



Model created in COMSOL Multiphysics 6.4

# Earthquake Analysis of a Building

## Introduction

---

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

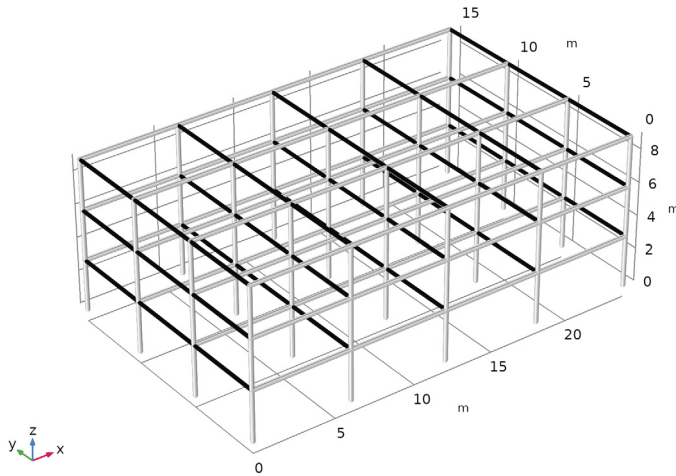
Earthquakes are, by their nature, not deterministic. If you were to check that a building can sustain an earthquake by time domain analysis, you would have to run a large number of analyses with different representative prescribed acceleration histories for the supports. Such an approach would require 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 provided 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,  $\nu = 0.30$ , and  $\rho = 7850$  kg/m<sup>3</sup>.

## **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.

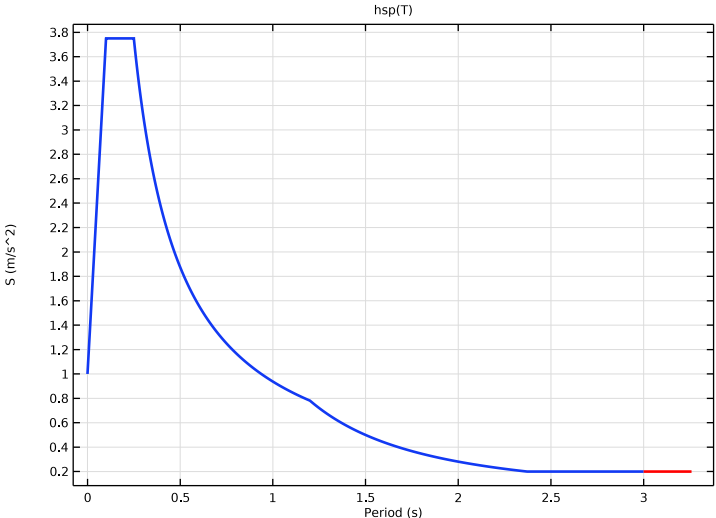


Figure 2: Horizontal design response spectrum.

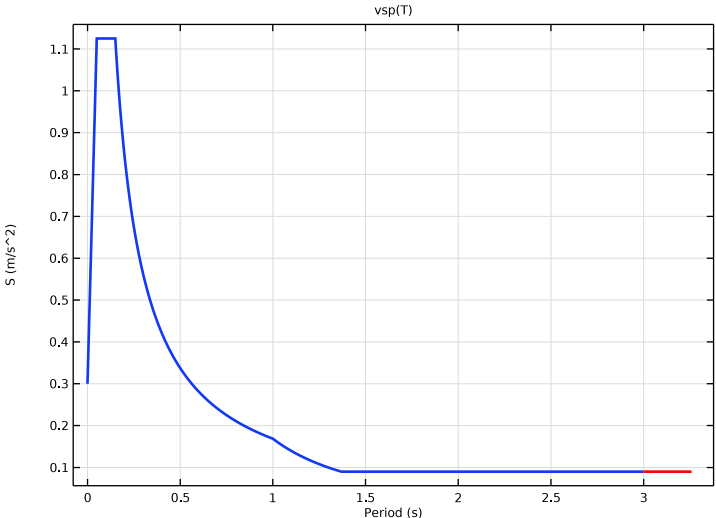
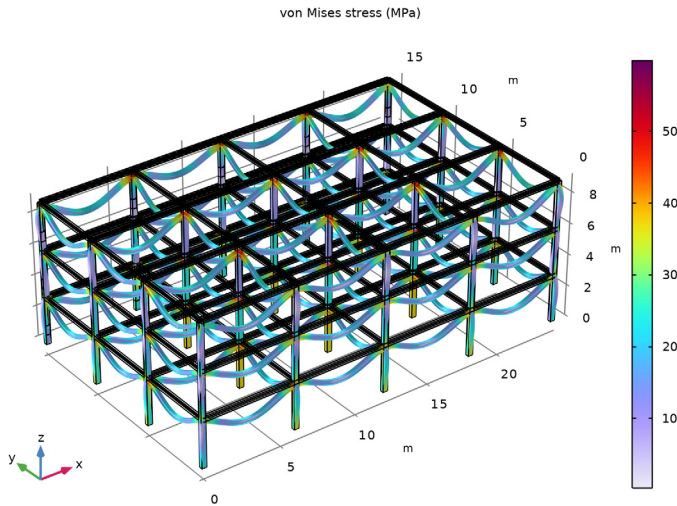


Figure 3: Vertical design response spectrum.

### **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](#).

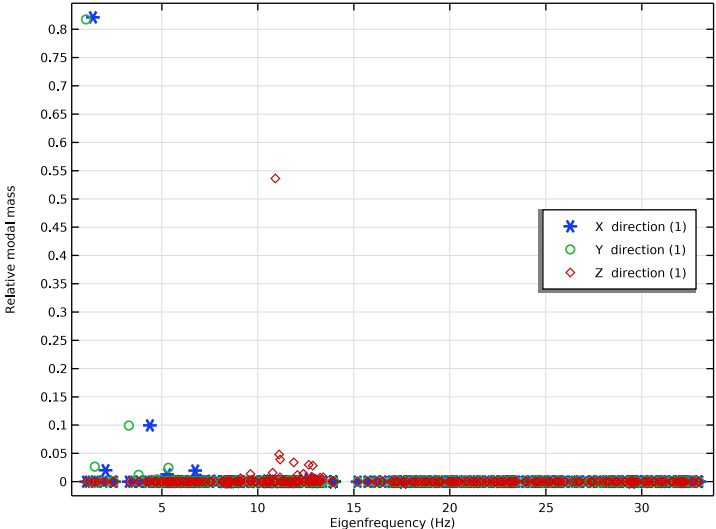


Figure 5: The relative modal mass as function of eigenfrequency.

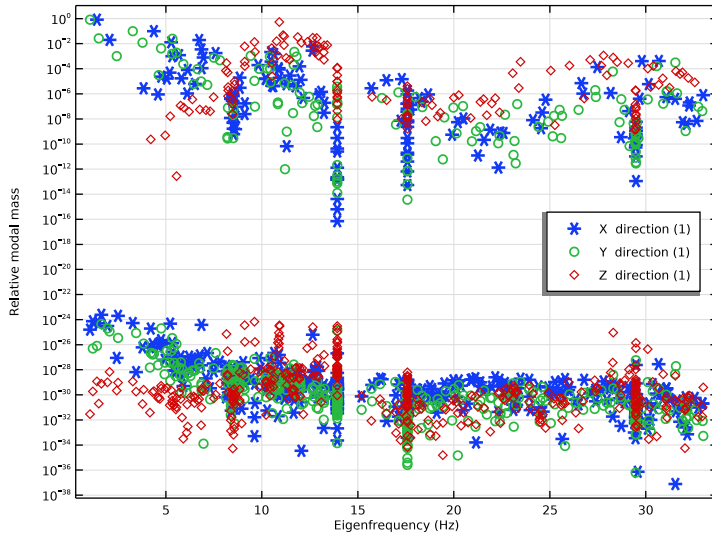


Figure 6: The relative modal mass as function of eigenfrequency; log scale.

Eigenfrequency=1.0508 Hz Line: Displacement magnitude (m) Arrow Point: Displacement field

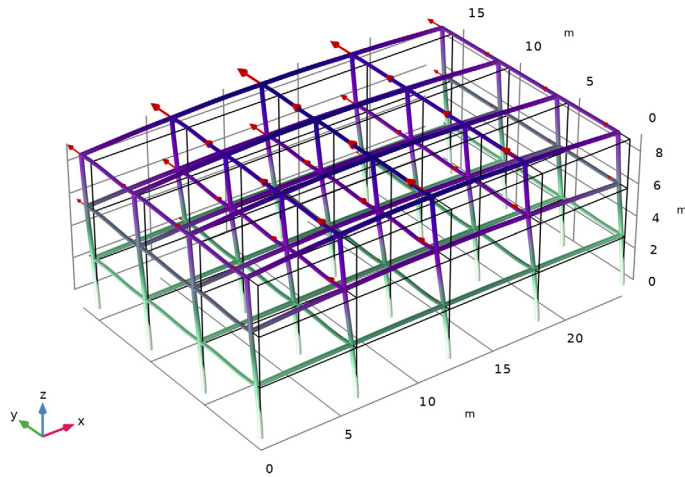


Figure 7: The first eigenmode; bending in the Y direction.

Eigenfrequency=1.4117 Hz Line: Displacement magnitude (m) Arrow Point: Displacement field

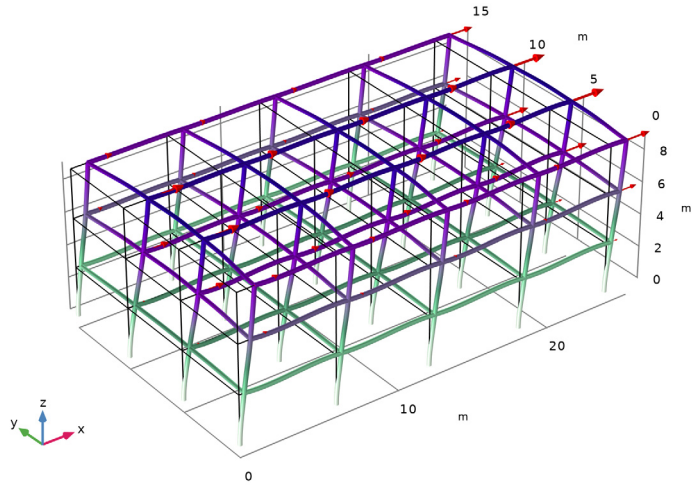


Figure 8: The second eigenmode; bending in the X direction.

Eigenfrequency=11.109 Hz Line: Displacement magnitude (m) Arrow Point: Displacement field

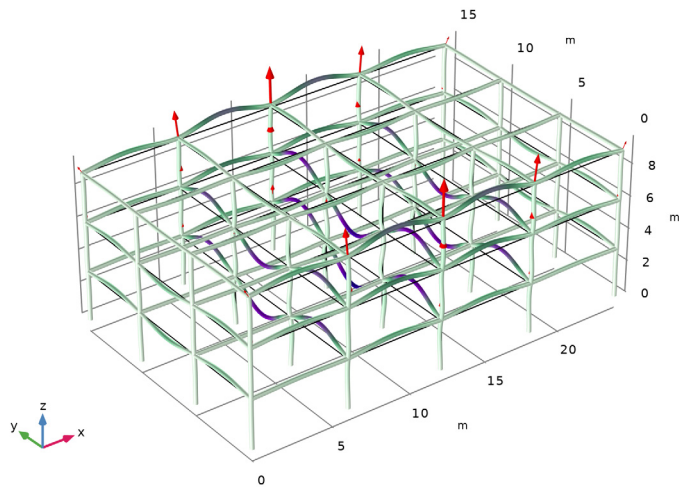


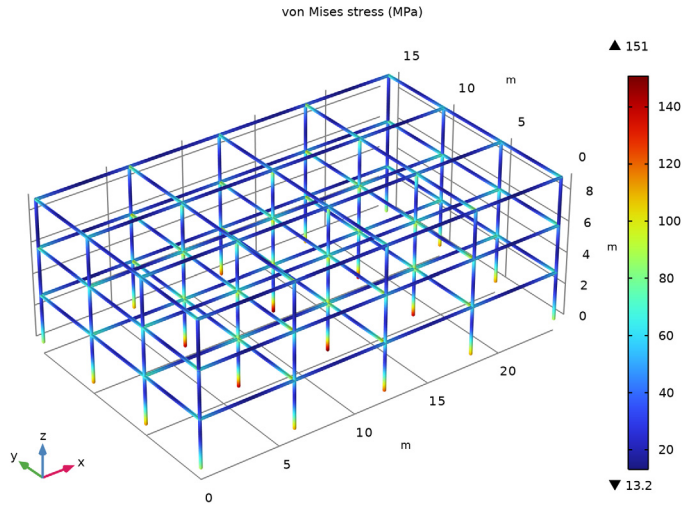
Figure 9: The first mode having a significant vertical displacement.

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

$$m_{\text{modal},k} > 0.01 m_{\text{total}} \quad (1)$$

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.*

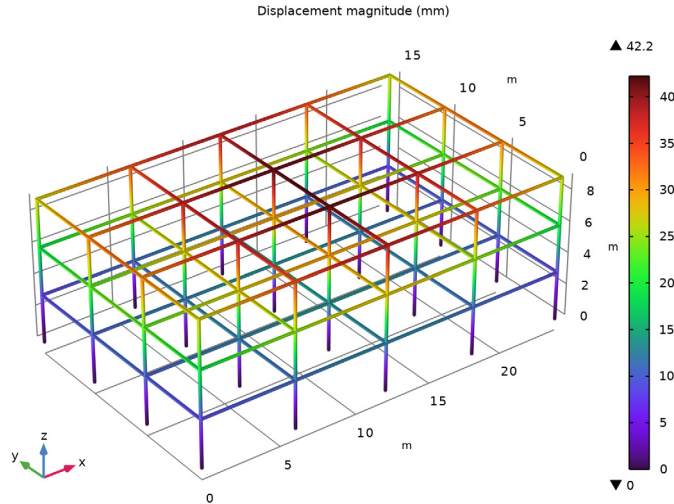


Figure 11: Total displacement with CQC solution.

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.

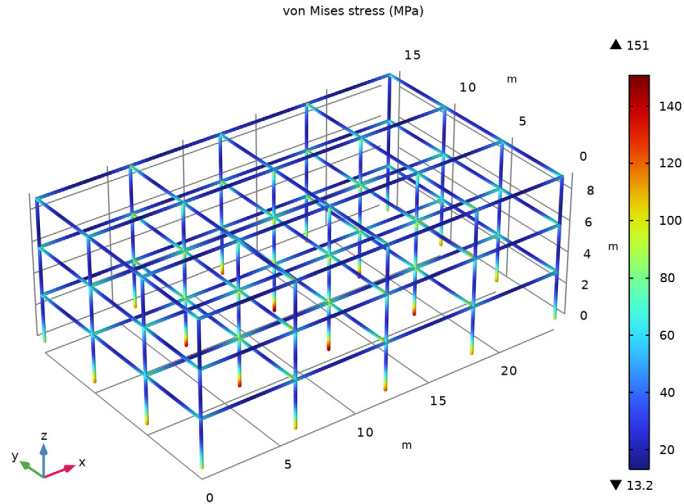


Figure 12: von Mises stress after including a missing mass correction.

---

**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 I**.

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  **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  **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  **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 **ID Plot Group**, type Horizontal Pseudoacceleration Spectrum in the **Label** text field.

2 Locate the **Plot Settings** section.

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

4 Select the **y-axis label** checkbox. In the associated 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  **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 **ID Plot Group**, type Vertical Pseudoacceleration Spectrum in the **Label** text field.
- 2 Locate the **Plot Settings** section.
- 3 Select the **x-axis label** checkbox. In the associated text field, type Period (s).
- 4 Select the **y-axis label** checkbox. In the associated 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, 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 **poll** 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, type Horizontal X in the **Name** text field.
- 6 Click **OK**.

### Array 2 (arr2)

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Array 1 (arr1)** and choose **Duplicate**.
- 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, type Horizontal Y in the **Name** text field.
- 6 Click **OK**.

### Array 3 (arr3)

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Array 2 (arr2)** and choose **Duplicate**.
- 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 I*


- 1 In the **Model Builder** window, click **Section Orientation I**.
- 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 Y (HEA220)*

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

#### *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  $\rho_L$  text field, type 1000.

#### *Added Mass - Horizontal Y*

- 1 Right-click **Added Mass - Horizontal X** and choose **Duplicate**.

- 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 **Physics** toolbar, click  **Points** and choose **Fixed Constraint**.
- 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 **Materials** 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 **Materials** 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 **Study** toolbar, click  **Compute**.

### RESULTS

Set default units for result presentation.

#### *Preferred Units I*

1 In the **Results** toolbar, click  **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Select the **Apply conversions to expressions with the same dimensions** checkbox.

4 Click  **Add Physical Quantity**.

5 In the **Physical Quantity** dialog, type stress in the text field.

6 In the tree, select **Solid Mechanics > Stress tensor (N/m<sup>2</sup>)**.

7 Click **OK**.

8 In the **Settings** window for **Preferred Units**, locate the **Units** section.

9 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m <sup>2</sup>	MPa


10 Click  **Apply**.

#### *Stress, 3D (beam)*

In the **Model Builder** window, expand the **Stress, 3D (beam)** node.

#### *Deformation*

1 In the **Model Builder** window, expand the **Results > Stress, 3D (beam) > Surface I** node, then click **Deformation**.


2 In the **Stress, 3D (beam)** toolbar, click  **Plot**.

### RESULT TEMPLATES

1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

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

- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

## RESULTS

*Geometry and Orientation (beam), Stress, 3D (beam)*

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 In the **Model Builder** window, click **Study 1**.
- 6 In the **Settings** window for **Study**, type Study: Eigenfrequency in the **Label** text field.

*Remove Low Mass Modes*

- 1 In the **Study** toolbar, click  **More Study Extensions** and choose **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  $(comp1.rsp1.mEffLX < comp1.rsp1.mass * massTo1) \&\& (comp1.rsp1.mEffLY < comp1.rsp1.mass * massTo1) \&\& (comp1.rsp1.mEffLZ < comp1.rsp1.mass * massTo1)$ .
- 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  **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 the **Table Graph** button in the window toolbar.

### RESULTS

#### Table Graph 1

- 1 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
- 2 Find the **Line style** subsection. From the **Line** list, choose **None**.
- 3 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

### Relative Modal Mass


- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 6**.
- 2 In the **Settings** window for **ID 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** checkbox. 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** checkbox.

#### *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 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**.

#### *Displacement, CQC Method*

- 1 Right-click **Stress, CQC Method** and choose **Duplicate**.
- 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. From the **Color table** list, choose **Spectrum**.
- 6 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**, locate the **Response Spectrum** section.
- 3 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** checkbox.
- 4 In the tree, select **Component 1 (comp1) > Beam (beam) > Gravity 1**.
- 5 Right-click and choose **Disable**.
- 6 In the **Study** toolbar, click  **Compute**.

## RESULTS

### *Response Spectrum 3D 2*



- 1 In the **Model Builder** window, under **Results > Datasets** right-click **Response Spectrum 3D 1** and choose **Duplicate**.
- 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 **Settings** window for **3D Plot Group**, type **Stress, CQC Method with Missing Mass Correction** in the **Label** text field.
- 2 Locate the **Data** section. From the **Dataset** list, choose **Response Spectrum 3D 2**.
- 3 In the **Stress, CQC Method with Missing Mass Correction** toolbar, click  **Plot**.
- 4 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**.