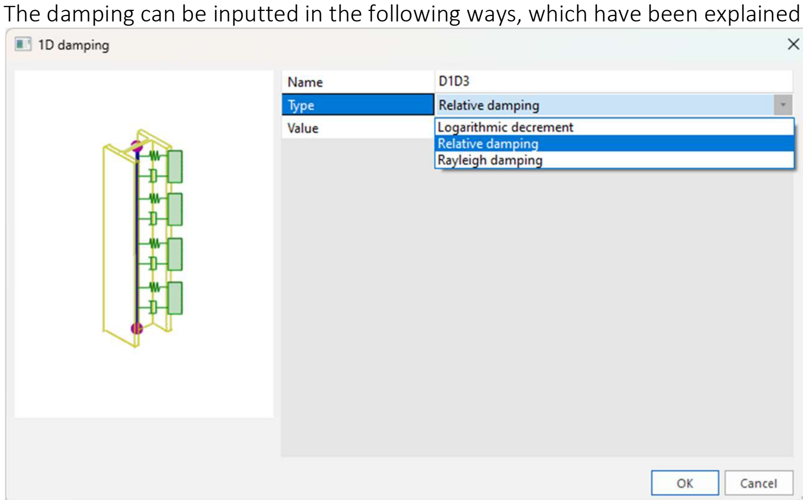


As specified in the theory, on each element a damping ratio needs to be inputted. When no damper is specified, a default value will be taken. In the properties of the Damping Group, this default can be set as either:

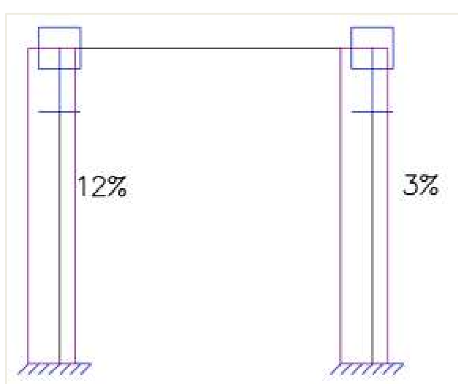
- « Global default »: the logarithmic decrement specified in the Damper Setup will be used.
- « Material default » : the logarithmic decrement of the material will be used.

In this example, the *Global default* is chosen.

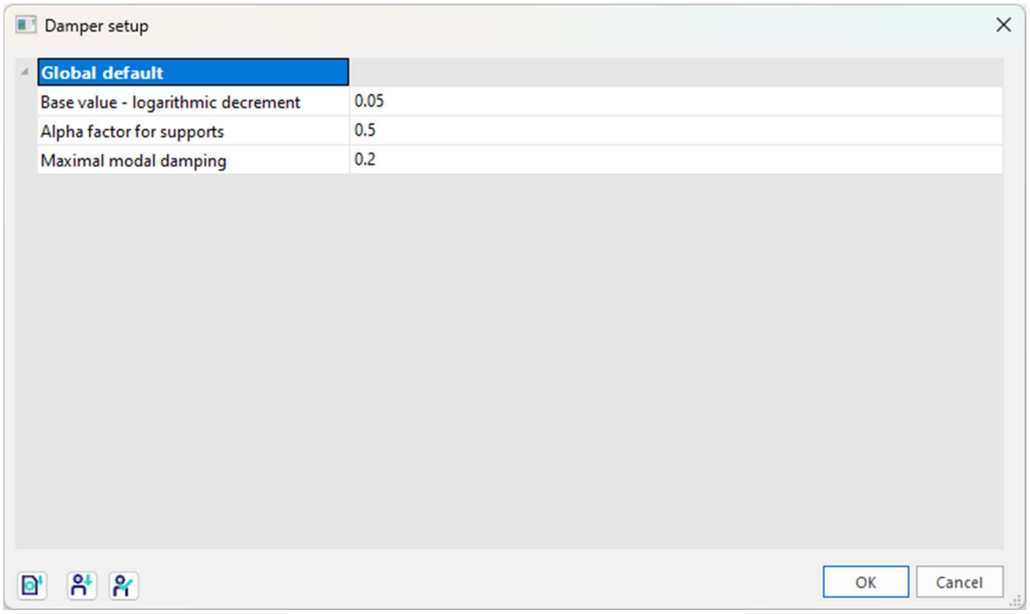
After the creation of a Damping Group, Dampers can be inputted. In this example, 1D Damping shall be inputted on the columns. The damping can be inputted in the following ways, which have been explained in the theory:



On the left column, a Relative damping of 0,12 is inputted.
On the right column, a Relative damping of 0,03 is inputted.



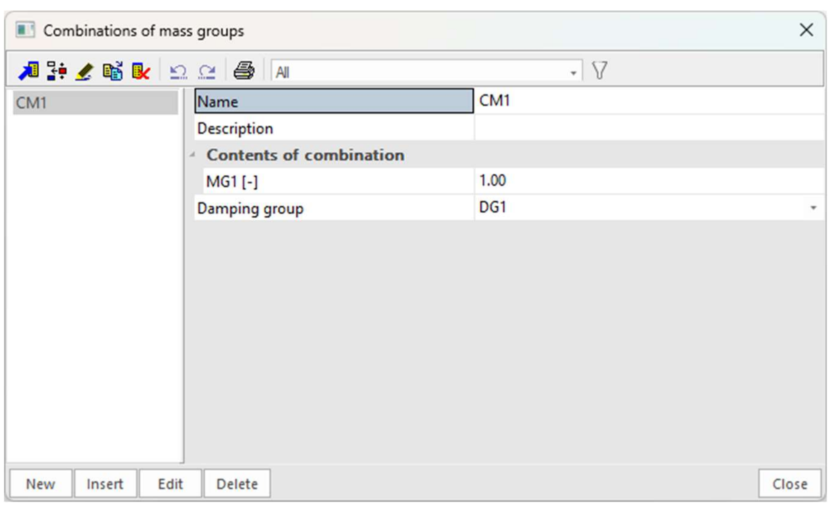
As a final step, the general parameters can be checked through Damper Setup:



The *Base value* specifies the default value when a Damping Group of type Global default is chosen and no damper is inputted on an element.
The *Alpha factor* is used in the damping calculation for supports as specified in the theory.
When the composite modal damping ratio is calculated, the value is checked with the *Maximal modal damping* value inputted here. If the calculated value is higher than the maximal value, the maximal value is used. In this example, the maximal value is set to 0,2 in accordance with the remark for formula (5.17)

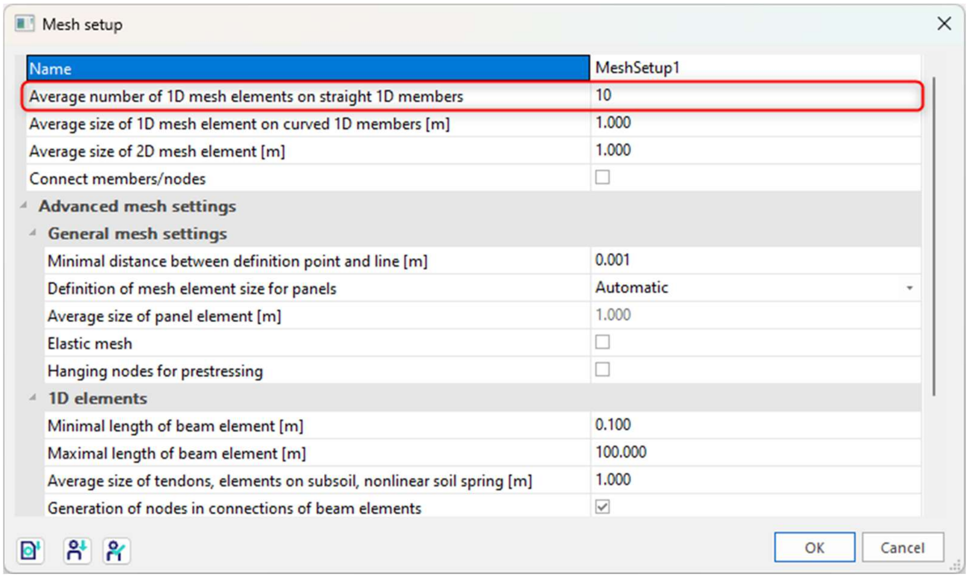
Step 4: mass matrix

A Combination of Mass Groups can now be created and the Damping Group can be specified:



Step 5: mesh setup

To obtain precise results for the dynamic calculation, the Finite Element Mesh is refined.
This can be done through the main menu Tools / Calculation & Mesh / Mesh settings:

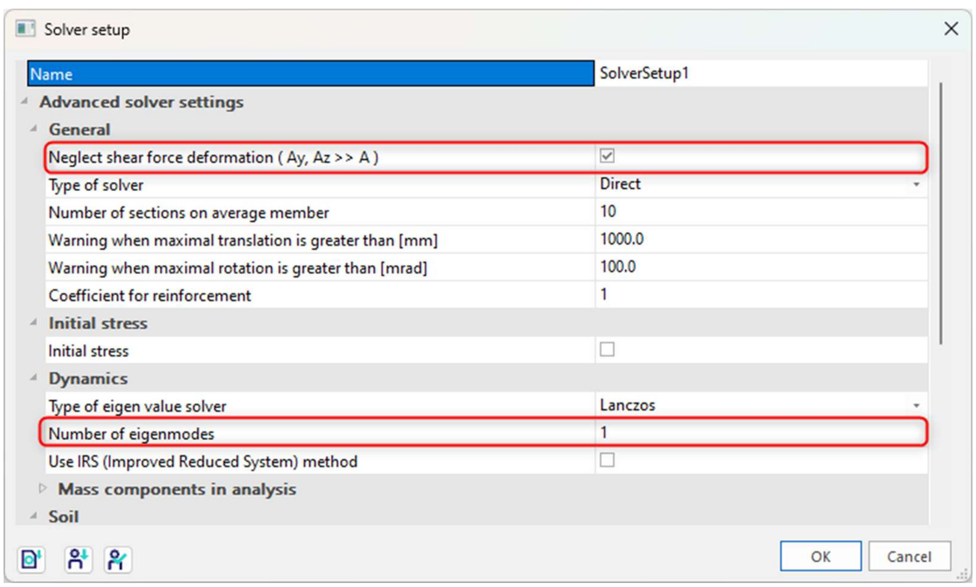


The Average number of tiles of 1D element is set to 10.

Step 6: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. For this example, only one eigenmode is required so in the main menu Tools / Calculation & Mesh / Solver Settings, the number of frequencies is set to 1.

To compare the results with a manual calculation, the shear force deformation is neglected.



Step 7: linear calculation and calculation protocol

All steps have been executed so the Free Vibration calculation can be started through the main menu Tools / Calculation & Mesh / Calculate.

The following results are obtained through the Calculation Protocol for the Eigen Frequency calculation:

Sum of masses

[kg]	X	Y	Z
Combination of mass groups 1	2500.00	0.00	2250.00

Relative modal masses

Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Wxi / Wxtot	Wyi / Wytot	Wzi / Wztot	Wxi_R / Wxtot_R	Wyi_R / Wytot_R	Wzi_R / Wztot_R	Damp ratio
1	89.0864	0.0705	14.1785	1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.1026
				1.0000	0.0000	0.0000	0.0000	0.0000	0.0000	

The calculated modal damping ratio is shown to be 0,1026 or 10,26%.

Step 8: manual calculation

In order to check the results of SCIA Engineer, a manual calculation is performed.
First, the calculated eigen frequency is checked using formula (2.3)
In this example, the two columns can be treated as fixed-fixed beams. Using default engineering tables [12], each column contributes the following stiffness to the frame:

$$k = \frac{12.EI}{L^3} \tag{5.19}$$

With for column 1:

E = 32000N/mm²
I = 5208300000mm⁴
L = 5000mm

And for column 2:

E = 32000N/mm²
I = 1250500000mm⁴
L = 5000mm

So:

$$k_1 = \frac{12 * \frac{32000N}{mm^2} * 5208300000mm^4}{(5000mm)^3} = 15999{,}8976N/mm$$

$$k_2 = \frac{12 * \frac{32000N}{mm^2} * 1250500000mm^4}{(5000mm)^3} = 3841{,}536N/mm$$

Both columns act in parallel since each column will displace the same amount due to the fact the beam is rigid. The beam itself does not bend so it does not contribute to the stiffness.

$$k_{tot} = k_1 + k_2 = \frac{15999{,}8976N}{mm} + \frac{3841{,}536N}{mm} = \frac{19841{,}4336N}{mm}$$

The vibrating mass is calculated as:

$$\frac{500kg}{m} * 5m = 2500kg$$

$$\omega = \sqrt{\frac{k}{m}} = \sqrt{\frac{19841433{,}6N/m}{2500kg}} = 89{,}087rad/s$$

$$f = \frac{\omega}{2\pi} = 14{,}1787Hz$$

These results correspond exactly to the results obtained by SCIA Engineer.

Next, the stiffness-weighted damping ratio is calculated. The first column has a damping ratio of 12%, the second column a damping ratio of 3%.
Using the elastic energy principle of formula (5.16) the modal damping ratio can be calculated as follows:

$$\xi = \frac{\xi_1 . k_1 + \xi_2 . k_2}{k_{tot}}$$

$$\xi = \frac{(0{,}12 * 15999{,}8976\mathrm{N/mm}) + (0{,}03 * 3841{,}536\mathrm{N/mm})}{19841{,}4336\mathrm{N/mm}}$$

$$\xi = 0{,}1026 = 10{,}26\%$$

This result corresponds exactly to the result obtained by SCIA Engineer.
The modal damping ratio can now be used to calculate the Damping Coefficient in a seismic calculation. This will be illustrated in the following examples.

Example 11-2.esa

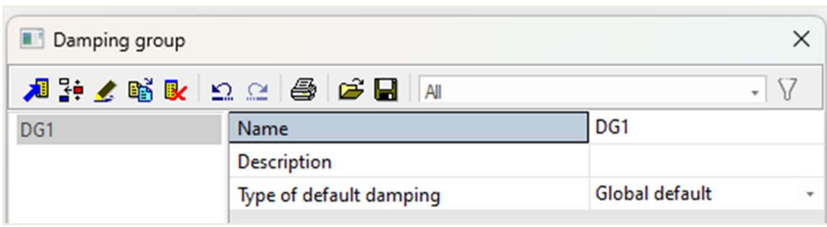
In this example, non-proportional damping is accounted for in a seismic calculation using the SRSS modal combination method. To this end, the example (04-2) from the previous chapter is extended with dampers.
More specifically, a relative damping of **12%, 3%** and **8%** is inputted on the three columns starting from the base of the structure.

Step 1: functionality

The first step to take into account the damping is to activate the functionality **Non-Proportional Damping** on the **Functionality** tab in the **Project Data**.

Step 2: damping group

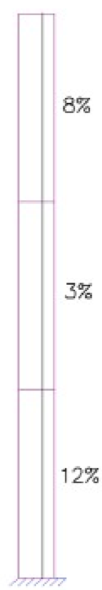
The second step is the creation of a **Damping Group**.



Since a damper will be inputted on all elements, the choice of the default damping type is not relevant.

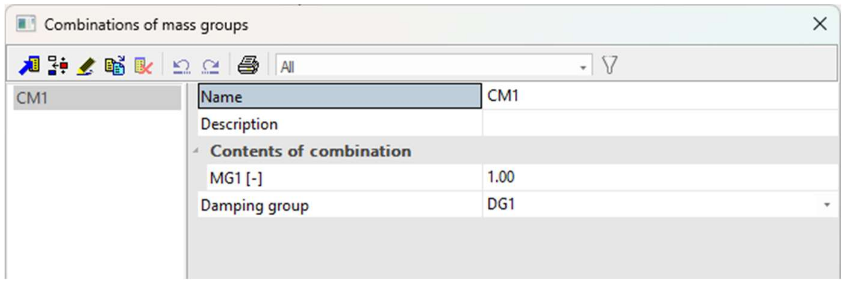
Step 3: dampers

After the creation of a **Damping Group**, **Dampers** can be inputted. A relative damping of **12%**, **3%** and **8%** is inputted on the three columns starting from the base of the structure:



Step 4: mass matrix

As a final step, the Damping Group is assigned to the Mass Combination:



Step 5: linear calculation and calculation protocol

The non-proportional damping has now been inputted so the Linear Calculation can be re-done to see the Seismic results.

The following results are obtained through the Calculation protocol of the Linear Calculation:

Dynamic loadcase: 2 : LC2

Mode	Freq. [Hz]	Damp ratio	Damp|coe	Sax [m/s²]	Say [m/s²]	Saz [m/s²]	G(j)	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.5253	0.0996	0.8176	0.1650	0.0000	0.0000	0.5001	0.1799	0.0000	0.0000	-1.7990
2	3.4262	0.0711	0.9086	0.3980	0.0000	0.0000	0.0154	0.1287	0.0000	0.0000	-0.3717
Level=	0.00							0.22	0.00	0.00	1.84

For both eigenmodes the Composite Modal Damping Ratio is calculated using equation (5.17).

As specified in the previous chapter, this Damping Ratio will be used to calculate the Damping Coefficient, which influences the spectral accelerations. Using equation (4.13):

$$\eta_1 = \sqrt{\frac{10}{(5+9{,}96)}} = 0{,}8176$$

$$\eta_2 = \sqrt{\frac{10}{(5+7{,}11)}} = 0{,}9087$$

As expected, since the modal damping ratios are higher than the default 5% used in the acceleration spectrum, they will have a positive effect thus lowering the response of the structure.

More specifically, for the first eigenmode only 81,7% of the spectral acceleration will be taken into account and for the second eigenmode 90,8%.

The spectral accelerations of the original example without damping can thus be multiplied by η :

$S_{ax,(1)}$ = 0,2019 m/s² * 0,8176 = 0,1651 m/s²
$S_{ax,(2)}$ = 0,4380 m/s² * 0,9087 = 0,3980 m/s²

These adapted spectral accelerations will thus influence the mode coefficients, the base shear, the overturning moment, the nodal displacements and accelerations,...

Example 11-3.esa:

In this example, non-proportional damping is accounted for in a seismic calculation using the CQC modal combination method. To this end, the example **Spectral_Analysis_3.esa** from the previous chapter is extended with dampers.
More specifically, a relative damping of **2%**, **5%** and **2%** is inputted on the three columns starting from the base of the structure.

As seen in the theory and the original example, the CQC method required the definition of a **Damping Spectrum**. This damping spectrum was used for the calculation of the **Modal Cross Correlation Coefficients** and to calculate the **Damping Coefficient** for each mode.

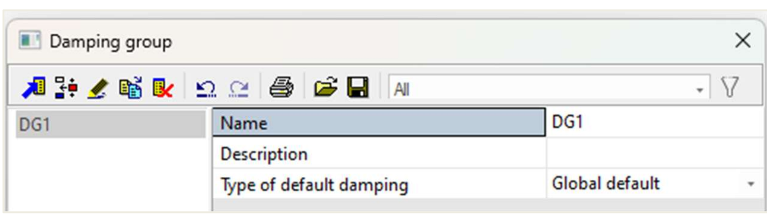
When however Non-Proportional Damping is used, the calculated Composite Modal Damping Ratios are used *instead* of the data of the Damping Spectrum. This is illustrated in this example.

Step 1: functionality

The first step to take into account the damping is to activate the functionality **Non-Proportional Damping** on the **Functionality** tab in the **Project Data**.

Step 2: damping group

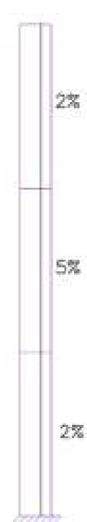
The second step is the creation of a **Damping Group**:



Since a damper will be inputted on all elements, the choice of the default damping type is not relevant.

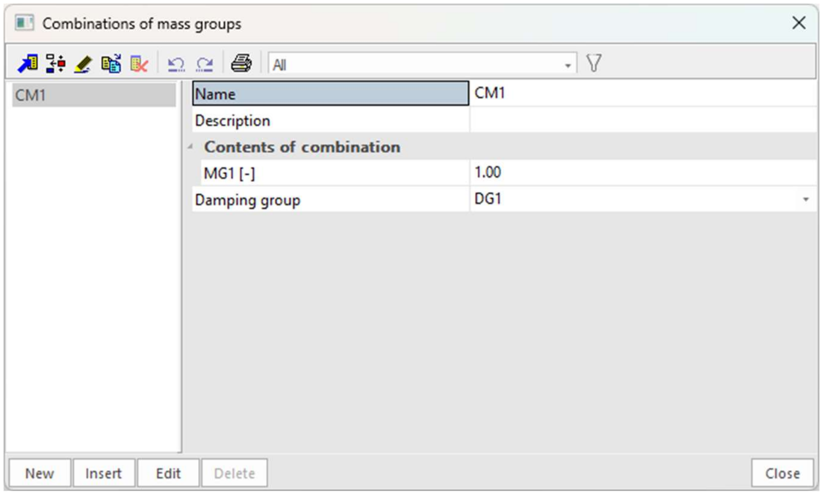
Step 3: dampers

After the creation of a **Damping Group**, **Dampers** can be inputted. A relative damping of **2%**, **5%** and **2%** is inputted on the three columns starting from the base of the structure:



Step 4: mass matrix

As a final step, the Damping Group is assigned to the Mass Combination:



Step 5: linear calculation and calculation protocol

The non-proportional damping has now been inputted so the Linear Calculation can be re-done to see the Seismic results.

The following results are obtained through the Calculation protocol of the Linear Calculation:

Dynamic loadcase: 2 : LC2

Mode	Freq. [Hz]	Damp ratio	Damp|coe	Sax [m/s²]	Say [m/s²]	Saz [m/s²]	G(j)	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.5253	0.0265	1.1432	0.2307	0.0000	0.0000	0.6993	0.2516	0.0000	0.0000	-2.5154
2	3.4262	0.0330	1.0979	0.4809	0.0000	0.0000	0.0187	0.1556	0.0000	0.0000	-0.4491
Level=	0.00							0.30	0.00	0.00	2.56

In the original example, a Damping Spectrum with a constant damping ratio of **2%** was used. Due to the inputted dampers, the calculated Composite Modal Damping Ratios of **2,64%** and **3,30%** are now used.

Using equation (4.13) the Damping Coefficients can be calculated:

$$\eta_1 = \sqrt{\frac{10}{(5 + 2{,}65)}} = 1{,}1432$$

$$\eta_2 = \sqrt{\frac{10}{(5 + 3{,}30)}} = 1{,}0976$$

As was the case in the original example, the damping ratios are lower than the default 5% used in the acceleration spectrum, they will have a negative effect thus augmenting the response of the structure.

Since the calculated damping ratios are higher than the original 2%, the response will be less when compared to the original example.

Second, the calculated Composite Modal Damping Ratios will be used for the calculation of the Modal Cross Correlation Coefficients of the CQC-method.

This will be illustrated in a manual calculation.

Step 6: manual calculation

In this paragraph, the application of the CQC-method using the calculated Composite Modal Damping Ratios is illustrated for the global response of the Base Shear.

Mode 1:

$$\omega_{(1)} = 3{,}3007\mathrm{rad/s}$$
$$F_{(1)} = 0{,}2701\mathrm{kN}$$

Mode 2:

$$\omega_{(2)} = 21{,}5192\mathrm{rad/s}$$
$$F_{(2)} = 0{,}1629\mathrm{kN}$$

Using a spreadsheet, the Modal Cross Correlation Coefficients $\boldsymbol{\rho_{i,j}}$ are calculated with a damping ratio $\boldsymbol{\xi_{i,j}}$ of 2,64% for the first eigenmode and 3,30% for the second eigenmode.

Mode	1	2
1	1	0,00055202
2	0,00055202	1

$$R_{tot} = \sqrt{\sum_{i=1}^{N}\sum_{j=1}^{N} R_{(i)}.\,\rho_{i,j}.\,R_{(j)}}$$

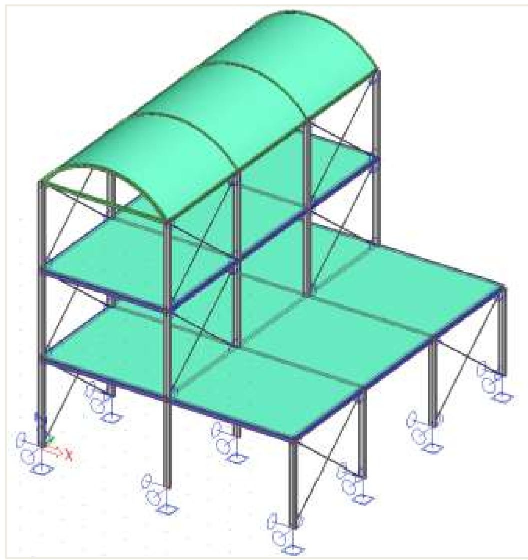
$$R_{tot} = \sqrt{\begin{matrix}(0{,}2701\mathrm{kN} * 1 * 0{,}2701\mathrm{kN}) + (0{,}2701\mathrm{kN} * 0{,}00055202 * 0{,}1629\mathrm{kN}) \\ +(0{,}1629\mathrm{kN} * 0{,}00055202 * 0{,}2701\mathrm{kN}) + (0{,}1629\mathrm{kN} * 1 * 0{,}1875\mathrm{kN})\end{matrix}}$$

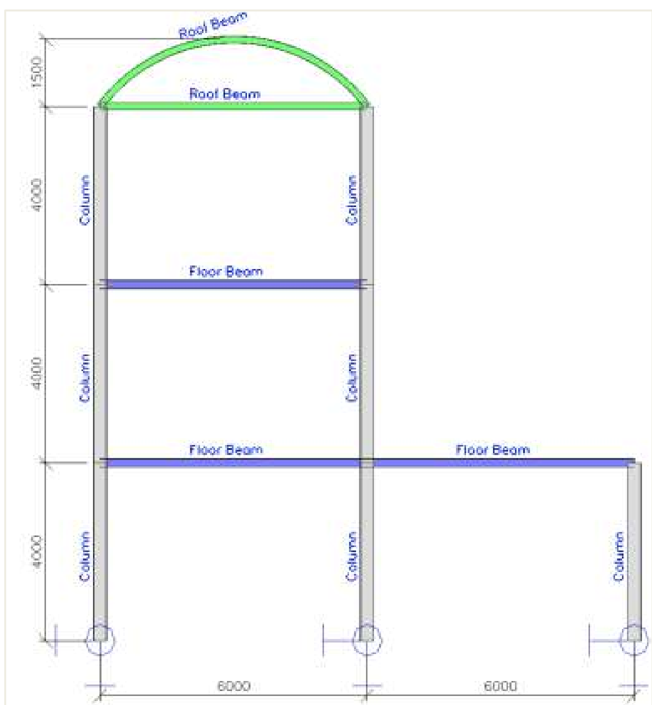
$$R_{tot} = 0{,}315\mathrm{kN}$$

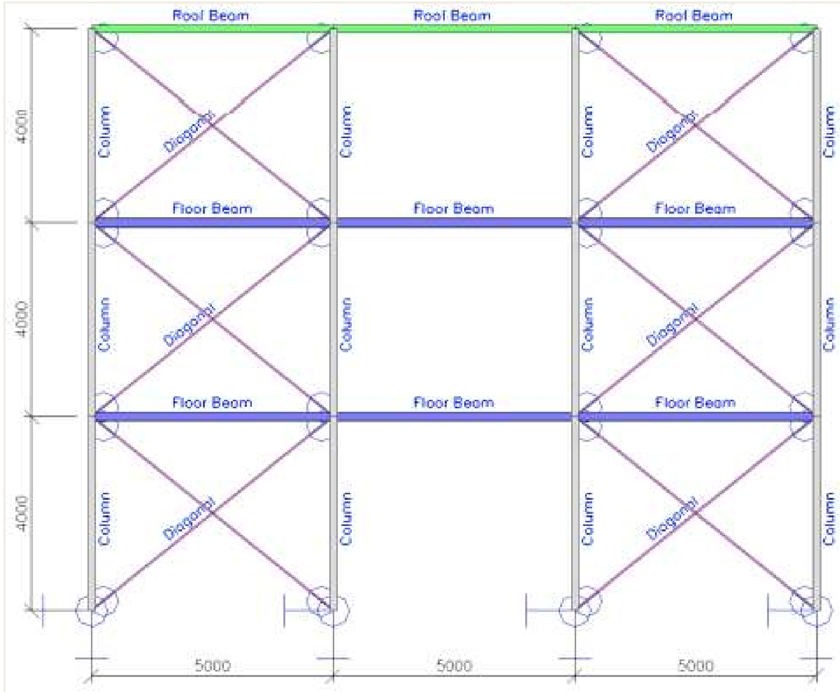
The difference between these Correlation Coefficients and the original is very small which was to be expected since the calculated damping ratios are close to the original 2%.

Example 11-4.esa

In this example, a seismic analysis is carried out on a storage depot. The layout of the structure is given in the pictures below. The depot is constructed with steel members manufactured of S235 according to EC-EN. On the upper roof, a steel shell is used with thickness 20 mm.
On each floor level, concrete slabs are used with thickness 200 mm. The slabs are manufactured in C25/30 according to EC-EN.







The diagonals are hinged in both directions. The column bases are also hinged though the anchors are spaced such that the rotation around the Z-axis is taken as fixed.

The steel members of the depot have following cross-sections:

- Columns: IPE300
- Floor beams: HE200A
- Roof beams: IPE160
- Diagonals: L(ARC) 40x40x4

The vertical loads acting on the structure are:

- Load case 1: the self-weight of the depot
- Load case 2: a category E (storage) imposed load of 5 kN/m² on all floor slabs.

The structure will be subjected to an earthquake loading in both X, Y and Z direction, using a Design Response Spectrum according to Eurocode 8 for Ground Type A with a behaviour factor of 1,5. This means that the spectrum for the internal forces will be divided by this value. The acceleration coefficient is 0,50.
For the dynamic calculation, the structural damping of the depot is taken into account. More specifically, a logarithmic decrement of 0,025 is used for steel and 0,056 for concrete [22].

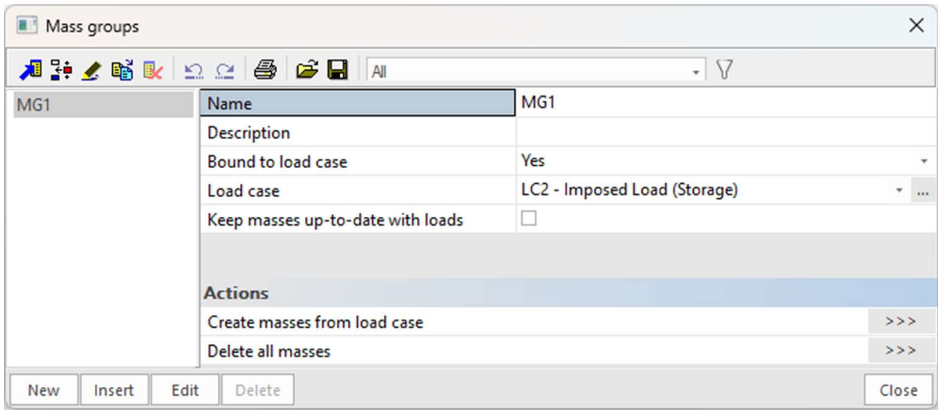
Step 1: functionality

The first step to take into account the damping is to activate the functionality Non-Proportional Damping on the Functionality tab in the Project Data.

Step 2: mass group and masses

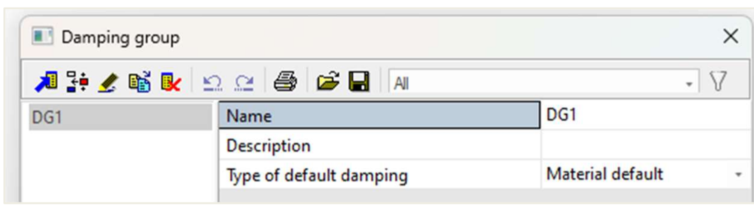
The second step is to create Mass Groups and then the creation of Masses.
Since the self-weight is automatically taken into account in a Combination of Mass Groups, only one Mass Group is created here, a group to take the mass of the imposed load into account.

Using the action “Create masses from load case” automatically generates masses from the already inputted loads.

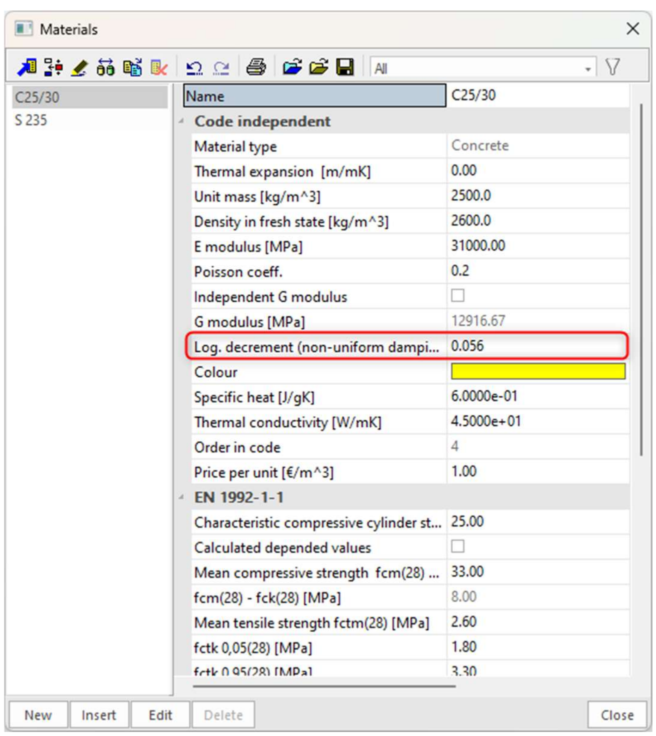


Step 3: damping groups

Before creating a **Combination of Mass Groups**, the damping is specified.
First of all, a **Damping Group** is created.



Since, in this example, the structural damping of the steel and concrete is taken into account, the *Type of default damping* is set to **Material default**. This way, when no damper is inputted on an element, the default damping value of the material will be used.
The damping values can be specified in the **Material** Library:



For the concrete, a logarithmic decrement of 0,056 is inputted, for the steel a value of 0,025.

Step 4: mass matrix

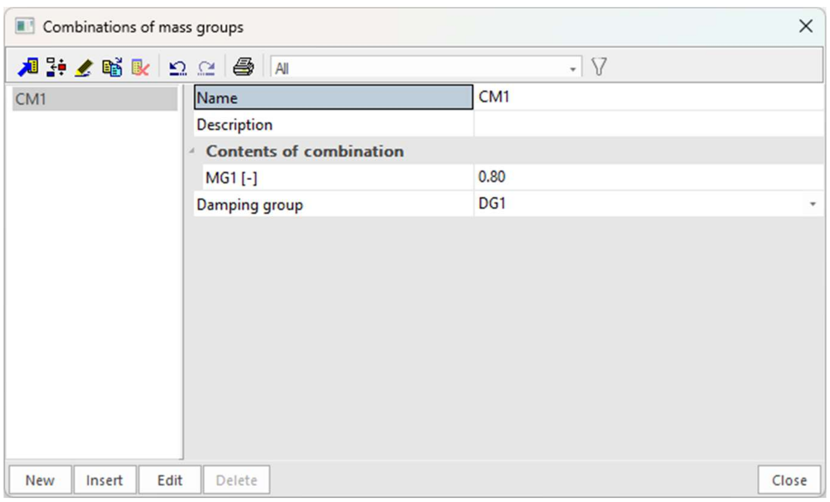
The Mass Group and Damping Group can now be combined in a Combination of Mass Groups.

As specified in formula (2.9) all gravity loads appearing in the following combination of actions need to be taken into account for an eigenmode calculation:

$$\sum G_k + \sum \psi_{E,i} . Q_{k,i}$$

For this example, with a Category E imposed load, φ is taken as 1,0 and $\psi_{2,i}$ as 0,8.
This gives a value of 0,8 for $\psi_{E,i}$

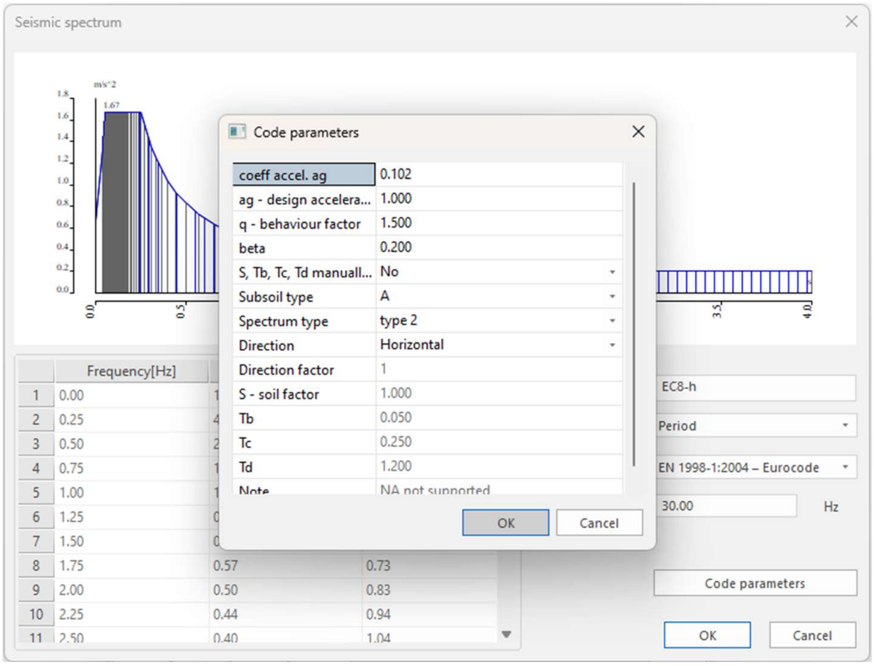
Since the self-weight is automatically taken into account, the Combination of Mass Groups CM1 can be formulated as 0,80 MG1:



As a final step, the Damping Group is assigned to the Combination of Mass Groups.

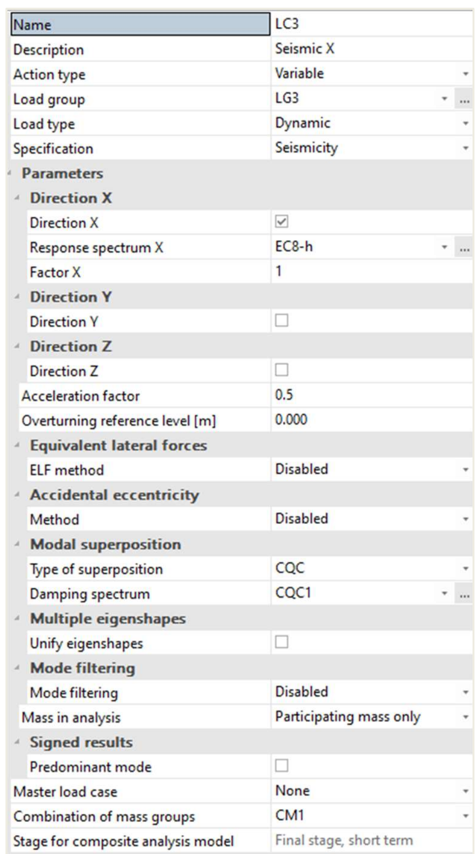
Step 5: seismic spectrum

Before creating the seismic load cases, the seismic spectrum needs to be defined through the main menu Library / Seismic spectrums.
Instead of inputting a spectrum manually, the spectrum according to EC8 is chosen. In this example, the spectrum for Ground Type A with a Behaviour Factor q = 1,5 is used for all directions:



Step 6: seismic load case

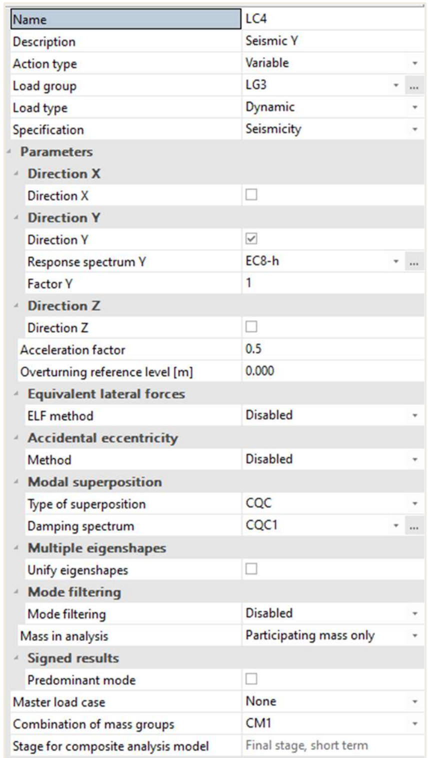
The **Seismic** load cases can now be defined through the workstation "**Load cases**", and "**Load Cases**". For the Seismic load case in the **X-direction**, the following parameters are used:

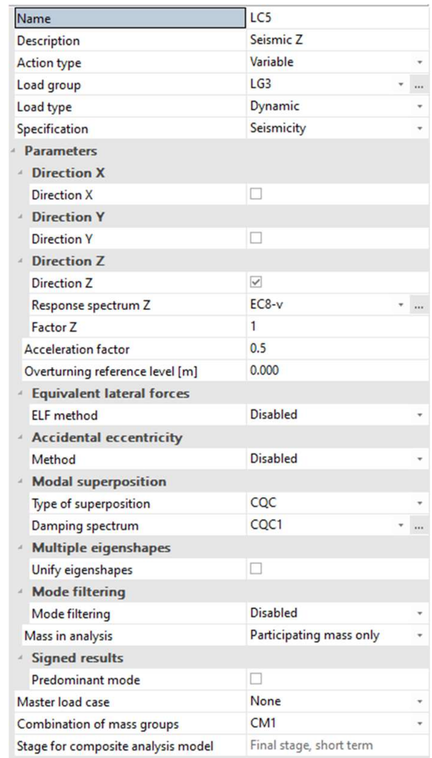


Name	LC3
Description	Seismic X
Action type	Variable
Load group	LG3
Load type	Dynamic
Specification	Seismicity
Parameters	
Direction X	
Direction X	☑
Response spectrum X	EC8-h
Factor X	1
Direction Y	
Direction Y	☐
Direction Z	
Direction Z	☐
Acceleration factor	0.5
Overturning reference level [m]	0.000
Equivalent lateral forces	
ELF method	Disabled
Accidental eccentricity	
Method	Disabled
Modal superposition	
Type of superposition	CQC
Damping spectrum	CQC1
Multiple eigenshapes	
Unify eigenshapes	☐
Mode filtering	
Mode filtering	Disabled
Mass in analysis	Participating mass only
Signed results	
Predominant mode	☐
Master load case	None
Combination of mass groups	CM1
Stage for composite analysis model	Final stage, short term

The **Coefficient of Acceleration** is set to **0,5**. As Type of evaluation the **CQC-method** is used.

In exactly the same way, the Seismic load cases in the Y and Z-direction are defined:





This steps have to be repeated for load cases that define the deformations (behaviour factor q is different).

NB:

For the load case Seismic Z a new spectrum has to be defined with type vertical.

Three other EN-Seismic load cases have to be defined, the first 3 are for internal forces and 3 new (with q-behaviour factor set to 1) for deformation. Each group of load cases has to get a load group with type "seismic" & "together" and they must be placed in separate combinations.

According to Eurocode 8 [7] the action effects due to the combination of the horizontal components of the seismic action may be computed using the following combinations:

$$\mathrm{E_{Edx}}\text{"+"}0{,}3.\mathrm{E_{Edy}}\text{" + "}0{,}3.\mathrm{E_{Edz}}$$
$$0{,}3.\mathrm{E_{Edx}}\text{"+"}\mathrm{E_{Edy}}\text{" + "}0{,}3.\mathrm{E_{Edz}}$$
$$0{,}3.\mathrm{E_{Edx}}\text{"+"}0{,}3.\mathrm{E_{Edy}}\text{" + "}\mathrm{E_{Edz}}$$

Where:

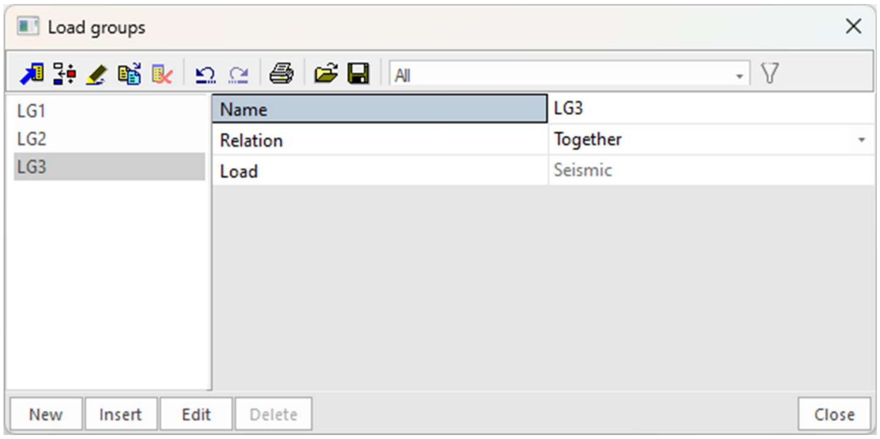
« + » implies « to be combined with ».

E_{Edx} represents the action effects due to the application of the seismic action along the chosen horizontal axis x of the structure.

E_{Edy} represents the action effects due to the application of the seismic action along the chosen horizontal axis y of the structure.

E_{Edz} represents the action effects due to the application of the seismic action along the chosen horizontal axis z of the structure.

First of all, this implies that all Load cases must always be considered together in a combination. In SCIA Engineer this can be done by putting both Seismic Load cases in a **Load Group** with relation **Together**.

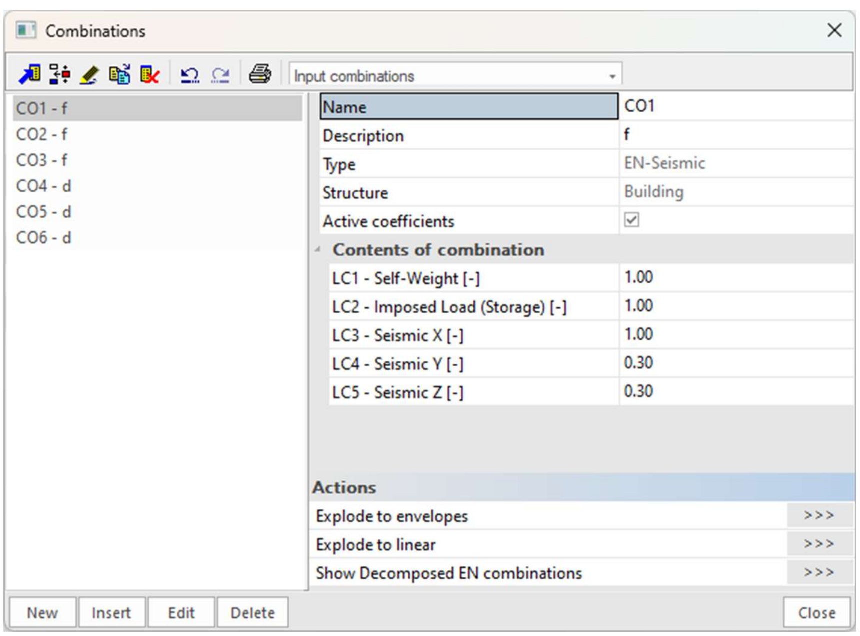


Next, the combination for the Seismic calculation can be inputted. According to Eurocode 8 [7] this combination is the following:

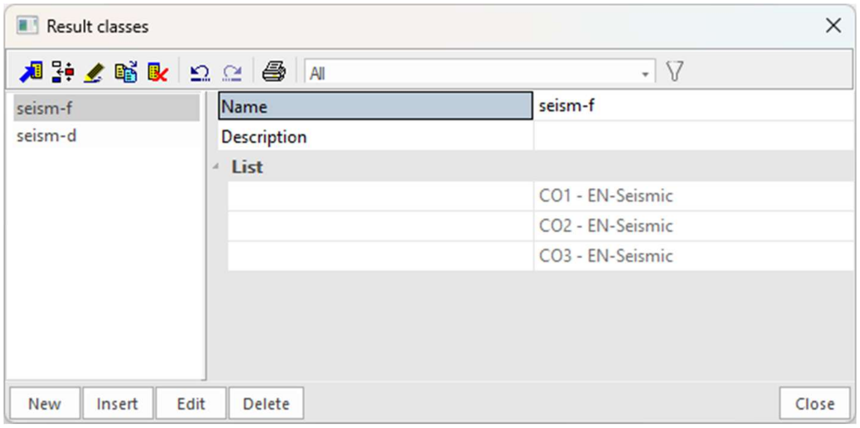
$$\sum G_k + P + A_{Ed} + \sum \psi_{2,i}.Q_{k,i} \tag{5.22}$$

Where A_{ed} represents the accidental action, which is in this case the combined seismic action.

In SCIA Engineer, the EN-seismic type can be used for this purpose.
To fulfil the conditions of the Eurocode, 6 load combinations of this type are created:

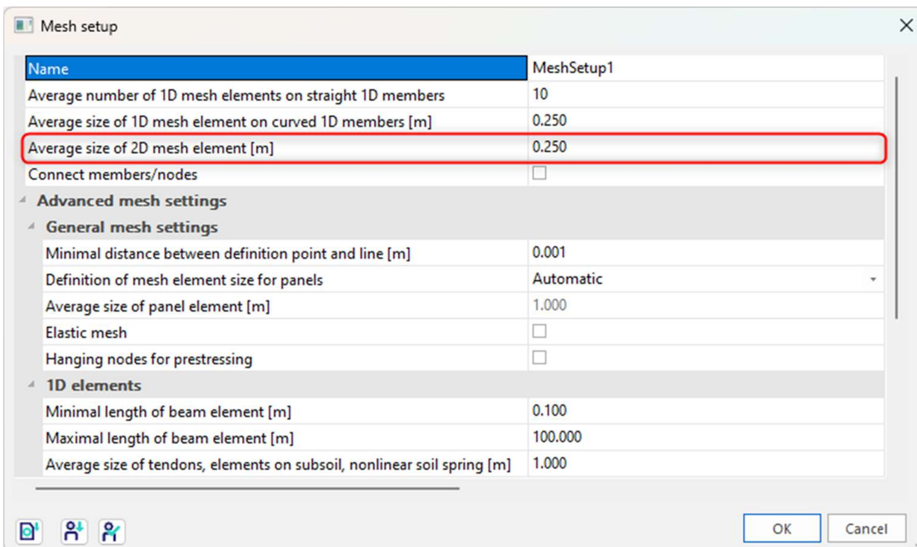


To be able to see the global extremum for the two combinations, two **Results classes** can be used:



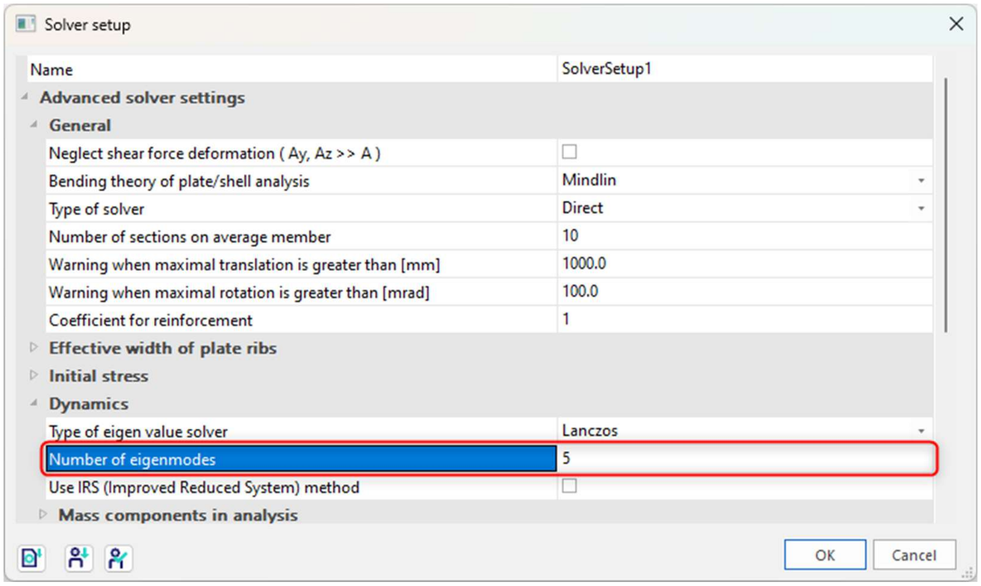
Step 7: mesh setup

To obtain precise results, the Finite Element Mesh is refined through the main menu Tools / Calculation & Mesh / Mesh Settings. The Average number of tiles of 1D element is set to 10; the Average size of 2D element is set to 0,25m.



Step 8: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. For this example, five eigenmodes are chosen.
In the main menu Tools / Calculation & Mesh / Solver Settings, the number of frequencies is thus set to 5.



Step 9: linear calculation and calculation protocol

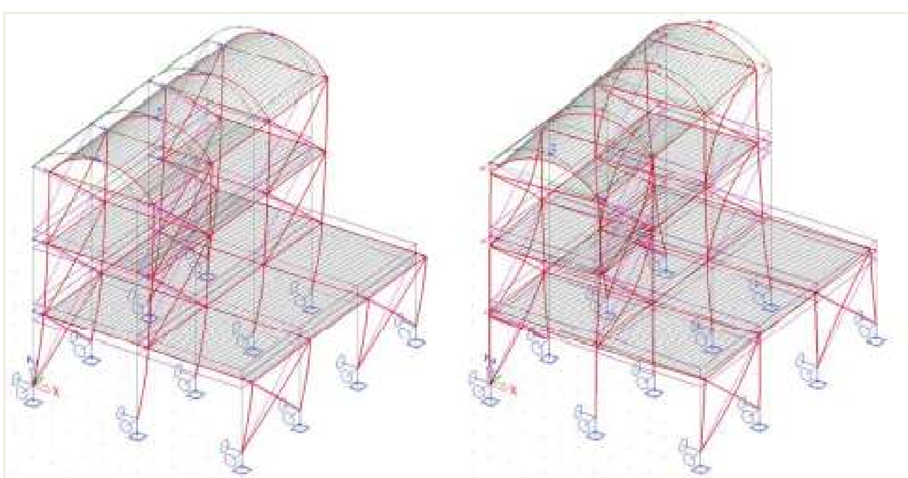
All steps have been executed so the Linear Calculation can be started through the main menu Tools / Calculation & Mesh / Calculate.

The Calculation Protocol for the Eigen Frequency calculation shows the following:

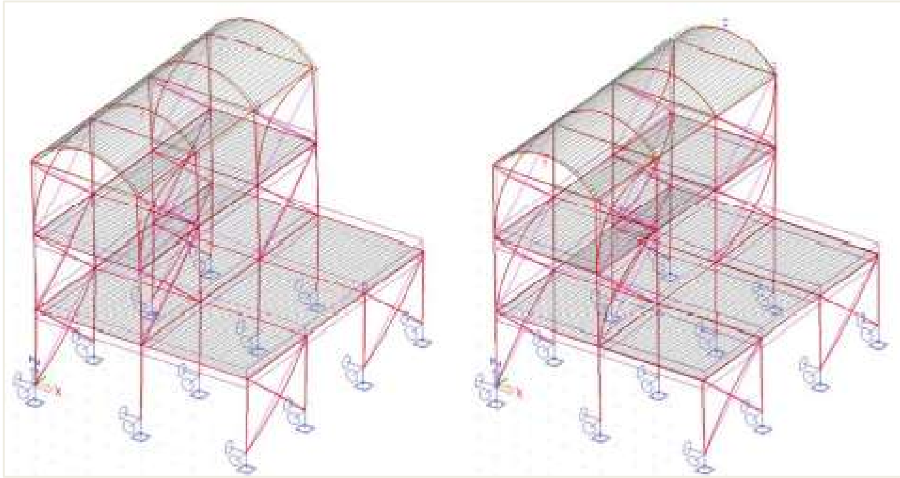
Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Wxi / Wxtot	Wyi / Wytot	Wzi / Wztot	Wxi_R / Wxtot_R	Wyi_R / Wytot_R	Wzi_R / Wztot_R	Damp ratio
1	4.5955	1.3672	0.7314	0.9686	0.0000	0.0000	0.0000	0.0103	0.0000	0.0081
2	11.4189	0.5502	1.8174	0.0000	0.6682	0.0000	0.0190	0.0000	0.2860	0.0080
3	13.6430	0.4605	2.1713	0.0271	0.0000	0.0002	0.0000	0.3694	0.0000	0.0081
4	13.8204	0.4546	2.1996	0.0000	0.2709	0.0000	0.0003	0.0000	0.5272	0.0080
5	14.9368	0.4207	2.3773	0.0000	0.0000	0.0000	0.0000	0.0000	0.0004	0.0080
				0.9957	0.9391	0.0002	0.0193	0.3797	0.8136	

It can be seen that for both horizontal directions more than 90% of the total mass is taken into account in these five modes so it is concluded that sufficient Eigenmodes have been calculated.

Through Deformation of nodes under 2D Members, the Deformed Mesh can be used to visualize the first four Eigenmodes:



Eigenmode 1: f = 0,73Hz Eigenmode 2: f = 1,82Hz



Eigenmode 3: f = 2,17Hz   Eigenmode 4: f = 2,20Hz

The Calculation Protocol for the Linear calculation shows the results of the seismic calculation:

Dynamic loadcase: 3 : LC3

Mode	Freq. [Hz]	Damp ratio	Damp|coe	Sax [m/s²]	Say [m/s²]	Saz [m/s²]	G(j)	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.7314	0.0081	1.3114	0.1809	0.0000	0.0000	4.4128	48.0082	0.0000	-0.0000	-302.1145
2	1.8174	0.0080	1.3134	0.4978	0.0000	0.0000	0.0000	0.0000	-0.0000	0.0000	-0.0000
3	2.1713	0.0081	1.3123	0.5940	0.0000	0.0000	-0.2748	4.4051	0.0000	-0.0000	42.6000
4	2.1996	0.0080	1.3131	0.6021	0.0000	0.0000	0.0000	0.0000	0.0000	-0.0000	-0.0000
5	2.3773	0.0080	1.3135	0.6508	0.0000	0.0000	0.0000	0.0000	0.0000	-0.0000	-0.0000
Level=	0.00							48.21	0.00	0.00	305.10

Dynamic loadcase: 4 : LC4

Mode	Freq. [Hz]	Damp ratio	Damp|coe	Sax [m/s²]	Say [m/s²]	Saz [m/s²]	G(j)	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.7314	0.0081	1.3114	0.0000	0.1809	0.0000	0.0000	0.0000	0.0000	-0.0000	-0.0000
2	1.8174	0.0080	1.3134	0.0000	0.4978	0.0000	-1.6335	-0.0000	91.1291	-609.0984	0.0000
3	2.1713	0.0081	1.3123	0.0000	0.5940	0.0000	-0.0000	0.0000	0.0000	-0.0000	0.0000
4	2.1996	0.0080	1.3131	0.0000	0.6021	0.0000	0.8587	0.0000	44.6772	-254.8292	-0.0000
5	2.3773	0.0080	1.3135	0.0000	0.6508	0.0000	0.0100	0.0000	0.0076	-0.0387	-0.0000
Level=	0.00							0.00	101.77	661.88	0.00

For each Eigenmode the Composite Damping Ratio has been calculated using the structural damping of the steel and concrete.

The combinations can now be used to verify the structural elements.

CHAPTER 13 : DIRECT TIME INTEGRATION

13.1 Theory

The title may be misleading because normally in the literature, this name is used for a dynamic computation without modal superposition. In SCIA Engineer, the eigenmodes are determined first and are used to uncouple the equilibrium equations into a set of m uncoupled second order differential equations which are solved one by one by direct time integration. The uncoupling is based on the properties given by equations.

$$\Phi_j^T . M . \Phi_i = 0 \qquad \text{si } i \neq j$$

$$\Phi_j^T . M . \Phi_i = 1 \qquad \text{si } i = j$$

$$\Phi_j^T . M . \Phi_i = \omega_i^2$$

In equation (3.1), a solution for y is assumed to be of the form:

$$y = \phi . Q \tag{7.1}$$

Where ϕ is the matrix of eigenvectors (n*n) and Q is a vector which is time dependant.

Substitution in equation (3.1) gives:

$$M . \phi . \ddot{Q} + C . \phi . \dot{Q} + K . \phi . Q = F \tag{7.2}$$

When the equation is pre-multiplied with ϕ^T and the above equations are taken into account, one obtains:

$$\ddot{Q} + \phi^T . C . \phi . \dot{Q} + \Omega^2 . Q = \phi^T . F \tag{7.3}$$

This set of equations is still coupled because of the damping term. If however C-orthogonality is assumed (this means that $\phi^T . C . \phi$ reduces to only diagonal terms), then the equations are uncoupled and can be solved separately. The global results are obtained by superposition of the individual results (7.1) is also the exact solution if the assumption of C-orthogonality holds. If however, only a few eigenvectors (m<n) are used in ϕ instead of all the eigenvectors, then the system of equations and the superposition of the solutions gives a solution y which is an approximation of the exact solution.

In SCIA Engineer, C-orthogonality is assumed and it is also assumed that all modal damping factors are constant. This means that:

$$\phi^T . C . \phi = 2 . \omega_i . \xi . \delta_{ij} \tag{7.4}$$

The value of ξ is one of the input data and is called damping factor.

The number of eigenvectors that is taken into account is also specified by the user. This value is equal to the number of eigenvectors computed in the eigenvalue computation.
The method used to solve each uncoupled second order differential equation is the *Newmark-method*. This method is unconditionally stable but the accuracy depends on the time step. This time step has to be given by the user. However, to help him in his choice, a value determined by the program will be used if the user does not specify a value. This proposed value is computed as: 0,01 T
Where T smallest period of all the modes which have to be taken into account.

This proposed value guarantees accuracy better than 1% over each period of integration of this highest mode. In most cases, a larger time step can be used because the contribution of this last mode is small.
This brings us to the question about the number of modes that should be used. When the time dependent terms on the left hand side of equation (7.3) are neglected, the solution for q_j (a term of Q) is:

$$q_j = \frac{1}{\omega_j^2} . \phi_j^T . F \tag{7.5}$$

This indicates that the lowest eigenmodes (ω_j small) will contribute more than the highest modes (ω_j large), if dynamic terms are neglected. This can give a first idea on how many modes to use.

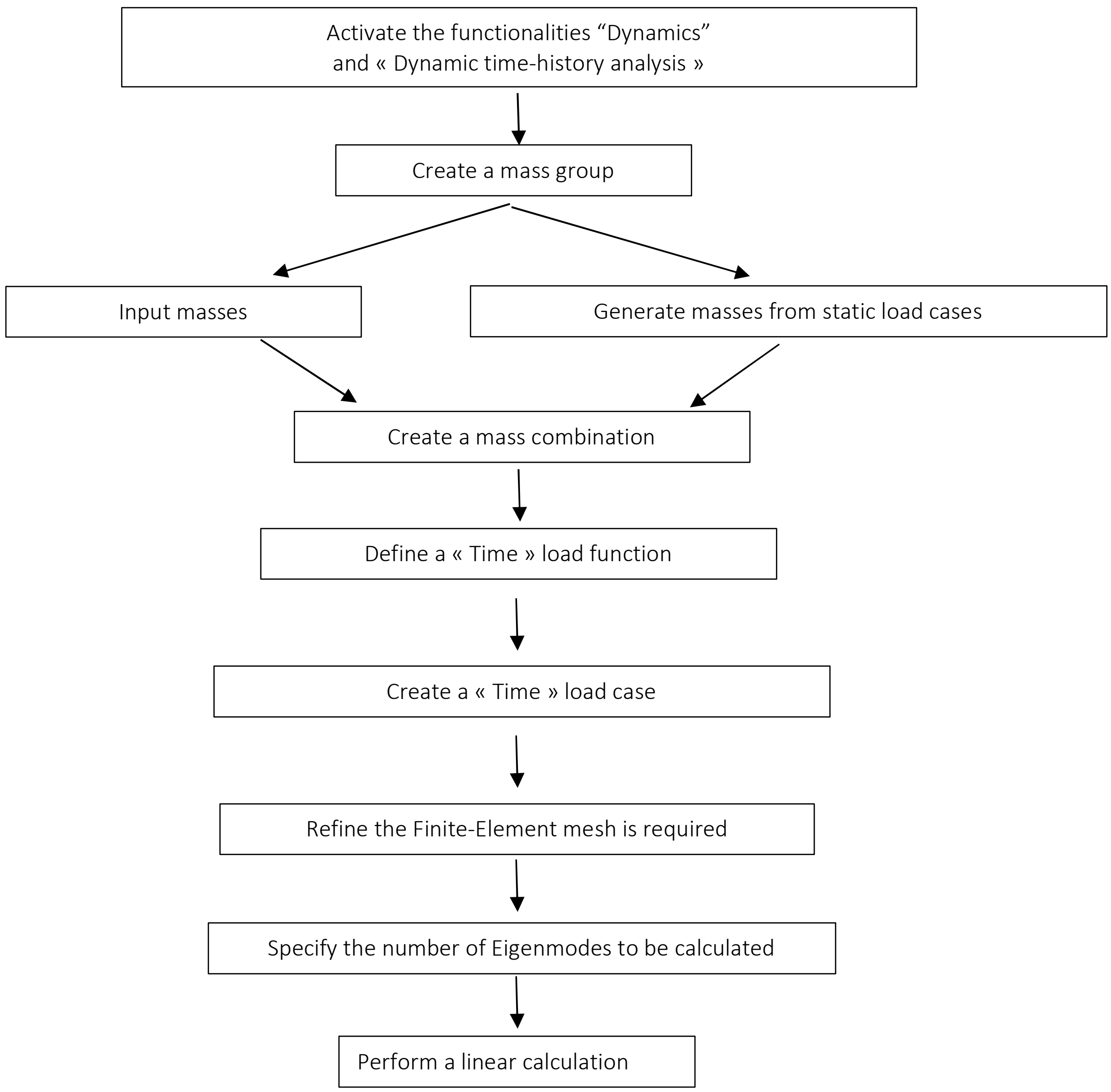
A second criterion is the periodicity of F. Any mode which coincides with the loading frequency should be taken into account.

Modal weight is a third criterion that can be used. If you add all modal weights in a particular direction together and divides this result by 9.81*sum of nodal masses in the same direction, you obtain a value smaller than 1. If this value is close to 1, it means that the higher modes will not contribute anymore. If, on the contrary, the value is smaller than 0,9, one can doubt about the value of a subsequent modal superposition.

13.2 Direct-time integration in SCIA Engineer

In SCIA Engineer, it's possible to input a dynamic function. This can be used for different purposes, for example: harmonic loads, explosions, … In this case, the user has to input a dynamic function which presents the frequency in function of the time.

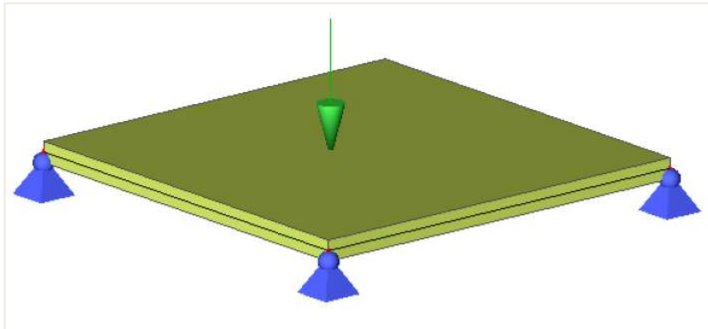
The following diagram shows the different steps which have to be performed for the time history calculation:



This functionality is only available in 32-bit version of SCIA Engineer!

Example 12-1.esa

In this example an explosion is simulated on a concrete plate.



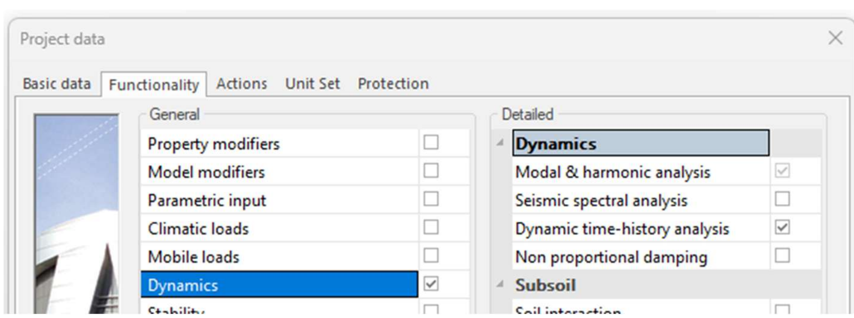
The plate has a dimension of 6x6 m and the thickness is 300 mm. The plate will be calculated according to the EC-EN and is made of concrete grade C30/37. The four corners are supported by hinged supports.

Three load cases are introduced:

- Self-weight
- Permanent surface load: -4 kN/m²
- Variable point load: blast of -11 kN

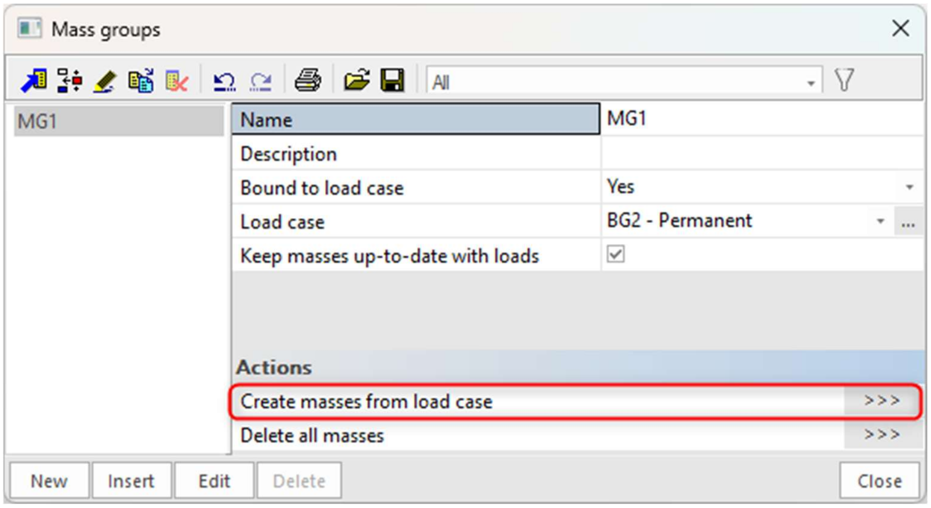
Step 1: functionality

In the “Project settings”, activate the options « Dynamics » and « Dynamic time-history analysis »:



Step 2: mass groups and masses

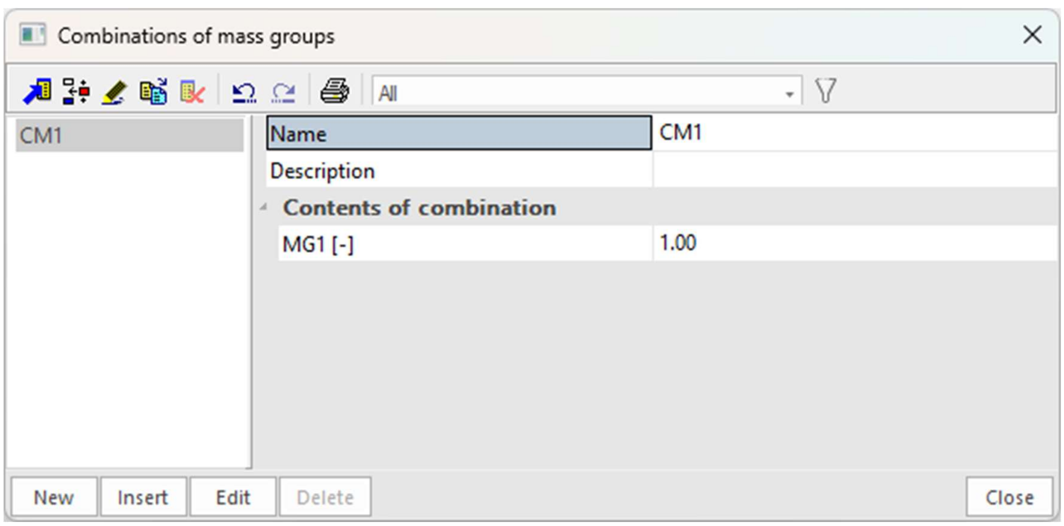
Open the menu ‘Dynamics’ and a mass group will be created here. For this, the permanent surface load of -4 kN/m² is used. For this, you can click on the ‘create masses from load case’ button.



A surface mass of 407,7 kg/m² is created.

Step 3: mass matrix

Next, a combination of mass groups can be created:

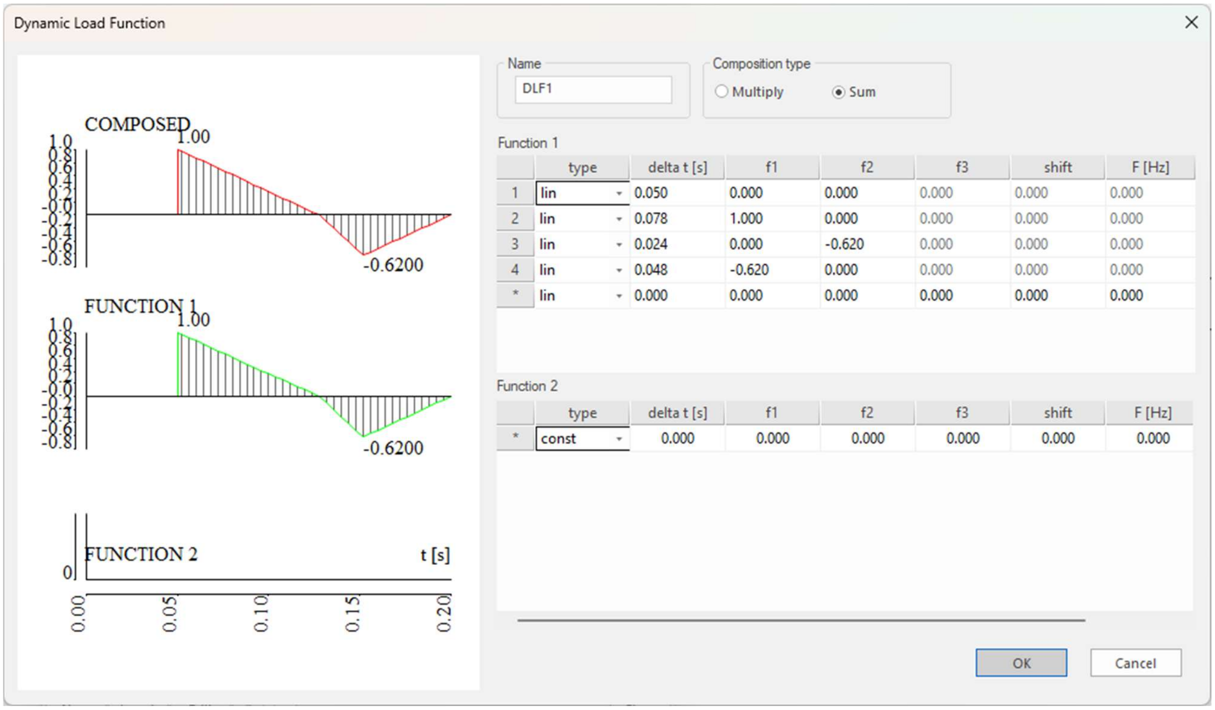


Step 4: dynamic load function

After the creation of masses, the explosion can be simulated by means of a dynamic load function.

Go to 'Libraries > Loads > Dynamic load functions'.
Here you can input the input of load coefficients in function in time.
Two types of functions can be input, namely a base and/or modal function. If both are introduced, the user can choose if these functions have to be multiplied or summarized.
4 types of functions can be chosen: constant, linear, parabolic or sinusoidal.

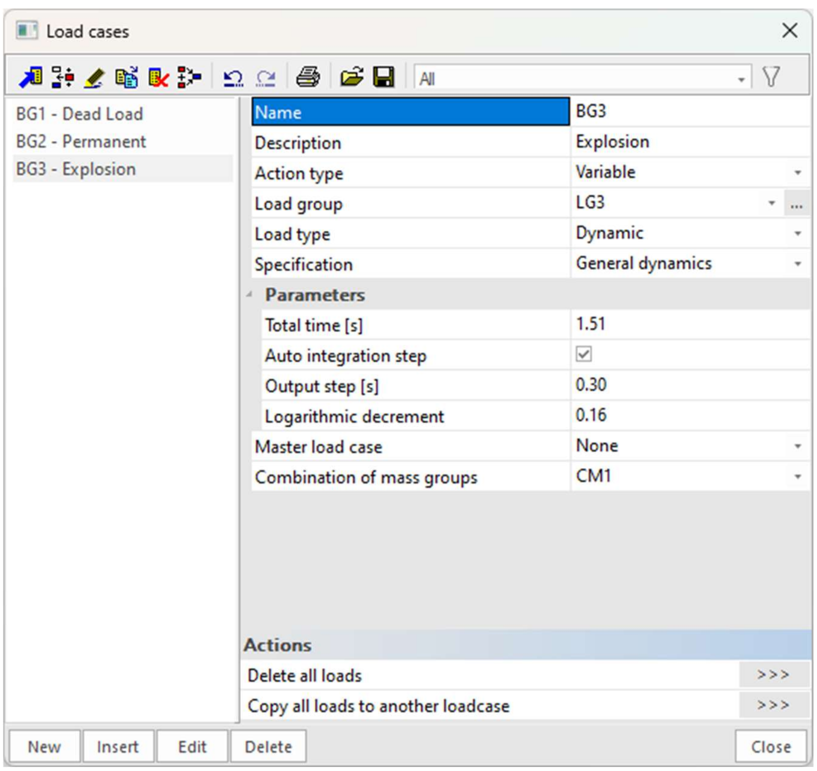
In our example a modal function is created with linear lines:



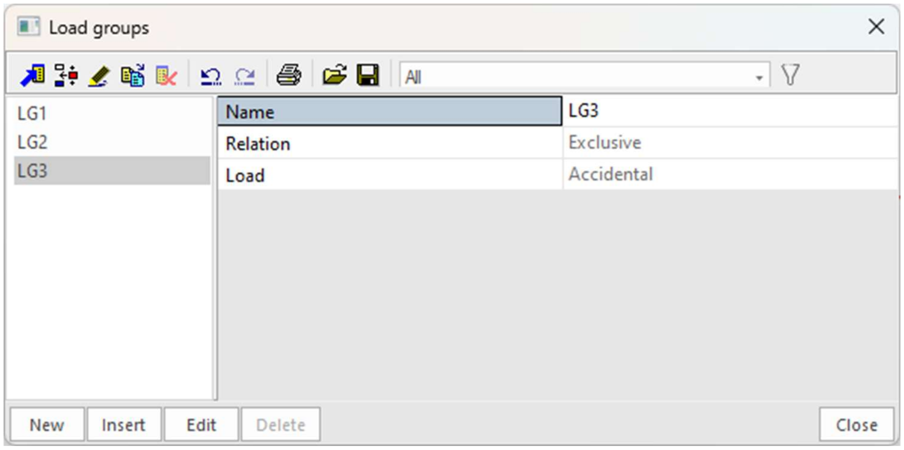
This function has to be attributed to a point load. We will do this in step 6.

Step 5: a "general dynamics" load case

A load case is introduced to simulate this explosion.
The action type is « Variable » and the type of load « Dynamic ».



For the load group, the user can choose a special case, namely « Accidental »:



Next, the specification has to be selected and the type **General dynamics** has to be chosen for a time history calculation.

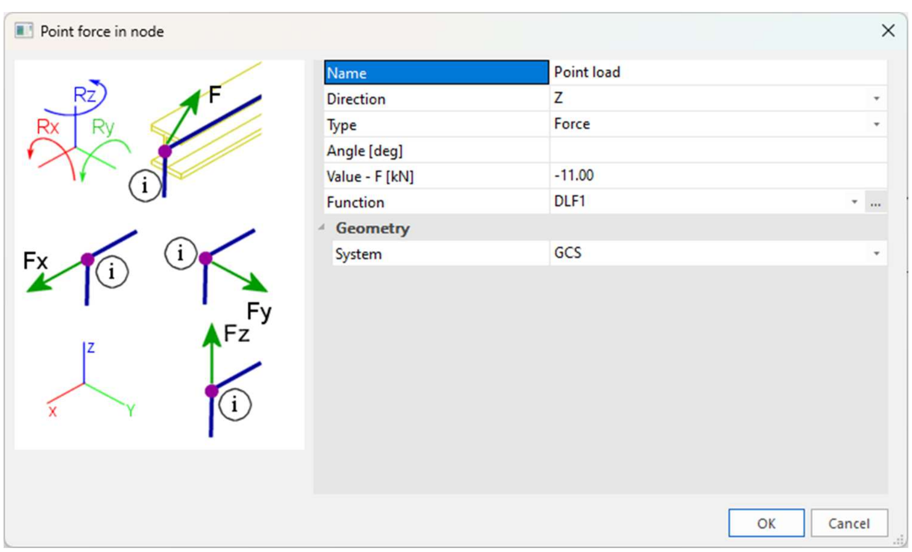
For this, we need some extra parameters:

- « Total time » : The total time of the dynamic analysis.
- « Integration step » : When “Auto” is checked, then 1/100 of the smallest period is taken. When “Auto” is not checked, then the user is allowed to select an integration step value.
- « Output step » : Step for generating the load cases. The value need to be bigger or equal at the integration step.
- « Log Decrement » : Damping defined as logarithmic decrement.

Step 6: input of loads which follow the load combination

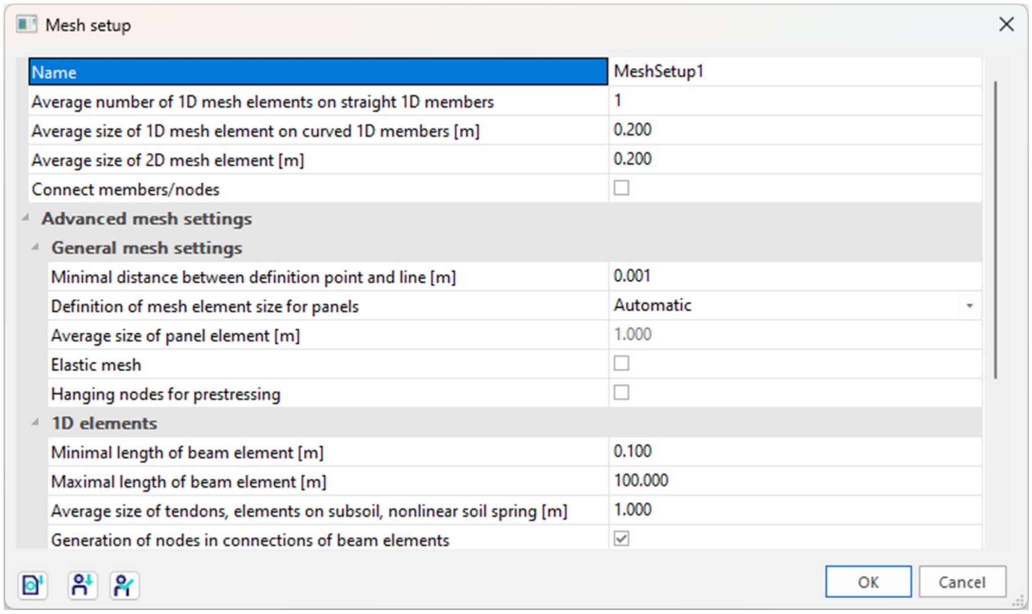
In this step, you will create of a nodal force. Only nodal forces can be linked to a dynamic function.
The value of the nodal force, will be multiplied with the coefficients in the function to achieve the final force in function of time.

A point force of -11 kN is input in the middle of the plate. The user has the option to attribute the dynamic function DLF1 to this load.



Step 7: mesh setup

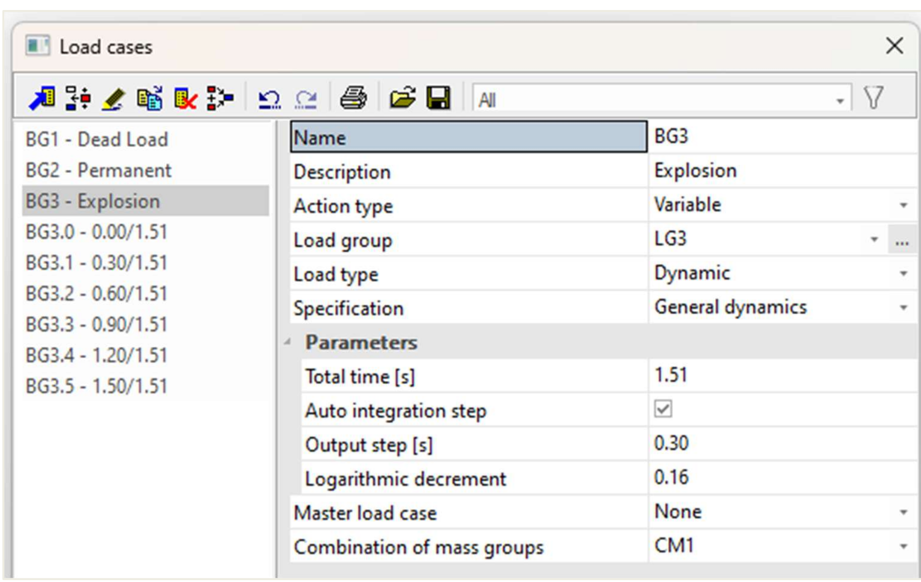
Before the calculation, the mesh is refined to get precise results.



Step 8: linear calculation

Now, the linear calculation can be performed.

When the calculation is finished, new load cases are created which present the influence of the blast on the structure on each output step (the output time must always be smaller than ‘Total time’, so in this example, we used 1,51 s as total time to get an output at 1,50 s):



To find the most extreme result, there load cases can be input in a Result class.

Step 9: results

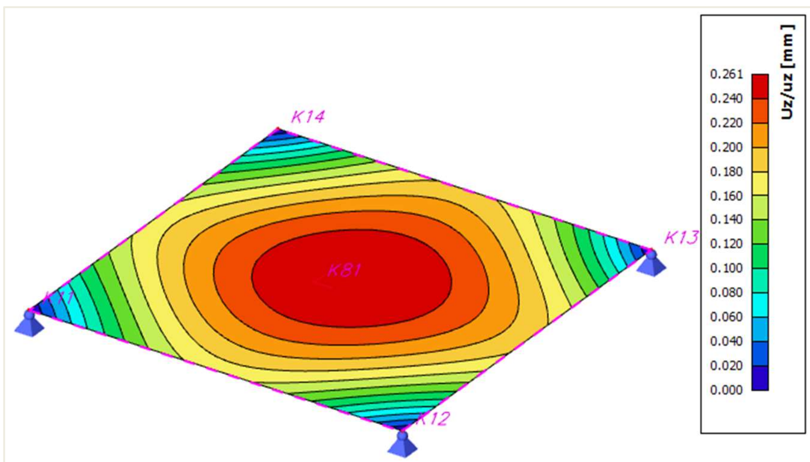
The eigen frequencies are shown in the “Results” menu:

Eigen frequencies

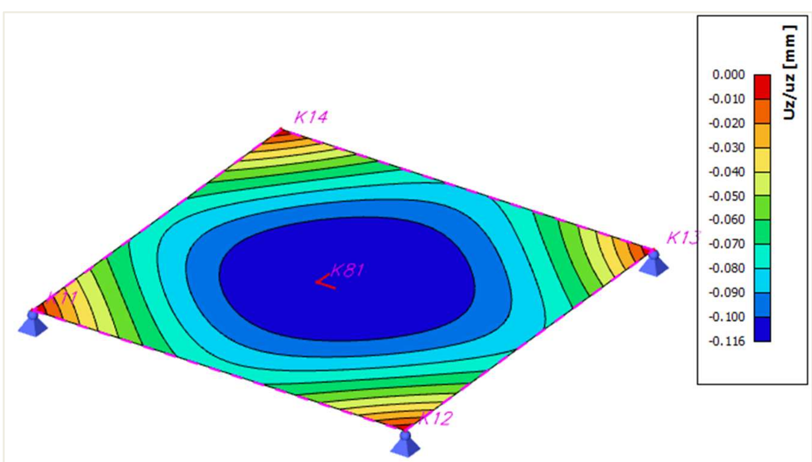
N	f [Hz]	ω [1/s]	ω^2 [1/s^2]	T [s]
Mass combination : CM1				
1	7.96	50.00	2500.06	0.13
2	17.98	112.95	12757.03	0.06
3	17.98	112.95	12757.07	0.06

Other results, like for example deformations, can be regarded for the different output steps:

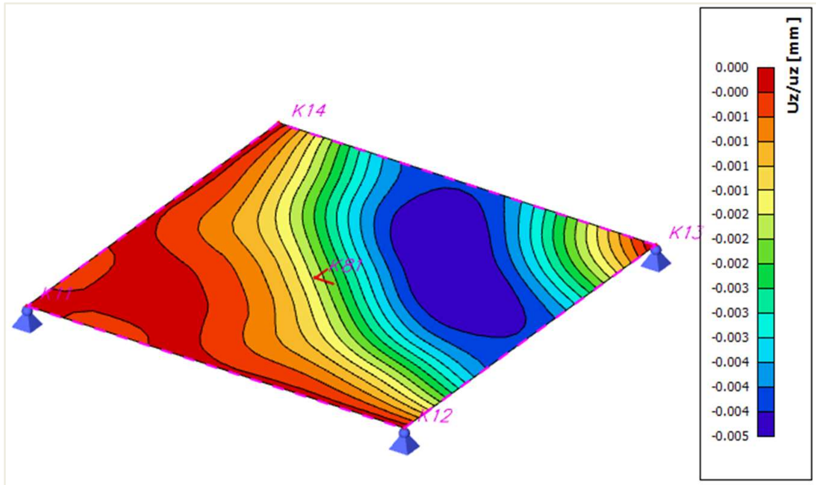
- After 0,3 seconds:



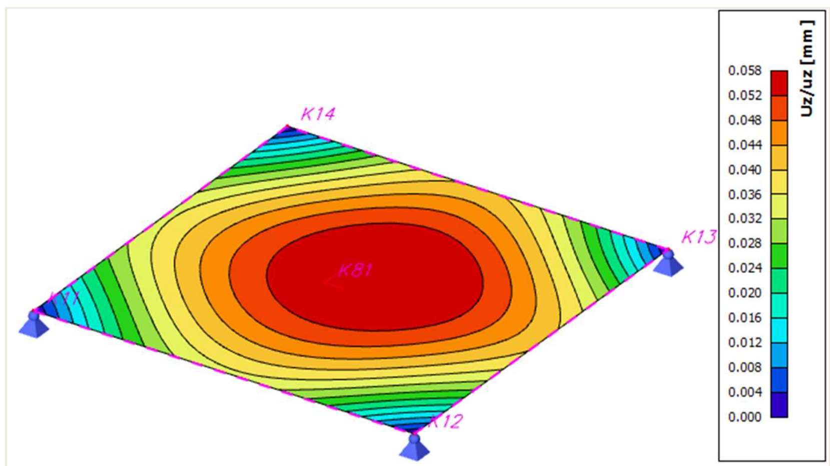
- After 0,6 seconds:



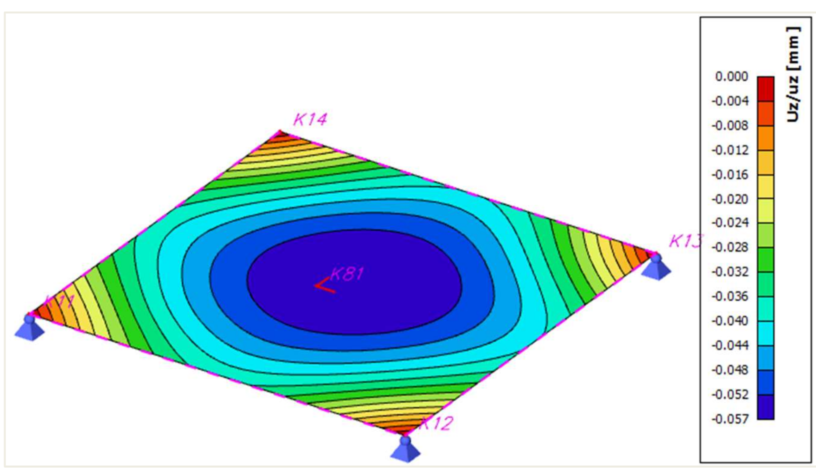
- After 0,9 seconds:



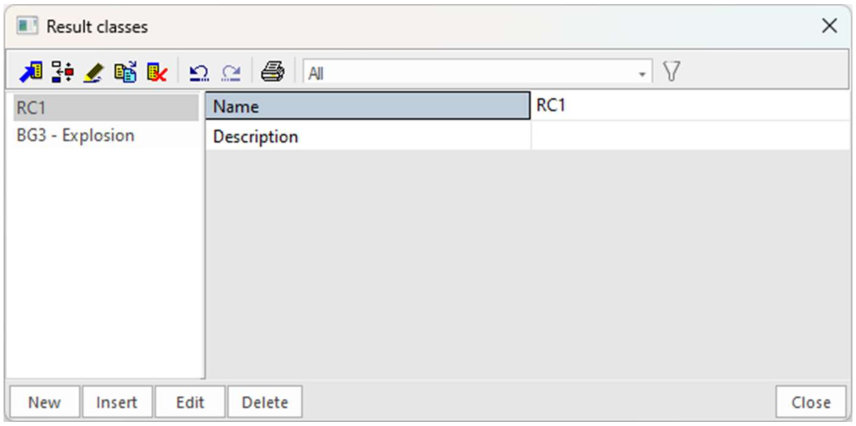
- After 1,2 seconds:

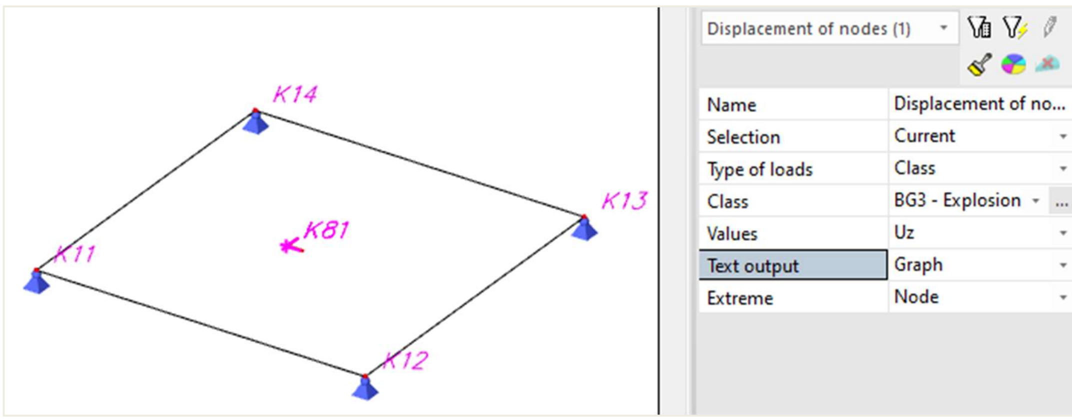


- After 1,5 seconds:

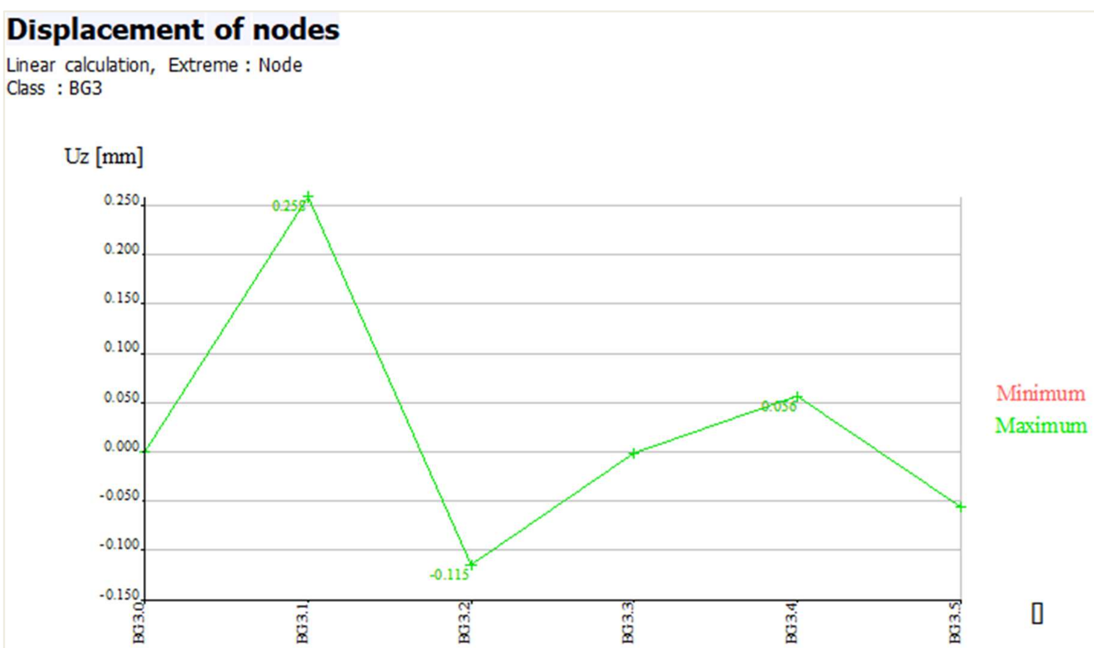


Or we can ask the result for the class which has been generated for the load cases.

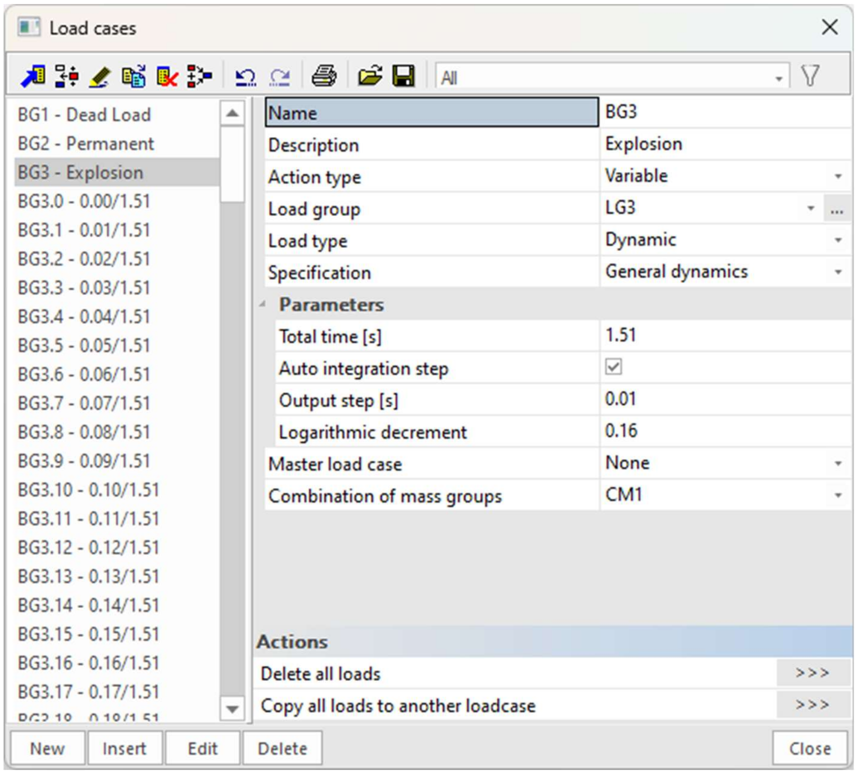




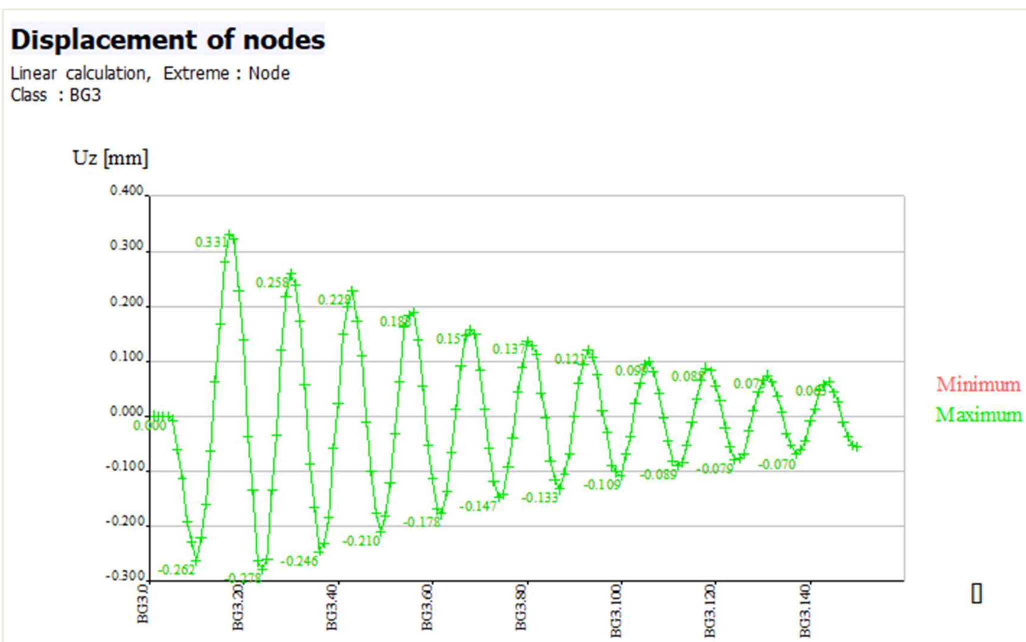
If you choose refresh, then you can see Uz for each 0,3 seconds in the selected node.



If we would set the output step to 0,01 s in the dynamics load case, then you would get 150 load cases.



And as a result, the "deformation in nodes" graph would give more detailed representation:



Example 12-2.esa

In this example a running load over a bar is simulated:

The beam has a length of 20 m and a section HE200A. The beam will be calculated according to the EC-EN and is made of steel S235. The edges are supported by hinged supports.

Two load cases are introduced:

- Self-weight
- Variable dynamic load: point loads of -100 kN on every 2 m over the beam

Step 1: functionality

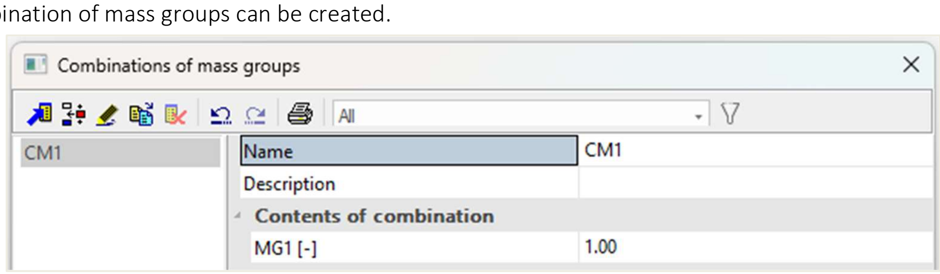
In the “Project settings”, activate the options « Dynamics » and « Dynamic time-history analysis ».

Step 2: mass groups

Open the menu ‘Dynamics’ and a mass group will be created here. For this, no mass is inputted. Only the self-weight is taken into account.

Step 3: mass matrix

Next, a combination of mass groups can be created.



Step 4: dynamic load functions

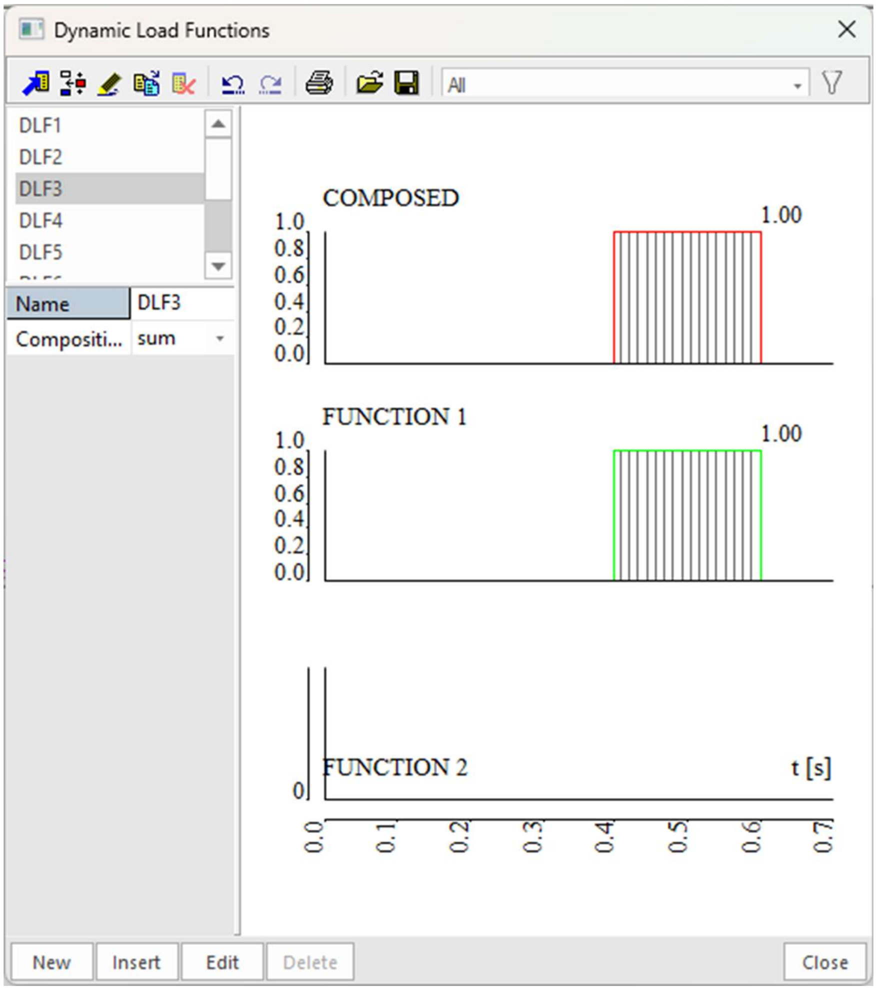
After the creation of masses, the running load can be simulated by means of dynamic load functions.

In the menu Library / Loads / Dynamic load functions, the input of frequencies in function in time is asked.

Two types of functions can be input, namely a base and/or modal function. If both are introduced, the user can choose if these functions have to be multiplied or summarized.
4 types of functions can be chosen: constant, linear, parabolic or sinusoidal.

In this example 9 modal functions are created with linear lines:

- DLF1 is 1,00 from 0,2s to 0,4s
- DLF2 is 1,00 from 0,4s to 0,6s
- …



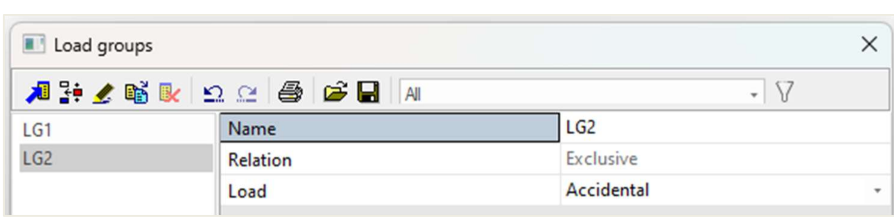
Each function will be attributed to a different point load (cf step 6):

- DLF1 to the first point load from the left.
- DLF2 to the second point load from the left.
- …

These 9 load functions will be used to simulate the effect of a point load moving from left to right over a time period is simulated. On each point (every 2m) the point load stays for a time of 0.20sec. So it takes 2 seconds for the point load to cross the whole beam.

Step 5: general dynamics load case

A load case is introduced to simulate this running load.
The action type is Variable and the type of load Dynamic
For the load group, the user can choose a special case, namely Accidental.



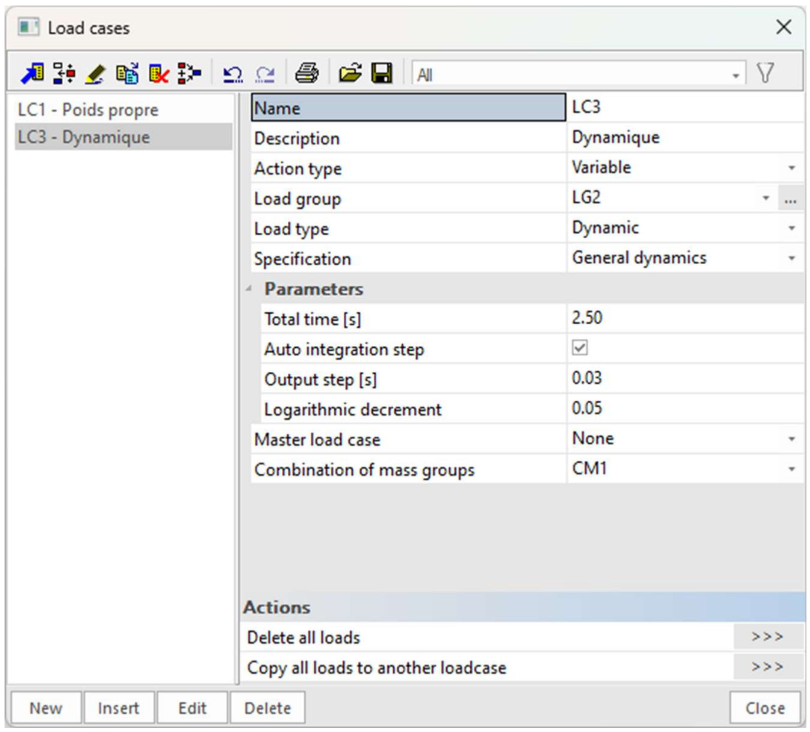
Next, the specification has to be selected and the type **General dynamics** has to be chosen for a time history calculation. After choosing general dynamics, some extra parameters have to be defined.

- **Total time [s]**: The total time of the dynamic analysis.

- **Integration step**: When "Auto" is checked, then 1/100 of the smallest period is taken. When "Auto" isn't checked, then the user is allowed to select an integration step value.

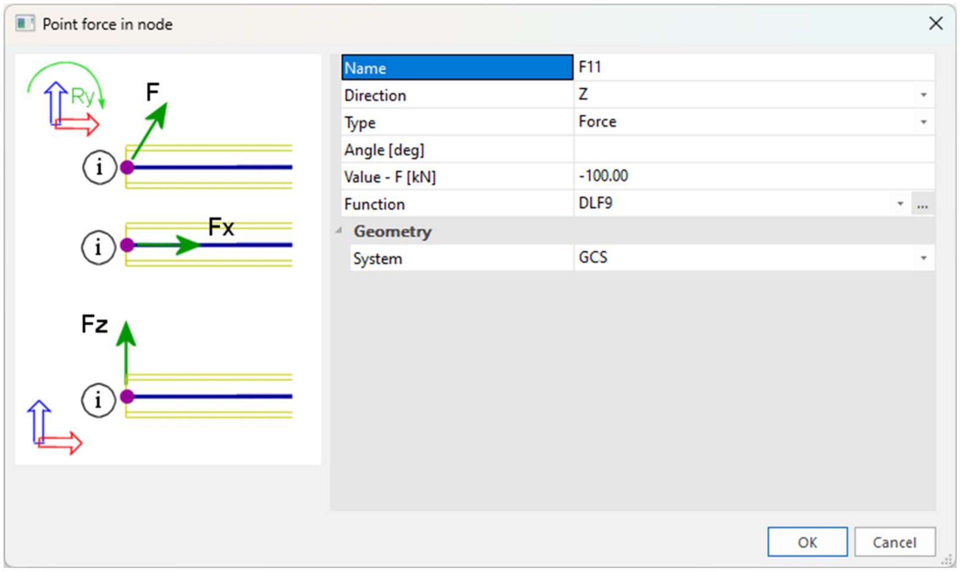
- **Output step [s]**: The step is used to determine on which points in time results must be generated. These will be saved in new generated load cases.

- **Log Decrement**: Damping defined as logarithmic decrement.



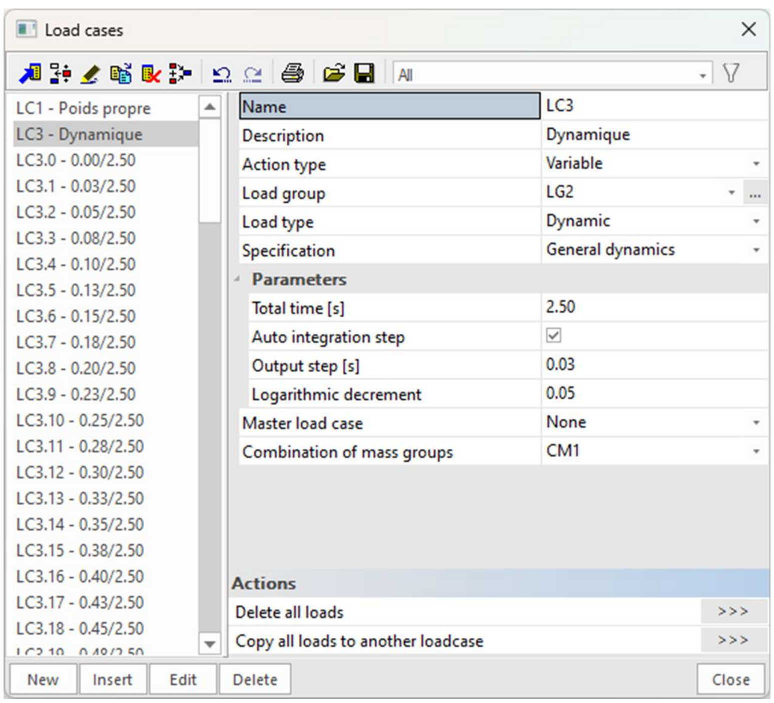
Step 6: input of loads

In this step, nodal forces will be inputted. Dynamic load functions can only be linked to nodal forces. Since they are 'nodal' forces, the user must provide internal nodes to place these internal forces on. Every 2 m an internal node has to be created on the beam. On each of these nodes a point force of -100 kN is set. The first point force from the left is linked to DLF1, the second to DLF2,...
This models the movement a single point load over the beam left to right over the beam in a total time of 2 sec.

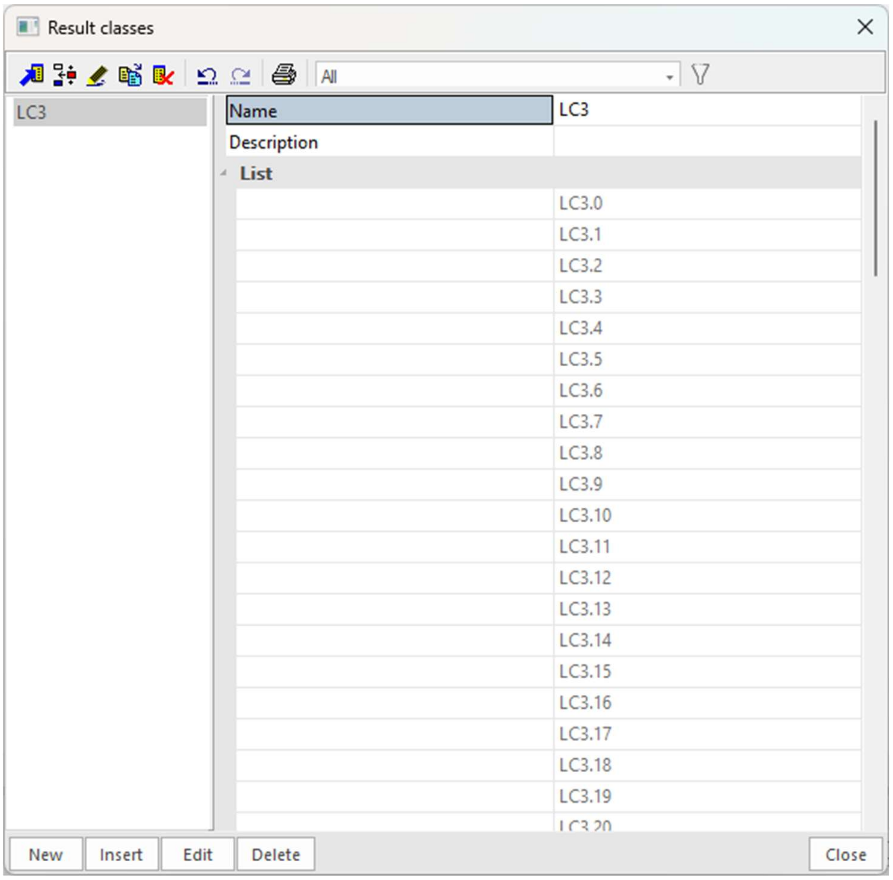


Step 7: linear calculation

Now, the calculation can be performed.
When the calculation is finished, new load cases are created which present each output step:



To find the most extreme result, these load cases are automatically input in a result class:



Step 8: results

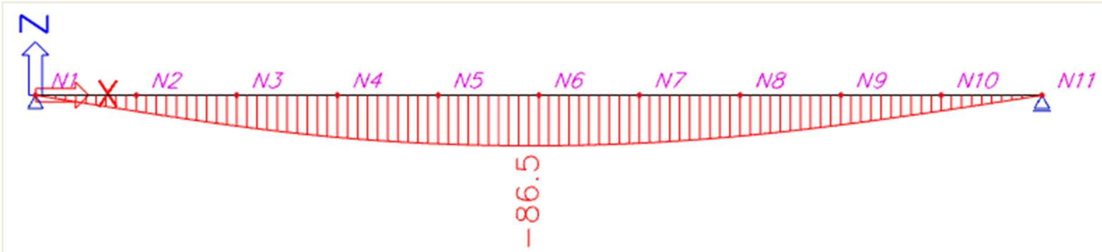
The eigen frequencies are shown in the results menu:

Eigen frequencies

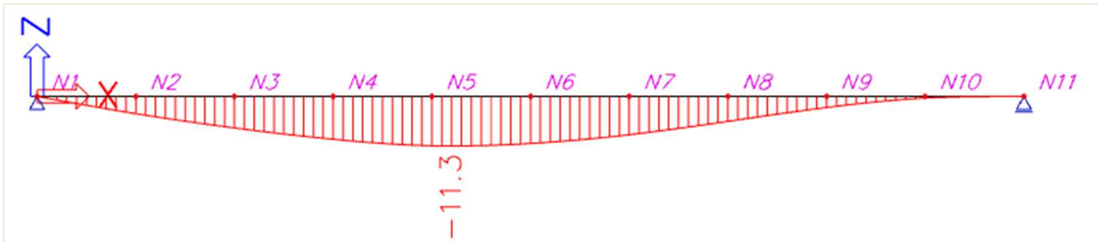
N	f [Hz]	ω [1/s]	ω^2 [1/s²]	T [s]
Mass combination : CM1				
1	5.03	31.62	999.93	0.20
2	19.78	124.28	15445.22	0.05
3	43.18	271.31	73608.14	0.02
4	64.59	405.79	164668.49	0.02

Other results, like for example deformations, can be regarded for the different output steps:

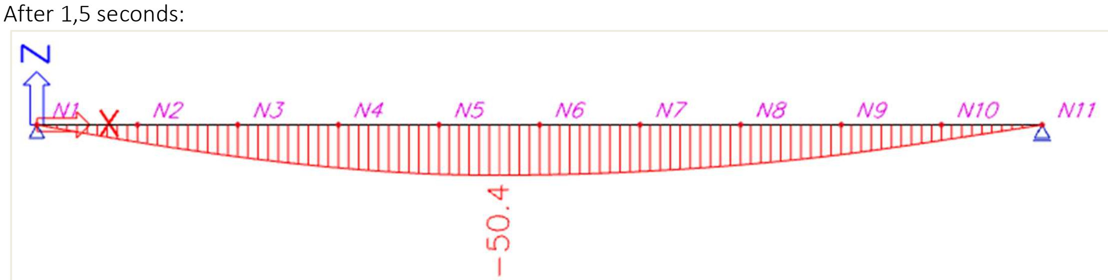
- After 0,5 second:



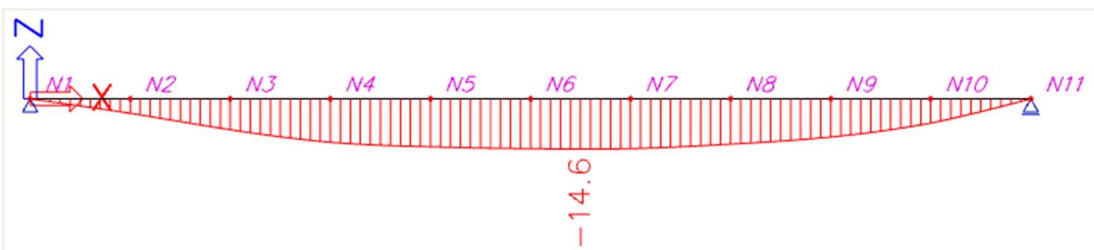
- After 1 second:



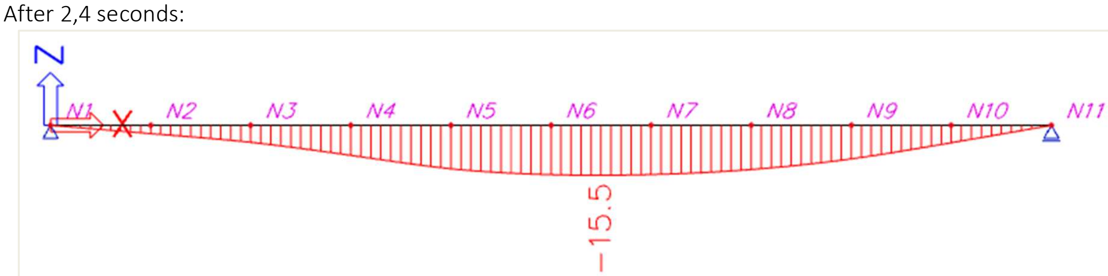
- After 1,5 seconds:



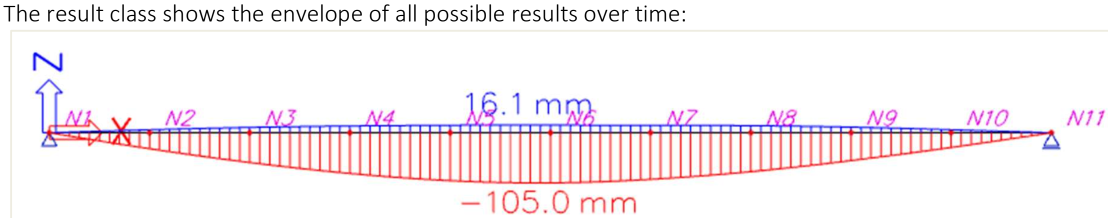
- After 2 seconds:



- After 2,4 seconds:

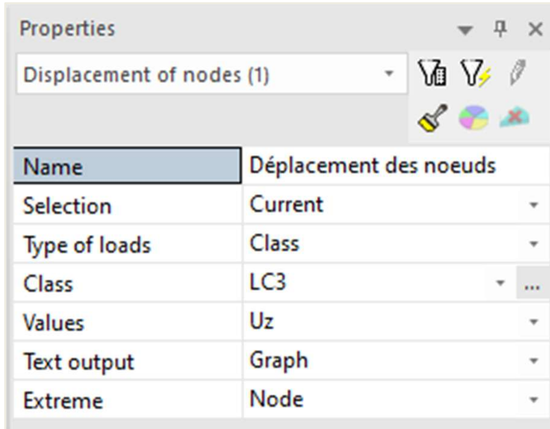


- The result class shows the envelope of all possible results over time:

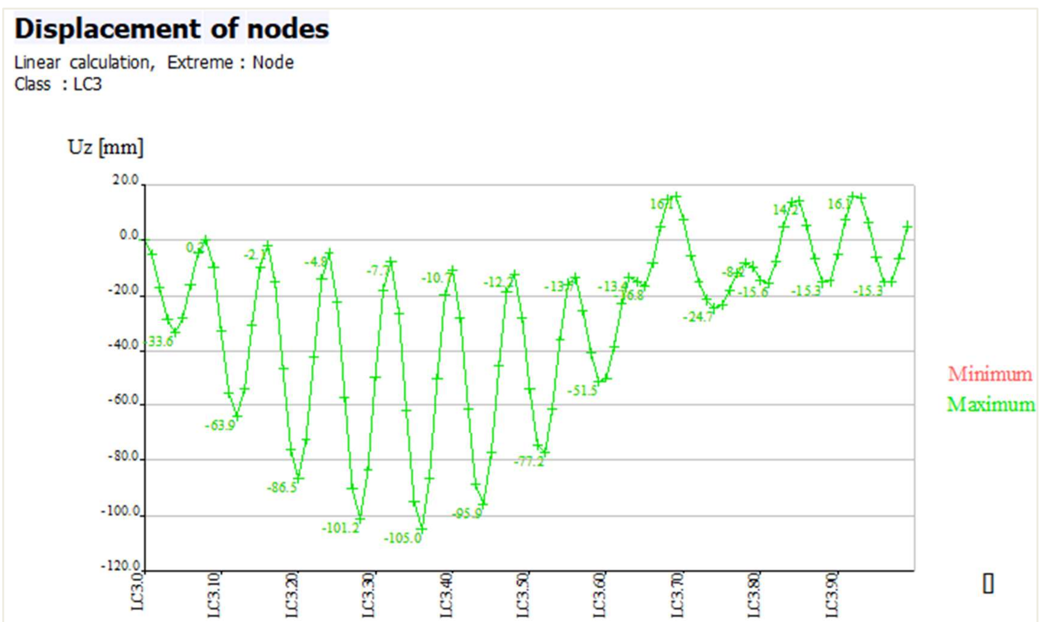


It is also possible to see the result in a certain point for all load cases in one picture. By this it is possible to see the result over the time.

Consider for instance the vertical displacement of the middle node N6.



The deformation of the middle node in function of the time is shown in the result preview:



his result clearly represents the vibration of the middle point over time.

CHAPTER 14 : VORTEX SHEDDING : KARMAN VIBRATION

In this chapter, the transverse vibration of cylindrical structures due to wind is examined.

First the theory is explained in which reference is made to the Harmonic load since Vortex shedding is a special case of harmonic loading.
The theory is then illustrated by an example of a steel chimney.

14.1 Theory

One of the most important mechanisms for wind-induced oscillations is the formation of vortices (concentrations of rotating fluid particles) in the wake flow behind certain types of structures such as chimneys, towers, suspended pipelines,…
At a certain (critical) wind velocity, the flow lines do not follow the contours of the body, but break away at some points, thus the vortices are formed.
These vortices are shed alternately from opposite sides of the structure and give rise to a fluctuating load perpendicular to the wind direction. The following figure illustrates the vortex shedding for flow past a circular cylinder. The created pattern is often referred to as the Karman Vortex Trail :

When a vortex is formed on one side of the structure, the wind velocity increases on the other side. This results in a pressure difference on the opposite sides and the structure is subjected to a lateral force away from the side where the vortex is formed. As the vortices are shed at the critical wind velocity alternately first from one side then the other, a harmonically varying lateral load with the same frequency as the frequency of the vortex shedding is formed.

The frequency of the vortex shedding f_v is given by:

$$f_v = \frac{S.v}{d} \tag{13.1}$$

With:

- S non-dimensional constant referred to as the « Strouhal Number ».
  For a cylinder, this is taken as 0,2.
- D width of the body loaded by the wind (m).
  For a cylinder, this equals the outer-diameter.
- v mean velocity of the wind flow (m/s).

The manner in which vortices are formed is a function of the Reynolds number Re, which is given by:

$$Re = 0,687.v.d.10^5 \tag{13.2}$$

In general, large Reynolds numbers mean turbulent flow.

The Reynolds number characterizes three major regions:

- Sub-critical:

$$300 \leq \mathrm{Re} \leq 10^5$$

- Super-critical:

$$10^5 \leq \mathrm{Re} \leq 3{,}5.10^6$$

- Trans-critical:

$$3{,}5.\,10^6 \leq \mathrm{Re}$$

For chimneys with circular cross section the flow is either in the supercritical or trans-critical range for wind velocities of practical interest.

If the vortex shedding frequency coincides with the natural frequency of the structure (resonance) quite large across-wind amplitudes of vibration will result unless sufficient damping is present. This principle was already discussed in a previous chapter.
In this case, formula (13.1) can be rewritten to calculate the critical wind velocity at which resonance occurs:

$$\mathrm{v_{crit}} = 5.\mathrm{d.f} \tag{13.3}$$

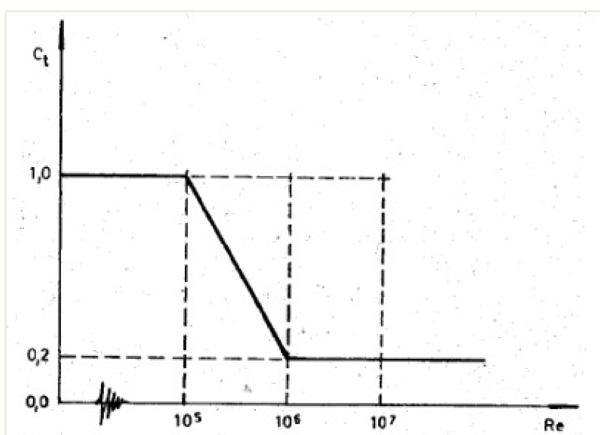
With:
f natural frequency of the structure.

The across-wind forces per unit length caused by the vortex shedding can be approximated by the following formula:

$$\mathrm{P_L(t)} = \frac{1}{2}.\rho.\mathrm{d.v_{crit}^2.C_t(t)} \tag{13.4}$$

With:

ρ	air density taken as 1,25kg/m^3
$C_t(t)$	lift coefficient that fluctuates in a harmonic of random manner and depends of the Reynolds number. The following figure shows this relation when C_t is proportional to the mode shape.



If the vortex shedding is taken as harmonic, equation (13.4) can be written as:

$$\mathrm{P_L(t)} = \mathrm{P_0.\sin(\omega_v t)} = \frac{1}{2}.\rho.\mathrm{d.v_{crit}^2.C_t.\sin(2.\pi.f_v)} \tag{13.5}$$

Assuming a constant wind profile, the equivalent modal force due to the fluctuating lift force of equation (13.5) is given by:

$$\mathrm{P(t)} = \mathrm{P_L.\sin(\omega_v t)} = \frac{1}{2}.\rho.\mathrm{d.v_{crit}^2.\sin(2.\pi.f_v)}.\int_0^H \mathrm{C_t(z)}\,\{\phi(\mathrm{z})\}\mathrm{dz} \tag{13.6}$$

With:

$\phi(z)$ modal shape at height z

H total height of the structure

As seen in a previous chapter, the dynamic amplitude Y at resonance can be written as :

$$Y = \frac{Y_S}{2\xi} \tag{13.7}$$

The static deformation Y_S is given by:

$$Y_S = \frac{P_0}{K} = \frac{P_0}{M.\omega^2} \tag{13.8}$$

M is the equivalent modal mass of a prismatic member given by:

$$M = \int_0^H m(z).\{\phi(z)\}^2 dz \tag{13.9}$$

With:

m(z) mass per unit height.

When combining formulas (13.7) and (13.8) the maximum response of a SDOF system subjected to a harmonic excitation may be written as:

$$Y = \frac{P_L}{M.\omega^2}.\frac{1}{2\xi} \tag{13.10}$$

It follows that when the vortex shedding occurs with the same frequency as the natural frequency of the structure, the maximal amplitude is given by:

$$Y = \frac{\frac{1}{2}.\rho.d.v_{crit}^2.\int_0^H C_t(z)\{\phi(z)\}dz}{\omega^2.\int_0^H m(z)\{\phi(z)\}^2 dz}.\frac{1}{2\xi} \tag{13.11}$$

When it is assumed that the mass per unit height is constant and that the lift coefficient is proportional to the mode shape, formula (13.11) can be simplified to the following:

$$Y = \frac{\rho.d^3.C_t}{16.\pi^2.S^2.m.\xi} \tag{13.12}$$

This equation may be used as a first estimate of likely response of the structure.

14.2 Vibration Karman in SCIA Engineer

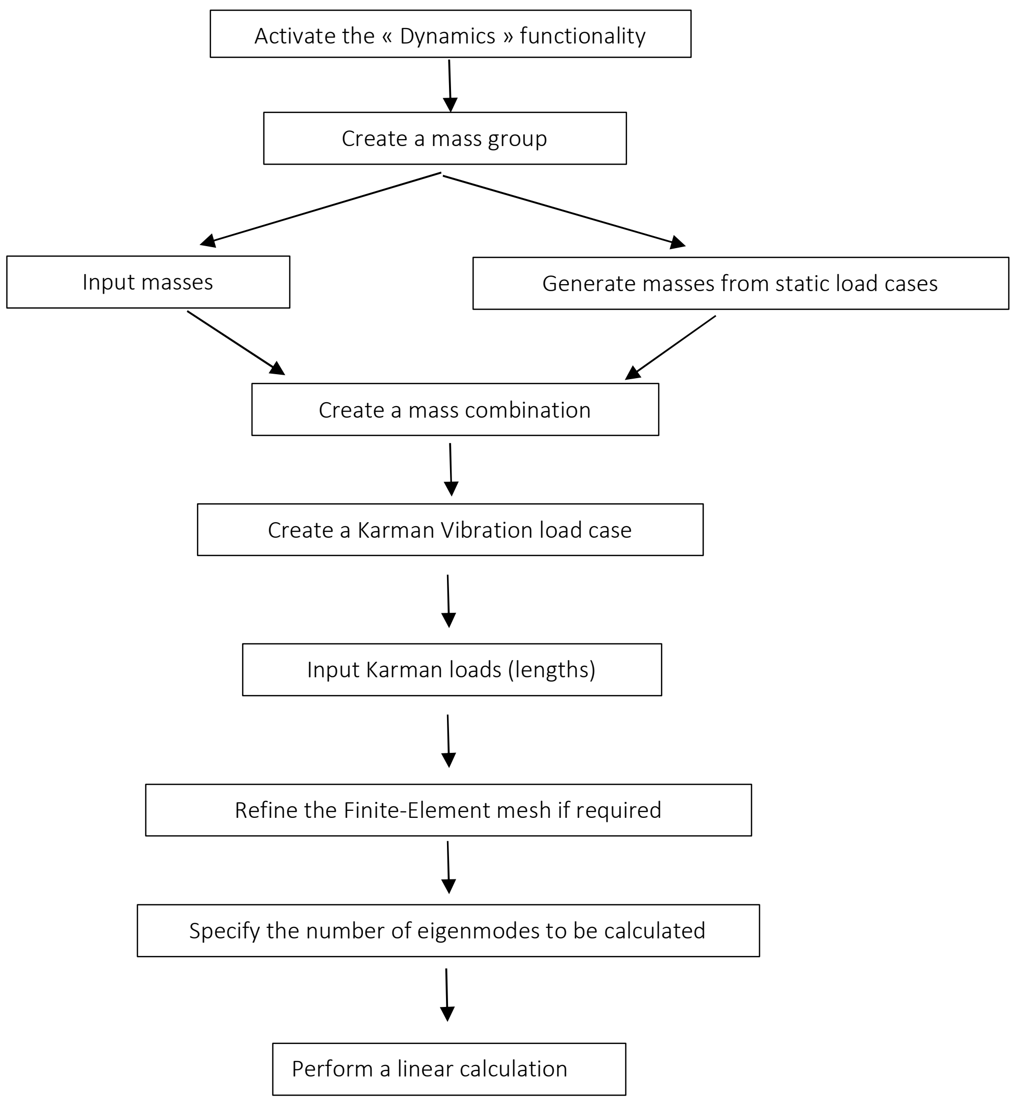
In SCIA Engineer, the Vortex Shedding was implemented according the Czech loading standard.

The effect is only taken into account if the critical wind velocity calculated by formula (13.3) is between a minimal and maximal value. These two extremes can be defined by the user. According to the Czech loading standard, these values are taken as 5 m/s and 20 m/s.

In addition to formula (13.11), in SCIA Engineer it is possible to specify the length of the structure where the Von Karman effect can occur. For each geometric node of the structure, it is possible to relate a length of the cylinder to the node. This implies that, in order to obtain precise results, the structure should be modeled with sufficient geometric nodes.

By default the effect can occur over the entire height of the structure however, when there are specific obstacles on the surface of a chimney for example, these obstacles will hamper the formation of the vortices and thus reduce the Von Karman effect. In practice, this is exactly the solution to suppress vortex-induced vibration: the fitting of special ribs on the surface of the cylinder.

The following diagram shows the required steps to perform a Vortex Shedding calculation :



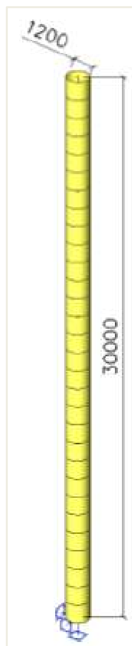
This diagram will be illustrated in the following example.

Example 13-1.esa:

In this example, a steel chimney is modeled with a fixed base.

The chimney has an outer-diameter of 1,2 m and a thickness of 6 mm. The total height is 30 m and the structure is manufactured in S 235 according to EC-EN.
To take into account the weight of insulation, electrical cables and other non-structural elements, a distributed mass of 55 kg/m is inputted.
No specific structural measures are taken to prevent the vortex shedding thus the entire length of the chimney must be taken into account for the Von Karman vibration.
To this end, the chimney is modeled as a cantilever built up as 30 members to create sufficient geometric nodes. Each node (except the base and top) will be assigned a chimney length of 1m.
For the logarithmic decrement of the chimney, a value of 0,025 is used.

One static load case is created: the self-weight of the structure.

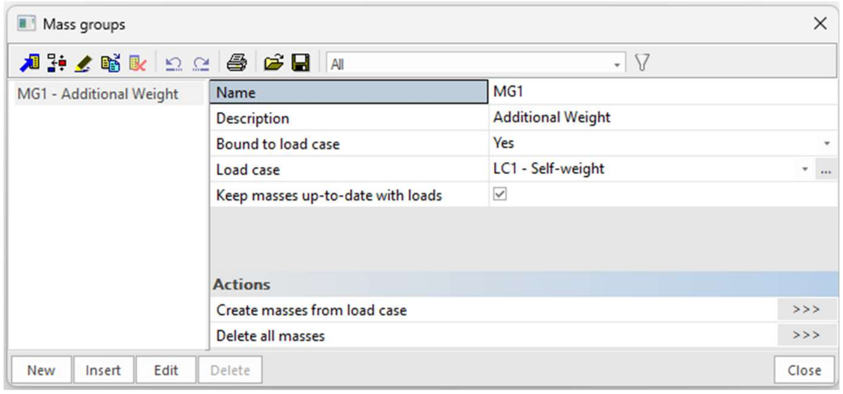


Step 1: functionality

The first step in the Karman Vibration calculation is to activate the functionality Dynamics on the Functionality tab in the Project Data.

Step 2: mass groups

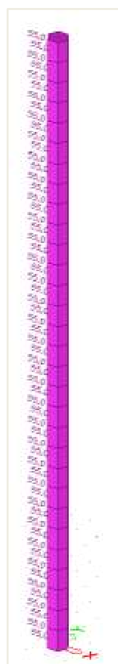
The second step is to create a Mass Group:



Actually, this mass group doesn't contain anything, since the self-weight is automatically taken into account. But to do dynamic calculation, at least one Mass Group needs to be defined.
In this mass group, we are going to place some additional masses. This will be added to the mass coming from the self-weight (which is always and automatically taken into account).

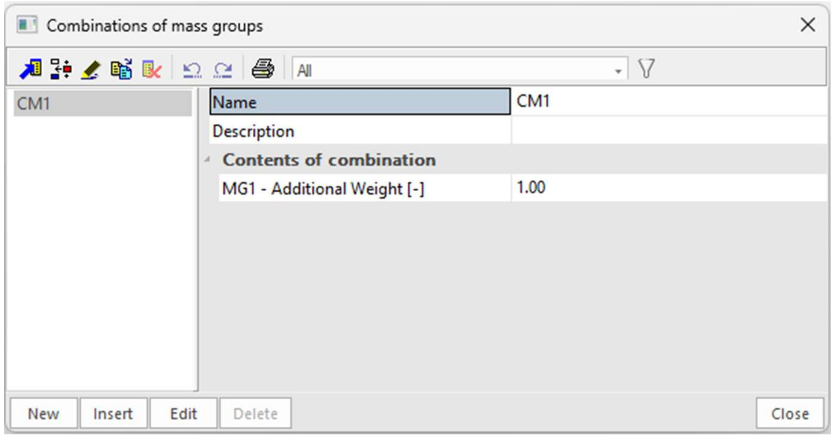
Step 3: masses

After the Mass Group has been created; the distributed mass of 55 kg/m can be inputted on all members:



Step 4: mass matrix

Next, the Mass Group is put within a "Combination of Mass Groups", which can be used for defining the harmonic load.



Step 5: Karman vibration load case

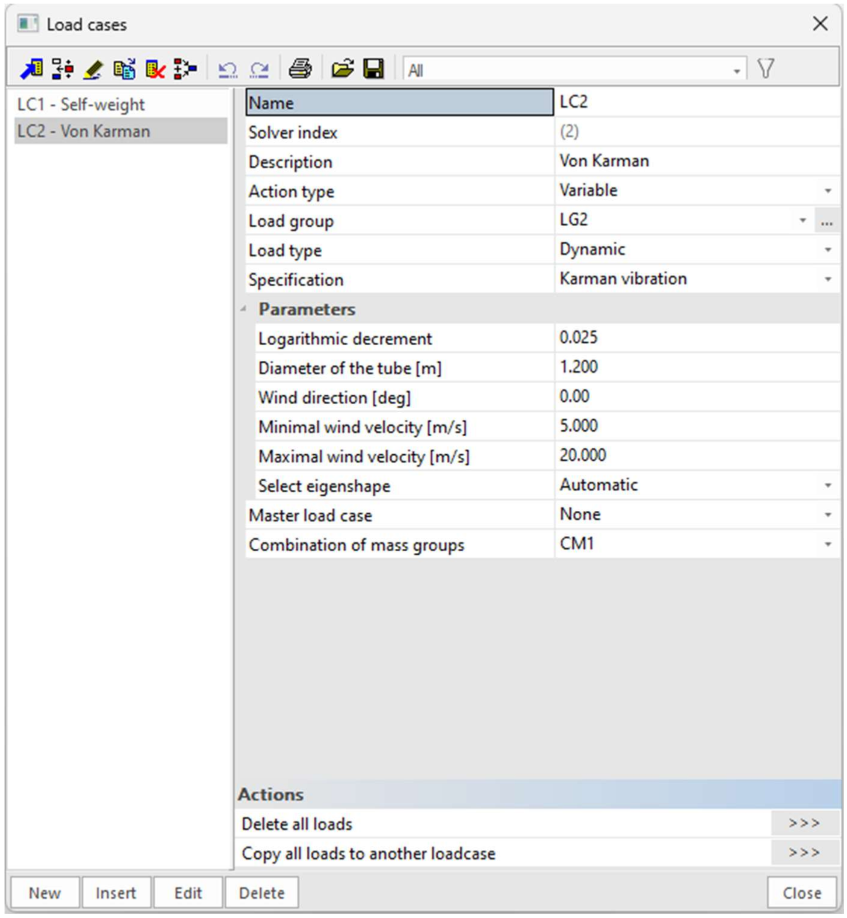
After creating a Mass Combination, a Karman vibration load case can be defined through Load cases, Combinations > Load Cases.

The « Action Type » of the Load Case is set to « Variable ».

The « Load Type » can then be changed to « Dynamic ».
Within the "Specification" field, the type of dynamic load can be set, in this case Karman vibration.

- The "logarithmic decrement" was given to be 0,025.
- The "Diameter of the tube" was 1,2m.
- The "Wind direction" is defined in the Global Coordinate System. A direction of 0,00 deg specifies the global X-axis. This implies that the Karman vibration will occur in a direction along the Y-axis (perpendicular to the wind direction).
- As specified in paragraph 6.2 the minimal and maximal wind velocities are set to 5 m/s and 20 m/s respectively. Vortex shedding will only occur if the critical wind velocity is between these two limits.
- The option "Select eigenshape" can be used to manually specify for which eigenmode the Vortex shedding needs to be calculated. When this option is left to 'Automatic', SCIA Engineer determines the representative mode automatically (which is the one with the biggest modal participation factor in the relevant direction).

Since the wind direction is set along the global X-axis, the representative mode will be a mode shape along the global Y-axis.



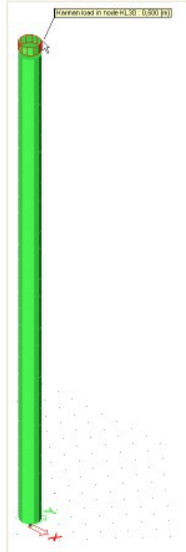
Step 6: Karman load

The parameters of the load case have been defined, what is left is to specify the length of the structure where the Von Karman effect can occur.

As indicated in a previous paragraph, SCIA Engineer allows relating a length of the chimney to each geometric node. This load can be inputted through Load > Point Force > Karman Load

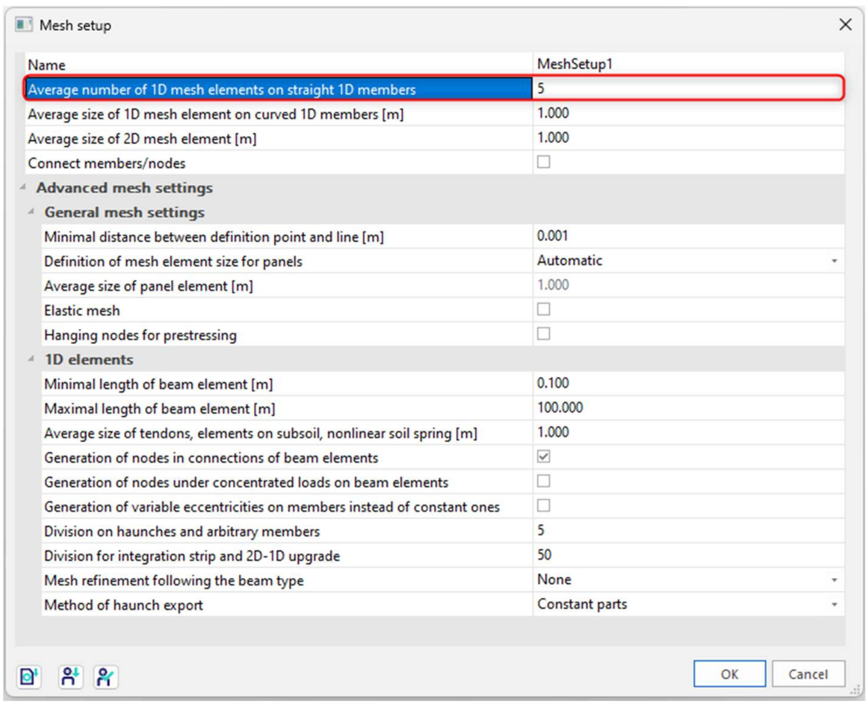
As no specific measures were taken to prevent vortex shedding and since the chimney was inputted as 30 members, each node is assigned a length of 1 m.

Additional nodes are made at 0,25m from the base and top. These nodes also get Karman loads assigned to them of 0,50 m.



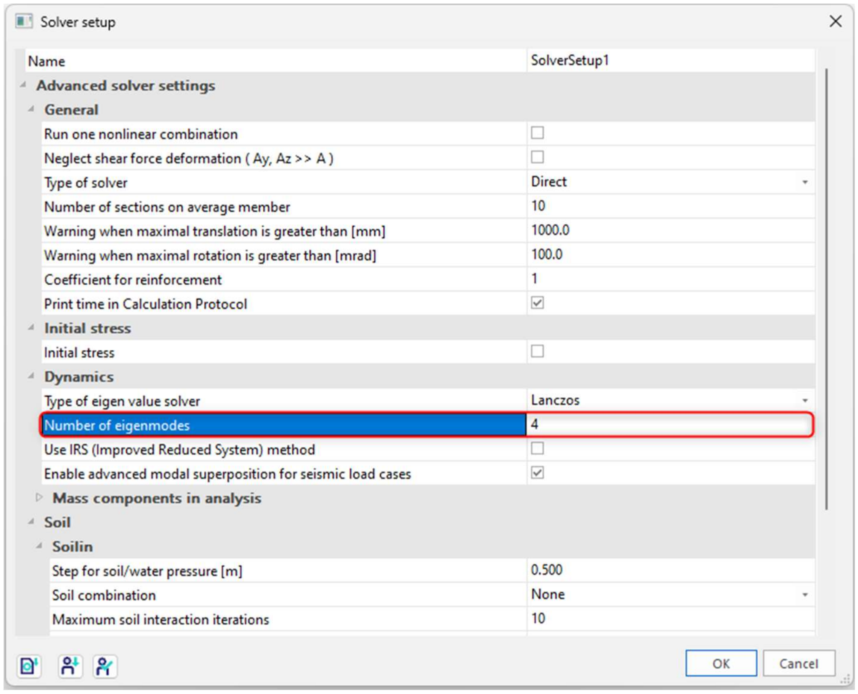
Step 7: mesh setup

To obtain precise results for the dynamic calculation, the Finite Element Mesh is refined. The “Average number of tiles of 1D element” is set to 5 through “ Mesh Setup”:



Step 8: solver setup

The last step before launching the calculation is setting the amount of eigenmodes to be calculated. The default value in “ Solver Setup” is 4. This is sufficient for this example.



Step 9: linear calculation and results

All steps have been executed so the "Linear Calculation" can be started through "Calculation".
The "Calculation Protocol" for the Eigen Frequency calculation shows the following:

Mode	Omega [rad/s]	Period [s]	Freq. [Hz]	Wxi / Wxtot	Wyi / Wytot	Wzi / Wztot	Wxi_R / Wxtot_R	Wyi_R / Wytot_R	Wzi_R / Wztot_R
1	7.4354	0.8450	1.1834	0.6152	0.0000	0.0000	0.0000	0.3755	0.0000
2	7.4354	0.8450	1.1834	0.0000	0.6152	0.0000	0.3755	0.0000	0.0000
3	46.0816	0.1363	7.3341	0.1905	0.0000	0.0000	0.0000	0.1977	0.0000
4	46.0816	0.1363	7.3341	0.0000	0.1905	0.0000	0.1977	0.0000	0.0000
				0.8057	0.8057	0.0000	0.5733	0.5733	0.0000

The details of the Karman Vibration calculation can be found in the "Calculation Protocol" for the linear calculation:

```
Karmans vibration is analyzed for eigen shape 2
Maximum horizontal modal translation [m] 0.02397
Critical wind velocity [m/s2] 7.10
Reynolds number 585346.91
Drag coefficient Ct 0.39
Load on cylinder at point of max displacement [N/m] 14.60
```

As expected, the Vortex shedding was analyzed for the second eigenmode, the mode with largest mass participation in the Y-direction.

The Maximum and Reduced loads are intermediate results used to calculate the across-wind forces.

The maximum horizontal translation for the second eigenmode can be found through “Deformation of Nodes” (note that the value has no relevance, the direction however is very important):

Displacement of nodes

Eigen solution, Extreme : Global
Selection : All
Mass combinations : CM1/2 - 1.18
Modal shapes are normalized, so that the generalized modal mass of each mode is equal to 1kg.

Node	Case	Ux [mm]	Uy [mm]	Uz [mm]	Fix [mrad]	Fiy [mrad]	Fiz [mrad]
N31	CM1/2 - 1.18	**0.0**	**24.0**	0.0	**-1.1**	0.0	0.0
N1	CM1/2 - 1.18	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**
N20	CM1/2 - 1.18	0.0	12.1	0.0	-1.0	**0.0**	0.0

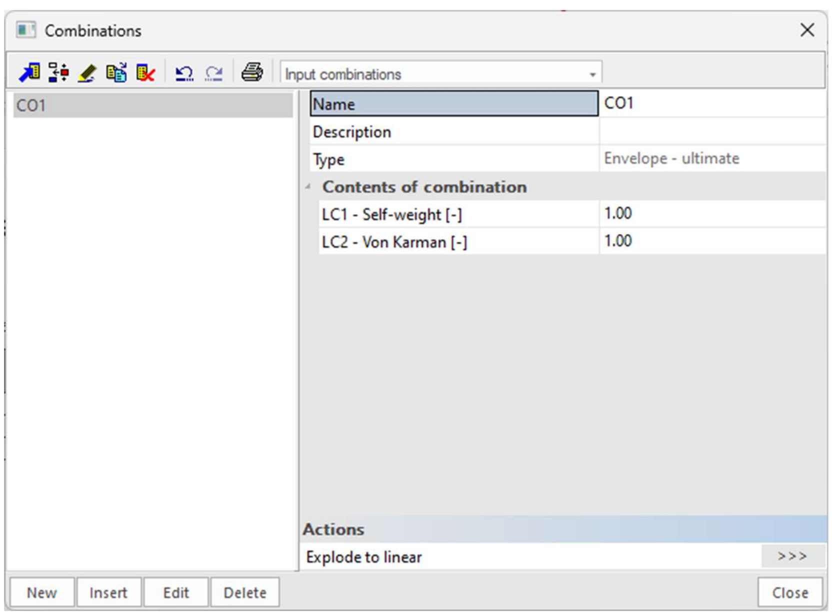
In the same way, the total deflection of the top of the chimney caused by the Karman Vibration can be shown:

Displacement of nodes

Linear calculation, Extreme : Global
Selection : All
Load cases : LC2

Node	Case	Ux [mm]	Uy [mm]	Uz [mm]	Fix [mrad]	Fiy [mrad]	Fiz [mrad]
N1	LC2	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**
N31	LC2	**0.0**	**-143.2**	0.0	**6.5**	0.0	0.0
N28	LC2	0.0	-123.5	0.0	6.5	**0.0**	0.0

Because of this large translation at the top, at the base of the chimney considerable stresses will occur.
As specified in a previous chapter, a combination of type “Envelope” provides the possibility for considering both sides of the vibration amplitude since a vibration is always in both directions.
An envelope combination is created for the chimney to evaluate the stresses at the base:



The “Member Stresses” for the lower member of the chimney give the following normal stresses for the combination:

Stress

Linear calculation, Extreme : Global
Selection : All
Combinations : CO1
Values : Normal -, Normal +, Shear, von Mises, Fatigue, Kappa, Sigma Y

Member	dx [m]	Case	Normal - [MPa]	Normal + [MPa]	Shear [MPa]	von Mises [MPa]	Fatigue [MPa]	Kappa [-]	Sigma Y [MPa]
B1	0.000	CO1/1	**-72.6**		0.0	**72.6**			0.0
B1	0.000	CO1/2		**68.0**	0.0	68.0			0.0
B1	0.000	CO1/1	-2.3	**-2.3**	**1.9**	4.0			**0.0**
B1	0.000	CO1	-72.6	68.0			**140.6**	-0.94	
B30	1.000	CO1	0.0	0.0			0.0	**-1.00**	
B30	0.938	CO1	0.0	0.0			0.0	**0.75**	

A stress range of 140,6 MPa will lead to significant fatigue problems after even a low amount of cycles. This is one of the most reported types of failures due to Vortex shedding.

A solution to this problem is the fitting of a helix type rib to prevent the correlation of the vortices (and thus lower the chimney-length which should be considered for the Von Karman effect). The disadvantage of such a rib is that it increases the drag force.

Since the Vortex shedding is a state of resonance, the amplitude is damping dependent as explained previously. Another solution is thus to increase the damping by installing a tuned mass damper system.
