Created in COMSOL Multiphysics 6.1

Prestressed Micromirror Vibrations: Thermoviscous-Thermoelasticity Coupling

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This model extends the analysis of the Prestressed Micromirror operation by including the losses from the thermoelastic effect and the interaction with surrounding air. This model also demonstrates the use of **Thermoviscous Acoustics-Thermoelasticity Boundary** multiphysics coupling.

Model Definition

Physics Interfaces and Couplings

This model uses the following interfaces and multiphysics couplings:

- **Thermoelasticity** — to compute the mechanical losses from irreversible heat transfer driven by thermoelastic effect, which can be particularly important for small structures.
- **Thermoviscous Acoustics, Frequency Domain** — to compute the acoustic variations of pressure, velocity, and temperature in geometries of small dimensions (microacoustics). This interface is used when modeling the response of transducers like microphones, miniature loudspeakers, and MEMS structures.
- **Pressure Acoustics, Frequency Domain** — to compute the pressure variations for the propagation of acoustic waves in fluids at quiescent background conditions.
- **Thermal Expansion** multiphysics coupling — to add an internal thermal strain caused by changes in temperature and account for the corresponding mechanical losses in the heat balance.
- **Thermoviscous Acoustics-Thermoelasticity Boundary** multiphysics coupling — to model thermoviscous losses in acoustic-structure interaction problems to great detail. It captures the effect of a non-ideal thermal condition at the fluid-structure interface which is important in MEMS.
- **Acoustic-Thermoviscous Acoustic Boundary** multiphysics coupling — to couple the **Thermoviscous Acoustics** interface to the **Pressure Acoustics** interface in both frequency and time domain.

Geometry

In addition to the solid domain (for the micromirror), the geometry model includes:

- A box enclosing the original micromirror, assigned to the **Thermoviscous Acoustics, Frequency Domain** interface.

- A half sphere surrounding the box, assigned to the **Pressure Acoustics, Frequency Domain** interface.
- An **Extra Dimension** for a **Perfectly Matched Boundary** around the sphere.

Studies

The full model includes both thermoelastic and thermoviscous losses. By disabling **Thermoviscous Acoustics** interface, the model includes only thermoelastic effect, or solid losses-only.

The model analyzes the operation of the micromirror through three studies. The first study computes stationary solutions for the initial normal stress, `sigma_pre`, of 3 GPa as well as the eigenfrequencies of the micromirror.

The second study computes the frequency response using the full model, including thermoviscous and solid losses. The study is done for 50–600 Hz (operating frequency) and for 13,150–13,500 Hz (near resonance).

The third study computes the frequency response using solid losses-only model. The study is done for 50–600 Hz (frequency of operation) and for 13,150–13,500 Hz (near resonance).

Results and Discussion

Figure 1 shows the fundamental eigenmode of the micromirror, f_0 = 13,339 Hz.

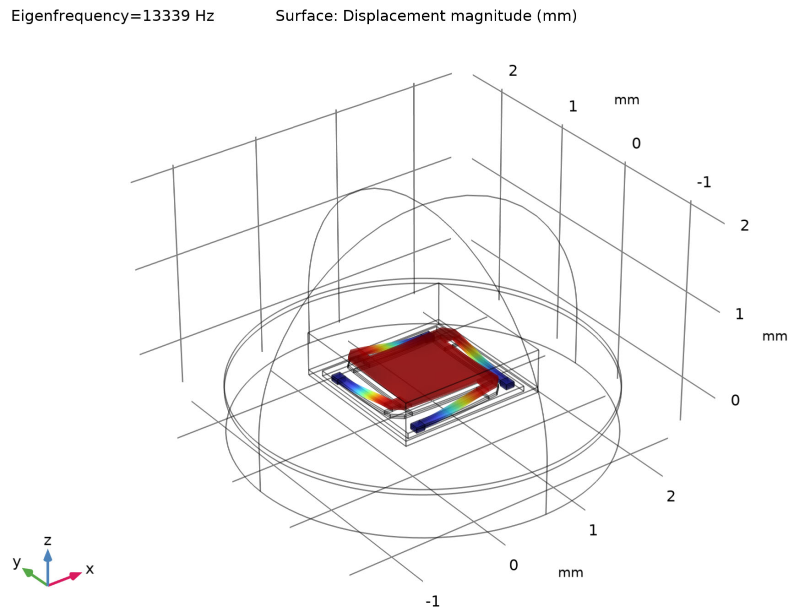


Figure 1: Fundamental eigenmode of the micromirror for initial normal stress = 3 GPa.

Figure 2 contains a surface plot of the displacement for $f = 50$ Hz.

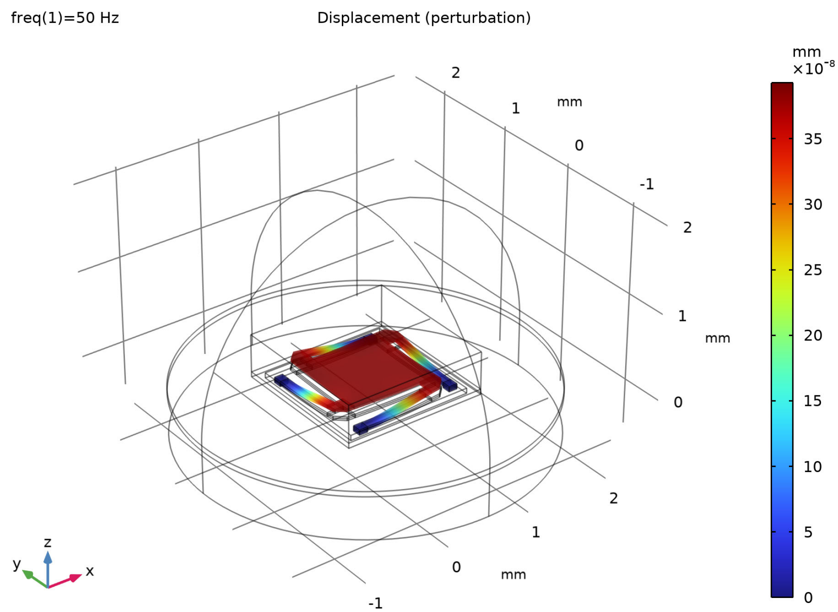


Figure 2: Surface plot of displacement for $f = 50$ Hz.

[Figure 3](#) visualizes the temperature distribution for f = 50 Hz using a combination of surface and slice plots.

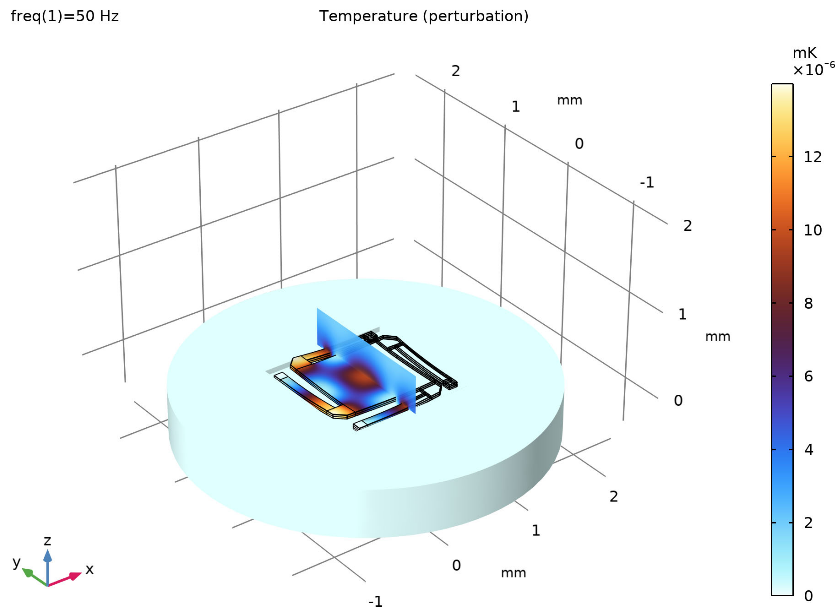


Figure 3: Surface and slice plots of temperature for f = 50 Hz.

Figure 4 shows the acoustic velocity field surrounding the micromirror for f = 50 Hz.

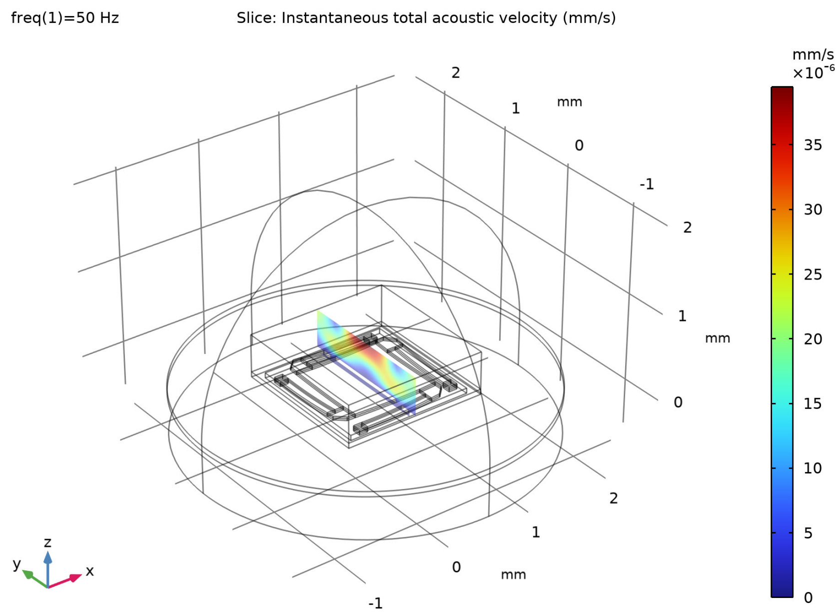


Figure 4: The acoustic velocity field around the micromirror for f = 50 Hz.

[Figure 5](#) shows the acoustic pressure field surrounding the micromirror for f = 50 Hz

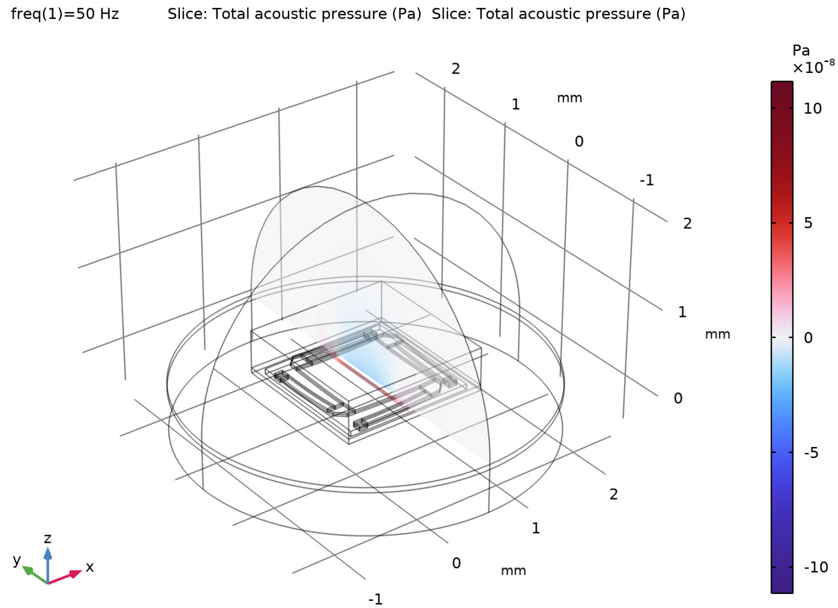


Figure 5: The acoustic pressure field around the micromirror for f = 50 Hz.

Figure 6 shows a comparison of the frequency response near resonance for the full versus the mechanical loss-only model.

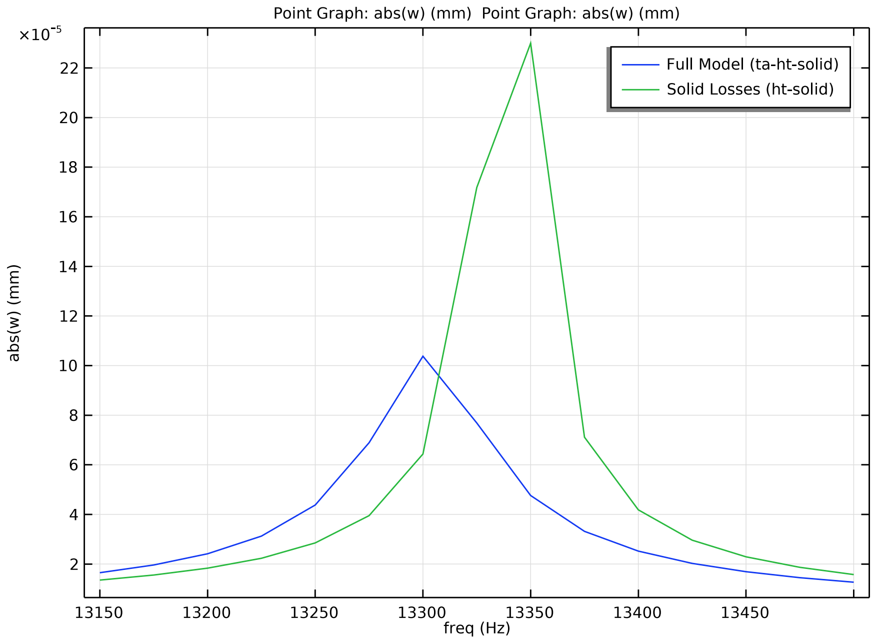


Figure 6: Frequency response around resonant frequency for the full model and the mechanical losses-only model.

Figure 7 shows a comparison of the frequency response in the range 50–600 Hz for the full versus the mechanical loss-only model.

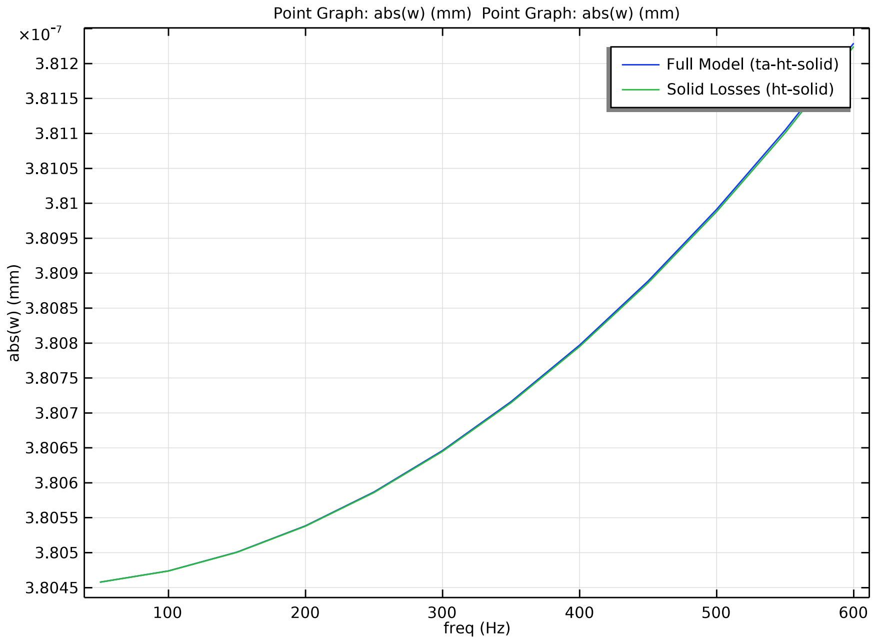


Figure 7: Frequency response around operating frequency (for 50–600 Hz) for the full model and the solid losses-only.

As expected, near resonance, the vibration frequency and amplitude are less for the full model than for the solid losses-only model. Around operating frequencies in the range 50–600 Hz, however, the difference in losses is negligible.

Reference

1. G. Kovacs, *Micromachined Transducers Sourcebook*, WCM McGraw-Hill, 1998.

Application Library path: `MEMS_Module/Actuators/micromirror_prestressed_vibration`

Modeling Instructions

Start by creating a new 3D model.

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>Thermal-Structure Interaction>Thermoelasticity**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics>Thermoviscous Acoustics, Frequency Domain (ta)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Solid Mechanics>Eigenfrequency, Prestressed**.

10 Click **Done**.

MULTIPHYSICS

Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic-Thermoviscous Acoustic Boundary**.

Thermoviscous Acoustic-Thermoelasticity Boundary 1 (tatb1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Thermoviscous Acoustic-Thermoelasticity Boundary**.

Define and enter the values for the following global parameters.

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 In the table, enter the following settings:

Name	Expression	Value	Description
sigma_pre	3[GPa]	3E9 Pa	Initial normal stress
fc	13400[Hz]	13400 Hz	Typical frequency
dvisc	0.22[mm]*sqrt(100[Hz]/fc)	1.9005E-5 m	Viscous boundary layer thickness at fc

Build geometry model of micromirror.

GEOMETRY 1

Set the geometry unit to mm for convenience.

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

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

3 From the **Length unit** list, choose **mm**.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Square 1 (sq1)

In the **Work Plane** toolbar, click **Square**.

Work Plane 1 (wp1)>Square 2 (sq2)

1 In the **Work Plane** toolbar, click **Square**.

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

3 In the **Side length** text field, type `0.2`.

4 Locate the **Position** section. In the **yw** text field, type `1`.

Work Plane 1 (wp1)>Chamfer 1 (cha1)

1 In the **Work Plane** toolbar, click **Chamfer**.

2 Click the Zoom Extents button in the Graphics toolbar.

3 On the object sq2, select Point 4 only.

4 In the Settings window for Chamfer, locate the Distance section.

5 In the Distance from vertex text field, type 0.1.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the Work Plane toolbar, click Rectangle.

2 In the Settings window for Rectangle, locate the Size and Shape section.

3 In the Width text field, type 0.9.

4 In the Height text field, type 0.1.

5 Locate the Position section. In the xw text field, type 0.2.

6 In the yw text field, type 1.1.

Work Plane 1 (wp1)>Square 3 (sq3)

1 In the Work Plane toolbar, click Square.

2 In the Settings window for Square, locate the Size section.

3 In the Side length text field, type 0.1.

4 Locate the Position section. In the xw text field, type 1.

5 In the yw text field, type 1.1.

Work Plane 1 (wp1)>Rotate 1 (rot1)

1 In the Work Plane toolbar, click Transforms and choose Rotate.

2 Select the objects cha1, r1, and sq3 only.

3 In the Settings window for Rotate, locate the Rotation section.

4 In the Angle text field, type range(90,90,360).

5 Locate the Center of Rotation section. In the xw text field, type 0.5.

6 In the yw text field, type 0.5.

Extrude 1 (ext1)

1 In the Model Builder window, right-click Geometry 1 and choose Extrude.

2 In the Settings window for Extrude, locate the Distances section.

3 In the table, enter the following settings:

Distances (mm)
0.02
0.04

Extrude 2 (ext2)

1 In the **Geometry** toolbar, click **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **General** section.
3 From the **Extrude from** list, choose **Faces**.
4 On the object **ext1**, select Boundaries 17, 26, 111, and 120 only.
5 Locate the **Distances** section. In the table, enter the following settings:

Distances (mm)
0.02

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type 1.45.
4 In the **Depth** text field, type 1.45.
5 In the **Height** text field, type 0.06.
6 Locate the **Position** section. In the **x** text field, type -0.225.
7 In the **y** text field, type -0.225.
8 In the **z** text field, type -0.02.

Add a cylindrical base for micromirror.

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 2.
4 In the **Height** text field, type 0.5.
5 Locate the **Position** section. In the **x** text field, type 0.5.
6 In the **y** text field, type 0.5.
7 In the **z** text field, type -0.52.

Cylinder 2 (cyl2)

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 2.
4 In the **Height** text field, type 0.06.

5 Locate the **Position** section. In the **x** text field, type `0.5`.

6 In the **y** text field, type `0.5`.

7 In the **z** text field, type `-0.02`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Click the **Zoom Extents** button in the **Graphics** toolbar.

3 Select the object **cyl2** only.

4 In the **Settings** window for **Difference**, locate the **Difference** section.

5 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

6 Select the object **blk1** only.

Add a box for thermoviscous acoustics domain.

Block 2 (blk2)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.65`.

4 In the **Depth** text field, type `1.65`.

5 In the **Height** text field, type `.5`.

6 Locate the **Position** section. In the **x** text field, type `-0.325`.

7 In the **y** text field, type `-0.325`.

8 In the **z** text field, type `-0.02`.

Add a sphere for pressure acoustics domain.

Sphere 1 (sph1)

1 In the **Geometry** toolbar, click **Sphere**.

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

3 In the **Radius** text field, type `2`.

4 Locate the **Position** section. In the **x** text field, type `0.5`.

5 In the **y** text field, type `0.5`.

6 In the **z** text field, type `0.04`.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

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

3 In the **z-coordinate** text field, type `0.04`.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **sph1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **par1**, select Domain 1 only.

Define selections for convenience when assigning material properties and boundary conditions.

DEFINITIONS

Air TA

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5 6` in the **Selection** text field.

5 Click **OK**.

Air ACPR

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3` in the **Selection** text field.

5 Click **OK**.

Substrate

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1 2 4` in the **Selection** text field.

5 Click **OK**.

Solid

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `7-36` in the **Selection** text field.

5 Click **OK**.

Solid Aluminum

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `9-18, 23, 24, 29-36` in the **Selection** text field.

5 Click **OK**.

Solid Steel

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `7, 8, 19-22, 25-28` in the **Selection** text field.

5 Click **OK**.

Solid-TA

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `24-28, 30, 31, 33, 34, 37-39, 41, 44, 47-52, 54, 55, 57, 58, 60-73, 75, 76, 78, 80-82, 84, 85, 87, 89-92, 94, 95, 98-106, 108, 109, 111, 113-115, 123, 124, 126, 128-130, 133-138, 140, 141, 143, 145-149, 152, 155, 157-162, 164, 165, 167, 168, 170-172, 174, 175, 178, 181-192` in the **Selection** text field.

6 Click **OK**.

Air

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type `Air` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
4 In the **Add** dialog box, in the **Selections to add** list, choose **Air TA** and **Air ACPR**.
5 Click **OK**.

Participation Factors 1 (mpf1)

In the **Definitions** toolbar, click **Physics Utilities** and choose **Participation Factors**.

Select material models from the material library and assign to the respective domains.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Aluminum**.
4 Click **Add to Component** in the window toolbar.
5 In the tree, select **Built-in>Structural steel**.
6 Click **Add to Component** in the window toolbar.
7 In the tree, select **Built-in>Air**.
8 Click **Add to Component** in the window toolbar.
9 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat1)

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

Structural steel (mat2)

1 In the **Model Builder** window, click **Structural steel (mat2)**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Solid Steel**.

Air (mat3)

1. In the **Model Builder** window, click **Air (mat3)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Air**.

Define Extra Dimension from Perfectly Matched Boundary.

ADD COMPONENT

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

GEOMETRY (EXTRA DIMENSION FROM PERFECTLY MATCHED BOUNDARY 1)

In the **Settings** window for **Geometry**, type `Geometry (Extra Dimension from Perfectly Matched Boundary 1)` in the **Label** text field.

Interval 1 (i1)

Right-click **Global Definitions>Extra Dimension 1 (xdim1)> Geometry (Extra Dimension from Perfectly Matched Boundary 1)** and choose **Interval**.

MESH (EXTRA DIMENSION FROM PERFECTLY MATCHED BOUNDARY 1)

In the **Settings** window for **Mesh**, type `Mesh (Extra Dimension from Perfectly Matched Boundary 1)` in the **Label** text field.

Distribution 1

1. Right-click **Global Definitions>Extra Dimension 1 (xdim1)> Mesh (Extra Dimension from Perfectly Matched Boundary 1)** and choose **Distribution**.
2. Select Domain 1 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `8`.

Edge 1

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

DEFINITIONS (XDIM1)

In the **Model Builder** window, expand the **Global Definitions>Extra Dimension 1 (xdim1)> Definitions** node.

Points to Attach 1

1. In the **Model Builder** window, expand the **Global Definitions>Extra Dimension 1 (xdim1)> Definitions>Extra Dimensions** node, then click **Points to Attach 1**.
2. Select Boundary 1 only.

EXTRA DIMENSION FROM PERFECTLY MATCHED BOUNDARY 1

1 In the **Model Builder** window, under **Global Definitions** click **Extra Dimension 1 (xdim1)**.

2 In the **Settings** window for **Extra Dimension**, type `Extra Dimension from Perfectly Matched Boundary 1` in the **Label** text field.

3 In the **Name** text field, type `acpr_pmb1_xdim`.

4 Locate the **Frames** section. Find the **Spatial frame coordinates** subsection. In the table, enter the following settings:

First	Second	Third
x1	y1	z1

5 Find the **Material frame coordinates** subsection. In the table, enter the following settings:

First	Second	Third
X_acpr_pmb1_xdim	Y_acpr_pmb1_xdim	Z_acpr_pmb1_xdim

6 Find the **Geometry frame coordinates** subsection. In the table, enter the following settings:

First	Second	Third
Xg_acpr_pmb1_xdim	Yg_acpr_pmb1_xdim	Zg_acpr_pmb1_xdim

7 Find the **Mesh frame coordinates** subsection. In the table, enter the following settings:

First	Second	Third
Xm_acpr_pmb1_xdim	Ym_acpr_pmb1_xdim	Zm_acpr_pmb1_xdim

Define Deforming Domain.

COMPONENT 1 (COMP1)

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

Deforming Domain 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Domains> Deforming Domain**.

2 In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5 6` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Deforming Domain**, locate the **Smoothing** section.

7 From the **Mesh smoothing type** list, choose **Hyperelastic**.

Setup the initial and boundary conditions for Solid Mechanics.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `10 18 24 36` in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

9 In the S_0 table, enter the following settings:

sigma_pre	0	0
0	sigma_pre	0
0	0	0

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Initial Stress and Strain 2

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `9 17 23 35` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

8 In the S_0 table, enter the following settings:

-sigma_pre	0	0
0	-sigma_pre	0
0	0	0

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 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `40 53 150 163` in the **Selection** text field.

5 Click **OK**.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `85` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Boundary Load**, locate the **Force** section.

7 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
linper(1)	z

8 In the **Model Builder** window, click **Solid Mechanics (solid)**.

9 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

10 From the **Selection** list, choose **Solid**.

Setup the initial and boundary conditions for Heat Transfer.

HEAT TRANSFER IN SOLIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

2 In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Solid**.

Temperature 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `40 53 150 163` in the **Selection** text field.

5 Click **OK**.

Setup the initial and boundary conditions for Thermoviscous Acoustics.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Frequency Domain (ta)**.

2 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5 6` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, locate the **Domain Selection** section.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog box, type `5 6` in the **Selection** text field.

9 Click **OK**.

10 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, locate the **Domain Selection** section.

11 From the **Selection** list, choose **Manual**.

12 Click **Clear Selection**.

13 Click **Paste Selection**.

14 In the **Paste Selection** dialog box, type `5 6` in the **Selection** text field.

15 Click **OK**.

Wall 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 15 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Wall**, locate the **Mechanical** section.

7 From the **Mechanical condition** list, choose **Slip**.

8 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

Setup the initial and boundary conditions for Pressure Acoustics.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air ACPR**.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, click **Cancel**.

Perfectly Matched Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfectly Matched Boundary**.

2 In the **Settings** window for **Perfectly Matched Boundary**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 7 9 118 119 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Perfectly Matched Boundary**, locate the **Geometry** section.

7 From the **Attenuation direction** list, choose **Normal**.

MULTIPHYSICS

Thermal Expansion 1 (te1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Thermal Expansion 1 (te1)**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Heat Sources** section.

3 Select the **Mechanical losses** check box.

4 Click the Transparency button in the **Graphics** toolbar.

Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)

1 In the **Model Builder** window, click **Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)**.

2 In the **Settings** window for **Acoustic-Thermoviscous Acoustic Boundary**, locate the **Boundary Selection** section.

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

Thermoviscous Acoustic-Thermoelasticity Boundary 1 (tatb1)

1 In the **Model Builder** window, click **Thermoviscous Acoustic-Thermoelasticity Boundary 1 (tatb1)**.

2 In the **Settings** window for **Thermoviscous Acoustic-Thermoelasticity Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Solid-TA**.

Define the mesh for the model.

MESH 1

Mapped 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `30, 37, 47, 60, 80, 89, 98, 113, 128, 145, 157, 170, 178` in the **Selection** text field.

5 Click **OK**.

Distribution 1

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

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

3 Click **Copy Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `45, 115, 129, 134, 160, 173, 219, 304` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 In the **Number of elements** text field, type `12`.

Distribution 2

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

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `56, 64, 74, 75, 95, 159, 199, 227, 243, 261, 288, 298` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `3`.

Distribution 3

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

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `37 152 194 309` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `6`.

Swept 1

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

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

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

4 From the **Selection** list, choose **Solid**.

Distribution 1

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

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

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

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

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

4 Click Paste Selection.

5 In the **Paste Selection** dialog box, type `6` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Free Tetrahedral**, click to expand the **Scale Geometry** section.

Size 1

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

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

3 Click the **Custom** button.

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `19 20 23 193` in the **Selection** text field.

7 Click **OK**.

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

9 Select the **Maximum element size** check box. In the associated text field, type `0.04`.

Free Tetrahedral 2

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size 1

1 Right-click **Free Tetrahedral 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** check box. In the associated text field, type `0.8`.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5 6` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.

8 From the **Handling of sharp edges** list, choose **No special handling**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Solid-TA**.
4 Locate the **Layers** section. In the **Number of layers** text field, type `3`.
5 From the **Thickness specification** list, choose **All layers**.
6 In the **Total thickness** text field, type `dvisc`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **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. In the **Maximum element size** text field, type `1`.
5 In the **Minimum element size** text field, type `0.01`.
6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

Add a study to compute eigenfrequencies of solid structure.

STUDY 1 - STRUCTURAL MODES (LOSSLESS)

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Study 1 - Structural Modes (lossless)` in the **Label** text field.
3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1 - Structural Modes (lossless)** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics (solid)	√	Automatic (Stationary)
Heat Transfer in Solids (ht)		Automatic (Stationary)

Physics interface	Solve for	Equation form
Thermoviscous Acoustics, Frequency Domain (ta)		Automatic (Frequency domain)
Pressure Acoustics, Frequency Domain (acpr)		Automatic (Frequency domain)
Moving mesh (Component 1)		Automatic

4 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Thermal Expansion 1 (te1)		Automatic (Stationary)

5 Right-click **Study 1 - Structural Modes (lossless)>Step 1: Stationary** and choose **Compute Selected Step**.

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `4`.

4 In the **Search for eigenfrequencies around** text field, type `8000[Hz]`.

5 From the **Eigenfrequency search method around shift** list, choose **Larger real part**.

6 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics (solid)	√	Automatic (Eigenfrequency)
Heat Transfer in Solids (ht)		Automatic (Stationary)
Thermoviscous Acoustics, Frequency Domain (ta)		Automatic (Frequency domain)
Pressure Acoustics, Frequency Domain (acpr)		Automatic (Frequency domain)
Moving mesh (Component 1)		Automatic

7 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Thermal Expansion 1 (te1)		Automatic (Stationary)
Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)		Automatic (Stationary)
Thermoviscous Acoustic-Thermoelasticity Boundary 1 (tatb1)		Automatic (Frequency domain perturbation)

8 Right-click **Step 2: Eigenfrequency** and choose **Compute Selected Step**.

Add a study to compute frequency response for the full model.

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>Solid Mechanics>Frequency Domain, Prestressed**.

4 Click **Add Study** in the window toolbar.

STUDY 2 - FREQUENCY RESPONSE: FULL MODEL (TA-HT-SOLID)

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Frequency Response: Full Model (ta-ht-solid)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2 - Frequency Response: Full Model (ta-ht-solid)** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sigma_pre (Initial normal stress)	`range(1,1,3)`	GPa

6 Right-click **Study 2 - Frequency Response: Full Model (ta-ht-solid)>Step 1: Stationary** and choose **Compute Selected Step**.

Step 2: Frequency Domain Perturbation

1 In the **Model Builder** window, click **Step 2: Frequency Domain Perturbation**.

2 In the **Settings** window for **Frequency Domain Perturbation**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(50,50,600) range(13150,25,13500)`.

4 From the **Reuse solution from previous step** list, choose **No**.

5 Find the **Values of linearization point** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Parameter value (sigma_pre (GPa))** list, choose **3 GPa**.

7 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics (solid)	√	Automatic (Frequency domain)
Heat Transfer in Solids (ht)	√	Automatic (Frequency domain perturbation)
Thermoviscous Acoustics, Frequency Domain (ta)	√	Automatic (Frequency domain)
Pressure Acoustics, Frequency Domain (acpr)	√	Automatic (Frequency domain)
Moving mesh (Component 1)		Automatic

8 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

9 From the **Use** list, choose **Solution Store 2 (sol4)**.

10 From the **Parameter value (sigma_pre (GPa))** list, choose **3 GPa**.

11 Right-click **Step 2: Frequency Domain Perturbation** and choose **Compute Selected Step**.

Add a study to compute frequency response for the solid losses-only model.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Solid Mechanics>Frequency Domain, Prestressed**.
3 Click **Add Study** in the window toolbar.

STUDY 3 - FREQUENCY RESPONSE SOLID LOSSES (HT-SOLID)

1 In the **Model Builder** window, click **Study 3**.
2 In the **Settings** window for **Study**, type `Study 3 - Frequency Response Solid Losses (ht-solid)` in the **Label** text field.
3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3 - Frequency Response Solid Losses (ht-solid)** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics (solid)	√	Automatic (Stationary)
Heat Transfer in Solids (ht)	√	Automatic (Stationary)
Thermoviscous Acoustics, Frequency Domain (ta)		Automatic (Frequency domain)
Pressure Acoustics, Frequency Domain (acpr)		Automatic (Frequency domain)
Moving mesh (Component 1)		Automatic

4 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
5 Click + **Add**.
6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sigma_pre (Initial normal stress)	`range(1,1,3)`	`GPa`

7 Right-click **Study 3 - Frequency Response Solid Losses (ht-solid)>Step 1: Stationary** and choose **Compute Selected Step**.

Step 2: Frequency Domain Perturbation

1 In the **Model Builder** window, click **Step 2: Frequency Domain Perturbation**.

2 In the **Settings** window for **Frequency Domain Perturbation**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(50,50,600) range(13150,25,13500)`.

4 Find the **Values of linearization point** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Parameter value (sigma_pre (GPa))** list, choose **3 GPa**.

6 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics (solid)	√	Automatic (Frequency domain)
Heat Transfer in Solids (ht)	√	Automatic (Frequency domain perturbation)
Thermoviscous Acoustics, Frequency Domain (ta)		Automatic (Frequency domain)
Pressure Acoustics, Frequency Domain (acpr)		Automatic (Frequency domain)
Moving mesh (Component 1)		Automatic

7 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)		Automatic (Stationary)
Thermoviscous Acoustic-Thermoelasticity Boundary 1 (tatb1)		Automatic (Frequency domain perturbation)

8 Right-click **Step 2: Frequency Domain Perturbation** and choose **Compute Selected Step**.

RESULTS

Mesh 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Datasets** and choose **Mesh**.

Create a mesh plot.

3 In the **Settings** window for **Mesh**, click **Plot**.

Mesh Plot

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mesh 1**.

4 Click to expand the **Selection** section. Click to expand the **Title** section. Locate the **Plot Settings** section. Select the **Propagate hiding to lower dimensions** check box.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Mesh 1

1 Right-click **Mesh Plot** and choose **Mesh**.

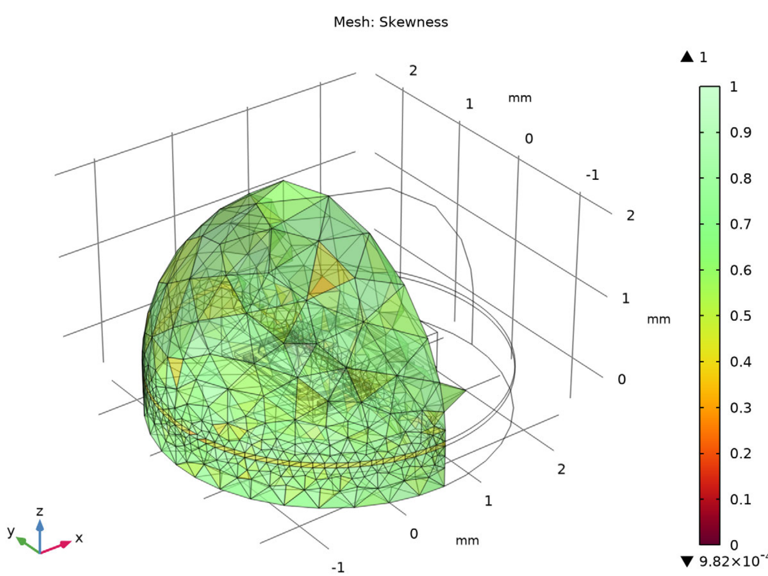
2 In the **Settings** window for **Mesh**, locate the **Level** section.

3 From the **Level** list, choose **Volume**.

4 Click to expand the **Element Filter** section. Select the **Enable filter** check box.

5 In the **Expression** text field, type `x<0.5[mm]`.

6 In the **Mesh Plot** toolbar, click **Plot**.



Plot mode shapes from eigenfrequency study.

Mode Shape (solid)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape (solid)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

4 Locate the **Color Legend** section. Clear the **Show legends** check box.

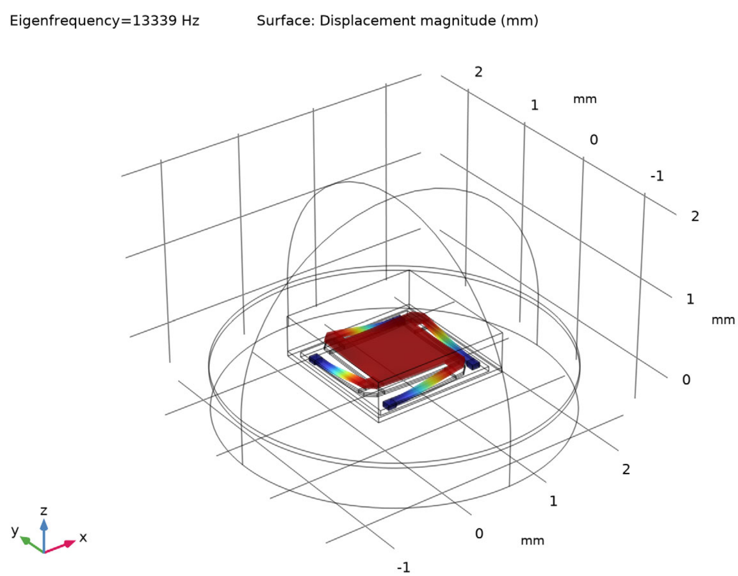
Surface 1

Right-click **Mode Shape (solid)** and choose **Surface**.

Deformation 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Deformation**.

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



Plot stress from stationary study for the full model.

Stress (stationary)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution Store 2 (sol4)**.

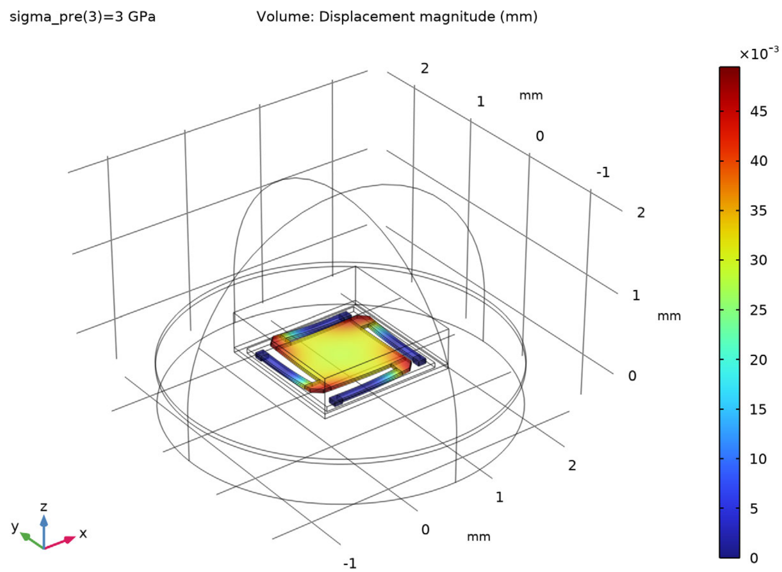
4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

5 Locate the **Data** section. From the **Parameter value (sigma_pre (GPa))** list, choose **3**.

Volume 1

1 Right-click **Stress (stationary)** and choose **Volume**.

2 In the **Stress (stationary)** toolbar, click **Plot**.



Plot displacement from stationary study for the full model.

Displacement (stationary)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution Store 2 (sol4)**.

4 From the **Parameter value (sigma_pre (GPa))** list, choose **3**.

5 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

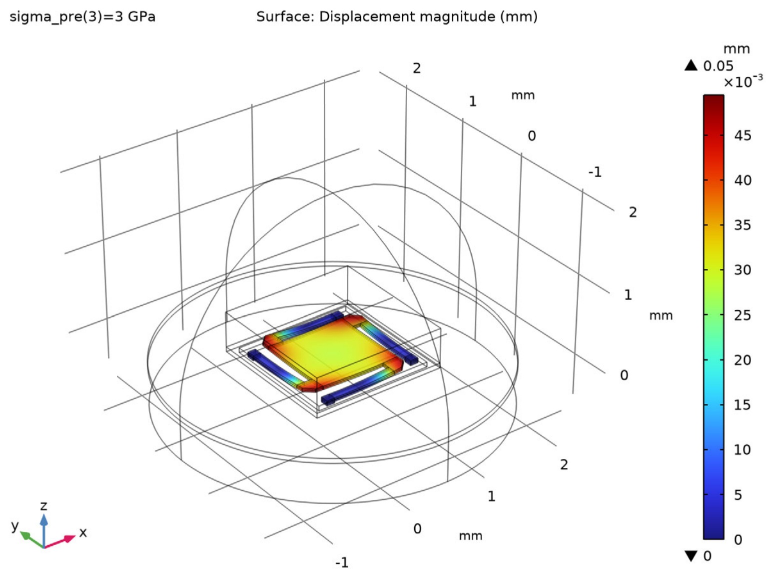
6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

7 Select the **Show units** check box.

8 In the **Displacement (stationary)** toolbar, click **Plot**.

Surface 1

1 Right-click **Displacement (stationary)** and choose **Surface**.
2 In the **Displacement (stationary)** toolbar, click **Plot**.



Plot temperature from stationary study for the full model.

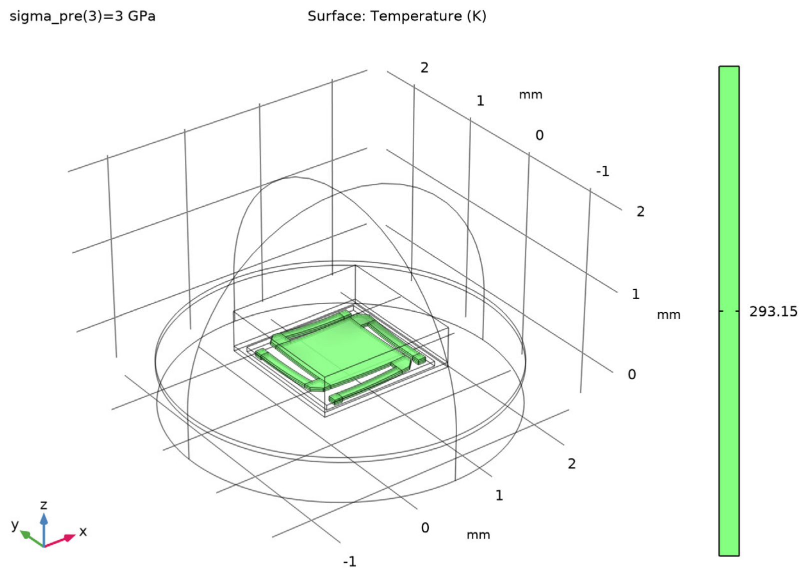
Temperature (stationary)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Temperature (stationary)` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution Store 2 (sol4)**.
4 From the **Parameter value (sigma_pre (GPa))** list, choose **3**.
5 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Surface 1

1 Right-click **Temperature (stationary)** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `T`.

4 In the **Temperature (stationary)** toolbar, click **Plot**.



Plot displacement from frequency domain perturbation study for the full model.

Displacement (perturbation)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution 3 (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

6 Locate the **Color Legend** section. Select the **Show units** check box.

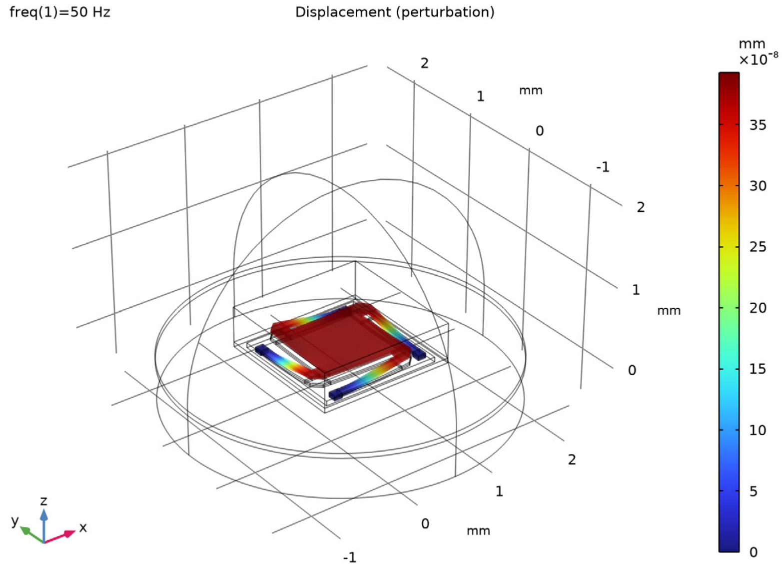
Surface 1

Right-click **Displacement (perturbation)** and choose **Surface**.

Deformation 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Deformation**.

2 In the **Displacement (perturbation)** toolbar, click **Plot**.



Plot temperature from frequency domain perturbation study for the full model.

Temperature (perturbation)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature (perturbation)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution 3 (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

6 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

1 Right-click **Temperature (perturbation)** and choose **Surface**.

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

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

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

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.

7 Click **OK**.

8 In the **Temperature (perturbation)** toolbar, click **Plot**.

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** check box. In the associated text field, type `1`.

Filter 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `x<0.5[mm]`.

Selection 1

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

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

3 From the **Selection** list, choose **Solid-TA**.

Slice 1

1 In the **Model Builder** window, right-click **Temperature (perturbation)** and choose **Slice**.

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

3 In the **Expression** text field, type `ta.T_t`.

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

5 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

6 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

8 In the **Temperature (perturbation)** toolbar, click **Plot**.

Transparency 1

1 Right-click **Slice 1** and choose **Transparency**.

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

3 In the **Transparency** text field, type `0.05`.

4 In the **Fresnel transmittance** text field, type `0.1`.

5 In the **Temperature (perturbation)** toolbar, click **Plot**.

Slice 2

1 In the **Model Builder** window, right-click **Temperature (perturbation)** and choose **Slice**.

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

3 In the **Expression** text field, type T.

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

5 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 In the **Temperature (perturbation)** toolbar, click **Plot**.

Volume 1

1 Right-click **Temperature (perturbation)** and choose **Volume**.

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

3 In the **Expression** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Custom**.

6 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

7 Click **Define custom colors**.

8 Set the RGB values to 224, 255, and 255, respectively.

9 Click **Add to custom colors**.

10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 1 2 4 in the **Selection** text field.

5 Click **OK**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Aluminum (anodized)**.

5 Locate the **Color** section. Select the **Use the plot's color** check box.

6 In the **Temperature (perturbation)** toolbar, click **Plot**.

Line 1

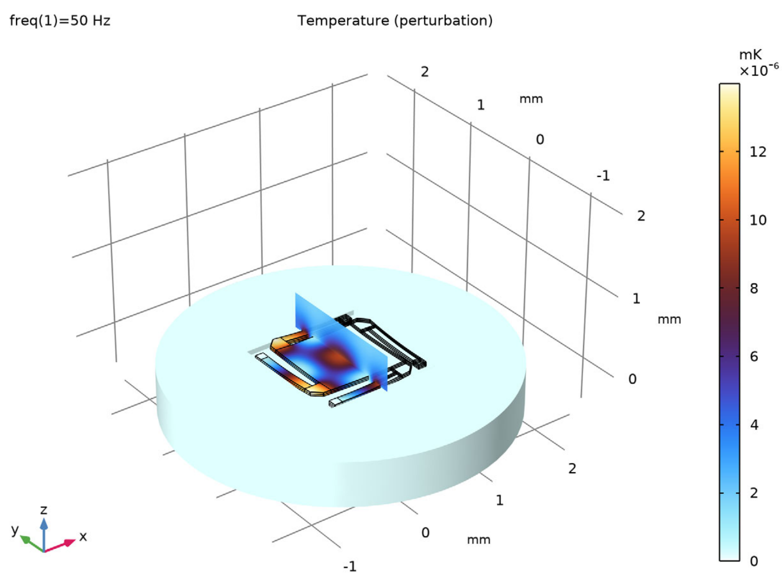
1 In the **Model Builder** window, right-click **Temperature (perturbation)** and choose **Line**.

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

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Black**.

5 In the **Temperature (perturbation)** toolbar, click **Plot**.



Create a slice plot of acoustic velocity from frequency domain perturbation study for the full model.

Acoustic Velocity (perturbation)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Velocity (perturbation)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution 3 (sol3)**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

5 Locate the **Color Legend** section. Select the **Show units** check box.

Slice 1

1 Right-click **Acoustic Velocity (perturbation)** and choose **Slice**.

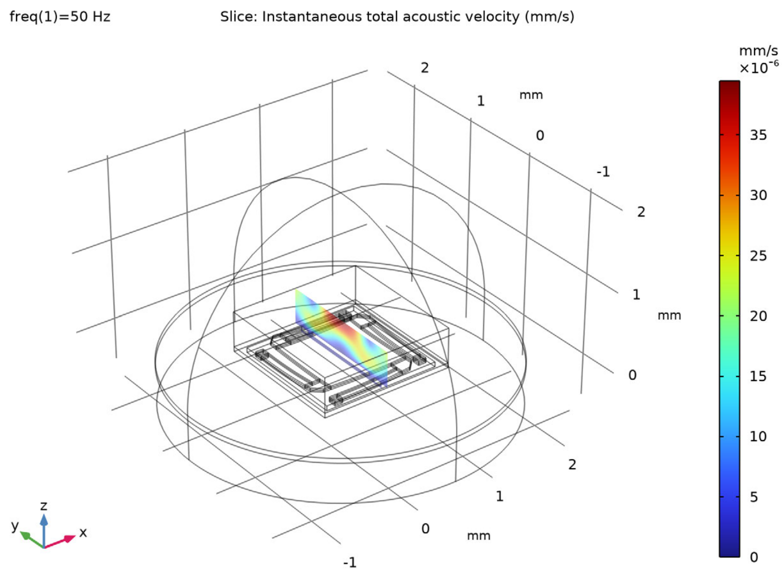
2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `ta.v_inst`.

4 From the **Unit** list, choose **mm/s**.

5 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

6 In the **Acoustic Velocity (perturbation)** toolbar, click **Plot**.



Create a slice plot of acoustic pressure from frequency domain perturbation study for the full model.

Acoustic Pressure (perturbation)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure (perturbation)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution 3 (sol3)**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

5 Locate the **Color Legend** section. Select the **Show units** check box.

Slice 1

1 Right-click **Acoustic Pressure (perturbation)** and choose **Slice**.

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

3 In the **Expression** text field, type `ta.p_t`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Wave>Wave** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Slice**, locate the **Coloring and Style** section.

9 From the **Scale** list, choose **Linear symmetric**.

10 In the **Acoustic Pressure (perturbation)** toolbar, click **Plot**.

Slice 2

1 In the **Model Builder** window, right-click **Acoustic Pressure (perturbation)** and choose **Slice**.

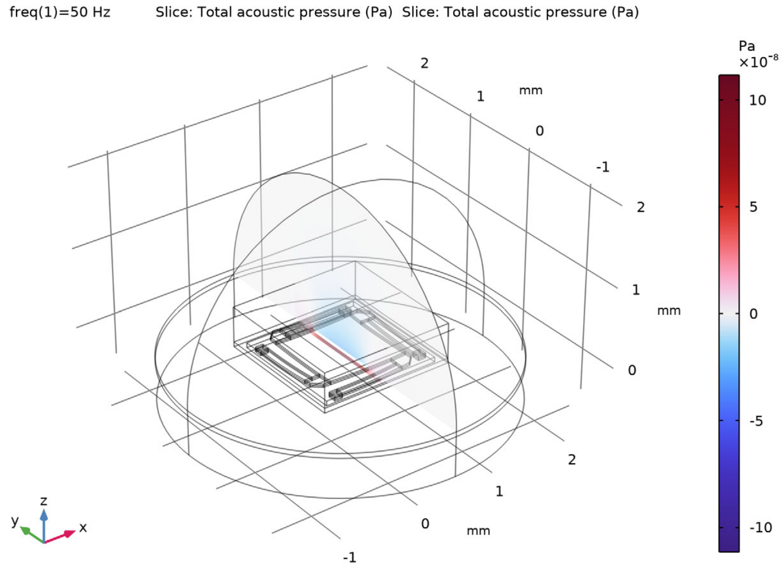
2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_t`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

5 Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

6 In the **Acoustic Pressure (perturbation)** toolbar, click **Plot**.



Plot displacement versus frequency to compare response between full model and solid losses-only model near resonance.

Response Comparison (at resonance)

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Response Comparison (at resonance)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution 3 (sol3)**.

4 From the **Parameter selection (freq)** list, choose **Manual**.

5 In the **Parameter indices (1-27)** text field, type `range(13,1,27)`.

Point Graph 1

1 Right-click **Response Comparison (at resonance)** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `97` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `abs(w)`.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **1**.

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

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

11 In the table, enter the following settings:

Legends
`Full Model (ta-ht-solid)`

12 In the **Response Comparison (at resonance)** toolbar, click **Plot**.

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 3 - Frequency Response Solid Losses (ht-solid)/ Solution 5 (sol5)**.

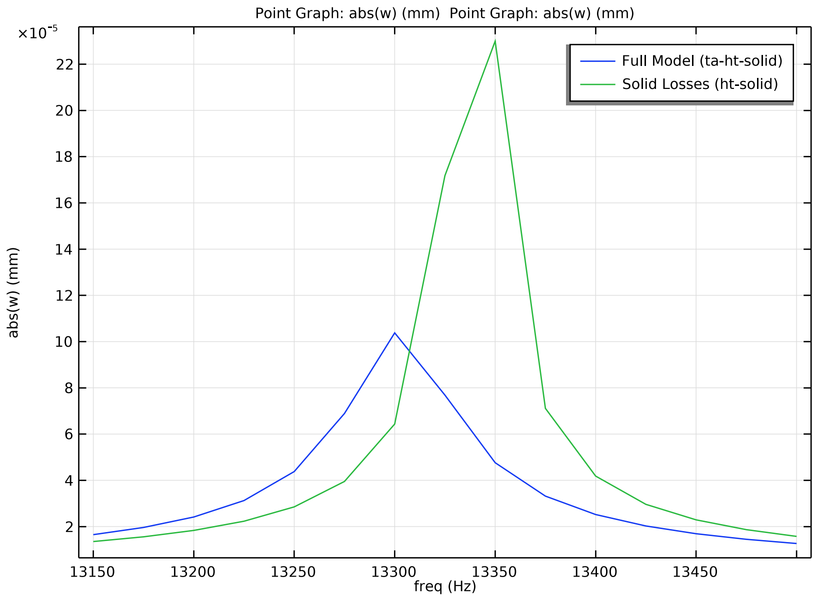
4 From the **Parameter selection (freq)** list, choose **Manual**.

5 In the **Parameter indices (1-27)** text field, type `range(13,1,27)`.

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

Legends
`Solid Losses (ht-solid)`

7 In the **Response Comparison (at resonance)** toolbar, click **Plot**.



Plot displacement versus frequency to compare response between full model and solid losses-only model for 50 to 600 Hz.

Response Comparison (typical operation)

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Response Comparison (typical operation)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Response: Full Model (ta-ht-solid)/Solution 3 (sol3)**.

4 From the **Parameter selection (freq)** list, choose **Manual**.

5 In the **Parameter indices (1-27)** text field, type `range(1,1,12)`.

Point Graph 1

1 Right-click **Response Comparison (typical operation)** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 97 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `abs(w)`.

8 Locate the **Coloring and Style** section. From the **Width** list, choose **1**.

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

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

11 In the table, enter the following settings:

Legends
`Full Model (ta-ht-solid)`

12 In the **Response Comparison (typical operation)** toolbar, click **Plot**.

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 3 - Frequency Response Solid Losses (ht-solid)/ Solution 5 (sol5)**.

4 From the **Parameter selection (freq)** list, choose **Manual**.

5 In the **Parameter indices (1-27)** text field, type `range(1,1,12)`.

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

Legends
`Solid Losses (ht-solid)`

7 In the **Response Comparison (typical operation)** toolbar, click **Plot**.