Model created in COMSOL Multiphysics 6.4

Shock Response of a Motherboard

Introduction

Various types of electronic equipment often need to certified to function after being subjected to a specified shock loading. Performing a numerical simulation is often cheaper than a physical test, and has a shorter turnaround time.

In many cases, it is not necessary to perform a full nonlinear contact analysis. The shock is studied not by dropping the component, but rather by attaching it to a rigid frame having a given acceleration history. A common such specification of a shock is that the acceleration behaves as a half sine function with a given period and peak amplitude.

In this example, the effect of a $50g$ half-sine shock load with a duration of 11 ms on a computer motherboard is investigated using two approaches: response spectrum analysis and direct time stepping using mode superposition.

Model Definition

GEOMETRY

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

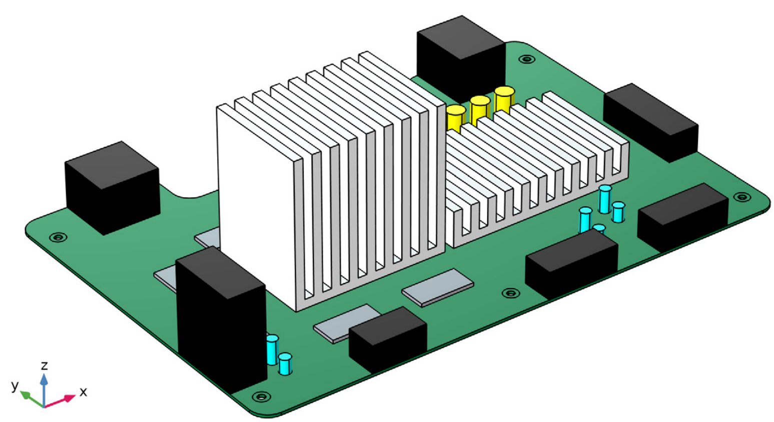

Figure 1: Motherboard geometry.

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 edges. The board is attached to the housing via six mounting bolts. The latter are not modeled explicitly.

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.

CONSTRAINTS

All six mounting holes have fixed constraints on their cylindrical boundaries.

LOADS

For the response spectrum analysis, the loading is represented by a pseudoacceleration spectrum as shown in Figure 2.

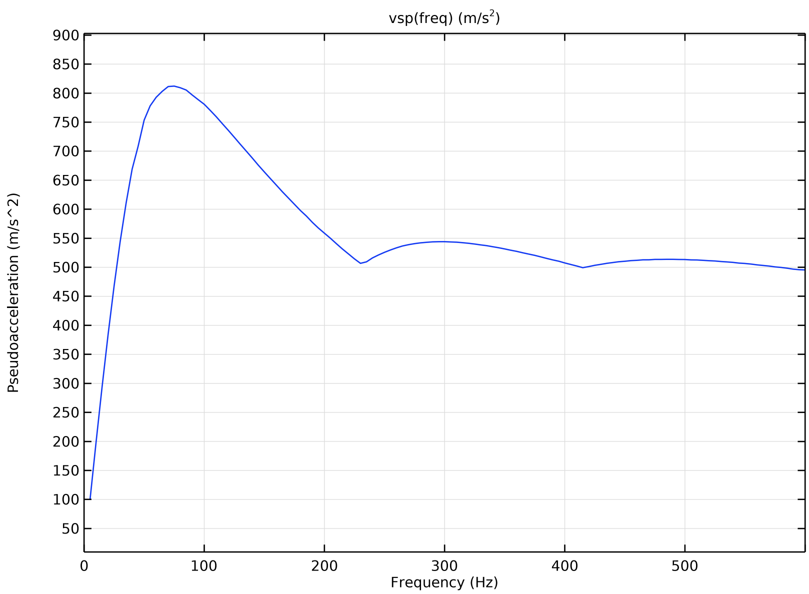

Figure 2: Vertical spectrum for 50g shock load with 11 ms duration.

During the transient computations, all parts of the motherboard are subjected to a base excitation with $50g$ magnitude and 11 ms duration:

$$f = 50 \cdot g \cdot \sin\left(\frac{2\pi t}{22\ \mathrm{ms}}\right) \qquad t < 11\ \mathrm{ms} \tag{1}$$

MESH

Figure 3 shows the mesh used.

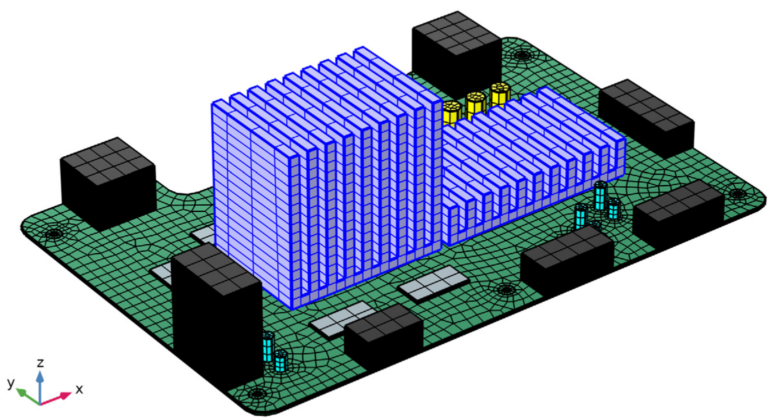

Figure 3: Mesh.

The discretization using this mesh together with quadratic serendipity elements results in approximately 120,000 degrees of freedom being used in the eigenfrequency analysis.

Results and Discussion

As a first step, the lowest fifteen eigenfrequencies have been computed. Figure 4 shows their distribution together with the values of the participation factors for the corresponding eigenmodes.

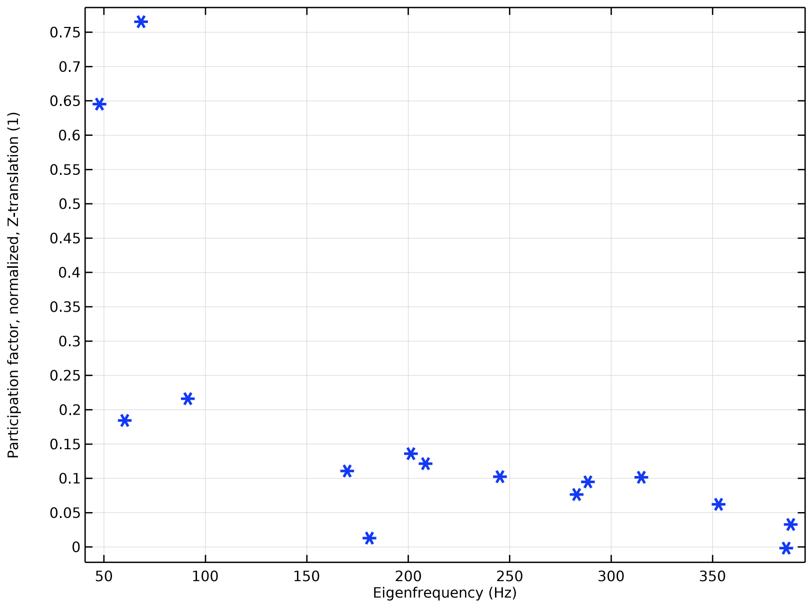

Figure 4: Participation factors for Z-translation for all the eigenmodes.

The sum of the effective modal mases for all computed eigenmodes is around 0.94. Thus, the modes account for about 94% of the vertical dynamic forces of the system.

Figure 5 shows the values of the vertical shock spectrum computed at the eigenfrequencies.

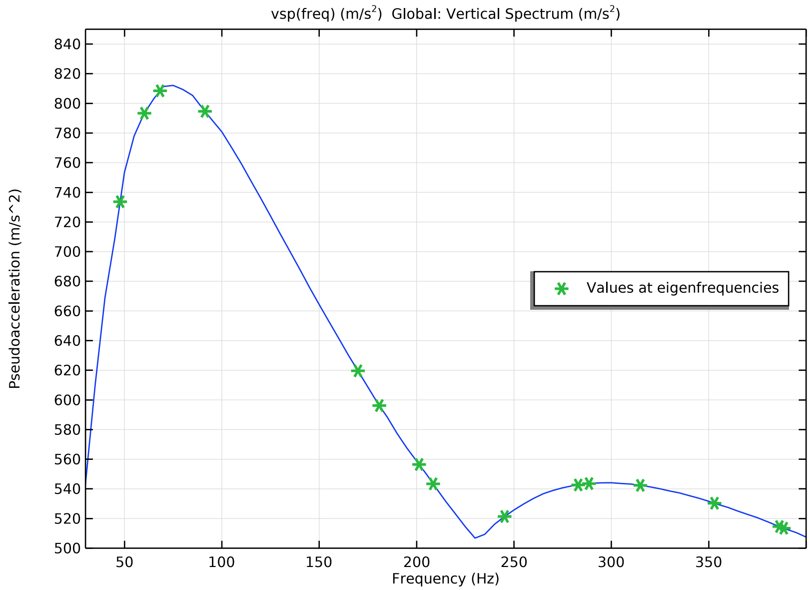

Figure 5: Vertical shock spectrum input at eigenfrequencies.

Thus, the first and third modes have the largest participation factor for vertical translation. The eigenfrequency for the third mode is also located close to the maximum of the input spectrum, so these two modes will be important for the response. The modes are shown in Figure 6 and Figure 7 below.

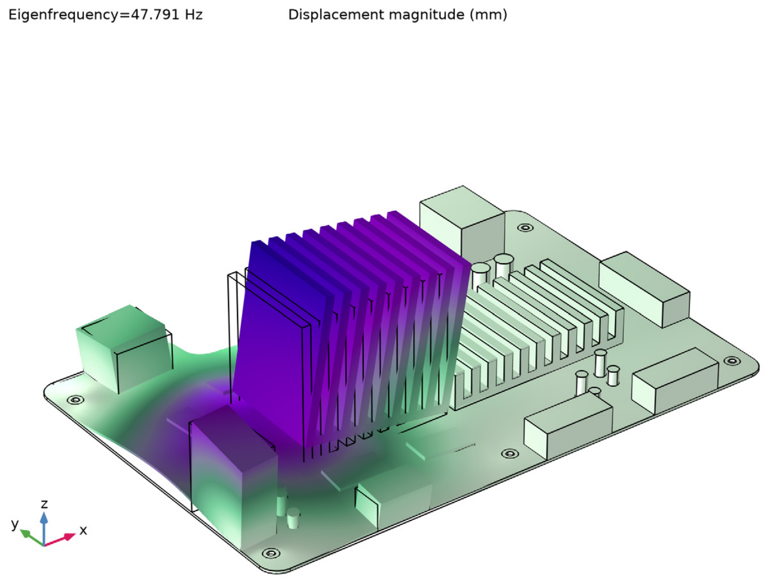

Figure 6: The 1st eigenmode.

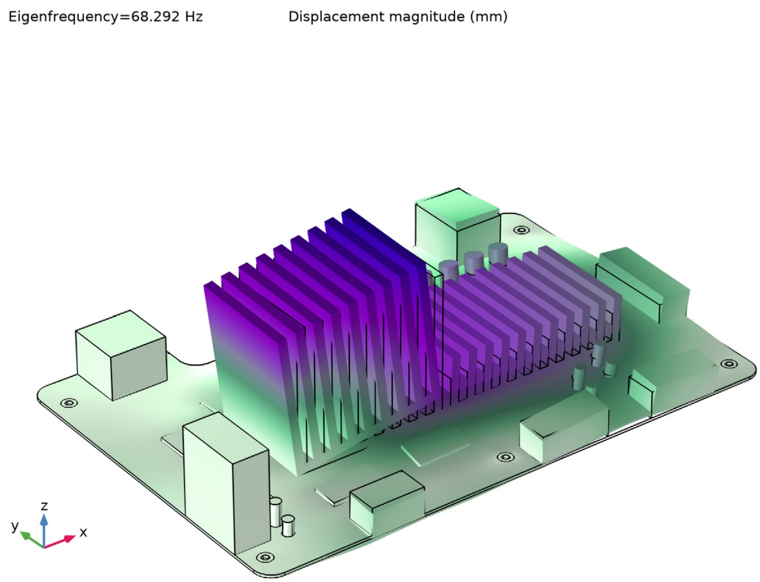

Figure 7: The 3rd eigenmode.

The response spectrum evaluation is based on the eigenvalue solution, and it is performed during postprocessing using a dedicated dataset called **Response Spectrum**.

In Figure 8, the vertical displacement response, as computed using the CQC mode combination method with damping of 5% is shown.

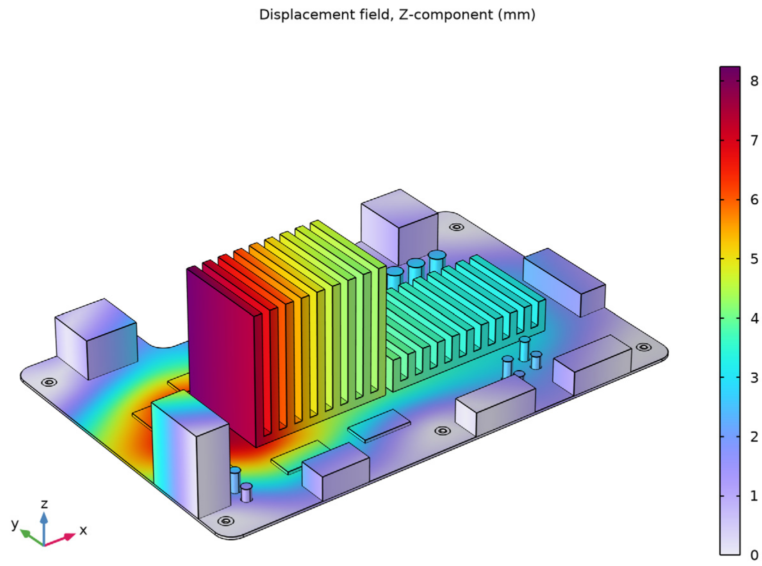

Figure 8: Vertical displacement, as a result of the response spectrum evaluation.

Figure 9 shows the stress distribution computed using the response spectrum analysis. The plot shows rather high stress level at the memory chips.

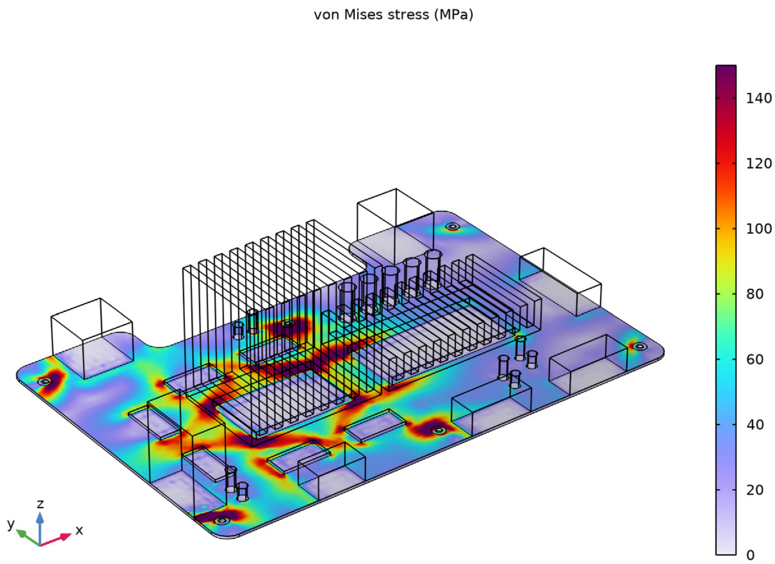

Figure 9: Stress in the motherboard components.

For comparison, the same analysis is also performed in the time domain, using a **Time Dependent, Modal** solver. Figure 10 shows the vertical displacement of the system after 11 ms of transient loading.

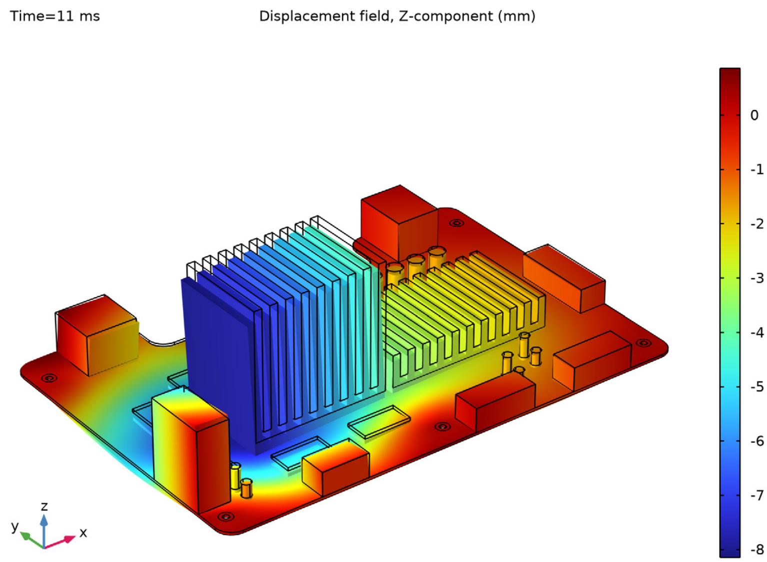

Figure 10: Dynamic response of the system at the end of the shock pulse.

The maximum displacement computed over a 60 ms long transient simulation is shown in Figure 11.

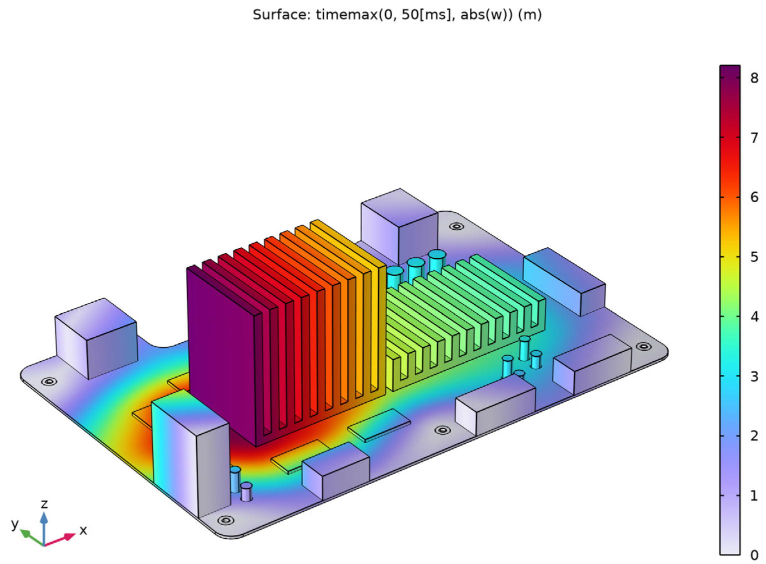

Figure 11: Maximum vertical displacement.

Note that the maximum is computed pointwise, so that Figure 11 does not correspond to a state of the system at some particular time instance. Rather, it gives an estimation of the largest deformation response in each point. In this sense, the plot presents a similar information as the computations of the shock spectrum response, and can be compared with Figure 8. Both methods indicate a maximum displacement of about 8 mm.

Notes About the COMSOL Implementation

The two methods used here has different merits. The response spectrum method only provides an approximate solution, whereas the time stepping is exact (within the limits given by time discretization and truncation of higher modes). On the other hand, you will have to store a full solution for all time steps in order to find the maximum values when using time stepping. This will cause large file sizes.

Note that the maximum response can occur significantly later than the end of the shock, particularly for systems with low damping. During the time stepping, a few periods of the lowest eigenfrequency should be covered. Here, the end of the time stepping is set to 60 ms, which is about three full periods of the first eigenmode.

For the time-dependent analysis, you enter the function for the load by entering the factor `sin(2*pi/22[ms]*t)*(t<11[ms])` in the **Load factor** input field available in the settings for the **Time Dependent, Modal** solver node. The load set up via the **Base Excitation** node will be multiplied by this factor, giving an effective load magnitude of 50g.

Application Library path: `MEMS_Module/Dynamics_and_Vibration/motherboard_shock_response`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Response Spectrum**.
6 Click **Done**.

GEOMETRY 1

Import the motherboard geometry from a file.

Import 1 (imp1)

1 In the **Geometry** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Source** section.
3 Click **Browse**.
4 Browse to the model's Application Libraries folder and double-click the file `motherboard_shock_response.mphbin`.

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Select the **Create imprints** checkbox.

5 In the **Geometry** toolbar, click **Build All**.

6 Click the **Go to Default View** button in the **Graphics** toolbar.

7 Click the **Show Grid** button in the **Graphics** toolbar.

The imported geometry should look similar to that shown in Figure 1.

DEFINITIONS

Prepare selections of the motherboard components to simplify the modeling process.

PCB

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

Select the printed circuit board.

2 Select Domain 1 only.

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

Heat sinks

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

Select both heat sinks.

2 Select Domains 17 and 21 only.

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

Chips

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

Select the CPU, GPU, and memory chips.

2 Select Domains 3, 4, 7–10, 18, and 22 only.

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

Connector blocks

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

Select all seven connector blocks located along the edges of the board.

2 Select Domains 2, 5, 6, and 11–14 only.

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

Capacitors (larger)

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

Select five capacitors of larger size placed in a row next to one of the heat sinks.

2 Select Domains 23–25, 28, and 31 only.

3 In the **Settings** window for **Explicit**, type `Capacitors (larger)` in the **Label** text field.

Capacitors (smaller)

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

Select all the remaining capacitors that are scattered over the motherboard.

2 Select Domains 15, 16, 19, 20, 26, 27, 29, and 30 only.

3 In the **Settings** window for **Explicit**, type `Capacitors (smaller)` in the **Label** text field.

Mounting holes

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

Select the boundaries for all six mounting holes. You can use the **Select Box** button to select one hole at a time and add its selection, then proceed to the next one.

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

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

4 Select Boundaries 7, 9, 11–14, 16–19, 38, 40, 42–49, 68, 70, 72–75, and 77–80 only.

5 In the **Label** text field, type `Mounting holes`.

View 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **View 1**.

2 In the **Settings** window for **View**, locate the **Colors** section.

3 Select the **Show material color and texture** checkbox.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **FR4 (Circuit Board)**.

4 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

FR4 (Circuit Board) (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **FR4 (Circuit Board) (mat1)**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **PCB**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in** > **Aluminum**.
3 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

Aluminum (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Aluminum (mat2)**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Heat sinks**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in** > **Silicon**.
3 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

. (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **. (mat3)**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Chips**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in** > **Acrylic plastic**.
3 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

Acrylic plastic (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Acrylic plastic (mat4)**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Connector blocks**.
4 Click to expand the **Appearance** section. From the **Material type** list, choose **Plastic (shiny)**.
5 From the **Color** list, choose **Black**.

The cylindrical capacitors have a complex internal structure with several layers of different materials. For the effective material, use the same elasticity modulus as for aluminum, which is the material of the capacitor's outer cylinder.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in** > **Aluminum**.
3 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

Capacitors (larger) composite material

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Aluminum 1 (mat5)**.
2 In the **Settings** window for **Material**, type `Capacitors (larger) composite material` in the **Label** text field.
3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Capacitors (larger)**.

   Set the effective material density for the larger capacitors.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`1500[kg/m^3]`	kg/m³	Basic

   Keep all the other mechanical properties the same as for aluminum.
5 Click to expand the **Appearance** section. From the **Material type** list, choose **Plastic (shiny)**.

6 From the **Color** list, choose **Yellow**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in** > **Aluminum**.

3 Right-click and choose **Add to Component 1 (comp1)**.

4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Capacitors (smaller) composite material

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Capacitors (smaller)**.

3 In the **Label** text field, type `Capacitors (smaller) composite material`.

Set the effective material density for the larger capacitors.

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

Property	Variable	Value	Unit	Property group
Density	rho	2000[kg/m^3]	kg/m³	Basic

5 Click to expand the **Appearance** section. From the **Material type** list, choose **Plastic (shiny)**.

6 From the **Color** list, choose **Cyan**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Mounting holes**.

MESH 1

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.

Explicitly prescribe the mesh size for the domains representing smaller capacitors.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Capacitors (smaller)**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

Free Quad 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

Select the upper face of the PCB. By using a selection by continuous tangent, you only need to click on a single boundary, even if the full selection is listed below.

2 In the **Settings** window for **Free Quad**, in the **Graphics** window toolbar, click next to **Select Boundaries**, then choose **Group by Continuous Tangent**.

3 Select Boundaries 4, 5, 8, 10, 21–28, 30, 34–37, 39, 41, 50–56, 58, 61–64, 66, 67, 69, and 71 only.

Size 1

1 Right-click **Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** checkbox. In the associated text field, type `6[mm]`.

6 Select the **Minimum element size** checkbox. In the associated text field, type `0.06[mm]`.

7 Click **Build Selected**.

Free Quad 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

Select one side boundary of each heat sink for quad meshing. This is to control the directions of the mesh sweep that you will set up in the coming modeling steps.

2 Select Boundaries 174 and 231 only.

Size 1

1 Right-click **Free Quad 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** checkbox. In the associated text field, type `5[mm]`.

6 Click **Build Selected**.

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

Do a similar thing using mapped meshes on the top boundaries of all rectangular components. You might need to hide the heat sink domains to select the top boundaries of the CPU and GPU components.

2 Select Boundaries 86, 92, 97, 102, 109, 114, 119, 128, 133, 140, 146, 151, 158, 215, and 280 only.

Size 1

1 Right-click **Mapped 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

5 Click **Build Selected**.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution: Default

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

2 In the **Settings** window for **Distribution**, type `Distribution: Default` in the **Label** text field.

3 Locate the **Distribution** section. In the **Number of elements** text field, type `3`.

Distribution: PCB

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `Distribution: PCB` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **PCB**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `2`.

Distribution: Heat sinks

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

2 In the **Settings** window for **Distribution**, type `Distribution: Heat sinks` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Heat sinks**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `4`.

5 Click **Build All**.

6 Click the **Go to Default View** button in the **Graphics** toolbar.

The final mesh should look similar to that shown in Figure 3.

GLOBAL DEFINITIONS

Acceleration (g) vs. Frequency (Hz)

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

2 In the **Settings** window for **Interpolation**, type `Acceleration (g) vs. Frequency (Hz)` in the **Label** text field.

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

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

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

Function	Unit
int1	1

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

Argument	Unit
t	Hz

Vertical Spectrum

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

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

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

4 Locate the **Definition** section. In the **Arguments** text field, type `freq`.

5 In the **Expression** text field, type `int1(freq)*g_const`.

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

Argument	Unit
freq	Hz

7 In the **Function** text field, type `m/s^2`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	freq	5	600	0	Hz

9 Click **Create Plot**.

RESULTS

Vertical Spectrum

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

2 Locate the **Plot Settings** section.

3 Select the **x-axis label** checkbox. In the associated text field, type `Frequency (Hz)`.

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

5 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

6 In the **x minimum** text field, type `0`.

7 In the **x maximum** text field, type `600`.

8 In the **Vertical Spectrum** toolbar, click **Plot**.

STUDY 1

Step 1: Eigenfrequency

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

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

3 From the **Eigenfrequency search method** list, choose **Around shift**.

4 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `15`.

5 Select the **Search for eigenfrequencies around shift** checkbox. In the associated text field, type `65`.

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

RESULTS

Preferred Units 1

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

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, type `disp` in the text field.

5 In the tree, select **General** > **Displacement (m)**.

6 Click **OK**.

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

8 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Displacement	m	mm

9 Click **Apply**.

Mode Shape (solid)

1 Click the **Go to Default View** button in the **Graphics** toolbar.

The default plot shows the eigenmode shape for the first natural frequency.

Evaluate the participation factors for vertical translation for all computed eigenmodes.

PARTICIPATION FACTORS (STUDY 1)

1 Go to the **Participation Factors (Study 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 **Data** section.

2 From the **Plot columns** list, choose **Manual**.

3 In the **Columns** list box, select **Participation factor, normalized, Z-translation (1)**.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

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

Participation Factors

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

2 In the **Settings** window for **1D Plot Group**, type `Participation Factors` in the **Label** text field.

Also, evaluate the values of the shock spectrum input at all computed eigenfrequencies.

Vertical Spectrum

1 In the **Model Builder** window, click **Vertical Spectrum**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 In the **x minimum** text field, type `30`.

4 In the **x maximum** text field, type `400`.

5 In the **y minimum** text field, type `500`.

6 In the **y maximum** text field, type `850`.

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

Global 1

1 Right-click **Vertical Spectrum** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Global definitions** > **Functions** > **vsp(freq) - Vertical Spectrum**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `freq`.

7 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

9 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Values at eigenfrequencies`

11 In the **Vertical Spectrum** toolbar, click **Plot**.

Note that the third eigenmode has the maximum participation factor for vertical translation. It is also close to the maximum of the shock spectrum input. Plot this eigenmode.

Mode Shape (solid)

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

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

3 From the **Eigenfrequency (Hz)** list, choose **68.292**.

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

5 Click the **Go to Default View** button in the **Graphics** toolbar.

As a final check, calculate the sum of relative modal masses for all computed eigenmodes.

Global Evaluation 1

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

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Response Spectrum 1** > **Effective modal mass** > **rsp1.mEffLZ - Effective modal mass, Z-translation - kg**.

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

Expression	Unit	Description
`rsp1.mEffLZ/rsp1.mass`	1	

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

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

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

The value shows that the computed eigenmodes account for more than 94% of the total mass.

Add a special dataset for response spectrum evaluation.

Response Spectrum 3D 1

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

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

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

Maximum Displacement, Response Spectrum

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

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

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

Surface 1

1 Right-click **Maximum Displacement, Response Spectrum** and choose **Surface**.

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

3 In the **Expression** text field, type `w`.

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

5 In the **Maximum Displacement, Response Spectrum** toolbar, click **Plot**.

6 Click the **Go to Default View** button in the **Graphics** toolbar.

SOLID MECHANICS (SOLID)

Next, perform a transient analysis of the response under a body load in the form of a half-sine pulse with 11 ms duration and 50 g magnitude.

Base Excitation 1

1 In the **Physics** toolbar, click **Global** and choose **Base Excitation**.

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

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

0	x
0	y
50*g_const	z

Use a Time Dependent, Modal solver, which computes the transient response by representing the solution as a superposition of precomputed eigenmodes.

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.

STUDY 2

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** checkbox.

Step 1: Time Dependent, Modal

1 In the **Study** toolbar, click **More Study Steps** and choose **Time Dependent > Time Dependent, Modal**.

2 In the **Settings** window for **Time Dependent, Modal**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **ms**.

4 In the **Output times** text field, type `range(0,0.5,60)`.

5 From the **Tolerance** list, choose **User controlled**.

6 In the **Relative tolerance** text field, type `0.0001`.

7 In the **Load factor** text field, type `sin(2*pi/22[ms]*t)*(t<11[ms])`.

This factor multiplies all applied loads, including that due to the base acceleration.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Modal Solver 1**.

3 In the **Settings** window for **Modal Solver**, locate the **Eigenpairs** section.

4 From the **Solution** list, choose **Solution 1 (sol1)**.

5 In the **Damping ratios** text field, type `0.05`.

6 Click to expand the **Output** section. Not storing the time derivatives reduces the size of the output data.

7 Clear the **Store the first time derivative** checkbox.

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

RESULTS

Maximum Displacement, Time History

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

2 In the **Settings** window for **3D Plot Group**, type `Maximum Displacement, Time History` 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 **Maximum Displacement, Time History** and choose **Surface**.

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

3 In the **Expression** text field, type `timemax(0,60[ms],abs(w))`.

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

Maximum Displacement, Time History

1 In the **Model Builder** window, click **Maximum Displacement, Time History**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Surface: timemax(0, 50[ms], abs(w)) (m)`.

5 Clear the **Parameter indicator** text field.

6 In the **Maximum Displacement, Time History** toolbar, click **Plot**.

7 Click the **Go to Default View** button in the **Graphics** toolbar.

Displacement, Time

To see how the displacement actually varies in time, plot and animate the transient analysis result.

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

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

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

4 From the **Time (ms)** list, choose **11**.

Surface 1

1 Right-click **Displacement, Time** and choose **Surface**.

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

3 In the **Expression** text field, type `w`.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

4 In the **Displacement, Time** toolbar, click **Plot**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

Animate the solution.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Displacement, Time**.

4 Locate the **Frames** section. In the **Number of frames** text field, type `100`.

5 Click **Show Frame**.

6 Click the **Play** button in the **Graphics** toolbar.

Finally, analyze the stress distribution using the response spectrum.

Stress, Response Spectrum

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

2 In the **Settings** window for **3D Plot Group**, type `Stress, Response Spectrum` in the **Label** text field.

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

4 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 1 only.

Surface 1

1 Right-click **Stress, Response Spectrum** and choose **Surface**.

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

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

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

5 Click to expand the **Range** section. Select the **Manual color range** checkbox.

6 In the **Maximum** text field, type `150`.

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

8 In the **Stress, Response Spectrum** toolbar, click **Plot**.

The plot shows a rather high stress level at the memory chips.