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).

Model Definition

The model geometry is represented in Figure 1.

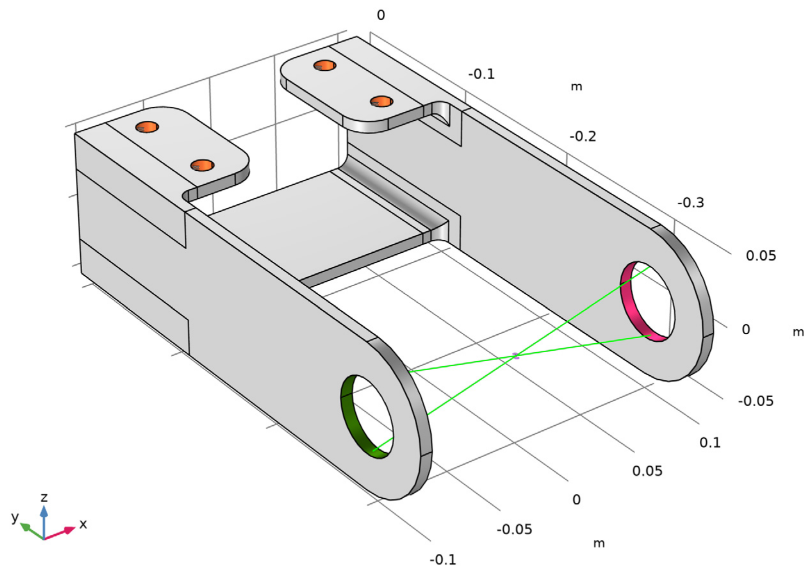

Figure 1: Bracket geometry. Rigid connector and its center of rotation are shown using special physics symbols.

The bracket is made of structural 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.

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 rigid body. In this tutorial, you use two components of such load as random excitations applied to the structure. The input PSD for the load components are shown in Figure 2.

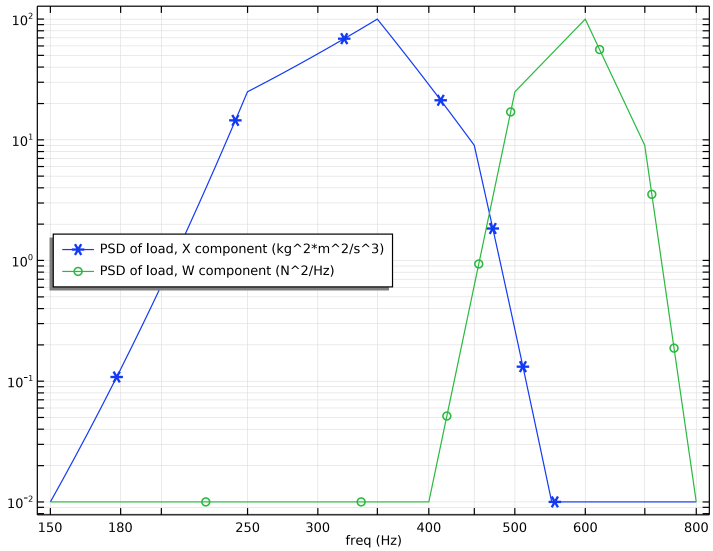

Figure 2: Input PSD for random load components. Log-log scale is used.

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

You will compute the displacement response at a point with coordinates: $X = 0$, $Y = 0$, Z =-0.45, which is a representative location in the lower plate of the bracket.

Results and Discussion

Figure 3 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 4.

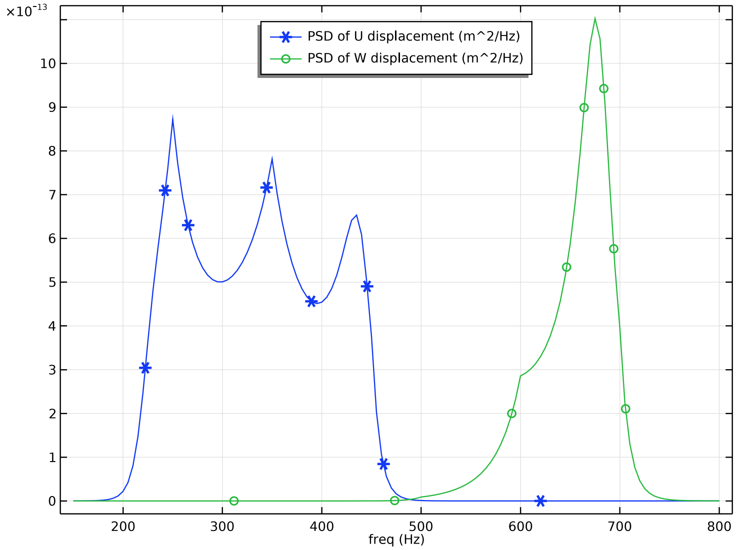

Figure 3: PSD for the horizontal (U) and vertical (W) displacement response at the selected location.

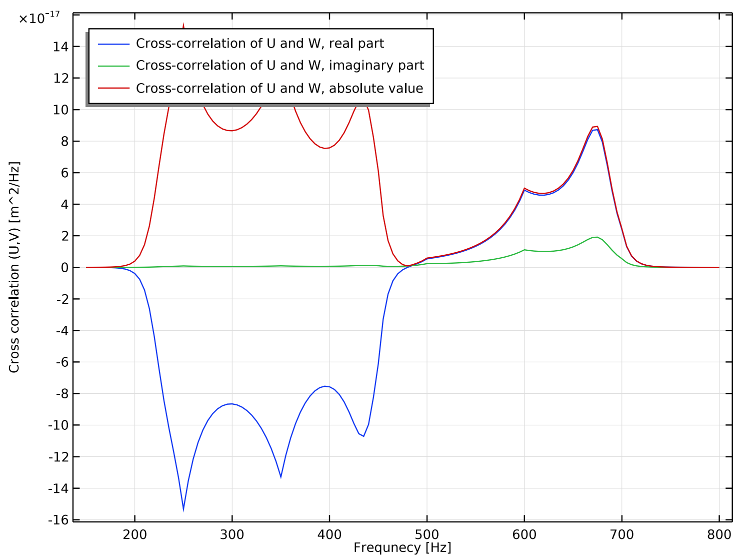

Figure 4: Cross-correlation between the horizontal and vertical displacement response at the selected location.

Figure 5 and Figure 6 show the PSD for the horizontal and vertical displacement, respectively, computed at a particular frequency.

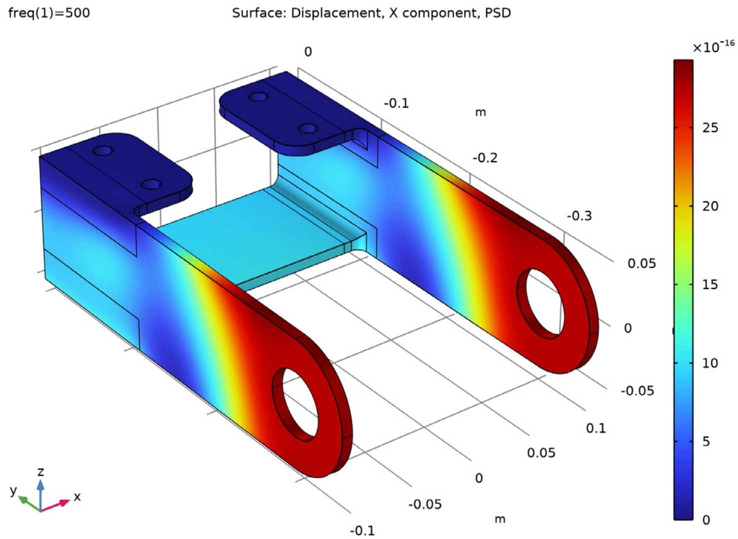

Figure 5: PSD of u-displacement component (m^2/Hz).

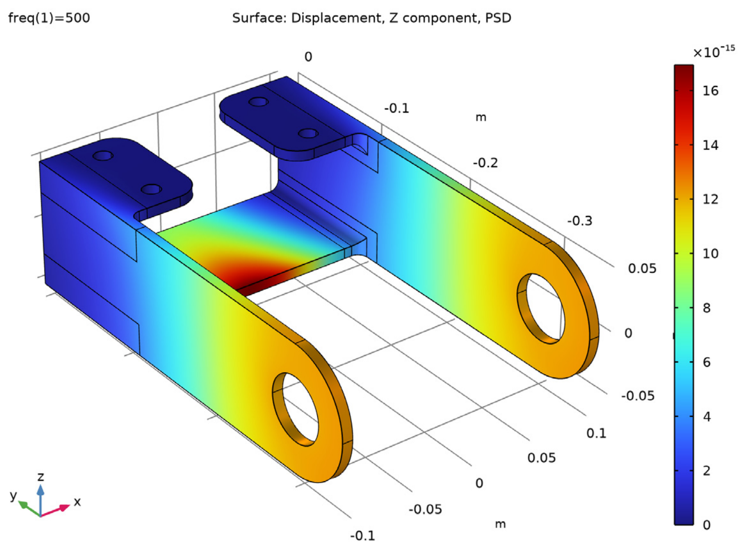

Figure 6: PSD of w-displacement component (m^2/Hz).

Figure 7, Figure 8 and Figure 9 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.

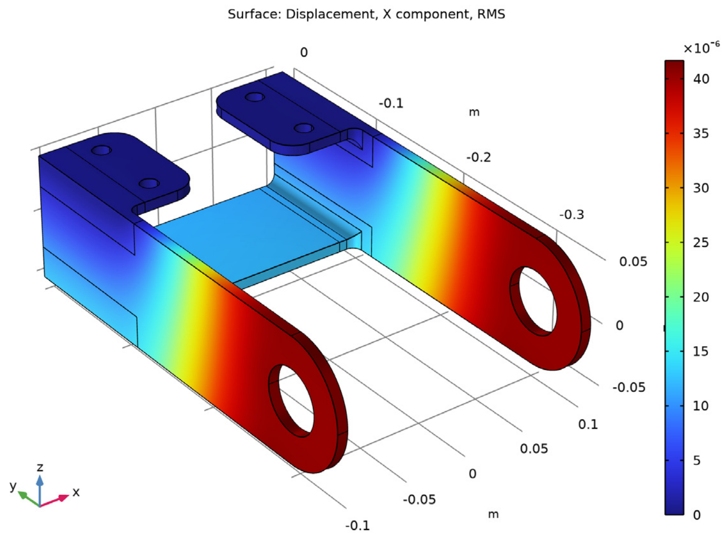

Figure 7: RMS of u-displacement component (m).

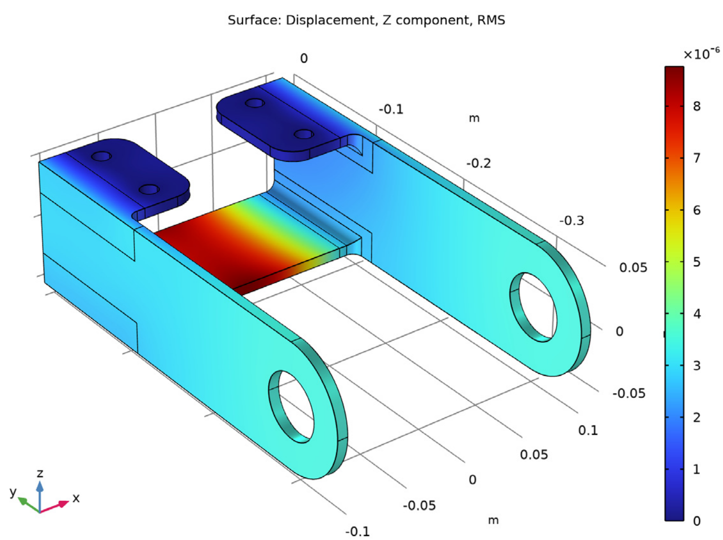

Figure 8: RMS of w-displacement component (m).

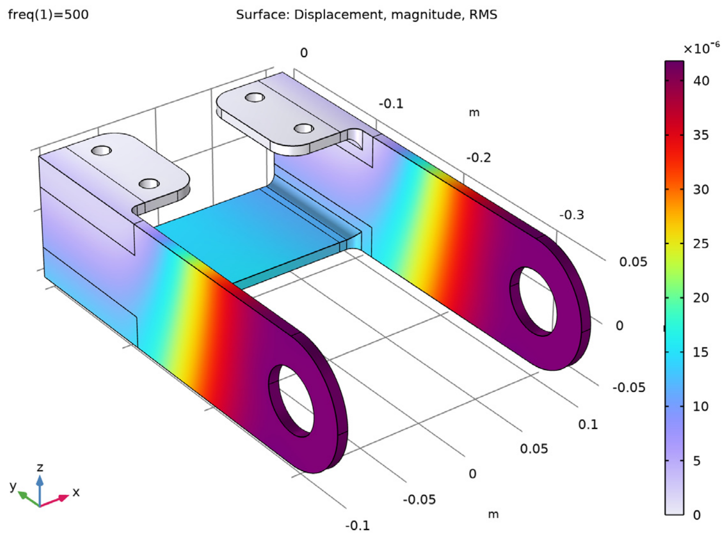

Figure 9: RMS of displacement magnitude (m).

Since the horizontal displacement is dominant, Figure 7 and Figure 9 are almost identical.

In Figure 10, 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 9). 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 9, the RMS of the von Mises equivalent stress is shown.

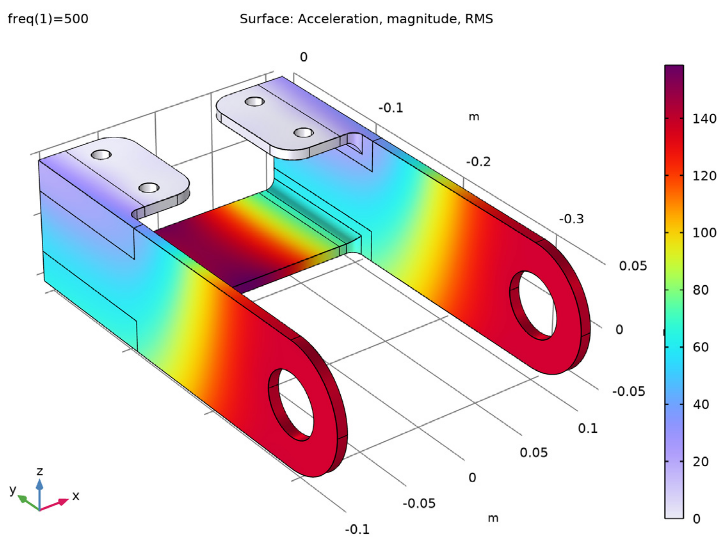

Figure 10: RMS of acceleration magnitude (m/s^2).

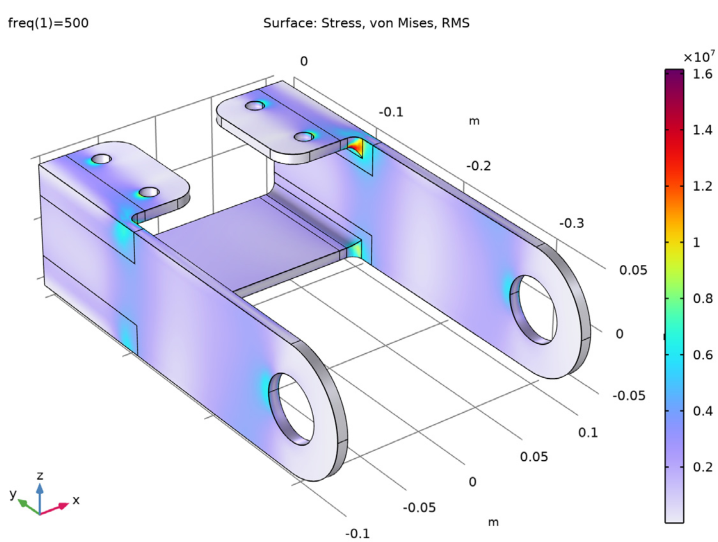

Figure 11: RMS of von Mises stress (Pa)

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

$$u_{\mathrm{RMS}} = \sqrt{\frac{1}{T}\int_{o}^{T} u(t)^2 dt} \quad (1)$$

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 a certain factor times the RMS value. 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. The model shows how to set up such functions.

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`.

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** check box.
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]

SOLID MECHANICS (SOLID)

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** check box.

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** check box.

10 Locate the **Prescribed Rotation** section. From the **By** list, choose **Constrained rotation**.

11 Select the **Constrain rotation around x-axis** check box.

12 Select the **Constrain rotation around y-axis** check box.

13 Select the **Constrain rotation around z-axis** check box.

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

Fx	x
0	y
Fz	z

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 **Home** 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.

Interpolation 1 (int1)

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

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

Piecewise 1 (pw1)

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

2 In the **Settings** window for **Piecewise**, locate the **Definition** section.

3 In the **Argument** text field, type `f`.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
150	800	exp(int1(log(f+250)))

5 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

6 In the **Function** text field, type `N^2/Hz`.

Piecewise 2 (pw2)

1 Right-click **Piecewise 1 (pw1)** and choose **Duplicate**.

2 In the **Settings** window for **Piecewise**, locate the **Definition** section.

3 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
150	800	exp(int1(log(f)))

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 From the **Correlation type** list, choose **Uncorrelated**.

4 In the table, enter the following settings:

Control name	Power spectral density
Fx	pw1(freq)
Fz	pw2(freq)

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

6 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 **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
pw1(freq)	kg^2*m^2/s^3	PSD of load, X component
pw2(freq)	N^2/Hz	PSD of load, W component

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

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 2** click **Table Graph 1**.

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

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

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

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 **1D Plot Group 2**.

2 In the **Settings** window for **1D 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 8.85 e-12 **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)`		`PSD of U displacement (m^2/Hz)`
`rvib1.psd(W)`		`PSD of W displacement (m^2/Hz)`

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

6 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 3** click **Table Graph 1**.

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

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

4 Locate the **Legends** section. Select the **Show legends** check box.

PSD of Displacements

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D 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 8.85 e-12 **More Derived Values** and choose **Other>Global Evaluation Sweep**.

Also evaluate and plot the cross-correlation for two different components of displacement responses at the chosen point. For verification, you also plot the non-random 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))		Cross-correlation of U and W, real part
imag(rvib1.cross(U,W))		Cross-correlation of U and W, imaginary part
abs(rvib1.cross(U,W))		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

1 Go to the **Table** window.

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

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 4** click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

Cross Correlation (U,V)

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Cross Correlation (U,V)` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 In the associated text field, type `Frequnecy [Hz]`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Cross correlation (U,V) [m^2/Hz]`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

8 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 **Home** toolbar, click **Add Plot Group** and choose **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** check box.

5 In the associated text field, type `Displacement, X component, PSD`.

6 In the **Unit** field, type `m^2/Hz`.

7 In the **PSD, u-displacement** toolbar, click **Plot**.

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.

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

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**.

Also 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** check box.

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** check box.

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`.

The arguments of the operator are the expression to evaluate, the minimum and maximum frequencies, and a number of evaluation points in between.

5 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 **PSD, w-displacement** and choose **Duplicate**.

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

3 Locate the **Title** section. From the **Title type** list, choose **Custom**.

4 Find the **Solution** subsection. Clear the **Solution** check box.

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 **Home** toolbar, click **Add Plot Group** and choose **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** check box.

5 In the associated text field, type `Stress, von Mises, RMS`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

7 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 Select the **Description** check box.

5 In the associated text field, type `Displacement, magnitude, RMS`.

6 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 Select the **Description** check box.

5 In the associated text field, type `Acceleration, magnitude, RMS`.

6 In the **RMS, acceleration** toolbar, click **Plot**.