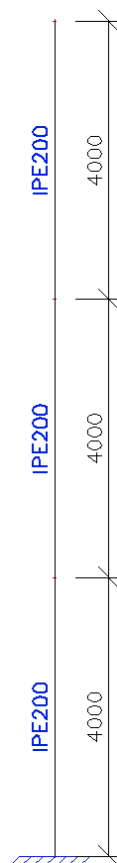


Benchmark: Spectral analysis

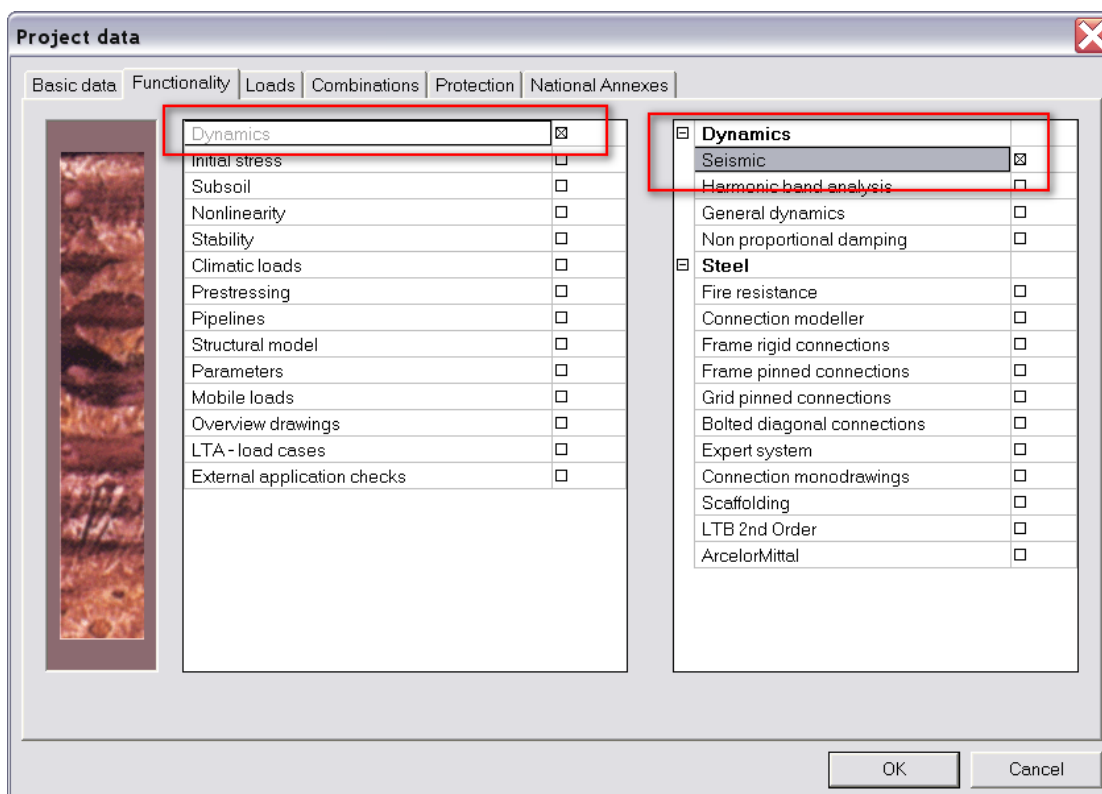
In this example a three-storey structure is modelled as a cantilever. The members have cross-section **IPE 200** and are manufactured in **S235** according to **EC-EN**. The height of each storey is **4m**. At each storey-level, the structure carries a mass of **500 kg**.



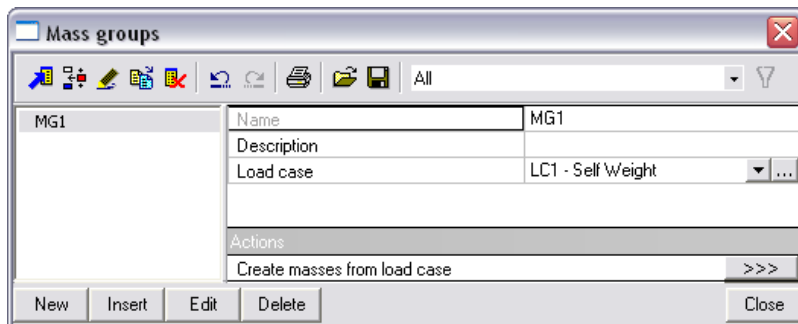
One static load case is created: the **self-weight** of the structure. However, in order not to take the self-weight into account for the dynamic calculation, the volumetric mass of **S235** can be set to **1 kg/m³** in the **Material** Library. This will render it easier to check the results through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

The structure will be subjected to an earthquake according to Eurocode 8, using a Design Response Spectrum for Ground Type **B** with a behaviour factor of **2**. The acceleration coefficient is **0,35**.

Step 1: 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** functionality needs to be activated:



Step 2: The second step is to create a **Mass Group**.



Step 3: After the Mass Group has been created; the **500 kg** masses can be inputted on each storey level:

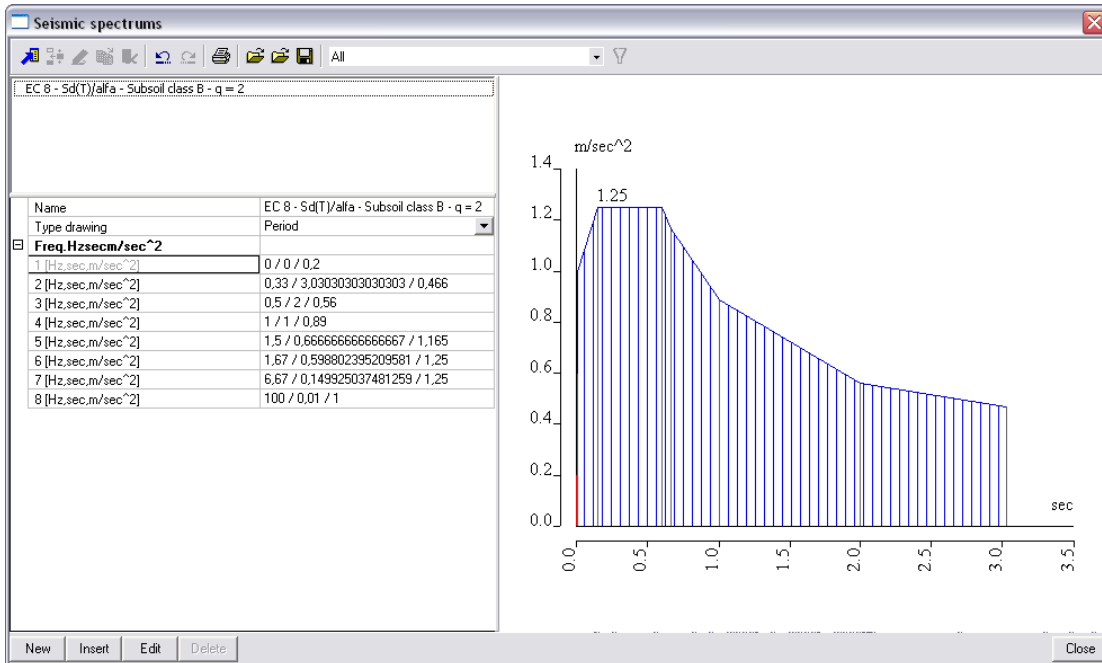


Step 4: Next, the Mass Group is put within a **Combination of Mass Groups**, which can be used for defining the Seismic load case.



Step 5: Before creating the Seismic load case, the Seismic Spectrum needs to be defined. This can be done through **Libraries > Seismic Spectrums**.

Instead of inputting a spectrum manually, the **System Database** is used which holds the predefined Eurocode 8 spectra. In this example, the spectrum for Ground Type **B** with a Behaviour Factor **q = 2** is used:



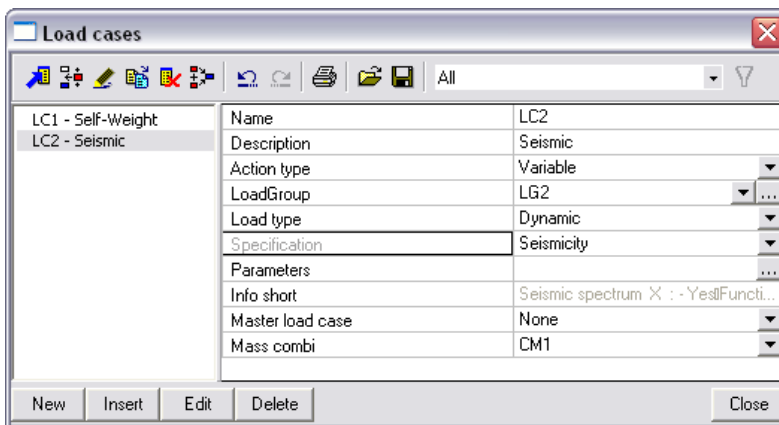
The values of the spectrum are given as **Sd(T)** divided by α . The acceleration coefficient will be defined in the seismic load case since the same spectrum can be used for different values of α .

Step 6: After creating a Mass Combination, a Seismic 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 **Seismicity**.



When this option is selected, the parameters of the seismic load can be inputted:

First of all, the options **Seismic spectrum X, Y, Z** can be activated. These options allow specifying which seismic spectrum has to be used in a certain global direction. In this example, the spectrum is defined in horizontal X-direction so **Seismic spectrum X** is activated and the specified spectrum of the previous step is chosen.

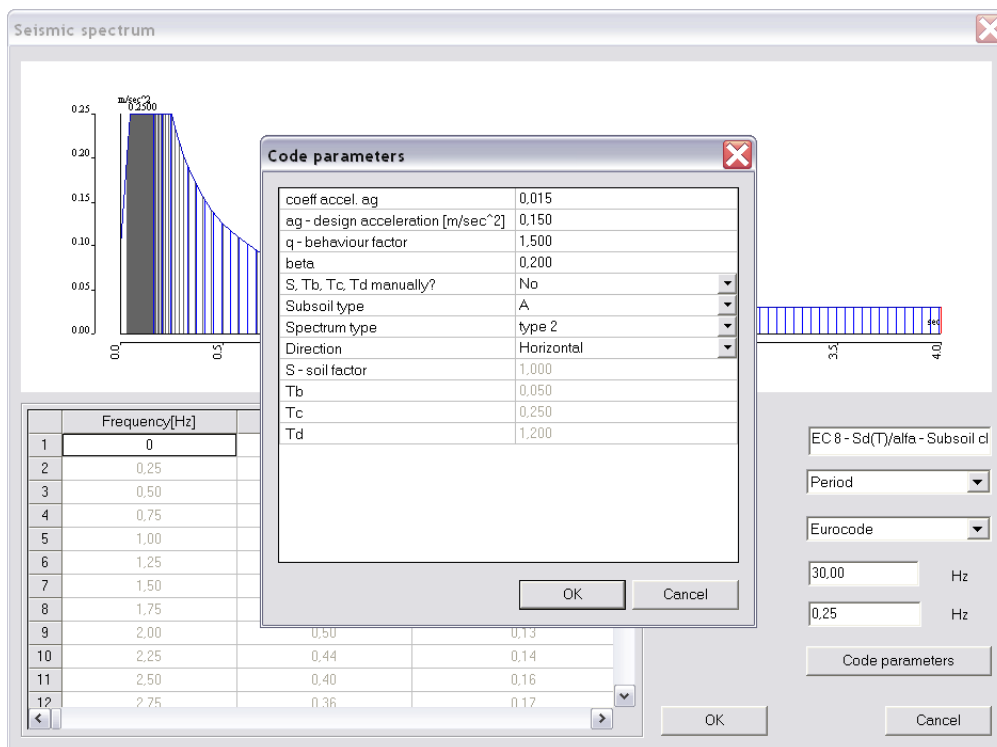
When no spectrum has been defined yet, a spectrum can be inputted through 

When the seismic action is not according to one of the global directions, the coefficients **direction X, Y, Z** can be used to specify a vector.

In this example the seismic action is according to the global X-direction so **direction X** is set to **1**.

As specified in the previous step, the values of the seismic spectra are given as **Sd(T)** divided by α . The **coefficient of acceleration** can now be inputted. In this example, the value of α is taken as **0,35**.

Remark: when a seismic spectrum according to the code is chosen, the value of α has to be 1. In that case, the coefficient of acceleration is inputted yet in the dialogue box of the parameters specifically for each code:



The **Overtuning** parameter is used when the supports of the structure are above ground level. By default, this value equals **0**. In the **Type of evaluation** box, the **Modal Combination Method** can be chosen. As specified in the theory, for the **CQC-method** an extra damping spectrum needs to be defined. This will be illustrated in a following example. For this example, the default **SRSS-method** will be applied.

Because of the square root in the formulas of the modal combination methods, the results are always positive. To use these results for a precise analysis of the structure, the option **signed results** can be used. When this property is checked, the user can select the mode shape which will be used to define the sign.

If the default is chosen, 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.

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

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

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

$$1 - \frac{\omega_{(i)}}{\omega_{(j)}} \leq \text{precision, where } i < j; \omega_{(i)} \leq \omega_{(j)}$$

Mode (i) and (j) are multiple. Then for example:

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

When the dynamic calculation is performed, the user has to verify if the 90% rule is satisfied. This can be done in the calculation protocol. In here the modal participation factors can be claimed. If the mass in vibration is lower than 90%, the number of eigen frequencies has to be increased. To avoid this check, one can activate the option **missing mass in modes** or **residual mode**. See further for more explanation.

Note:

The options under Multiple 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.

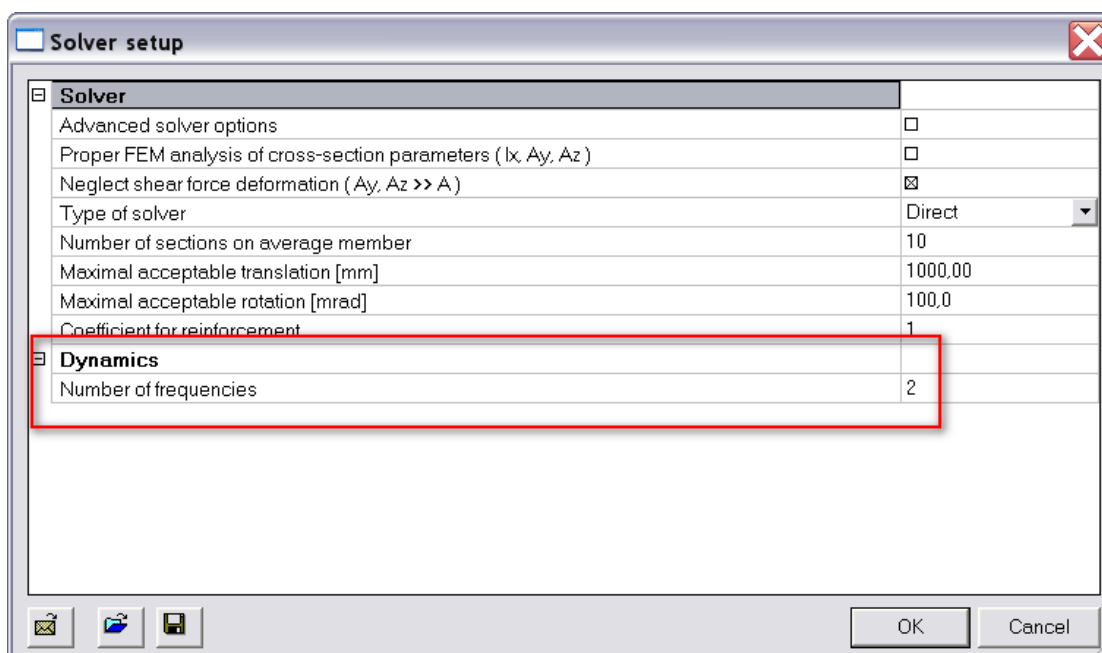
Step 7: As specified in the previous chapters, the finite element mesh needs to be refined to obtain precise results.

This can be done through **Calculation, Mesh > Mesh Setup**.

For this example however, the default mesh is not refined which will make it easier to verify the results by a manual calculation.

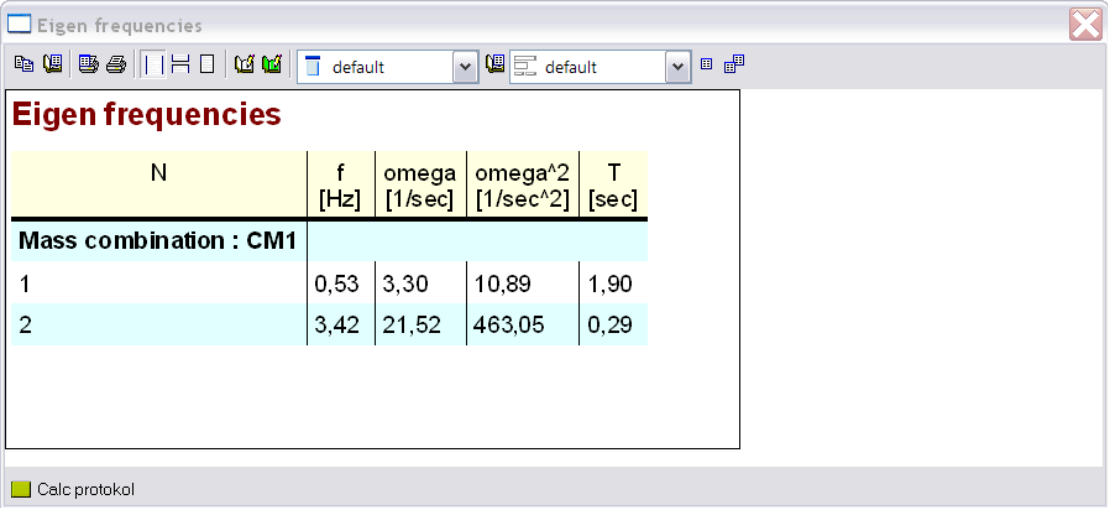
Step 8: The last step before launching the calculation is setting the amount of eigenmodes to be calculated. For this example, two eigenmodes are chosen.

In **Calculation, Mesh > Solver Setup** the number of frequencies is thus set to **2**.



Step 9: All steps have been executed so the **Linear Calculation** can be started through **Calculation, Mesh > Calculation**.

The following results are obtained:



Eigen frequencies

N	f [Hz]	omega [1/sec]	omega^2 [1/sec^2]	T [sec]
Mass combination : CM1				
1	0,53	3,30	10,89	1,90
2	3,42	21,52	463,05	0,29

Calc protokol

Using **Deformation of Nodes**, the **Deformed Structure** can be shown to view the eigenmodes:

Eigenmode 1:

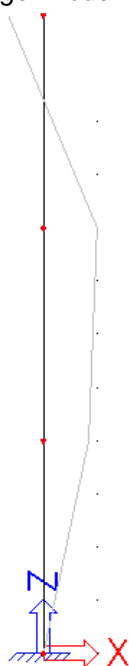


Deformation of nodes

Eigen solution, Extreme : No
 Selection : All
 Mass combinations : CM1/1 - 0,53
 Values are multiplied by 10000 for better numeric representation.

Node	Case	Ux []	Uz []	Fiy []
N1	CM1/1 - 0,53	0,00	0,00	0,00
N2	CM1/1 - 0,53	61,28	0,00	28,21
N3	CM1/1 - 0,53	208,03	0,00	42,96
N4	CM1/1 - 0,53	391,11	0,00	47,14

Eigenmode 2:



Deformation of nodes

Eigen solution, Extreme : No
 Selection : All
 Mass combinations : CM1/2 - 3,42
 Values are multiplied by 10000 for better numeric representation.

Node	Case	Ux []	Uz []	Fiy []
N1	CM1/2 - 3,42	0,00	0,00	0,00
N2	CM1/2 - 3,42	257,55	0,00	72,39
N3	CM1/2 - 3,42	304,51	0,00	-64,93
N4	CM1/2 - 3,42	-202,33	0,00	-156,78

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

Modal participation factors							
Number	Omega	Period	Freq. [Hz]	Damp ratio	Wxi / Wxtot	Wyi / Wytot	Wzi / Wztot
1	3.3007	1.9036	0.5253	0.0000	0.7269	0.0000	0.0000
2	21.5192	0.2920	3.4249	0.0000	0.2157	0.0000	0.0000
					0.9426	0.0000	0.0000

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. For the first Eigenmode, **73%** of the total mass is vibrating. For the second Eigenmode, **22%** is vibrating. In total, these two Eigenmodes account for about **94%** 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 two Eigen modes are sufficient for this example.

The details of the seismic calculation can be found in the **Calculation Protocol** for the **Linear Calculation**:

Dynamic loadcase: 2											
Number	Freq. [Hz]	Damp ratio	Damp coef.	Sax [m/s ²]	Say [m/s ²]	Saz [m/s ²]	G(j)	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.5253	0.0000	1.0000	0.2019	0.0000	0.0000	0.6119	0.2201	0.0000	0.0000	-2.2008
2	3.4249	0.0000	1.0000	0.4380	0.0000	0.0000	0.0170	0.1417	0.0000	0.0000	-0.4090
Level=	0.00							0.26	0.00	0.00	2.24

The **Damping ratio** shows the manually inputted damping ratio for the respective Eigenmode. 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**.

In case of a design spectrum is used, then the damping and the non-linearity of the connections are taken into account in the behaviour factor q .

Sax, **Say** and **Saz** show the spectral accelerations.

G(j) shows 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 last line in the table shows the global response. This response was obtained by means of the SRSS-method for combining the modal responses.

The formulas for these parameters have been shown in the theory and will now be illustrated by means of a manual calculation.

Manual Calculation

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.

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

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

$$\{\phi_1\} = \begin{Bmatrix} 0,039111 \\ 0,020803 \\ 0,006128 \\ 0 \end{Bmatrix} \quad \{\phi_2\} = \begin{Bmatrix} -0,020233 \\ 0,030451 \\ 0,025755 \\ 0 \end{Bmatrix}$$

Participation factor: $\gamma_{k,(j)} = \{\phi_k\}^T \cdot \{m\}$

$$\Rightarrow \gamma_{x,(1)} = 0,039111 \cdot 500 + 0,020803 \cdot 500 + 0,006128 \cdot 500 = 33,021$$

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

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

$$\Rightarrow M_{x,ef,(1)} = (33,021)^2 = 1090,39$$

$$\Rightarrow M_{x,ef,(2)} = (17,984)^2 = 323,42$$

Participation mass ratio: $L_{k,(j)} = \frac{M_{k,ef,(j)}}{M_{k,tot}}$

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

$$\Rightarrow L_{x,(2)} = \frac{323,42}{500 + 500 + 500} = \mathbf{0,2156}$$

These results correspond to the results obtained by Scia Engineer

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

Number	Freq. [Hz]	Damp ratio	Damp coef.	Sax [m/s ²]	Say [m/s ²]	Saz [m/s ²]	G(j)	Fx [kN]	Fy [kN]	Mx [kNm]	My [kNm]
1	0.5253	0.0000	1.0000	0.2019	0.0000	0.0000	0.6119	0.2201	0.0000	0.0000	-2.2008
2	3.4249	0.0000	1.0000	0.4380	0.0000	0.0000	0.0170	0.1417	0.0000	0.0000	-0.4090
Level=	0.00							0.26	0.00	0.00	2.24

The spectral acceleration **Sax** 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 $S_d(T)/\alpha$

Freq./Period/Accel. [Hz.sec]
0,00/0,00/0,200
0,33/3,03/0,466
0,50/2,00/0,560
1,00/1,00/0,890
1,50/0,67/1,165
1,67/0,60/1,250
6,67/0,15/1,250
100,00/0,01/1,000

The first mode has a period T_1 of **1,9036 s** $\Rightarrow S_d(T_1)/\alpha = 0,5918 \text{ m/s}^2$

The second mode has a period T_2 of **0,2920 s** $\Rightarrow S_d(T_2)/\alpha = 1,25 \text{ m/s}^2$

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

$$\Rightarrow S_{ax,(1)} = 0,5918 \frac{\text{m}}{\text{s}^2} \cdot 0,35 = 0,2071 \frac{\text{m}}{\text{s}^2}$$

$$\Rightarrow S_{ax,(2)} = 1,25 \frac{\text{m}}{\text{s}^2} \cdot 0,35 = 0,4375 \frac{\text{m}}{\text{s}^2}$$

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:

$$G_{k,(j)} = \frac{S_{a,k,(j)} \cdot \gamma_{k,(j)}}{\omega_{(j)}^2}$$

$$\Rightarrow G_{x,(1)} = \frac{0,2019 \cdot 33,021}{(3,3007)^2} = 0,6119$$

$$\Rightarrow G_{x,(2)} = \frac{0,4380 \cdot 17,984}{(21,5192)^2} = 0,0170$$

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: $F_{i,k,(j)} = m_{i,k,(j)} \cdot S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \phi_{i,k,(j)}$

$$F_{4,x,(1)} = 500 \text{ kg} \cdot 0,2019 \frac{\text{m}}{\text{s}^2} \cdot 33,021 \cdot 0,039111 = 130,38 \text{ N}$$

$$F_{3,x,(1)} = 500 \text{ kg} \cdot 0,2019 \frac{\text{m}}{\text{s}^2} \cdot 33,021 \cdot 0,020803 = 69,35 \text{ N}$$

$$F_{2,x,(1)} = 500 \text{ kg} \cdot 0,2019 \frac{\text{m}}{\text{s}^2} \cdot 33,021 \cdot 0,006128 = 20,43 \text{ N}$$

$$F_{1,x,(1)} = 0 \text{ N}$$

$$\text{Base Shear Force: } F_{k,(j)} = \sum_i F_{i,k,(j)}$$

$$\Rightarrow F_{x,(1)} = 130,38 \text{ N} + 69,35 \text{ N} + 20,43 \text{ N} = 220,129 \text{ N} = \mathbf{0,2201 \text{ kN}}$$

$$\text{Overturning moment in node i: } M_{i,k,(j)} = F_{i,k,(j)} \cdot z_i$$

$$M_{4,y,(1)} = -130,38 \text{ N} \cdot 12 \text{ m} = -1564,50 \text{ Nm}$$

$$M_{3,y,(1)} = -69,35 \text{ N} \cdot 8 \text{ m} = -554,77 \text{ Nm}$$

$$M_{2,y,(1)} = -20,43 \text{ N} \cdot 4 \text{ m} = -81,71 \text{ Nm}$$

$$M_{1,y,(1)} = 0 \text{ Nm}$$

$$\text{Overturning Moment: } M_{k,(j)} = \sum_i M_{i,k,(j)}$$

$$\Rightarrow M_{y,(1)} = -1564,50 \text{ Nm} - 554,77 \text{ Nm} - 81,71 \text{ Nm} = -2200,89 \text{ Nm} = \mathbf{-2,2009 \text{ kNm}}$$

Note:

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.

However, as stated in the previous chapters, the signs have no absolute importance since vibration amplitudes always occurs on both sides of the equilibrium position.

Mode 2:

$$\text{Lateral force in node i: } F_{i,k,(j)} = m_{i,k,(j)} \cdot S_{a,k,(j)} \cdot \gamma_{k,(j)} \cdot \phi_{i,k,(j)}$$

$$F_{4,x,(2)} = 500 \text{ kg} \cdot 0,4380 \frac{\text{m}}{\text{s}^2} \cdot 17,984 \cdot -0,020233 = -79,69 \text{ N}$$

$$F_{3,x,(2)} = 500 \text{ kg} \cdot 0,4380 \frac{\text{m}}{\text{s}^2} \cdot 17,984 \cdot 0,030451 = 119,93 \text{ N}$$

$$F_{2,x,(2)} = 500 \text{ kg} \cdot 0,4380 \frac{\text{m}}{\text{s}^2} \cdot 17,984 \cdot 0,025755 = 101,44 \text{ N}$$

$$F_{1,x,(2)} = 0 \text{ N}$$

$$\text{Base Shear Force: } F_{k,(j)} = \sum_i F_{i,k,(j)}$$

$$\Rightarrow F_{x,(2)} = -79,69 \text{ N} + 119,93 \text{ N} + 101,44 \text{ N} = 141,68 \text{ N} = \mathbf{0,1417 \text{ kN}}$$

$$\text{Overturning moment in node i: } M_{i,k,(j)} = F_{i,k,(j)} \cdot z_i$$

$$M_{4,y,(2)} = 79,69 \text{ N} \cdot 12 \text{ m} = 956,25 \text{ Nm}$$

$$M_{3,y,(2)} = -119,93 \text{ N} \cdot 8 \text{ m} = -959,45 \text{ Nm}$$

$$M_{2,y,(2)} = -101,44 \text{ N} \cdot 4 \text{ m} = -405,74 \text{ Nm}$$

$$M_{1,y,(2)} = 0 \text{ Nm}$$

$$\text{Overturning Moment: } M_{k,(j)} = \sum_i M_{i,k,(j)}$$

$$\Rightarrow M_{y,(2)} = 956,25 \text{ Nm} - 959,45 \text{ Nm} - 405,74 \text{ Nm} = -408,94 \text{ Nm} = \mathbf{-0,4089 \text{ kNm}}$$

To obtain the global response, the modal responses need to be combined. In this example the **SRSS-method** was used:

$$F_x = \sqrt{(F_{x,(1)})^2 + (F_{x,(2)})^2} = \sqrt{(0,2201 \text{ kN})^2 + (0,1417 \text{ kN})^2} = \mathbf{0,2618 \text{ kN}}$$

$$M_y = \sqrt{(M_{y,(1)})^2 + (M_{y,(2)})^2} = \sqrt{(-2,2009 \text{ kNm})^2 + (-0,4089 \text{ kNm})^2} = \mathbf{2,238 \text{ kNm}}$$

These results correspond exactly to the results obtained by Scia Engineer.

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:

$$\text{Displacement in node } i: u_{i,k,(j)} = G_{k,(j)} \cdot \phi_{i,k,(j)}$$

$$u_{4,x,(1)} = 0,6119 \cdot 0,039111 = 0,02393 \text{ m} = \mathbf{23,93 \text{ mm}}$$

$$u_{3,x,(1)} = 0,6119 \cdot 0,020803 = 0,01273 \text{ m} = \mathbf{12,73 \text{ mm}}$$

$$u_{2,x,(1)} = 0,6119 \cdot 0,006128 = 0,00375 \text{ m} = \mathbf{3,75 \text{ mm}}$$

$$u_{1,x,(1)} = \mathbf{0 \text{ mm}}$$

$$\text{Acceleration in node } i: \ddot{u}_{i,k,(j)} = \omega_{(j)}^2 \cdot G_{k,(j)} \cdot \phi_{i,k,(j)}$$

$$a_{4,x,(1)} = (3,3007)^2 \cdot 0,6119 \cdot 0,039111 = 0,26073 \frac{\text{m}}{\text{s}^2} = \mathbf{260,73 \text{ mm/s}^2}$$

$$a_{3,x,(1)} = (3,3007)^2 \cdot 0,6119 \cdot 0,020803 = 0,13868 \frac{\text{m}}{\text{s}^2} = \mathbf{138,68 \text{ mm/s}^2}$$

$$a_{2,x,(1)} = (3,3007)^2 \cdot 0,6119 \cdot 0,006128 = 0,04085 \frac{\text{m}}{\text{s}^2} = \mathbf{40,85 \text{ mm/s}^2}$$

$$a_{1,x,(1)} = \mathbf{0 \text{ mm/s}^2}$$

Mode 2:

$$\text{Displacement in node } i: u_{i,k,(j)} = G_{k,(j)} \cdot \phi_{i,k,(j)}$$

$$u_{4,x,(2)} = 0,0170 \cdot (-0,020233) = -0,00034 \text{ m} = \mathbf{-0,34 \text{ mm}}$$

$$u_{3,x,(2)} = 0,0170 \cdot 0,030451 = 0,00052 \text{ m} = \mathbf{0,52 \text{ mm}}$$

$$u_{2,x,(2)} = 0,0170 \cdot 0,025755 = 0,00044 \text{ m} = \mathbf{0,44 \text{ mm}}$$

$$u_{1,x,(2)} = \mathbf{0 \text{ mm}}$$

$$\text{Acceleration in node } i: \ddot{u}_{i,k,(j)} = \omega_{(j)}^2 \cdot G_{k,(j)} \cdot \phi_{i,k,(j)}$$

$$a_{4,x,(2)} = (21,5192)^2 \cdot 0,0170 \cdot (-0,020233) = -0,15928 \frac{m}{s^2} = \mathbf{-159,28 \text{ mm/s}^2}$$

$$a_{3,x,(2)} = (21,5192)^2 \cdot 0,0170 \cdot 0,030451 = 0,23972 \frac{m}{s^2} = \mathbf{239,72 \text{ mm/s}^2}$$

$$a_{2,x,(2)} = (21,5192)^2 \cdot 0,0170 \cdot 0,025755 = 0,20275 \frac{m}{s^2} = \mathbf{202,75 \text{ mm/s}^2}$$

$$a_{1,x,(2)} = \mathbf{0 \text{ mm/s}^2}$$

To obtain the global response, the modal responses need to be combined. In this example the **SRSS-method** was used:

Displacements:

$$u_{4,x} = \sqrt{(u_{4,x,(1)})^2 + (u_{4,x,(2)})^2} = \sqrt{(23,93)^2 + (-0,34)^2} = \mathbf{23,93 \text{ mm}}$$

$$u_{3,x} = \sqrt{(u_{3,x,(1)})^2 + (u_{3,x,(2)})^2} = \sqrt{(12,73)^2 + (0,52)^2} = \mathbf{12,74 \text{ mm}}$$

$$u_{2,x} = \sqrt{(u_{2,x,(1)})^2 + (u_{2,x,(2)})^2} = \sqrt{(3,75)^2 + (0,44)^2} = \mathbf{3,78 \text{ mm}}$$

$$u_{1,x} = \mathbf{0 \text{ mm}}$$

Accelerations:

$$a_{4,x} = \sqrt{(a_{4,x,(1)})^2 + (a_{4,x,(2)})^2} = \sqrt{(260,73)^2 + (-159,28)^2} = \mathbf{305,53 \text{ mm/s}^2}$$

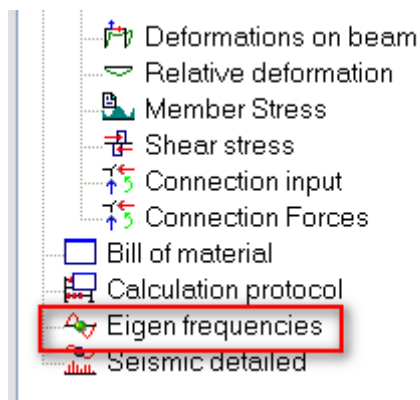
$$a_{3,x} = \sqrt{(a_{3,x,(1)})^2 + (a_{3,x,(2)})^2} = \sqrt{(138,68)^2 + (239,72)^2} = \mathbf{276,94 \text{ mm/s}^2}$$

$$a_{2,x} = \sqrt{(a_{2,x,(1)})^2 + (a_{2,x,(2)})^2} = \sqrt{(40,85)^2 + (202,75)^2} = \mathbf{206,82 \text{ mm/s}^2}$$

$$a_{1,x} = \mathbf{0 \text{ mm/s}^2}$$

In Scia Engineer, a specific result menu **Seismic Detailed** was designed to view these modal displacements and accelerations.

In the Properties Window, the options for viewing the modal results can be set:



Properties	
Seismic detailed (1)	
Name	Seismic detailed
Selection	All
Type of loads	Load cases
Load cases	LC2 - Seismic
Filter	No
Modal results	Accelerations
Evaluation for	Sum
Values	Deformed mesh
Extreme	No

- In the field **Load Cases**, a Seismic Load case can be selected.
- The field **Modal results** allows 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, **Summarised** results.

The following results are obtained:

Modal Displacements:

Seismic detailed											
Linear calculation, Extreme : No											
Selection : All											
Load cases : LC2											
Case	Node	X [m]	Y [m]	Z [m]	Mode	Ux [mm]	Uy [mm]	Uz [mm]	Fix [mrad]	Fiy [mrad]	Fiz [mrad]
LC2	1	0,000	0,000	0,000	1	0,00	0,00	0,00	0,0	0,0	0,0
LC2	1	0,000	0,000	0,000	2	0,00	0,00	0,00	0,0	0,0	0,0
LC2	2	0,000	0,000	4,000	1	3,75	0,00	0,00	0,0	1,7	0,0
LC2	2	0,000	0,000	4,000	2	0,44	0,00	0,00	0,0	0,1	0,0
LC2	3	0,000	0,000	8,000	1	12,73	0,00	0,00	0,0	2,6	0,0
LC2	3	0,000	0,000	8,000	2	0,52	0,00	0,00	0,0	-0,1	0,0
LC2	4	0,000	0,000	12,000	1	23,93	0,00	0,00	0,0	2,9	0,0
LC2	4	0,000	0,000	12,000	2	-0,34	0,00	0,00	0,0	-0,3	0,0

Modal Accelerations:

Seismic detailed											
Linear calculation, Extreme : No											
Selection : All											
Load cases : LC2											
Case	Node	X [m]	Y [m]	Z [m]	Mode	Ax [mm/sec²]	Ay [mm/sec²]	Az [mm/sec²]	Alphax [mrad/sec²]	Alphay [mrad/sec²]	Alphaz [mrad/sec²]
LC2	1	0,000	0,000	0,000	1	0,00	0,00	0,00	0,0	0,0	0,0
LC2	1	0,000	0,000	0,000	2	0,00	0,00	0,00	0,0	0,0	0,0
LC2	2	0,000	0,000	4,000	1	40,85	0,00	0,00	0,0	18,8	0,0
LC2	2	0,000	0,000	4,000	2	202,91	0,00	0,00	0,0	57,0	0,0
LC2	3	0,000	0,000	8,000	1	138,68	0,00	0,00	0,0	28,6	0,0
LC2	3	0,000	0,000	8,000	2	239,90	0,00	0,00	0,0	-51,2	0,0
LC2	4	0,000	0,000	12,000	1	260,72	0,00	0,00	0,0	31,4	0,0
LC2	4	0,000	0,000	12,000	2	-159,40	0,00	0,00	0,0	-123,5	0,0

Summarized Displacements:

Seismic detailed										
Linear calculation, Extreme : No										
Selection : All										
Load cases : LC2										
Case	Node	X [m]	Y [m]	Z [m]	Ux [mm]	Uy [mm]	Uz [mm]	Fix [mrad]	Fiy [mrad]	Fiz [mrad]
LC2	1	0,000	0,000	0,000	0,00	0,00	0,00	0,0	0,0	0,0
LC2	2	0,000	0,000	4,000	3,78	0,00	0,00	0,0	1,7	0,0
LC2	3	0,000	0,000	8,000	12,74	0,00	0,00	0,0	2,6	0,0
LC2	4	0,000	0,000	12,000	23,93	0,00	0,00	0,0	2,9	0,0

Summarized Accelerations:

Seismic detailed										
Linear calculation, Extreme : No										
Selection : All										
Load cases : LC2										
Case	Node	X [m]	Y [m]	Z [m]	Ax [mm/sec ²]	Ay [mm/sec ²]	Az [mm/sec ²]	Alphax [mrad/sec ²]	Alphay [mrad/sec ²]	Alphaz [mrad/sec ²]
LC2	1	0,000	0,000	0,000	0,00	0,00	0,00	0,0	0,0	0,0
LC2	2	0,000	0,000	4,000	206,98	0,00	0,00	0,0	60,0	0,0
LC2	3	0,000	0,000	8,000	277,10	0,00	0,00	0,0	58,6	0,0
LC2	4	0,000	0,000	12,000	305,59	0,00	0,00	0,0	127,4	0,0

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.