Model created in COMSOL Multiphysics 6.4

Random Vibration Test of a Motherboard

Introduction

Electronic components are often placed in an environment, where they will be subjected to vibrations. For many such components, it may be mandatory to perform vibration tests. The tests are performed by attaching the component to a shaker table, where it will be subjected to a pseudorandom acceleration. The acceleration input has a frequency content given by a specified power spectral density (PSD). The input is sometimes also called acceleration spectral density (ASD).

In order to predict the outcome of such a test, it is possible to perform a simulation using random vibration analysis. This example shows how such an analysis can be done.

The analyzed structure is a motherboard with some components attached.

Model Definition

GEOMETRY

Figure 1 shows a motherboard with a design that is typical for smaller computer devices such as game consoles, for example.

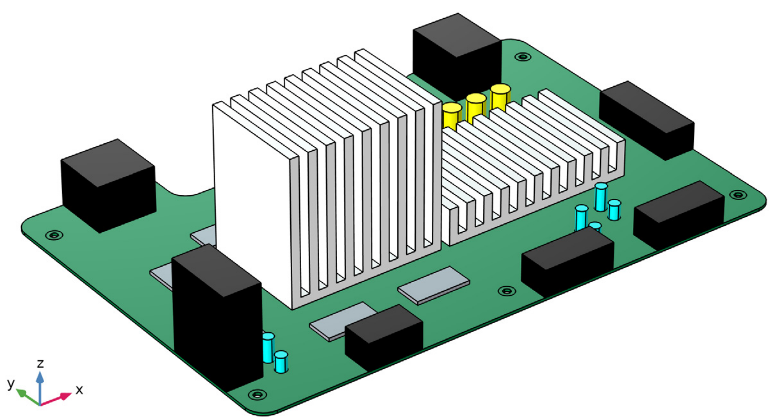

Figure 1: Motherboard geometry.

A processor (CPU) and a graphics chip (GPU) are covered by massive heat sinks used for passive cooling. Memory chips are located next to the CPU unit. A number of cylindrical capacitors of various sizes are scattered over the motherboard. Several connectors for peripherals are located along the motherboard's edges. The board is intended to be attached to the housing via six mounting bolts. The latter are not modeled explicitly. The structure is attached to the shaker table during vibration testing at the same locations.

MATERIAL

The board itself is made of a generic PCB material. The heat sinks are made of aluminum. The chips are modeled as made of silicon. The connectors are modeled as rectangular blocks made of plastic. Some effective material properties are used to represent the capacitors.

Rayleigh damping is assumed, with a relative damping of 0.04 at the frequencies 40 and 1000 Hz.

CONSTRAINTS

All six mounting holes are considered to be fixed to the shaker table. In order to be able to measure the bolt forces, rigid connectors are used instead of ordinary fixed constraints.

LOADS

The loading is provided by an acceleration of the shaker table. The power spectral density for this test is shown in Figure 2. The test is performed three times, where the structure is independently accelerated in each of the three global directions. Thus, three different **Random Vibration** nodes are also used. The acceleration cannot be explicitly prescribed for a ground motion when a modal based study is used. Instead, the **Base Excitation** feature is used. It provides a uniform body force to the structure.

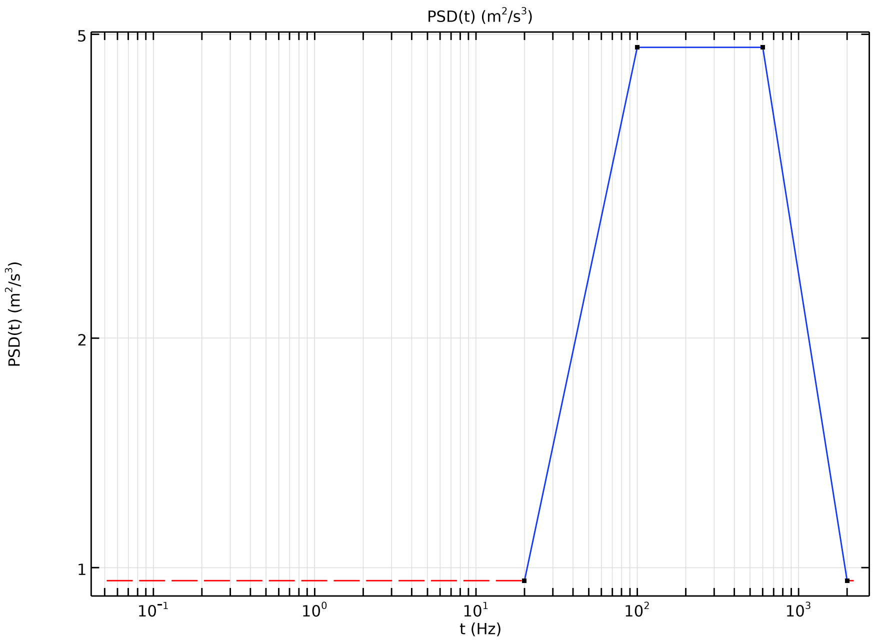

Figure 2: The power spectral density for the applied acceleration.

MOUNTING BOLTS

The motherboard is designed to be mounted with six M3 bolts. The following properties are assumed:

- Stress area: 5 mm^2
- Mounting prestress: 200 MPa (giving a bolt force of 1 kN)
- Friction coefficient between bolt and PCB: 0.12

Given these numbers, the maximum allowable shear force without sliding is $F_{s,max}$ = 120 N.

Each bolt is modeled by a rigid connector constrained in all degrees of freedom (DOFs), so that it is possible to measure the individual reaction forces.

MESH

Figure 3 shows the mesh used.

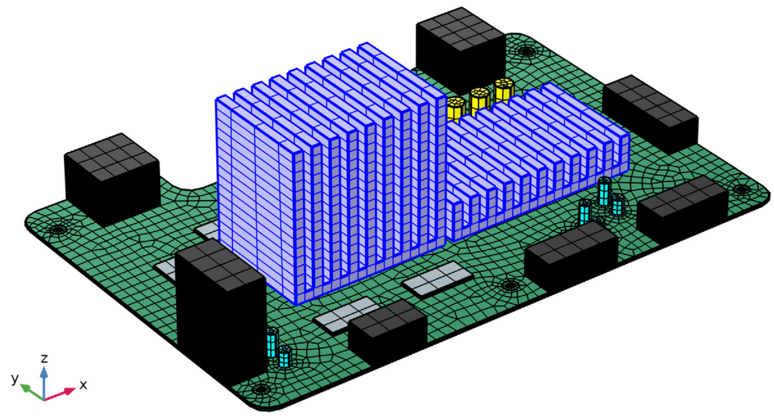

Figure 3: Mesh.

The discretization using this mesh together with quadratic serendipity elements results in approximately 130,000 DOFs being used in the eigenfrequency analysis.

Results and Discussion

The excitation PSD covers a large frequency range, 20 – 2000 Hz. As an effect, a large number of eigenfrequencies and corresponding eigenmodes (about 100) have to be computed. Using all these eigenmodes in the reduced-order model may however lead to long evaluation times. It is thus useful to investigate their relative importance. A good indicator is the modal mass. In Figure 4, the relative modal mass in each direction is shown for all eigenmodes.

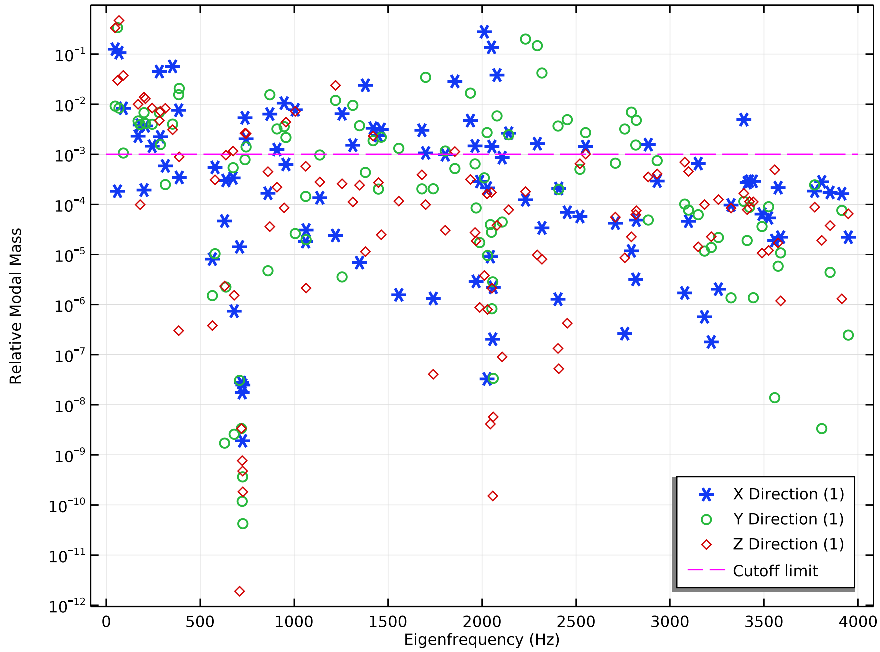

Figure 4: The distribution of modal mass for all computed eigenmodes.

As can be seen, the relative modal mass can differ by orders of magnitude between different modes. In the analyses, only modes with a relative modal mass larger than 0.0001 are included. It should however be noted that a mode which mainly consists of the movement of a single small component can have a low value of the participation factor, so this type of truncation should be used with some care.

In Figure 5 to Figure 7, the RMS accelerations (in units of g) are plotted for each of the three excitation directions. One important note here, is that since the actual acceleration from the shaker table is transformed into an equivalent inertial force, all displacements, velocities, and accelerations are computed relative to the fixation points. While this often is relevant for displacements, you will in most cases be interested in the accelerations relative to a space-fixed frame (*absolute accelerations*). This is what would be measured by an attached accelerometer. To compute the PSD and RMS values for the absolute accelerations, the PSD of the excitation must be added. There are special built-in acceleration variables that take care of this transformation.

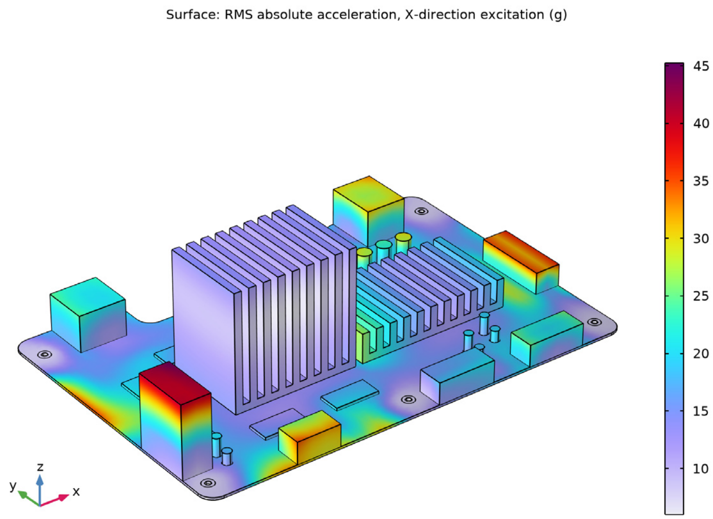

Figure 5: RMS of absolute acceleration caused by excitation in the X direction.

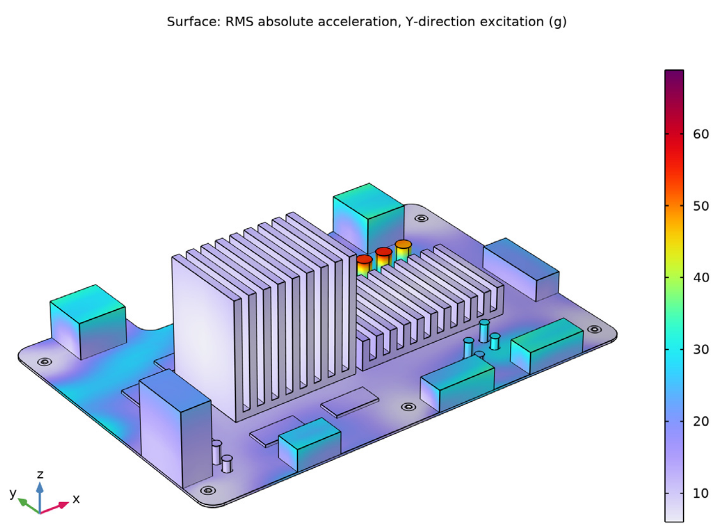

Figure 6: RMS of absolute acceleration caused by excitation in the Y direction.

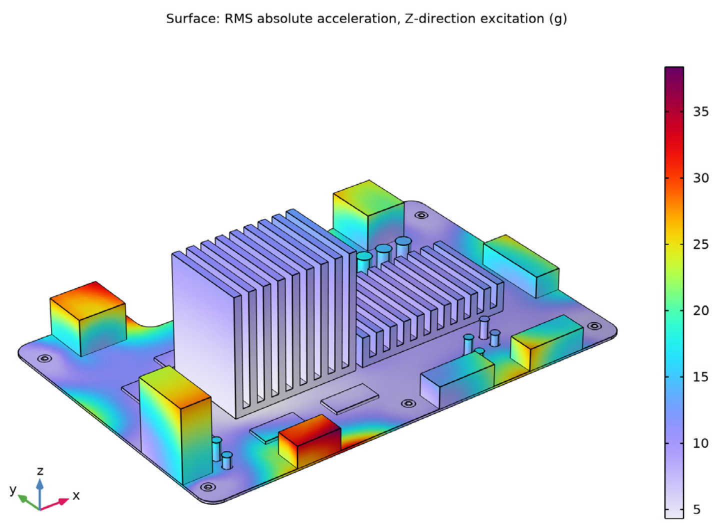

Figure 7: RMS of absolute acceleration caused by excitation in the Z direction.

For the worst case (shown in Figure 6), the RMS acceleration in some of the capacitors reaches about $70g$. Assuming that the peak value is three times larger than the RMS, the peak acceleration can be estimated to about $200g$. This is a high value, so the allowed accelerations for the affected components should be checked.

In order to get a better understanding of how different parts of the spectrum contribute to the RMS value, it is instructive to plot the PSD at locations with high RMS values. In Figure 8 through Figure 10, the PSD is plotted as function of frequency for a point on top of one of the small capacitors showing large accelerations in Figure 6. You can clearly see how the input spectrum equals the output spectrum for the excitation direction at low frequencies.

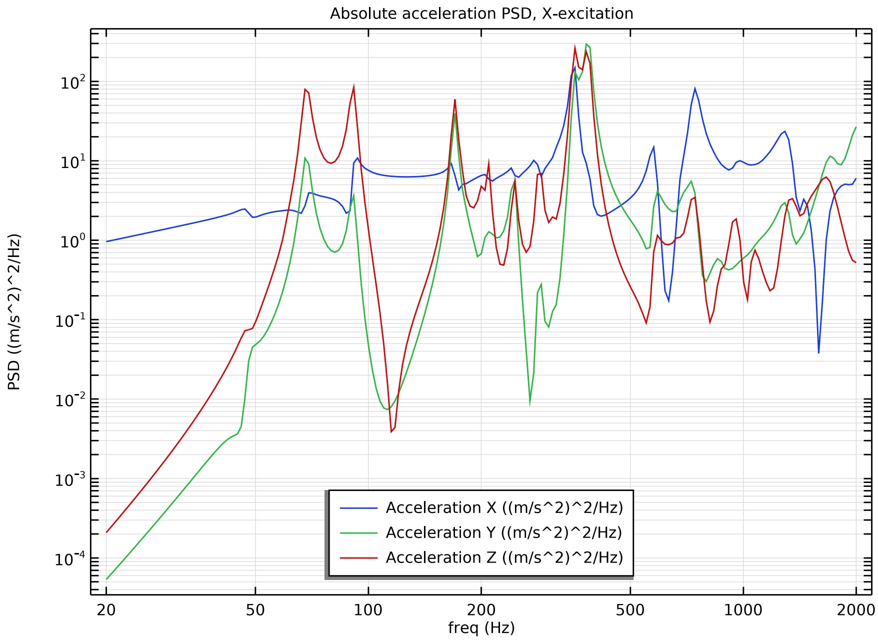

Figure 8: Computed PSD of the absolute acceleration for one of the capacitors when subjected to X direction excitation.

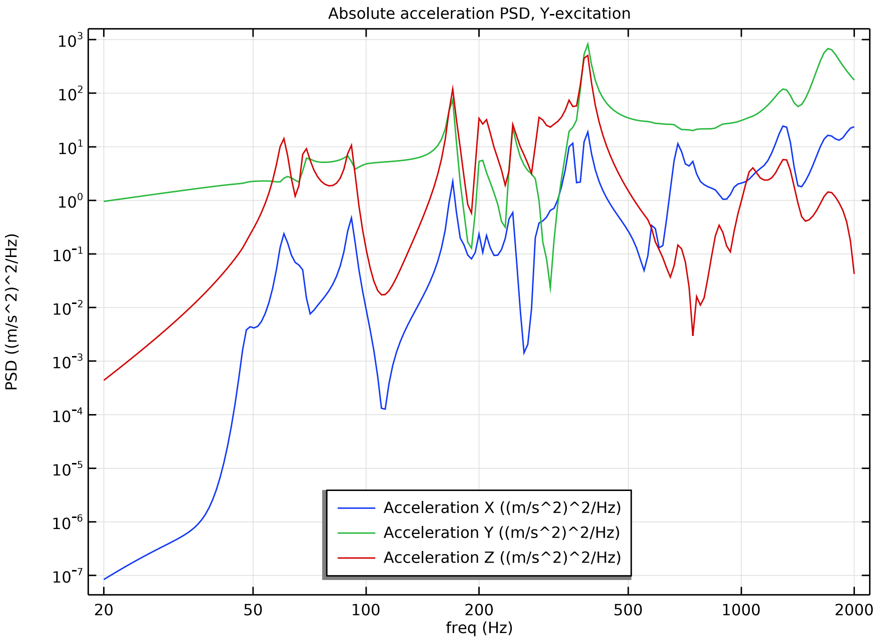

Figure 9: Computed PSD of the absolute acceleration for one of the capacitors when subjected to Y direction excitation.

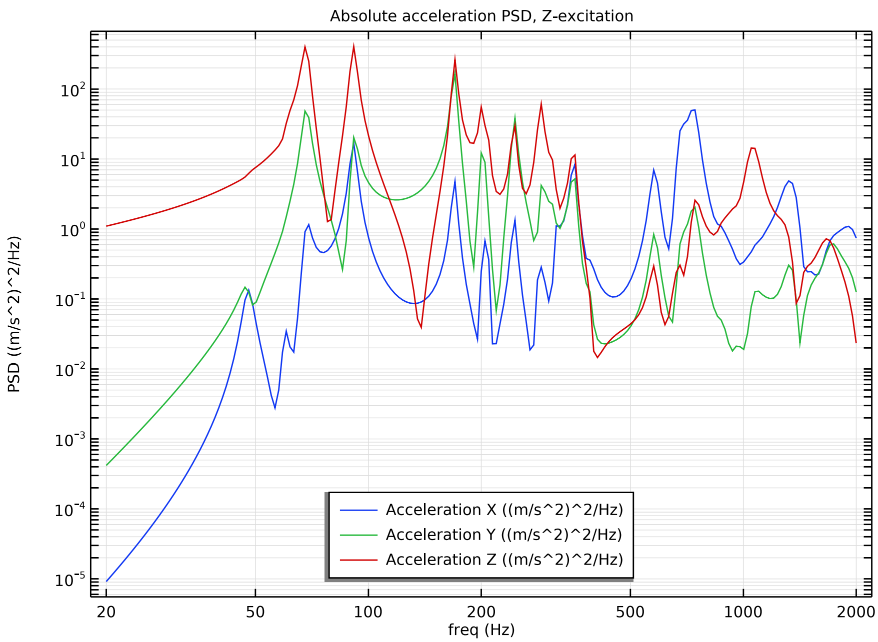

Figure 10: Computed PSD of the absolute acceleration for one of the capacitors when subjected to Z direction excitation.

Finally, the capacity of the mounting bolts are checked. In particular, the risk of sliding under the bolt heads is considered. Given the largest allowed shear force for a single bolt, $F_{\mathrm{s,\,max}}$, a margin of safety can be defined as

$$\mathrm{MoS} = \frac{F_{\mathrm{s,\,max}}}{3F_{\mathrm{RMS}}} - 1$$

where the assumption that the peak value exceeds the RMS value by a factor of 3 has been introduced.

Tables of all bolt forces and corresponding margins of safety are generated in the **Bolt Forces X-excitation**, **Bolt Forces Y-excitation**, and **Bolt Forces Z-excitation** nodes under **Derived Values**. The margin is positive for all bolts, however it is rather small in all three excitation directions for the bolts labeled 'Bolt 2' and 'Bolt 5'. The smallest value, for *Z*-direction excitation, is about 0.13.

If there is sliding under the bolt head, there is a risk that the bolt will lose its grip. If that would happen, the whole analysis would be invalidated, since the boundary conditions are no longer the same. The margin of safety is, however, determined with respect to the estimated peak value, so the force will not be that large for many load cycles. The bolts can thus be considered as validated.

Notes About the COMSOL Implementation

The given acceleration spectrum is that of the shaker table, that is what the bolt locations experience. The load is applied using a **Base Excitation** node. This is usually the preferred approach if all supports move synchronously, as is the case in vibration testing. Otherwise, the approximate *large mass method* must be used.

When computing the RMS value of the accelerations, the `q2` and `q2sq` operators are used. These operators are similar to the `rms` operator, but act on a quadratic form. The simpler of these operators, `q2sq`, takes predefined variables defined for random vibration as input, and directly returns the RMS. When supplying user-defined quadratic forms, the `q2` operator is used instead. This operator returns the square of the RMS. The following three cases would give the same result for a single variable `u`:

```
rvib1.rms(u,...)
sqrt(rvib1.q2(u^2,...)
rvib1.q2sq(sqrt(u^2+eps),...)
```

For the total relative displacement, the following cases are equivalent for computing the RMS:

```
rvib1.q2sq(solid.disp_rv,...)
sqrt(rvib1.q2(u^2+v^2+w^2,...)
rvib1.q2sq(sqrt(u^2+v^2+w^2+eps),...)
```

Note that the argument to the `q2sq` operator must never evaluate to zero, hence the addition of the small number `eps`.

One of the desired results is the clamping forces in the bolts. In a modal-based study, it is not possible to directly evaluate reaction forces in fixed constraints. For this reason, the holes are constrained using six rigid connectors, in which forces can be computed.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/motherboard_random_vibration`

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Structural Mechanics Module** > **Dynamics and Vibration** > **motherboard_shock_response** in the tree.

3 Click **Open**.

Delete nodes related to the previous studies.

GLOBAL DEFINITIONS

Acceleration (g) vs. Frequency (Hz) (int1), Vertical Spectrum (vsp)

1 In the **Model Builder** window, under **Global Definitions**, Ctrl-click to select **Acceleration (g) vs. Frequency (Hz) (int1)** and **Vertical Spectrum (vsp)**.

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

STUDY 2

In the **Model Builder** window, right-click **Study 2** and choose **Delete**.

RESULTS

In the **Model Builder** window, expand the **Results** > **Datasets** node.

Grid 1D 1, Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, under **Results** > **Datasets**, Ctrl-click to select **Grid 1D 1** and **Study 1/Solution 1 (sol1)**.

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

GLOBAL DEFINITIONS

Parameters 1

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

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

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

Replace the constraint by rigid connectors, so that the bolt forces can be evaluated easily.

DEFINITIONS

Hole 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** node.

2 Right-click **Component 1 (comp1)** > **Definitions** and choose **Selections** > **Explicit**.

3 In the **Settings** window for **Explicit**, type `Hole 1` in the **Label** text field.

4 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 7, 11, 12, 16, and 17 only.

Hole 2

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Hole 2` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 38, 42, 43, 46, and 47 only.

Hole 3

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Hole 3` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 68, 72, 73, 77, and 78 only.

Hole 4

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Hole 4` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 9, 13, 14, 18, and 19 only.

Hole 5

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Hole 5` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 40, 44, 45, 48, and 49 only.

Hole 6

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Hole 6` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 70, 74, 75, 79, and 80 only.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Solid Mechanics (solid)** node.

2 Right-click **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint 1** and choose **Delete**.

Rigid Connector, Hole 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 In the **Settings** window for **Rigid Connector**, type `Rigid Connector, Hole 1` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hole 1**.

4 Locate the **Prescribed Displacement at Center of Rotation** section. Select the **Prescribed in x direction** checkbox.

5 Select the **Prescribed in y direction** checkbox.

6 Select the **Prescribed in z direction** checkbox.

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

8 Select the **Constrain rotation around x-axis** checkbox.

9 Select the **Constrain rotation around y-axis** checkbox.

10 Select the **Constrain rotation around z-axis** checkbox.

11 Click to expand the **Reaction Force Settings** section. Select the **Evaluate reaction forces** checkbox.

Rigid Connector, Hole 2

1 Right-click **Rigid Connector, Hole 1** and choose **Duplicate**.

2 In the **Settings** window for **Rigid Connector**, type `Rigid Connector, Hole 2` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hole 2**.

Rigid Connector, Hole 3

1 Right-click **Rigid Connector, Hole 2** and choose **Duplicate**.

2 In the **Settings** window for **Rigid Connector**, type `Rigid Connector, Hole 3` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hole 3**.

Rigid Connector, Hole 4

1 Right-click **Rigid Connector, Hole 3** and choose **Duplicate**.

2 In the **Settings** window for **Rigid Connector**, type `Rigid Connector, Hole 4` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hole 4**.

Rigid Connector, Hole 5

1 Right-click **Rigid Connector, Hole 4** and choose **Duplicate**.

2 In the **Settings** window for **Rigid Connector**, type `Rigid Connector, Hole 5` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hole 5**.

Rigid Connector, Hole 6

1 Right-click **Rigid Connector, Hole 5** and choose **Duplicate**.

2 In the **Settings** window for **Rigid Connector**, type `Rigid Connector, Hole 6` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hole 6**.

Set up the eigenvalue solver so that all eigenfrequencies in the interesting frequency range are captured. Then, run it. Note that also eigenfrequencies outside the range of the applied spectrum can contribute to the dynamic response, so the range is extended by a factor of two in each direction.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 From the **Eigenfrequency search method** list, choose **Rectangle**.

4 In the **Approximate number of eigenfrequencies** text field, type `120`.

5 Find the **Rectangle search region** subsection. In the **Smallest real part (Eigenfrequency)** text field, type `fL/2`.

6 In the **Largest real part (Eigenfrequency)** text field, type `fU*2`.

7 In the **Model Builder** window, click **Study 1**.

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

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

RESULTS

Mode Shape (solid)

Add a plot showing how the mass is distributed within the computed eigenmodes.

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 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 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.

5 In the **1D Plot Group 2** toolbar, click **Plot**.

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

Modal Mass Distribution

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

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** checkbox. In the associated text field, type `Relative Modal Mass`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

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

Line Segments 1

1 Right-click **Modal Mass Distribution** and choose **Line Segments**.

2 In the **Settings** window for **Line Segments**, locate the **x-Coordinates** section.

3 In the table, enter the following settings:

Expression	Unit	Description
0		
fU*2		Upper frequency limit of input spectrum

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

Expression	Unit	Description
0.001	1	
0.001	1	

5 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Magenta**.

6 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Cutoff limit

10 Locate the **Data** section. From the **Dataset** list, choose **Study: Eigenfrequency/ Solution 1 (sol1)**.

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

12 In the **Modal Mass Distribution** toolbar, click **Plot**.

Add a filter so that the random vibration analyses can be performed using only relevant eigenmodes. This will speed up result evaluations significantly. Here, the already existing **Response Spectrum** node is used to obtain the structural mass. In general, you would have to add a **Mass Properties** node to compute the mass of the structure.

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 **Empty Study**.

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

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

MODE FILTER

In the **Settings** window for **Study**, type `Mode Filter` in the **Label** text field.

Step 1: Combine Solutions

1 In the **Study** toolbar, click **More Study Extensions** and choose **Combine Solutions**.

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

3 From the **Solution operation** list, choose **Remove solutions**.

4 From the **Solution** list, choose **Study: Eigenfrequency/Solution 1 (sol1)**.

5 From the **Exclude method** list, choose **Implicit**.

6 In the **Excluded if** text field, type `comp1.rsp1.mEffLX+comp1.rsp1.mEffLY+comp1.rsp1.mEffLZ<comp1.rsp1.mass*1e-4`.

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

Add studies for a random vibration analysis. The eigenmodes are already computed, so the newly generated **Study 3** can be removed.

ADD STUDY

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

STUDY: ROM BUILDING

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

Step 1: Model Reduction

1 In the **Model Builder** window, under **Study: ROM building** click **Step 1: Model Reduction**.

2 In the **Settings** window for **Model Reduction**, locate the **Model Reduction Settings** section.

3 From the **Training study for eigenmodes** list, choose **Mode Filter**.

STUDY 3

In the **Model Builder** window, right-click **Study 3** and choose **Delete**.

Add functions describing the acceleration spectrum. Since the original spectrum is defined in units of g^2/Hz, it should be converted to SI units.

GLOBAL DEFINITIONS

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 In the **Function name** text field, type PSD.

4 In the table, enter the following settings:

t	f(t)
20	0.01*g_const^2
100	0.05*g_const^2
600	0.05*g_const^2
2000	0.01*g_const^2

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

Function	Unit
PSD	m^2/s^3

6 In the **Argument** table, enter the following settings:

Argument	Unit
t	Hz

In these types of spectra, it is usually assumed that the functions are linear in a log-log space. An ordinary linear interpolation between the given values would not give the intended result.

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

8 From the **Function** list, choose **Logarithmic**.

9 Click **Create Plot**.

RESULTS

Acceleration Spectrum

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

2 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

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

Create the parameters (ROM controls) that are going to be used for the acceleration spectra.

GLOBAL DEFINITIONS

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
accX	1[m/s^2]
accY	1[m/s^2]
accZ	1[m/s^2]

Add the loads corresponding to the acceleration from the shaker table.

SOLID MECHANICS (SOLID)

Base Excitation 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Base Excitation 1**.

2 In the **Settings** window for **Base Excitation**, locate the **Base Excitation** section.

3 Specify the $\mathbf{a}_b$ vector as

accX	x

accY	y
accZ	z

Add damping for the response analysis.

Linear Elastic Material 1

In the **Model Builder** window, 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 40.

5 In the ζ_1 text field, type 0.04.

6 In the f_2 text field, type 1000.

7 In the ζ_2 text field, type 0.04.

Make sure that the damping is not used if the eigenfrequency study is run again.

STUDY: EIGENFREQUENCY

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study: Eigenfrequency** 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**.

Create a ROM based on the most relevant eigenmodes.

STUDY: ROM BUILDING

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

Provide random vibration spectra, and update the ROM with it.

GLOBAL DEFINITIONS

Random Vibration, X

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**, type `Random Vibration, X` in the **Label** text field.

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

Control name	Power spectral density
accX	`PSD(freq)`

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

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

6 From the **Integration method** list, choose **User defined**.

7 In the **Number of integration points** text field, type `nF`.

STUDY: ROM BUILDING

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

RESULTS

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.

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

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** checkbox.

4 Locate the **Save Data in the Model** section. From the **Save plot data** list, choose **On**.

RMS absolute acceleration X-excitation

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

2 In the **Settings** window for **3D Plot Group**, type `RMS absolute acceleration X-excitation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: ROM building/ Solution 3 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Surface: RMS absolute acceleration, X-direction excitation (g)`.

6 Click to expand the **Selection** section.

Surface 1

1 Right-click **RMS absolute acceleration X-excitation** and choose **Surface**.

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

3 In the **Expression** text field, type `sqrt(rvib1.q2(solid.a_abs_rv2))/g_const`.

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

Switching off the refinement in the plot will give faster evaluations.

5 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

6 From the **Resolution** list, choose **No refinement**.

7 In the **RMS absolute acceleration X-excitation** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Random Vibration, Y

1 In the **Model Builder** window, under **Global Definitions** > **Reduced-Order Modeling** right-click **Random Vibration, X (rvib1)** and choose **Duplicate**.

2 In the **Settings** window for **Random Vibration**, type `Random Vibration, Y` in the **Label** text field.

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

Control name	Power spectral density
accX	0
accY	PSD(freq)

Random Vibration, Z

1 Right-click **Random Vibration, Y** and choose **Duplicate**.

2 In the **Settings** window for **Random Vibration**, type `Random Vibration, Z` in the **Label** text field.

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

Control name	Power spectral density
accY	0
accZ	PSD(freq)

STUDY: ROM BUILDING

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

RESULTS

RMS absolute acceleration Y-excitation

1 Right-click **RMS absolute acceleration X-excitation** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `RMS absolute acceleration Y-excitation` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Surface: RMS absolute acceleration, Y-direction excitation (g)`.

Surface 1

1 In the **Model Builder** window, expand the **RMS absolute acceleration Y-excitation** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `sqrt(rvib2.q2(solid.a_abs_rv2))/g_const`.

4 In the **RMS absolute acceleration Y-excitation** toolbar, click **Plot**.

RMS absolute acceleration Z-excitation

1 In the **Model Builder** window, right-click **RMS absolute acceleration Y-excitation** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `RMS absolute acceleration Z-excitation` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Surface: RMS absolute acceleration, Z-direction excitation (g)`.

Surface 1

1 In the **Model Builder** window, expand the **RMS absolute acceleration Z-excitation** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `sqrt(rvib3.q2(solid.a_abs_rv2))/g_const`.

4 In the **RMS absolute acceleration Z-excitation** toolbar, click **Plot**.

After the examination of the RMS plots, select a point where the acceleration is high in all directions and study the PSD in that location.

DEFINITIONS

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 608 only.

5 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
a_absX	aveop1(solid.a_abs_rvX)	m/s²	Acceleration X
a_absY	aveop1(solid.a_abs_rvY)	m/s²	Acceleration Y
a_absZ	aveop1(solid.a_abs_rvZ)	m/s²	Acceleration Z

After having added new variables, the reduced-order models must be rebuilt. In this case, it would have been possible to avoid the creation of new variables, since an `at3()` operator could be used instead. That, however, requires that you find the coordinates of the point where you want to examine the value.

STUDY: ROM BUILDING

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

RESULTS

Global Evaluation Sweep, X-excitation Acceleration PSD

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

Evaluate the acceleration PSD functions. In order to get values in a space fixed frame, the input PSD must be added in the excitation direction.

2 In the **Settings** window for **Global Evaluation Sweep**, type `Global Evaluation Sweep, X-excitation Acceleration PSD` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: ROM building/ Solution 3 (sol3)**.

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

Parameter name	Parameter value list
freq	10^range(log10(20),1/nFd,log10(2000))[Hz]

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

Expression	Unit	Description
rvib1.psd(a_absX)	(m/s^2)^2/Hz	Acceleration X
rvib1.psd(a_absY)	(m/s^2)^2/Hz	Acceleration Y
rvib1.psd(a_absZ)	(m/s^2)^2/Hz	Acceleration Z

6 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 **Legends** section.

2 Select the **Show legends** checkbox.

3 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

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

Absolute acceleration PSD X-excitation

1 In the **Model Builder** window, under **Results** click **1D Plot Group 7**.

2 In the **Settings** window for **1D Plot Group**, type `Absolute acceleration PSD X-excitation` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** checkbox. In the associated text field, type `PSD ((m/s^2)^2/Hz)`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Absolute acceleration PSD, X-excitation`.

8 In the **Absolute acceleration PSD X-excitation** toolbar, click **Plot**.

Global Evaluation Sweep, Y-excitation Acceleration PSD

1 In the **Model Builder** window, right-click **Global Evaluation Sweep, X-excitation Acceleration PSD** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation Sweep**, type `Global Evaluation Sweep, Y-excitation Acceleration PSD` in the **Label** text field.

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

Expression	Unit	Description
rvib2.psd(a_absX)	(m/s^2)^2/Hz	Acceleration X
rvib2.psd(a_absY)	(m/s^2)^2/Hz	Acceleration Y
rvib2.psd(a_absZ)	(m/s^2)^2/Hz	Acceleration Z

4 Click ▾ next to **Evaluate**, then choose **New Table**.

Absolute acceleration PSD Y-excitation

1 In the **Model Builder** window, right-click **Absolute acceleration PSD X-excitation** and choose **Duplicate**.

2 In the **Model Builder** window, click **Absolute acceleration PSD X-excitation 1**.

3 In the **Settings** window for **1D Plot Group**, type `Absolute acceleration PSD Y-excitation` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Absolute acceleration PSD, Y-excitation`.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Table 4**.

4 In the **Absolute acceleration PSD Y-excitation** toolbar, click **Plot**.

Global Evaluation Sweep, Z-excitation Acceleration PSD

1 In the **Model Builder** window, right-click **Global Evaluation Sweep, Y-excitation Acceleration PSD** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation Sweep**, type `Global Evaluation Sweep, Z-excitation Acceleration PSD` in the **Label** text field.

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

Expression	Unit	Description
rvib3.psd(a_absX)	(m/s^2)^2/Hz	Acceleration X
rvib3.psd(a_absY)	(m/s^2)^2/Hz	Acceleration Y
rvib3.psd(a_absZ)	(m/s^2)^2/Hz	Acceleration Z

4 Click next to **Evaluate**, then choose **New Table**.

Absolute acceleration PSD Z-excitation

1 In the **Model Builder** window, right-click **Absolute acceleration PSD Y-excitation** and choose **Duplicate**.

2 In the **Model Builder** window, click **Absolute acceleration PSD Y-excitation 1**.

3 In the **Settings** window for **1D Plot Group**, type `Absolute acceleration PSD Z-excitation` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Absolute acceleration PSD, Z-excitation`.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Table 5**.

4 In the **Absolute acceleration PSD Z-excitation** toolbar, click **Plot**.

Check the bolt forces.

Bolt Forces X-excitation

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

2 In the **Settings** window for **Global Evaluation**, type `Bolt Forces X-excitation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: ROM building/ Solution 3 (sol3)**.

4 Locate the **Expressions** section. Click **Load from File**.

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

6 Click **Evaluate**.

Bolt Forces Y-excitation

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

2 In the **Settings** window for **Global Evaluation**, type `Bolt Forces Y-excitation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: ROM building/ Solution 3 (sol3)**.

4 Locate the **Expressions** section. Click **Load from File**.

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

6 Click **Evaluate**.

Bolt Forces Z-excitation

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

2 In the **Settings** window for **Global Evaluation**, type `Bolt Forces Z-excitation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: ROM building/ Solution 3 (sol3)**.

4 Locate the **Expressions** section. Click **Load from File**.

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

6 Click **Evaluate**.