



Model created in COMSOL Multiphysics 6.4

Bracket – Random Vibration Analysis

Introduction

The various examples based on a bracket geometry form a suite of tutorials that summarizes the fundamentals of modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

This example shows how to perform a random vibration analysis for a structure using power spectral density (PSD). The computations are based on the modal reduced-order model (ROM).



In the *Structural Mechanics Modeling* chapter of the *Structural Mechanics Module User's Guide*:

- *Random Vibration (PSD) Study*
- *Performing a Random Vibration Analysis*

Model Definition

The model geometry is represented in [Figure 1](#).

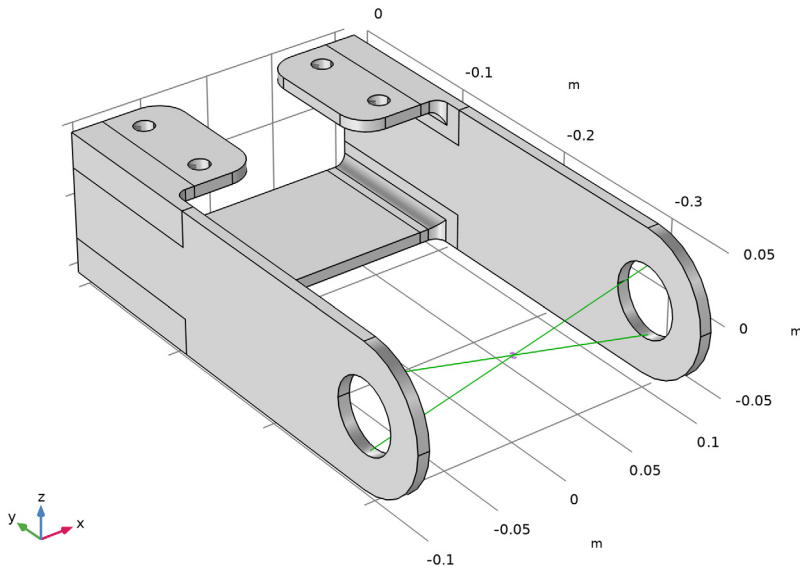


Figure 1: Bracket geometry. A rigid connector and its center of rotation are shown using special physics symbols.

The bracket is made of steel. Four smaller holes in the bracket are fully constrained to represent a bolted connection.

A pin connects two larger holes in the bracket arms. The pin is assumed to be perfectly rigid and is modeled with a rigid connector. Thus, the pin is not represented in the model geometry.

In reality, the mass and moment of inertia of the pin should also be taken into account in the analysis. It is ignored here, in order to keep the example shorter.

The rigid connector has the translational and rotational degrees of freedom of a rigid body. Options for this feature include applying an external load to such a rigid body. In this tutorial, you use two components of the load as random excitations applied to the structure. The input PSD for the load components are shown in [Figure 2](#).

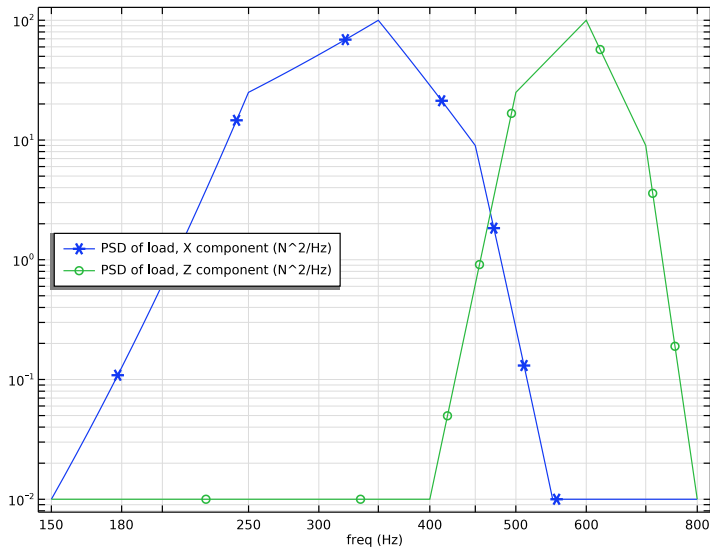


Figure 2: Input PSD for random load components. Log-log scale is used.

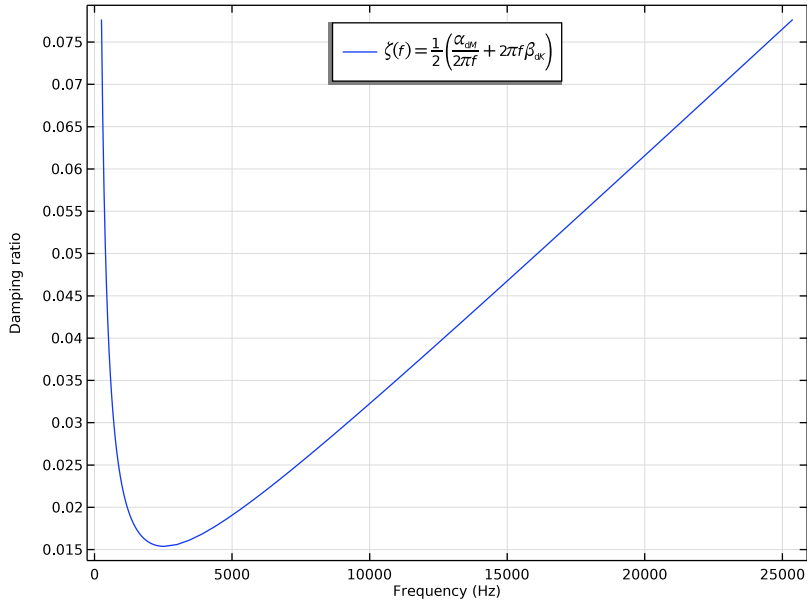


Figure 3: The damping ratio curve.

Rayleigh damping is chosen here with relative damping set to 0.05 and 0.03 at the frequencies 400 Hz and 700 Hz, respectively. The damping ratio curve is shown in Figure 3.

The two applied load components are assumed to be uncorrelated. Physically, this means that it is two different sources of excitation which cause the load in the two directions.

You will compute the displacement response at a point with coordinates: $X = 0$, $Y = 0$, $Z = -0.045$, which is a representative location in the lower plate of the bracket.

Results and Discussion

Figure 4 shows the PSD for two displacement components, calculated for the frequency range that contains the first three natural frequencies of the structure. Note that there are three peaks on the resulting PSD. The left and right ones correspond to two natural frequencies of the structure, while the middle peak corresponds to the maximum of the input excitation.

The cross-correlation between the two displacement components is shown in Figure 5.

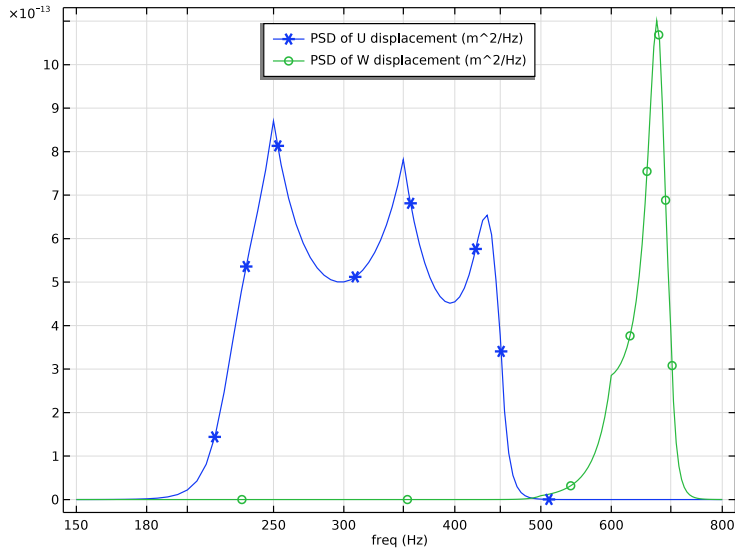


Figure 4: PSD for the horizontal (U) and vertical (W) displacement response at the selected location.

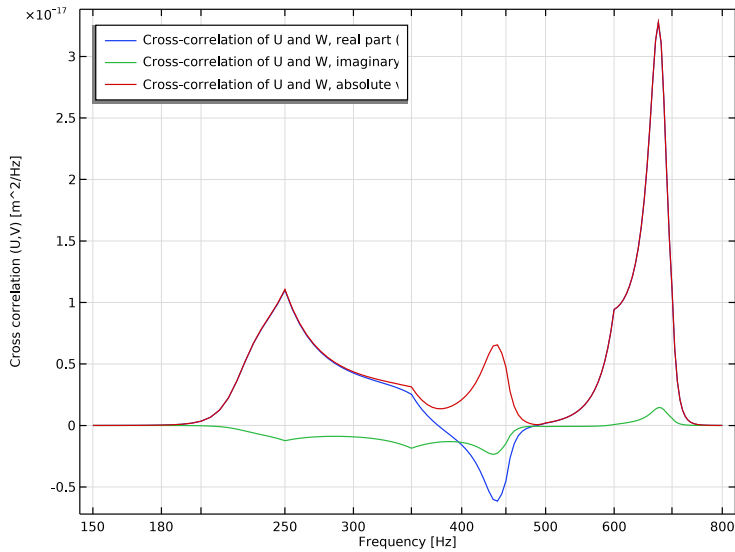


Figure 5: Cross-correlation between the horizontal and vertical displacement response at the selected location.

Figure 6 and Figure 7 show the PSD for the horizontal and vertical displacement, respectively, computed at a particular frequency.

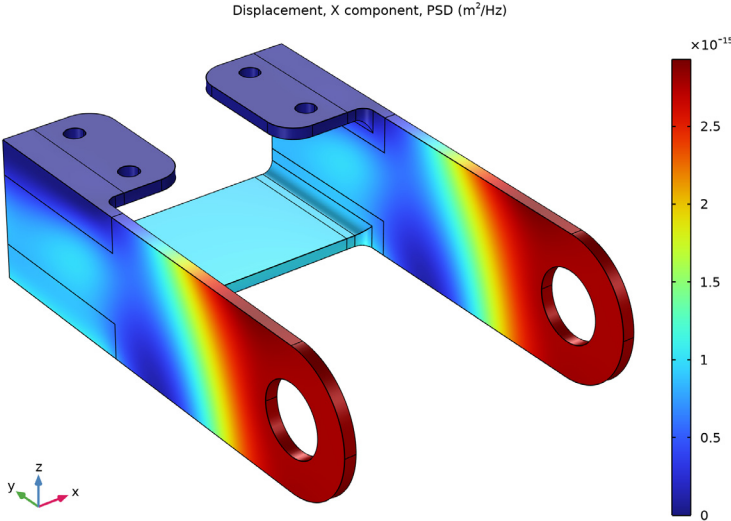


Figure 6: PSD of u -displacement component (m^2/Hz).

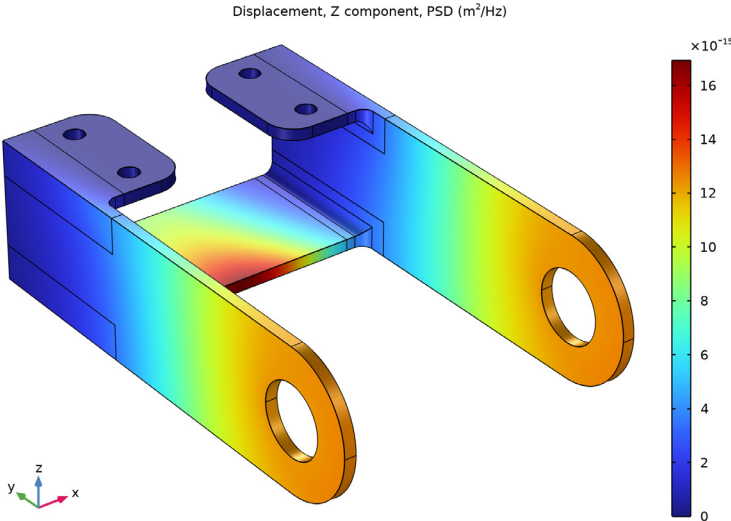


Figure 7: PSD of w -displacement component (m^2/Hz).

Figure 8, Figure 9, and Figure 10 show the RMS for the horizontal and vertical displacements, as well as for the displacement magnitude, computed using integration of their corresponding PSD over the frequency range from 150 to 800 Hz.

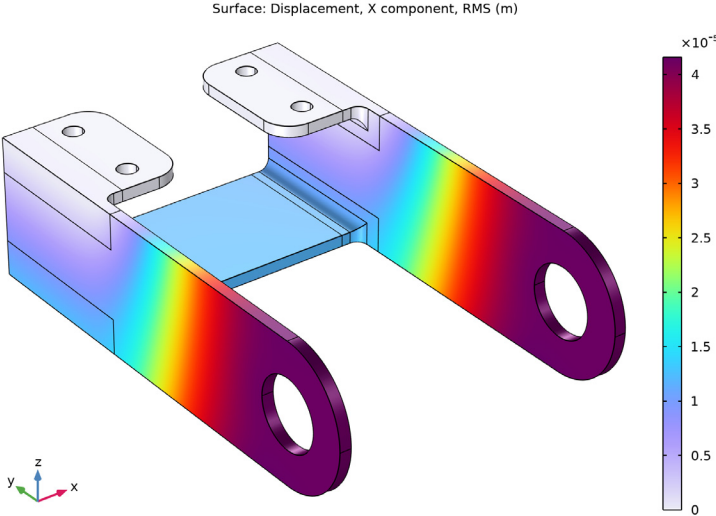


Figure 8: RMS of u -displacement component (m).

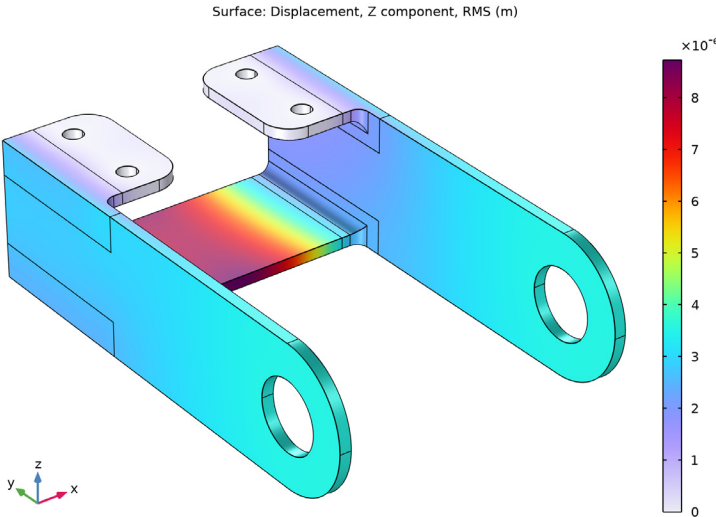


Figure 9: RMS of w -displacement component (m).

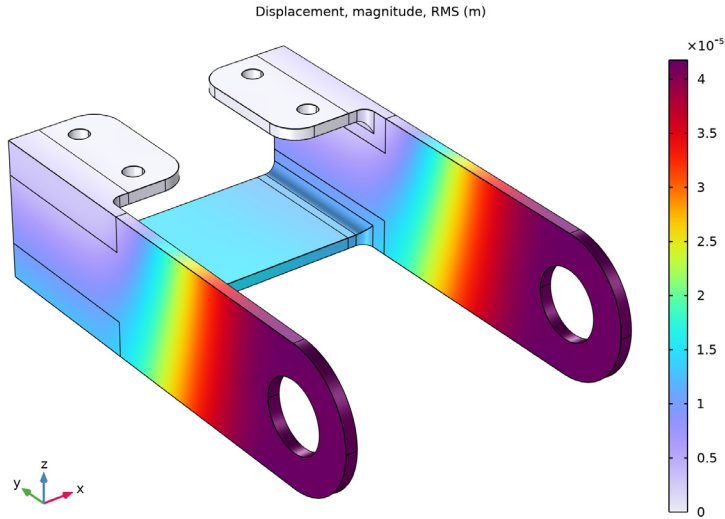


Figure 10: RMS of displacement magnitude (m).

Since the horizontal displacement is dominant, [Figure 8](#) and [Figure 10](#) are almost identical.

In [Figure 11](#), the RMS of the total acceleration is shown. It can be noted that the distribution over the structure differs fundamentally from that of the displacement ([Figure 10](#)). The reason is that the vertical motion of the central section of the bracket is related to a higher eigenfrequency than the horizontal motion of the two arms. This is why accelerations are more pronounced than displacements in this part of the bracket.

In [Figure 10](#), the RMS of the von Mises equivalent stress is shown.

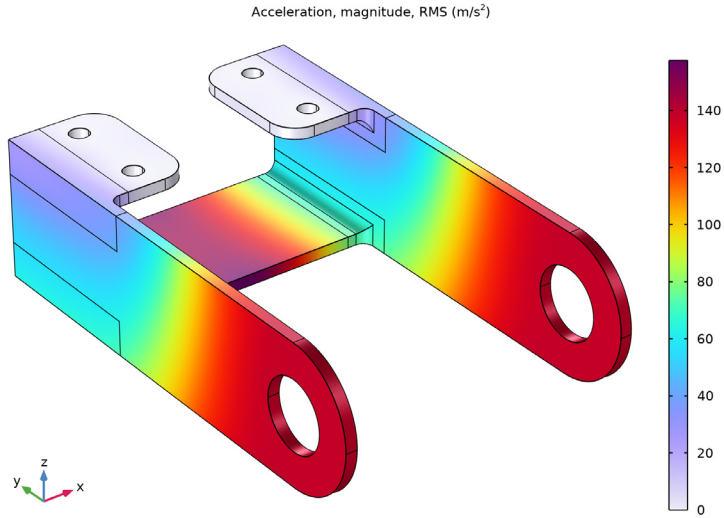


Figure 11: RMS of acceleration magnitude (m/s^2).

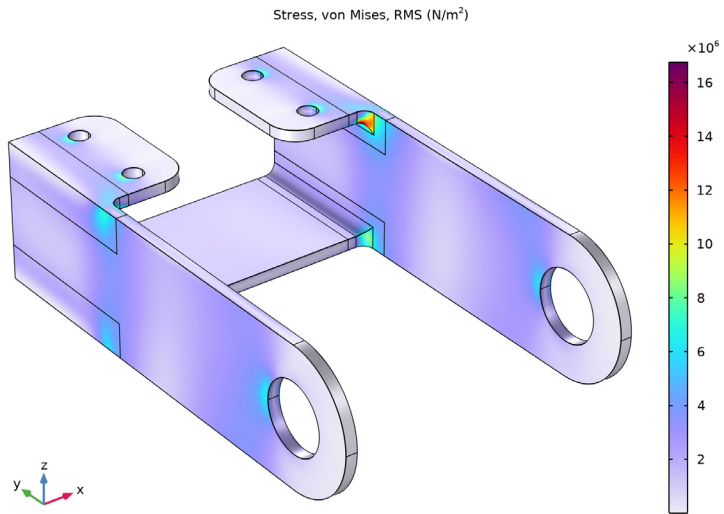


Figure 12: RMS of von Mises stress (Pa).

The interpretation of results from a random vibration analysis requires special attention. Usually, the RMS is the important result. The RMS is defined as

$$u_{\text{RMS}} = \sqrt{\frac{1}{T} \int_0^T u(t)^2 dt} \quad (1)$$

for some long time T . In many cases, you are more interested in the maximum value than in the RMS. For many statistical distributions, for example the normal distribution, the tail of the distribution extends to infinity. Mathematically the response can then reach an arbitrarily high value with a probability that increases with time.

In practice, it is customary to assume that the maximum value does not exceed the RMS value multiplied by a certain factor. This factor is often taken as 3 or 4. Thus, the maximum stress can be estimated to 50 to 65 MPa.

For vibrating structures, fatigue is often a more important aspect than the maximum stress. In the Fatigue Module, you can find an example where this model is extended by a fatigue analysis.

Notes About the COMSOL Implementation

The simplest way to set up a random vibration analysis in COMSOL Multiphysics is to add a predefined study called **Random Vibration (PSD)**. This will automatically add two study nodes configured to act as a sequence.

The first added study is an eigenfrequency analysis that serves as a basis for the modal reduced-order model (ROM) required for the system response analysis. Note that any **Damping** nodes should be manually disabled under this study node for all structural interfaces in the model. This is needed because the eigenmode extraction should be always done for undamped system.

It is enough to compute the last of the two studies. This will automatically compute the requested number of eigenfrequencies and then create a ROM based on the computed eigenmodes.

When you add a **Random Vibration (PSD)** study, a **Reduced-Order Modeling** node with subnodes will be also added automatically under **Global Definitions**. These nodes are used for specifying the model control parameters (inputs) and to enter expressions for the input PSD to be used in the random vibration computations. Such computations are performed as postprocessing steps. However, if the PSD expressions are changed under **Global Definitions**, a solution update might be needed.

The evaluations are done using special operators that become available as part of the random vibration analysis.

The input PSD as functions of the frequency are often specified using linear interpolation in the log-log representation of the data. You can see how to set up such functions in the model.

Two action buttons are provided in the **Damping Settings** section in order to visualize the damping ratio with respect to frequency. The first button shows a dynamic preview plot of the damping ratio, while the second button generates a plot in the **Results** node.


Application Library path: Structural_Mechanics_Module/Tutorials/
bracket_random_vibration

Modeling Instructions



ROOT

This file serves as starting point for various examples based on a bracket geometry. It sets up the model geometry, material, and mesh. It also adds a Solid Mechanics interface with a fixed constraint applied to the bolt holes.

APPLICATION LIBRARIES

- 1 From the **File** menu, choose **Application Libraries**.
- 2 In the **Application Libraries** window, select **Structural Mechanics Module > Tutorials > bracket_basic** in the tree.
- 3 Click  **Open**.

ADD STUDY


- 1 In the **Home** toolbar, click  **Add Study** to open the **Add Study** window.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Random Vibration (PSD)**.
- 4 Right-click and choose **Add Study**.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1) > Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

Damping 1


- 1 In the **Physics** toolbar, click  **Attributes** and choose **Damping**.
- 2 In the **Settings** window for **Damping**, locate the **Damping Settings** section.
- 3 From the **Input parameters** list, choose **Damping ratios**.
- 4 In the f_1 text field, type 400.
- 5 In the ζ_1 text field, type 0.05.
- 6 In the f_2 text field, type 700.
- 7 In the ζ_2 text field, type 0.03.

In order to visualize the damping ratio curve, preview the **Damping Ratio** plot through an action button from the **Damping Settings** section.

- 8 Click **Damping Ratio Preview** in the upper-right corner of the **Damping Settings** section. From the menu, choose **Damping Ratio Preview**.

Now add a rigid connector to the holes in the bracket arms to simulate the presence of a connecting pin.

Rigid Connector 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Rigid Connector**.
- 2 In the **Settings** window for **Rigid Connector**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Pin Holes**.

By default the location of the center of rotation is computed automatically. You can also manually specify its location.

To visualize its position you need to enable the physics symbols.


- 4 In the **Model Builder** window, click **Solid Mechanics (solid)**.
- 5 In the **Settings** window for **Solid Mechanics**, locate the **Physics Symbols** section.
- 6 Select the **Enable physics symbols** checkbox.

The displacement of the rigid connector in the y direction and all rotations are constrained.

- 7 In the **Model Builder** window, click **Rigid Connector 1**.

- 8 In the **Settings** window for **Rigid Connector**, locate the **Prescribed Displacement at Center of Rotation** section.
- 9 Select the **Prescribed in y direction** checkbox.
- 10 Locate the **Prescribed Rotation** section. From the **By** list, choose **Constrained rotation**.
- 11 Select the **Constrain rotation around x-axis** checkbox.
- 12 Select the **Constrain rotation around y-axis** checkbox.
- 13 Select the **Constrain rotation around z-axis** checkbox.

Applied Force 1

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Applied Force**.
Apply a load at the center of rotation of the rigid body.
- 2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.
- 3 Specify the **F** vector as

F _x	x
0	y
F _z	z

STUDY 1

Step 1: Eigenfrequency

You need to disable the damping for this study because the eigenmode computation should be performed for the undamped system. The damping will however be used in the following modal reduced-order model and random response analysis.

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.
- 2 In the **Settings** window for **Eigenfrequency**, 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) > Solid Mechanics (solid) > Linear Elastic Material 1 > Damping 1**.
- 5 Right-click and choose **Disable**.

GLOBAL DEFINITIONS

Set up two control parameters to be used as loads.

Global Reduced-Model Inputs 1

- 1 In the **Model Builder** window, expand the **Global Definitions > Reduced-Order Modeling** node, then click **Global Reduced-Model Inputs 1**.
- 2 In the **Settings** window for **Global Reduced-Model Inputs**, locate the **Reduced-Model Inputs** section.
- 3 In the table, enter the following settings:

Control name	Expression
Fx	100[N]
Fz	100[N]

DEFINITIONS

Variables 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
Set up variables to evaluate the displacement components at a selected point.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:


Name	Expression	Unit	Description
U	at3(0, 0, -0.045, u)	m	Displacement, X component
W	at3(0, 0, -0.045, w)	m	Displacement, Z component

STUDY 2

Frequency Domain 1

- 1 In the **Model Builder** window, expand the **Step 1: Model Reduction** node, then click **Frequency Domain 1**.
- 2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
- 3 In the **Frequencies** text field, type 500.

STUDY 1

In the **Study** toolbar, click  **Compute**.

STUDY 2


The computation of the solution for Study 2 will find the requested number of eigenfrequencies and create a reduced-order model based on the computed eigenmodes.

1 Click  **Compute**.

GLOBAL DEFINITIONS

Next, set up the input PSD for the random loads.

PSD of load

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, type PSD of load in the **Label** text field.
- 3 Locate the **Definition** section. In the **Function name** text field, type PSD.
- 4 In the table, enter the following settings:

t	f(t)
400	0.01
500	5 ²
600	10 ²
700	3 ²
800	0.01

- 5 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	Hz

- 6 In the **Function** table, enter the following settings:

Function	Unit
PSD	N ² /Hz

- 7 Click to expand the **Data Transformation for Interpolation** section. From the **Argument** list, choose **Logarithmic**.
- 8 From the **Function** list, choose **Logarithmic**.

Random Vibration 1 (rvib1)

Assume that the two random loads are uncorrelated, and have different power spectral densities.

- 1 In the **Model Builder** window, under **Global Definitions > Reduced-Order Modeling** click **Random Vibration 1 (rvib1)**.
- 2 In the **Settings** window for **Random Vibration**, locate the **Power Spectrum** section.

3 In the table, enter the following settings:

Control name	Power spectral density
rom1.Fx	PSD(freq+250 [Hz])
rom1.Fz	PSD(freq)

4 Locate the **Output Operator Settings** section. In the **Lower frequency limit** text field, type 150.

5 In the **Upper frequency limit** text field, type 800.

Update the study to make the input change available for the solution.

STUDY 2

In the **Study** toolbar, click  **Update Solution**.

RESULTS

Global Evaluation Sweep 1

1 In the **Results** toolbar, click  **More Derived Values** and choose **Other > Global Evaluation Sweep**.

Plot the input PSD for the load components in a range of frequencies containing first three natural frequencies of the structure.

2 In the **Settings** window for **Global Evaluation Sweep**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Parameters** section. In the table, enter the following settings:

Parameter name	Parameter value list
freq	range(150,5,800) [Hz]

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

Expression	Unit	Description
PSD(freq+250[Hz])	N ² /Hz	PSD of load, X component
PSD(freq)	N ² /Hz	PSD of load, Z component

6 Click  **Evaluate**.



TABLE 1

1 Go to the **Table 1** 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 markers** subsection. From the **Marker** list, choose **Cycle**.
- 3 From the **Positioning** list, choose **Interpolated**.
- 4 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 5 Click the  **x-Axis Log Scale** button in the **Graphics** toolbar.
- 6 Click the  **y-Axis Log Scale** button in the **Graphics** toolbar.

PSD of Loads

- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 2**.
- 2 In the **Settings** window for **ID Plot Group**, type PSD of Loads in the **Label** text field.
- 3 Locate the **Legend** section. From the **Position** list, choose **Middle left**.

The random response computations can be performed as postprocessing steps using the updated solution. First, evaluate and plot the PSD for the displacement response at the chosen point.

Global Evaluation Sweep 2

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Other > Global Evaluation Sweep**.
- 2 In the **Settings** window for **Global Evaluation Sweep**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Parameter name	Parameter value list
freq	range(150,5,800) [Hz]

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

Expression	Unit	Description
rvib1.psd(U)	m ² /Hz	PSD of U displacement
rvib1.psd(W)	m ² /Hz	PSD of W displacement


- 5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 6 Click  **Evaluate**.


TABLE 2

- 1 Go to the **Table 2** 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 markers** subsection. From the **Marker** list, choose **Cycle**.
- 3 From the **Positioning** list, choose **Interpolated**.
- 4 Click the  **x-Axis Log Scale** button in the **Graphics** toolbar.
- 5 Locate the **Legends** section. Select the **Show legends** checkbox.

PSD of Displacements

- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 3**.
- 2 In the **Settings** window for **ID Plot Group**, type PSD of Displacements in the **Label** text field.
- 3 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

Global Evaluation Sweep 3

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Other > Global Evaluation Sweep**.

Evaluate and plot the cross-correlation for two different components of displacement responses at the chosen point. For verification, you also plot the nonrandom frequency response result.

- 2 In the **Settings** window for **Global Evaluation Sweep**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Parameter name	Parameter value list
freq	range(150,5,800) [Hz]

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

Expression	Unit	Description
real(rvib1.cross(U,W))	m ² /Hz	Cross-correlation of U and W, real part
imag(rvib1.cross(U,W))	m ² /Hz	Cross-correlation of U and W, imaginary part
abs(rvib1.cross(U,W))	m ² /Hz	Cross-correlation of U and W, absolute value




- 5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 6 Click  next to  **Evaluate**, then choose **New Table**.

TABLE 3


- 1 Go to the **Table 3** window.
- 2 Click the **Table Graph** button in the window toolbar.

RESULTS

Table Graph 1


- 1 Click the  **x-Axis Log Scale** button in the **Graphics** toolbar.
- 2 In the **Settings** window for **Table Graph**, locate the **Legends** section.
- 3 Select the **Show legends** checkbox.

Cross Correlation (U,V)

- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 4**.
- 2 In the **Settings** window for **ID Plot Group**, type Cross Correlation (U,V) in the **Label** text field.
- 3 Locate the **Plot Settings** section.
- 4 Select the **x-axis label** checkbox. In the associated text field, type Frequency [Hz].
- 5 Select the **y-axis label** checkbox. In the associated text field, type Cross correlation (U,V) [m^2/Hz].
- 6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.
- 7 In the **Cross Correlation (U,V)** toolbar, click  **Plot**.



Plot the distribution over the structure of the PSD for two displacement components.

PSD, u-displacement

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type PSD, u-displacement in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 1


- 1 Right-click **PSD, u-displacement** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `rvib1.psd(u)`.

- 4 Select the **Description** checkbox. In the associated text field, type Displacement , X component , PSD.
- 5 In the **Unit** field, type m^2/Hz .
- 6 In the **PSD, u-displacement** toolbar, click  **Plot**.
- 7 Click the  **Show Grid** button in the **Graphics** toolbar.

PSD, w-displacement

- 1 In the **Model Builder** window, right-click **PSD, u-displacement** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type PSD, w-displacement in the **Label** text field.

Surface 1

- 1 In the **Model Builder** window, expand the **PSD, w-displacement** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `rvib1.psd(w)`.
- 4 In the **Description** text field, type Displacement , Z component , PSD.
- 5 In the **PSD, w-displacement** toolbar, click  **Plot**.

Plot the distribution over the structure of the RMS of the displacement components.

Random vibration plots can take a long time to generate, so it is a good idea not to replot unless explicitly requested. Also, storing the plots in the saved file can save time when reopening the model.


- 6 In the **Model Builder** window, click **Results**.
- 7 In the **Settings** window for **Results**, locate the **Update of Results** section.
- 8 Select the **Only plot when requested** checkbox.
- 9 Locate the **Save Data in the Model** section. From the **Save plot data** list, choose **On**.

RMS, u-displacement

- 1 In the **Model Builder** window, right-click **PSD, u-displacement** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type RMS, u-displacement in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.
- 4 Find the **Solution** subsection. Clear the **Solution** checkbox.

Surface 1


- 1 In the **Model Builder** window, expand the **RMS, u-displacement** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.

- 3 In the **Expression** text field, type `rvib1.rms(u)`.
- 4 In the **Description** text field, type Displacement, X component, RMS.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.
- 6 In the **RMS, u-displacement** toolbar, click  **Plot**.
The plotting will take some time because it implies integration over the whole selected frequency range at each location in the structure.


RMS, w-displacement

- 1 In the **Model Builder** window, right-click **RMS, u-displacement** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type RMS, w-displacement in the **Label** text field.


Surface 1

- 1 In the **Model Builder** window, expand the **RMS, w-displacement** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `rvib1.rms(w)`.
- 4 In the **Description** text field, type Displacement, Z component, RMS.
- 5 In the **RMS, w-displacement** toolbar, click  **Plot**.

RMS, von Mises stress

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type RMS, von Mises stress in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 1


- 1 Right-click **RMS, von Mises stress** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `rvib1.q2sq(solid.mises_rv)`.
- 4 Select the **Description** checkbox. In the associated text field, type Stress, von Mises, RMS.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.
- 6 In the **RMS, von Mises stress** toolbar, click  **Plot**.

RMS, displacement

- 1 In the **Model Builder** window, right-click **RMS, von Mises stress** and choose **Duplicate**.

- 2 In the **Settings** window for **3D Plot Group**, type RMS, displacement in the **Label** text field.


Surface 1

- 1 In the **Model Builder** window, expand the **RMS, displacement** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `rvib1.q2sq(solid.disp_rv)`.
- 4 In the **Description** text field, type Displacement, magnitude, RMS.
- 5 In the **RMS, displacement** toolbar, click  **Plot**.

RMS, acceleration

- 1 In the **Model Builder** window, right-click **RMS, displacement** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type RMS, acceleration in the **Label** text field.

Surface 1

- 1 In the **Model Builder** window, expand the **RMS, acceleration** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `rvib1.q2sq(solid.utt_rv)`.
- 4 In the **Description** text field, type Acceleration, magnitude, RMS.
- 5 In the **RMS, acceleration** toolbar, click  **Plot**.