Created in COMSOL Multiphysics 6.2

Earthquake Analysis of a Building

Introduction

This example shows how to perform an earthquake analysis of a structure using response spectrum analysis.

Earthquakes are, by their nature, not deterministic. If you were to check that a building can sustain an earthquake by time domain analysis, you would have to run a large number of analyses with different representative prescribed acceleration histories for the supports. Such an approach would require large computational resources, and it would still not be guaranteed that the worst possible case is covered.

A response spectrum analysis, on the other hand, requires only an eigenfrequency analysis and possibly the solution of few linear stationary load cases. The severity of the earthquake is represented by a *design response spectrum*, which is provide by various national and international building codes.

Model Definition

The structure is a regular beam framework. The geometry is parameterized so that it can have any number of columns in both horizontal directions, and any number of floors. Also, the column spacing and floor height are parameters. The geometry used in the analysis is shown in Figure 1.

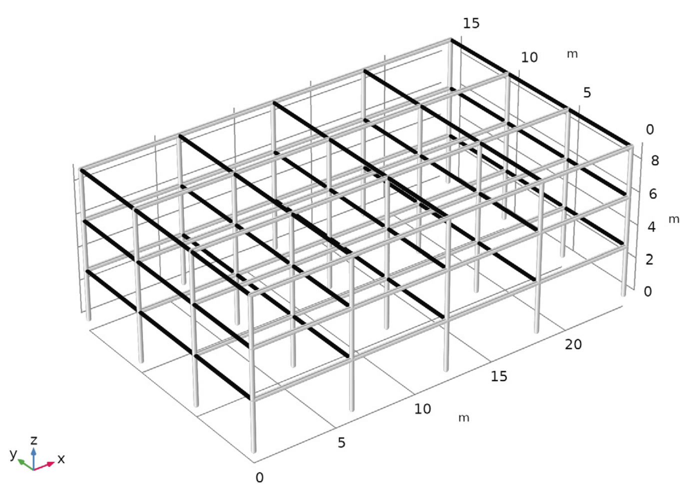

Figure 1: The geometry of the steel frame.

GEOMETRY

The selected geometrical parameters are:

- Number of columns in the X direction: 5
- Column spacing in the X direction: 6 m
- Number of columns in the Y direction: 4
- Column spacing in the Y direction: 5 m
- Number of floors: 4
- Floor height: 3 m

The following beam cross sections are used:

- Columns: Box 300-by-200 mm with wall thickness 10 mm. The stiff direction is along the global X-axis.
- Horizontal beams in the X direction: HEA 260.
- Horizontal beams in the Y direction: HEA 220.

MATERIAL

Structural steel with E = 200 GPa, ν = 0.30, and ρ = 7850 kg/m^{3}.

MASS

The horizontal beams are assigned an additional mass of 1000 kg/m, representing the mass of each floor.

CONSTRAINTS

The end of all columns are fixed in the ground plane.

LOADS

In one study, the deflection under a pure gravitational loading is analyzed.

For the response spectrum evaluations, design response spectra inspired by Eurocode (EN1998-1) are used. The horizontal pseudoacceleration spectrum is shown in Figure 2, and the vertical spectrum is shown in Figure 3. The spectra represent a rather mild earthquake.

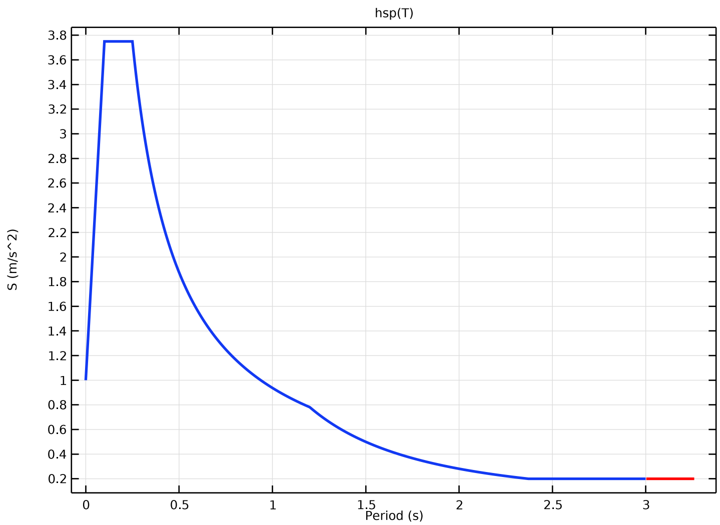

Figure 2: Horizontal design response spectrum.

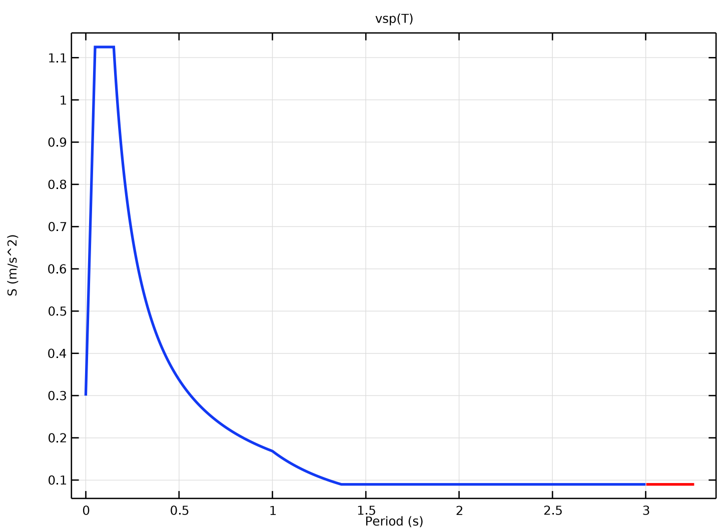

Figure 3: Vertical design response spectrum.

GRAVITY LOAD

In the first analysis, a pure gravity load is applied to the structure. The stress distribution is shown in Figure 4. The maximal stress occurs at the top of the outermost columns, and it is approximately 60 MPa.

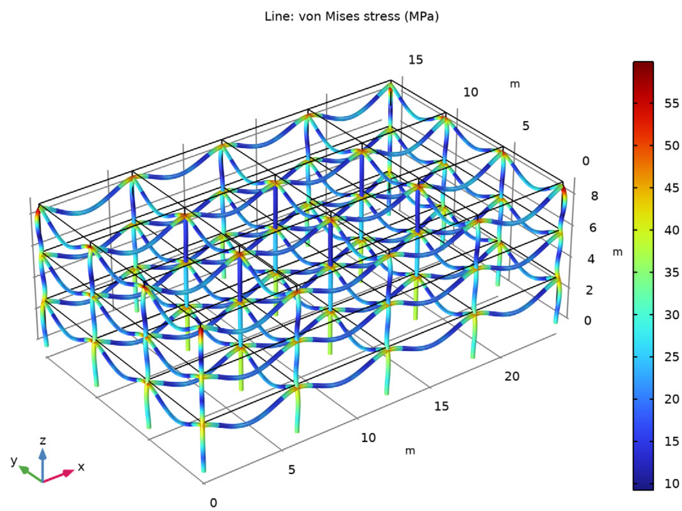

Figure 4: Stress distribution with only gravity load.

EIGENFREQUENCY ANALYSIS

Within the computed range of eigenfrequencies (up to 33 Hz), there are over 500 natural frequencies. The reason for this is the large number of symmetries and near symmetries which such a structure possesses. However, most of the corresponding eigenmodes will not contribute to the earthquake response because they cannot be excited by a homogeneous base movement.

The time for evaluating a result quantity in a response spectrum analysis is proportional to the square of the number of included eigenmodes. For this reason, it is useful to limit the analysis to those eigenmodes that actually contribute significantly to the response.

An investigation of the mass associated with each eigenmode can give an indication of which modes are important. In Figure 5 and Figure 6, the ratio between the modal mass and the total mass is shown for all computed eigenmodes on a linear and logarithmic scale, respectively. It can be seen that over 80% of the mass in both horizontal directions is represented already by the first two modes.

The most important modes for each direction are shown in Figure 7, Figure 8, and Figure 9.

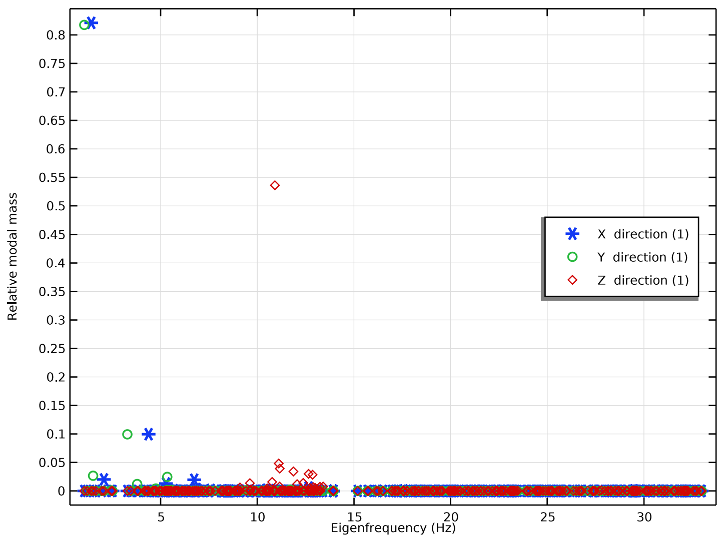

Figure 5: The relative modal mass as function of eigenfrequency.

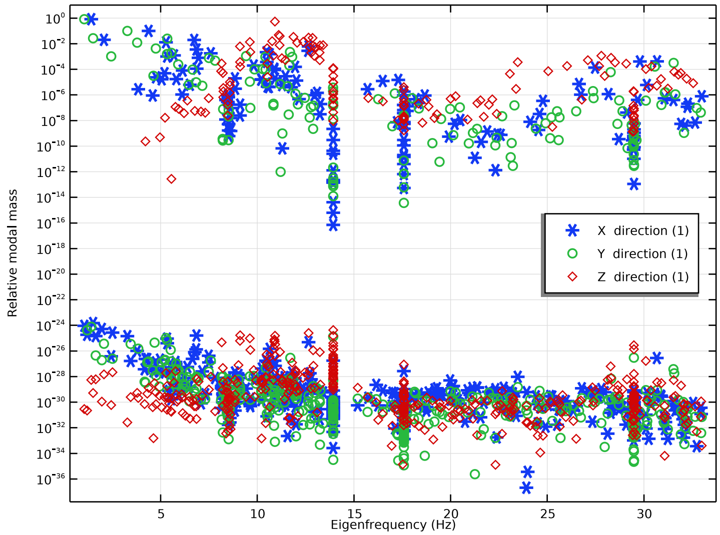

Figure 6: The relative modal mass as function of eigenfrequency; log scale.

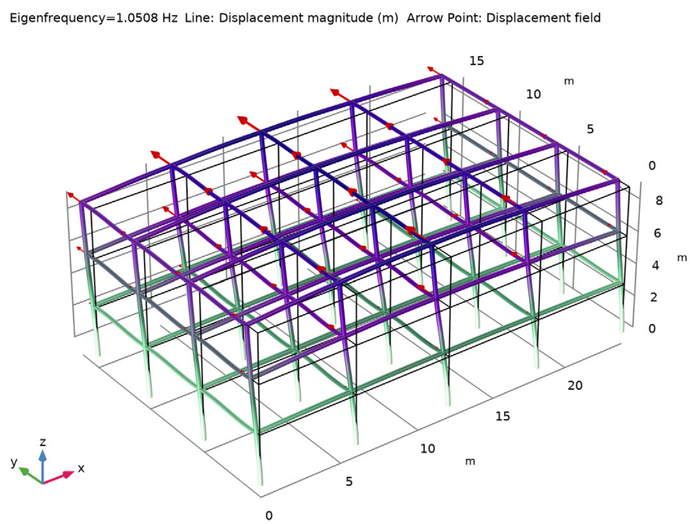

Figure 7: The first eigenmode; bending in the Y direction.

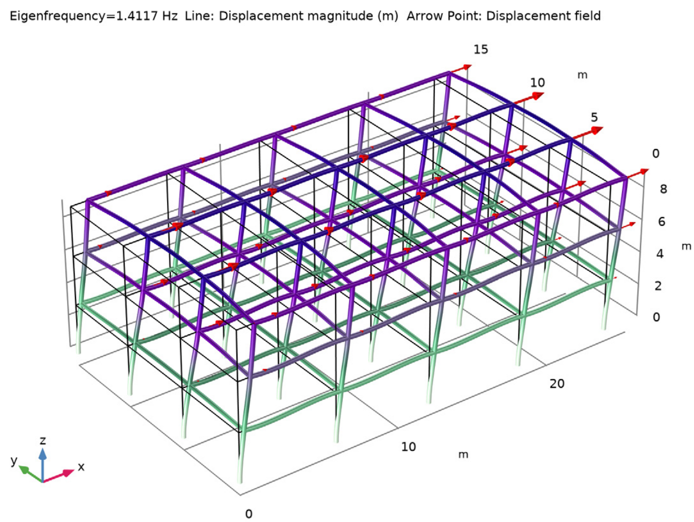

Figure 8: The second eigenmode; bending in the X direction.

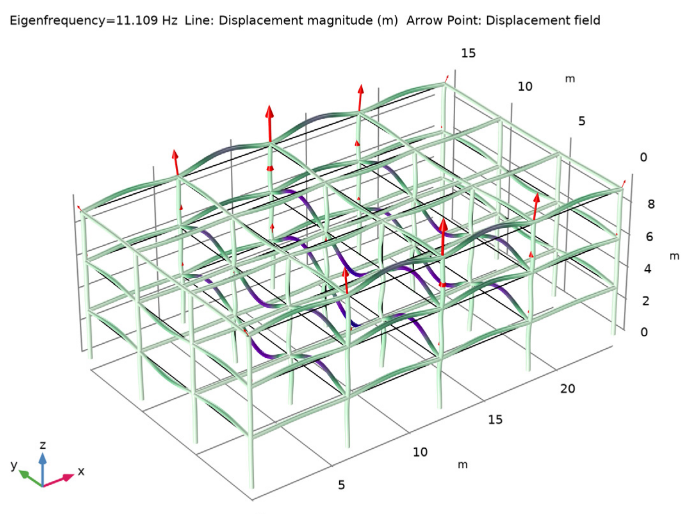

Figure 9: The first mode having a significant vertical displacement.

A **Combine Solutions** node is used to filter the set of computed eigenmodes into a set of modes to be included in the response spectrum analysis. The criterion for inclusion of a mode is

$$m_{\mathrm{modal},k} > 0.01 m_{\mathrm{total}} \tag{1}$$

for at least one direction k. After such filtering, only 23 modes are retained.

First, a standard response spectrum analysis is performed, for which all modes are considered as being fully periodic. The stress and displacement results are shown in Figure 10 and Figure 11, respectively. Note that displacements in a response spectrum analysis are relative to the fixed base which is the ground in this case.

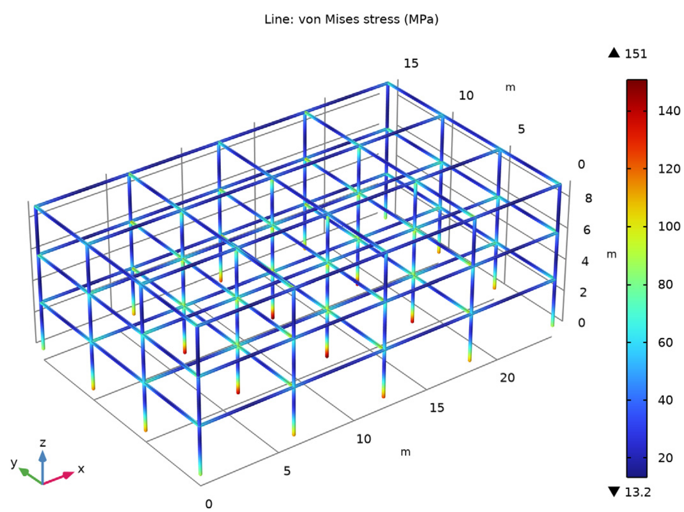

Figure 10: von Mises stress with CQC solution.

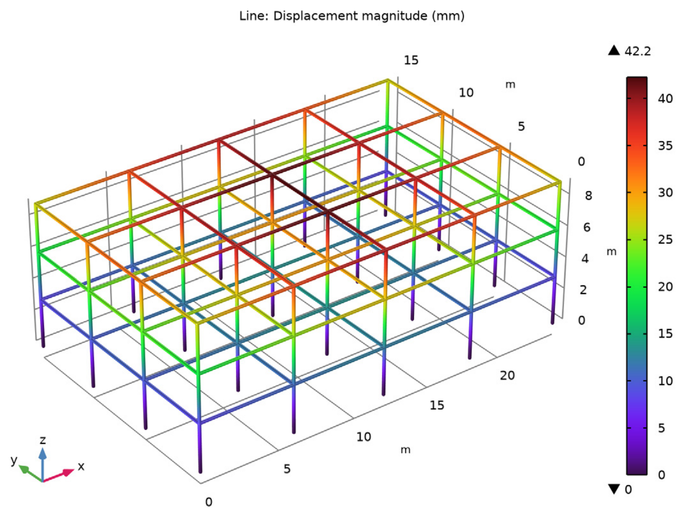

Figure 11: Total displacement with CQC solution.

Certain standards require more sophisticated analysis methods. In the next step, two extensions are made:

- A split is made between *periodic* and *rigid* modes, using the Gupta method. Modes with higher natural frequencies are called rigid. The highest frequency at which the modes are fully periodic is set to 10 Hz, and the lowest frequency where the modes are fully in phase with each other is set to 20 Hz.
- A missing mass correction is used, which provides an additional stationary load containing the mass forces from masses not represented by the eigenmodes. The zero period frequency (the frequency where the acceleration in the response spectrum equals the peak acceleration of the earthquake) is set to 30 Hz.

The stress results from such evaluation are shown in Figure 12. In this case, the deviation from the first evaluation is small; the largest stress is still 151 MPa. The reason is that in this case, the response is dominated by the periodic modes.

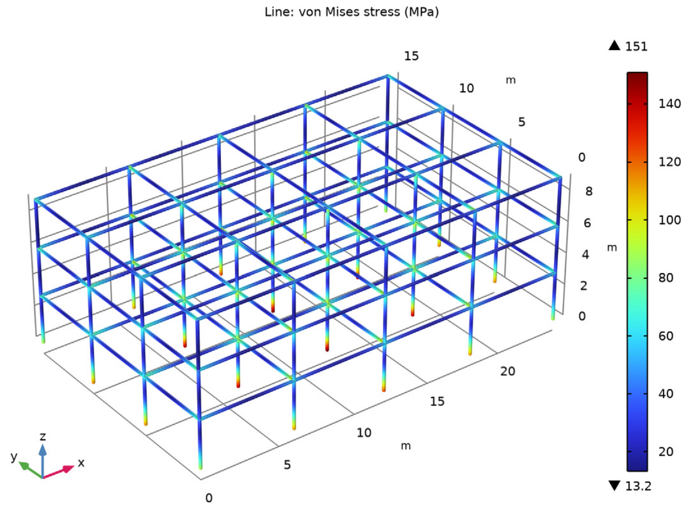

Figure 12: von Mises stress after including a missing mass correction.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/building_response_spectrum`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Response Spectrum**.

6 Click **Done**.

GLOBAL DEFINITIONS

Geometry Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Geometry Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `building_response_spectrum_parameters.txt`.

Horizontal Spectrum Parameters

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Horizontal Spectrum Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `building_response_spectrum_horspec_parameters.txt`.

Vertical Spectrum Parameters

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Vertical Spectrum Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `building_response_spectrum_vertspec_parameters.txt`.

Horizontal Spectrum

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Piecewise**.

2 In the **Settings** window for **Piecewise**, type `Horizontal Spectrum` in the **Label** text field.

3 In the **Function name** text field, type `hsp`.

4 Locate the **Definition** section. In the **Argument** text field, type `T`.

5 Find the **Intervals** subsection. Click **Load from File**.

6 Browse to the model's Application Libraries folder and double-click the file `building_response_spectrum_horspec_function.txt`.

7 Click **Create Plot**.

RESULTS

Horizontal Pseudoacceleration Spectrum

1 In the **Settings** window for **1D Plot Group**, type `Horizontal Pseudoacceleration Spectrum` in the **Label** text field.

2 Locate the **Plot Settings** section.

3 Select the **x-axis label** check box. In the associated text field, type `Period (s)`.

4 In the **y-axis label** text field, type `S (m/s^2)`.

Function 1

1 In the **Model Builder** window, expand the **Horizontal Pseudoacceleration Spectrum** node, then click **Function 1**.

2 In the **Settings** window for **Function**, locate the **Output** section.

3 From the **Extrapolation** list, choose **Right**.

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

5 In the **Horizontal Pseudoacceleration Spectrum** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Vertical Spectrum

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Piecewise**.

2 In the **Settings** window for **Piecewise**, type `Vertical Spectrum` in the **Label** text field.

3 In the **Function name** text field, type `vsp`.

4 Locate the **Definition** section. In the **Argument** text field, type `T`.

5 Find the **Intervals** subsection. Click **Load from File**.

6 Browse to the model's Application Libraries folder and double-click the file `building_response_spectrum_vertspec_function.txt`.

7 Click **Create Plot**.

RESULTS

Vertical Pseudoacceleration Spectrum

1 In the **Settings** window for **1D Plot Group**, type `Vertical Pseudoacceleration Spectrum` in the **Label** text field.

2 Locate the **Plot Settings** section.

3 Select the **x-axis label** check box. In the associated text field, type `Period (s)`.

4 In the **y-axis label** text field, type `S (m/s^2)`.

Function 1

1 In the **Model Builder** window, expand the **Vertical Pseudoacceleration Spectrum** node, then click **Function 1**.

2 In the **Settings** window for **Function**, locate the **Output** section.

3 From the **Extrapolation** list, choose **Right**.

4 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

5 In the **Vertical Pseudoacceleration Spectrum** toolbar, click **Plot**.

Horizontal Pseudoacceleration Spectrum, Vertical Pseudoacceleration Spectrum

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Horizontal Pseudoacceleration Spectrum** and **Vertical Pseudoacceleration Spectrum**.

2 Right-click and choose **Group**.

Design Response Spectra

1 In the **Settings** window for **Group**, type `Design Response Spectra` in the **Label** text field.

2 In the **Model Builder** window, collapse the **Design Response Spectra** node.

GEOMETRY 1

Column

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, type `Column` in the **Label** text field.

3 Locate the **Coordinates** section. In the table, enter the following settings:

x (m)	**y (m)**	**z (m)**
0	0	0
0	0	WZ

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Columns` in the **Name** text field.

6 Click **OK**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **pol1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `colX`.

5 In the **y size** text field, type `colY`.

6 In the **z size** text field, type `nFloors`.

7 Locate the **Displacement** section. In the **x** text field, type `WX`.

8 In the **y** text field, type `WY`.

9 In the **z** text field, type `WZ`.

10 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Columns**.

11 Click **Build Selected**.

12 Click the **Zoom Extents** button in the **Graphics** toolbar.

Horizontal X direction

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, type `Horizontal X direction` in the **Label** text field.

3 Locate the **Coordinates** section. In the table, enter the following settings:

x (m)	**y (m)**	**z (m)**
0	0	WZ
WX	0	WZ

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Horizontal X` in the **Name** text field.

6 Click **OK**.

Array 1 (arr1)

In the **Model Builder** window, right-click **Array 1 (arr1)** and choose **Duplicate**.

Array 2 (arr2)

1 In the **Model Builder** window, click **Array 2 (arr2)**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 Click to select the **Activate Selection** toggle button for **Input objects**.

4 From the **Input objects** list, choose **Horizontal X**.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Horizontal X**.

6 Locate the **Size** section. In the **x size** text field, type `colX-1`.

7 Click **Build Selected**.

Horizontal Y direction

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, type `Horizontal Y direction` in the **Label** text field.

3 Locate the **Coordinates** section. In the table, enter the following settings:

x (m)	y (m)	z (m)
0	0	WZ
0	WY	WZ

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Horizontal Y` in the **Name** text field.

6 Click **OK**.

Array 2 (arr2)

Right-click **Array 2 (arr2)** and choose **Duplicate**.

Array 3 (arr3)

1 In the **Model Builder** window, click **Array 3 (arr3)**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 Click to select the **Activate Selection** toggle button for **Input objects**.

4 From the **Input objects** list, choose **Horizontal Y**.

5 Locate the **Size** section. In the **x size** text field, type `colX`.

6 In the **y size** text field, type `colY-1`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Horizontal Y**.

8 Click **Build All Objects**.

Foundation

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Foundation` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Point**.

4 Locate the **Box Limits** section. In the **z maximum** text field, type `WZ/2`.

BEAM (BEAM)

Cross Section Data - Columns

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross-Section Data 1**.

2 In the **Settings** window for **Cross-Section Data**, type `Cross Section Data - Columns` in the **Label** text field.

3 Locate the **Cross-Section Definition** section. From the **Section type** list, choose **Box**.

4 In the h_y text field, type `300[mm]`.

5 In the h_z text field, type `200[mm]`.

6 In the t_y text field, type `10[mm]`.

7 In the t_z text field, type `10[mm]`.

Section Orientation 1

1 In the **Model Builder** window, click **Section Orientation 1**.

2 In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

1	X
0	Y
0	Z

Cross Section: Horizontal X (HEA260)

1 In the **Physics** toolbar, click **Edges** and choose **Cross-Section Data**.

2 In the **Settings** window for **Cross-Section Data**, type `Cross Section: Horizontal X (HEA260)` in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **Horizontal X**.

4 Locate the **Cross-Section Definition** section. From the **Section type** list, choose **H-profile**.

5 In the h_y text field, type `250[mm]`.

6 In the h_z text field, type `260[mm]`.

7 In the t_y text field, type `12.5[mm]`.

8 In the t_z text field, type `7.5[mm]`.

Section Orientation 1

1 In the **Model Builder** window, click **Section Orientation 1**.

2 In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the V vector as

0	X
0	Y
1	Z

Cross Section: Horizontal X (HEA260)

In the **Model Builder** window, right-click **Cross Section: Horizontal X (HEA260)** and choose **Duplicate**.

Cross Section: Horizontal Y (HEA220)

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross Section: Horizontal X (HEA260) 1**.

2 In the **Settings** window for **Cross-Section Data**, type `Cross Section: Horizontal Y (HEA220)` in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **Horizontal Y**.

4 Locate the **Cross-Section Definition** section. In the h_y text field, type `210[mm]`.

5 In the h_z text field, type `220[mm]`.

6 In the t_y text field, type `11[mm]`.

7 In the t_z text field, type `7[mm]`.

Fixed Constraint 1

In the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

Added Mass - Horizontal X

1 In the **Physics** toolbar, click **Edges** and choose **Added Mass**.
2 In the **Settings** window for **Added Mass**, type `Added Mass - Horizontal X` in the **Label** text field.
3 Locate the **Edge Selection** section. From the **Selection** list, choose **Horizontal X**.
4 Locate the **Added Mass** section. In the ρ_L text field, type `1000`.
5 Right-click **Added Mass - Horizontal X** and choose **Duplicate**.

Added Mass - Horizontal Y

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Added Mass - Horizontal X 1**.
2 In the **Settings** window for **Added Mass**, type `Added Mass - Horizontal Y` in the **Label** text field.
3 Locate the **Edge Selection** section. From the **Selection** list, choose **Horizontal Y**.

Fixed Constraint 1

1 In the **Model Builder** window, click **Fixed Constraint 1**.
2 In the **Settings** window for **Fixed Constraint**, locate the **Point Selection** section.
3 From the **Selection** list, choose **Foundation**.

You will start by computing the effects of gravity as a model verification.

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Structural steel**.
4 Right-click and choose **Add to Component 1 (comp1)**.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.
2 In the **Settings** window for **Edge**, locate the **Edge Selection** section.
3 From the **Selection** list, choose **All edges**.

Distribution 1

1 Right-click **Edge 1** and choose **Distribution**.

2 Right-click **Distribution 1** and choose **Build All**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

Add and run a stationary study to check the gravity load.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Right-click and choose **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: GRAVITY

1 In the **Settings** window for **Study**, type `Study: Gravity` in the **Label** text field.

2 In the **Home** toolbar, click **Compute**.

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Stress (beam)** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

Deformation

1 In the **Model Builder** window, expand the **Line 1** node, then click **Deformation**.

2 In the **Stress (beam)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study: Gravity/Solution 1 (sol1)>Beam>Beam Orientation (beam)**.

4 Click **Add Plot** in the window toolbar.

5 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Beam Orientation (beam), Stress (beam)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (beam)** and **Beam Orientation (beam)**.
2 Right-click and choose **Group**.

Gravity Results

1 In the **Settings** window for **Group**, type `Gravity Results` in the **Label** text field.
2 In the **Model Builder** window, collapse the **Gravity Results** node.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3 In the **Approximate number of eigenfrequencies** text field, type `600`.
4 In the **Maximum number of eigenfrequencies** text field, type `1000`.
5 Find the **Symmetry and consistency settings** subsection. Clear the **Perform consistency check** check box.
6 In the **Model Builder** window, click **Study 1**.
7 In the **Settings** window for **Study**, type `Study: Eigenfrequency` in the **Label** text field.

Remove Low Mass Modes

1 In the **Study** toolbar, click **Combine Solutions**.
2 In the **Settings** window for **Combine Solutions**, type `Remove Low Mass Modes` in the **Label** text field.
3 Locate the **Combine Solutions Settings** section. From the **Solution operation** list, choose **Remove solutions**.
4 From the **Exclude method** list, choose **Implicit**.
5 In the **Excluded if** text field, type `(comp1.rsp1.mEffLX<comp1.rsp1.mass*massTol)&&(comp1.rsp1.mEffLY<comp1.rsp1.mass*massTol)&&(comp1.rsp1.mEffLZ<comp1.rsp1.mass*massTol)`.
6 In the **Study** toolbar, click **Compute**.

Once the study is run, set the **Combined Solutions** node to point to the stored full eigenvalue solution as input. This is necessary in case you want to modify the filtering, and then just rerun the second study step.

7 In the **Model Builder** window, click **Step 2: Remove Low Mass Modes**.

8 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

9 From the **Solution** list, choose **Study: Eigenfrequency/Solution 2 (sol2)**.

10 From the **Use** list, choose **Study: Eigenfrequency/Solution Store 1 (sol3)**.

RESULTS

Examine how the modes contribute to the total mass.

Total Modal Mass

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Total Modal Mass` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Eigenfrequency/ Solution 2 (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Response Spectrum 1> Effective modal mass>rsp1.mEffLX - Effective modal mass, X-translation - kg**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
rsp1.mEffLY	kg	Effective modal mass, Y-translation
rsp1.mEffLZ	kg	Effective modal mass, Z-translation

6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Integral**.

7 From the **Method** list, choose **Summation**.

8 Click **Evaluate**.

True Mass

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `True Mass` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Eigenfrequency/ Solution 2 (sol2)**.

4 From the **Eigenfrequency selection** list, choose **First**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Response Spectrum 1>rsp1.mass - Mass - kg**.

6 Click **Evaluate**.

Relative Modal Mass Contribution

1 In the **Results** toolbar, click (8.5) **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Relative Modal Mass Contribution` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`rsp1.mEffLX/rsp1.mass`	1	`X direction`
`rsp1.mEffLY/rsp1.mass`	1	`Y direction`
`rsp1.mEffLZ/rsp1.mass`	1	`Z direction`

4 Locate the **Data** section. From the **Dataset** list, choose **Study: Eigenfrequency/Solution Store 1 (sol3)**.

5 Click **Evaluate**.

TABLE 3

1 Go to the **Table 3** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 6** click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

Relative Modal Mass

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Relative Modal Mass` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Relative modal mass`.

6 In the **Relative Modal Mass** toolbar, click **Plot**.

7 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

8 In the **Relative Modal Mass** toolbar, click **Plot**.

Mode Shape (beam)

In the **Model Builder** window, click **Mode Shape (beam)**.

Arrow Point 1

1 In the **Mode Shape (beam)** toolbar, click **More Plots** and choose **Arrow Point**.

2 In the **Settings** window for **Arrow Point**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Line 1**.

Deformation 1

1 Right-click **Arrow Point 1** and choose **Deformation**.

2 In the **Mode Shape (beam)** toolbar, click **Plot**.

Mode Shape (beam)

1 In the **Model Builder** window, under **Results** click **Mode Shape (beam)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **1.4117**.

4 In the **Mode Shape (beam)** toolbar, click **Plot**.

5 From the **Eigenfrequency (Hz)** list, choose **11.109**.

6 In the **Mode Shape (beam)** toolbar, click **Plot**.

Mode Shape (beam), Relative Modal Mass

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Mode Shape (beam)** and **Relative Modal Mass**.

2 Right-click and choose **Group**.

Eigenfrequency Results

1 In the **Settings** window for **Group**, type `Eigenfrequency Results` in the **Label** text field.

2 In the **Model Builder** window, collapse the **Eigenfrequency Results** node.

Add a response spectrum dataset, and enter the evaluation settings.

Response Spectrum 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Response Spectrum 3D**.

2 In the **Settings** window for **Response Spectrum 3D**, locate the **Data** section.

3 From the **Eigenfrequency dataset** list, choose **Study: Eigenfrequency/Solution 2 (sol2)**.

4 Locate the **Spectra** section. From the **Depends on** list, choose **Period time**.

5 From the **Primary horizontal spectrum** list, choose **Horizontal Spectrum (hsp)**.

6 From the **Secondary horizontal spectrum** list, choose **Horizontal Spectrum (hsp)**.

7 From the **Vertical spectrum** list, choose **Vertical Spectrum (vsp)**.

3D Plot Group 7

In the **Results** toolbar, click **3D Plot Group**.

Line 1

Right-click **3D Plot Group 7** and choose **Line**.

Stress, CQC Method

1 In the **Settings** window for **3D Plot Group**, type `Stress, CQC Method` in the **Label** text field.

2 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Line 1

1 In the **Model Builder** window, click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 In the **Expression** text field, type `beam.mises`.

4 From the **Unit** list, choose **MPa**.

5 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

Stress, CQC Method

1 In the **Model Builder** window, click **Stress, CQC Method**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Response Spectrum 3D 1**.

4 In the **Stress, CQC Method** toolbar, click **Plot**.

5 Right-click **Stress, CQC Method** and choose **Duplicate**.

Displacement, CQC Method

1 In the **Model Builder** window, under **Results** click **Stress, CQC Method 1**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement, CQC Method` in the **Label** text field.

Line 1

1 In the **Model Builder** window, expand the **Displacement, CQC Method** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 In the **Expression** text field, type `beam.disp`.

4 From the **Unit** list, choose **mm**.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>Spectrum** in the tree.

7 Click **OK**.

8 In the **Displacement, CQC Method** toolbar, click **Plot**.

DEFINITIONS

Investigate the influence of incorporating a correction for the mass not represented by the eigenmodes. In order to do so, you need to set up a particular structure of load cases. This is done from the **Response Spectrum** node.

Response Spectrum 1 (rsp1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Response Spectrum 1 (rsp1)**.

2 In the **Settings** window for **Response Spectrum**, click in the upper-right corner of the **Response Spectrum** section. Locate the **Response Spectrum** section. Click **Create Missing Mass Correction Study** in the upper-right corner of the section.

   This action added load groups under **Global Definitions** (**Load Groups for Missing Mass Correction**), loads in the Beam interface (**Loads for Missing Mass Correction**), and a new study (**Study: Missing Mass Load Cases**).

STUDY: MISSING MASS LOAD CASES

Step 4: Missing Mass Static Load Cases

The gravity load (or any other external loads) should not be part of the missing mass analysis.

1 In the **Model Builder** window, expand the **Study: Missing Mass Load Cases** node, then click **Step 4: Missing Mass Static Load Cases**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Beam (beam)>Gravity 1**.

5 Right-click and choose **Disable**.

6 In the **Home** toolbar, click **Compute**.

RESULTS

Response Spectrum 3D 1

In the **Model Builder** window, under **Results>Datasets** right-click **Response Spectrum 3D 1** and choose **Duplicate**.

Response Spectrum 3D 2

1 In the **Model Builder** window, click **Response Spectrum 3D 2**.

2 In the **Settings** window for **Response Spectrum 3D**, locate the **Combination** section.

3 From the **Rigid modes** list, choose **Gupta**.

4 In the **Frequency limit for pure periodic modes** text field, type `10`.

5 In the **Frequency limit for pure rigid modes** text field, type `20`.

6 In the **Zero period acceleration frequency** text field, type `30`.

7 From the **Mass correction** list, choose **Missing mass method**.

8 Locate the **Data** section. From the **Missing mass load cases dataset** list, choose **Study: Missing Mass Load Cases/Solution 4 (sol4)**.

Displacement, CQC Method, Stress, CQC Method

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress, CQC Method** and **Displacement, CQC Method**.

2 Right-click and choose **Duplicate**.

Stress, CQC Method with Missing Mass Correction

1 In the **Model Builder** window, under **Results** click **Stress, CQC Method 1**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, CQC Method with Missing Mass Correction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Response Spectrum 3D 2**.

4 In the **Stress, CQC Method with Missing Mass Correction** toolbar, click **Plot**.

5 Click **Plot**.

Displacement, CQC Method with Missing Mass Correction

1 In the **Model Builder** window, under **Results** click **Displacement, CQC Method 1**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement, CQC Method with Missing Mass Correction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Response Spectrum 3D 2**.

4 In the **Displacement, CQC Method with Missing Mass Correction** toolbar, click **Plot**.