



Vibrating Micromirror with Viscous and Thermal Damping

Introduction

Micromirror systems are part of small MEMS (microelectromechanical systems) systems for integrated scanning micromirror systems in telecommunications and consumer applications. The mirrors have low power consumption and the manufacturing cost is low. Their applications range from microscopy applications, medical imaging, scanners, and heads-up displays, to some fiber optics applications.

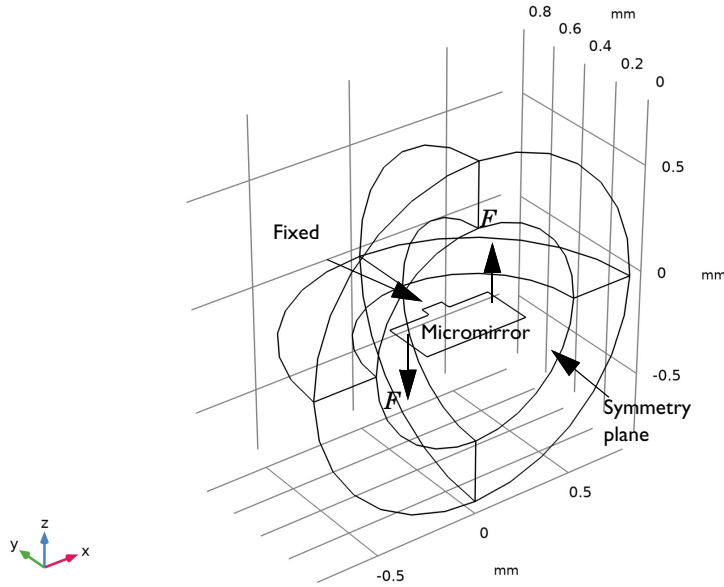


Figure 1: Geometry of the micromirror system including the fixed constraint, the torquing force components \vec{F} , and the symmetry plane.

The dynamic vibrating behavior of the micromirror (or other MEMS resonators) and especially the damping is an important factor affecting the design of their actuator systems. In the development process of new micromirror systems, modeling the damping correctly, and thus predicting the performance of the system, can save both time and money.

This model presents a highly simplified micromirror geometry modeled with the Shell physics interface coupled to Thermoviscous Acoustics, Frequency Domain. The model compares the simulated damped vibration results to a simple oscillator model tuned using the results of an eigenfrequency study. For a more advanced micromirror model see the *Prestressed Micromirror Vibrations: Thermoviscous–Thermoelasticity Coupling* tutorial model also found in the Application Library.

Model Definition

A sketch of the idealized micromirror system modeled here is shown in [Figure 1](#). The mirror is made of silicon and is surrounded by air. It is 0.5 mm by 0.5 mm (only half is shown in the figure because of symmetry) and has a thickness of 1 μm .

The solid mirror is modeled using shell elements and the surrounding air is modeled using thermoviscous acoustics. A pressure acoustics layer is used to truncate the computational domain. The model combines the Shell interface, the Thermoviscous Acoustics, Frequency Domain interface, and the Pressure Acoustics, Frequency Domain interface. The three physics interfaces are set up and combined using multiphysics couplings: the Acoustic-Thermoviscous Acoustics Boundary and the Thermoviscous Acoustic-Structure Boundary.

One goal with the model is to get a correct assessment of the damping the mirror experiences. Therefore, the detailed thermoviscous acoustics is used to model the air domain surrounding the micromirror. The interface includes thermal and viscous damping explicitly as it solves the full linearized Navier–Stokes, continuity, and energy equations, implemented in the Thermoviscous Acoustics, Frequency Domain interface.

An important parameter in such models is the thickness of the viscous and thermal boundary layers (also called the penetration depths). It is in these layers that the energy is dissipated (viscous drag and thermal conduction). For air, the layers are of approximately the same size. The thickness of the viscous boundary layer δ_v is given by the expression

$$\delta_v = \sqrt{\frac{2\mu}{\omega_0 \rho_0}} = 0.22[\text{mm}] \sqrt{\frac{100[\text{Hz}]}{f_0}} \quad (1)$$

since $\delta_v = 0.22$ mm at 100 Hz. The boundary layer thickness can also be evaluated in postprocessing through the variables `tash.d_visc` and `tash.d_therm`.

The model is both solved using a frequency-domain sweep and using an eigenfrequency study. The frequency sweep results in a frequency response where the displacement (or velocity) is evaluated at the tip of the mirror (on the symmetry plane).

The response at the resonance yields the Q-factor Q_r and the resonance frequency f_r . Their expected values are computed based on the eigenfrequency analysis results. The Q-factor is defined as

$$Q = \frac{f_0}{\Delta f}$$

where f_0 is the resonance frequency and Δf is the peak width at half power (which corresponds to the 3 dB down width). The value of the band width is evaluated in the results using the **Graph Marker** functionality, see [Figure 7](#) (bottom). The Q-factor is related to the attenuation rate α through

$$\alpha = \frac{\omega_0}{2Q} \quad \omega_0 = 2\pi f_0$$

The result of the eigenfrequency study is a complex-valued eigenfrequency f_c which relates to the other parameters through

$$f_0 = \text{Re}(f_c) \quad \alpha = 2\pi \text{Im}(f_c) \quad (2)$$

Finally, the response of an underdamped harmonic oscillator can be written as

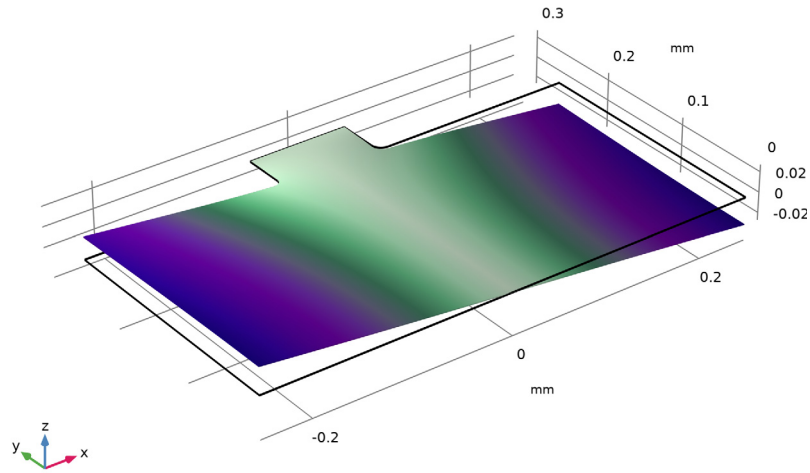
$$\frac{v}{\bar{F}} = \frac{\omega_0^2}{(i\omega)^2 + 2\alpha i\omega + \omega_0^2} \quad (3)$$

where \bar{F} is the actuating force and v is the velocity of the oscillator. This expression is used as a model equation for the frequency response in the vicinity of the resonance frequency, based on parameters identified in the eigenfrequency analysis (using the relations in [Equation 2](#)).

Results and Discussion

The first two modes from the eigenfrequency study are shown in [Figure 2](#). The torquing mode studied in the subsequent frequency domain analysis is the top figure where f_c is at about 10.5 kHz, while a symmetric vibrating mode is found at about 13 kHz. The next mode (not shown) is at nearly 40 kHz. The modal von Mises stress for the first mode is shown in [Figure 3](#). It is important to note the regions of high stress, these are also the regions where a finer mesh needs to be used to capture the full dynamics.

Eigenfrequency= $10547+145.43i$ Hz Surface: Displacement magnitude (mm)



Eigenfrequency= $13071+166.85i$ Hz Surface: Displacement magnitude (mm)

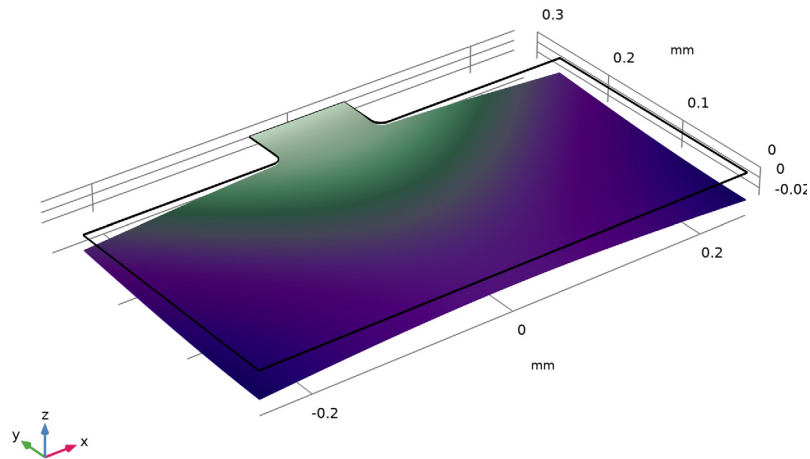


Figure 2: Displacement plot of the torquing (top) and symmetric (bottom) vibrating modes of the micromirror.

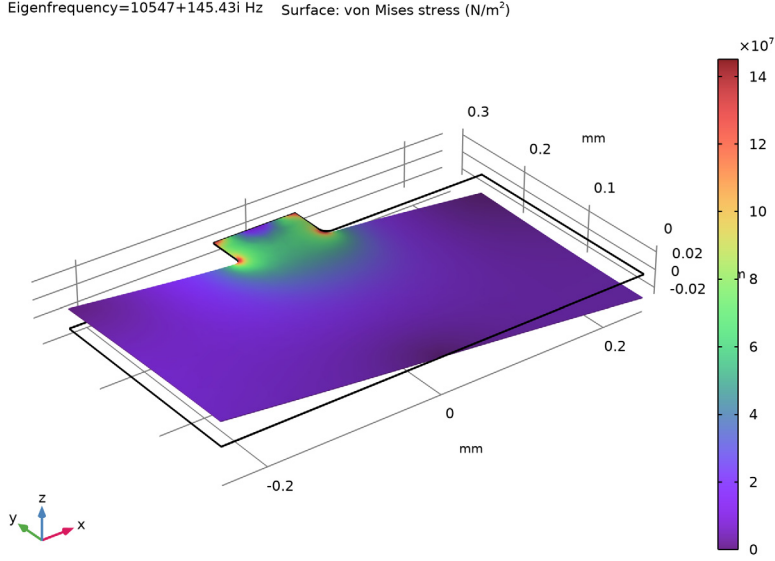


Figure 3: Von Mises stress distribution in the shell for first (torquing) mode.

The displacement of the micromirror with the torquing actuation is shown in [Figure 4](#) at a frequency of 10 kHz. The acoustic temperature variations and the acoustic pressure distribution also at 10 kHz is shown in [Figure 5](#). The instantaneous local velocity (acoustic variations in the velocity) is shown in [Figure 6](#). A high velocity region is seen near the edge of the micromirror. The extent of this region into the surrounding air is given by the scale of the viscous boundary layer (also known as the viscous penetration depth, [Equation 1](#)). The thermal boundary layer (thermal penetration depth) can in the same way be identified in [Figure 5](#) (top).

The displacement response of the system is plotted in [Figure 7](#) (top). This plot shows the absolute value of the z-component of the displacement field $|w_{\text{shell}}|$ evaluated at the tip of the micromirror. The bottom part of this figure shows the power response, on the dB scale (normalized by the maximum), by plotting

$$20\log(|i\omega w_{\text{shell}}|)$$

and comparing to the model expression given in [Equation 3](#). The parameters used in the analytical model expression stem from the eigenfrequency derived in the eigenfrequency

study. The value of the eigenfrequency f_c and the other parameters used in the analytical model are defined as variables (see the **Component 1>Definitions>Variables 1**). The values of the variables evaluated for the first eigenfrequency, are computed in the Evaluation Group in the model. Using the computed value leads to good agreement between the two power response curves, as seen in Figure 7 (bottom). The figure also shows the bandwidth of the frequency response, this value agrees well with the value extracted from the eigenfrequency analysis and evaluated in the Evaluation Group.

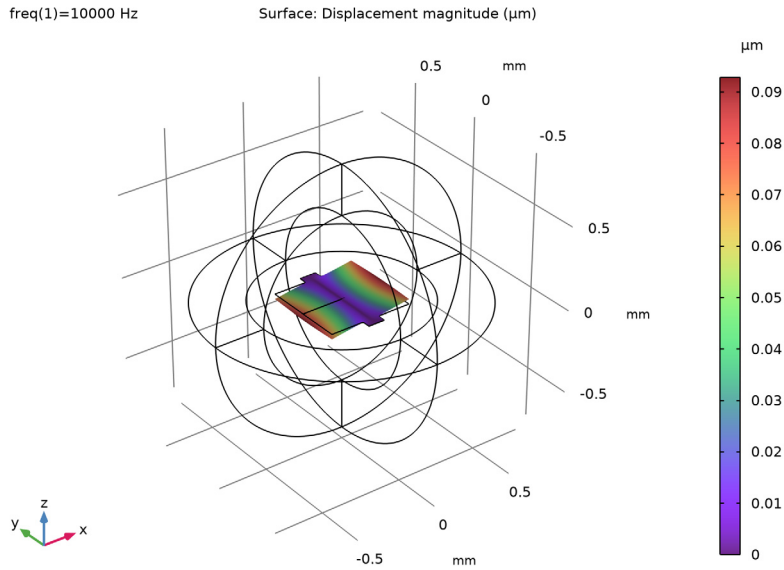


Figure 4: Displacement of the micromirror at 10 kHz subject to the torquing force.

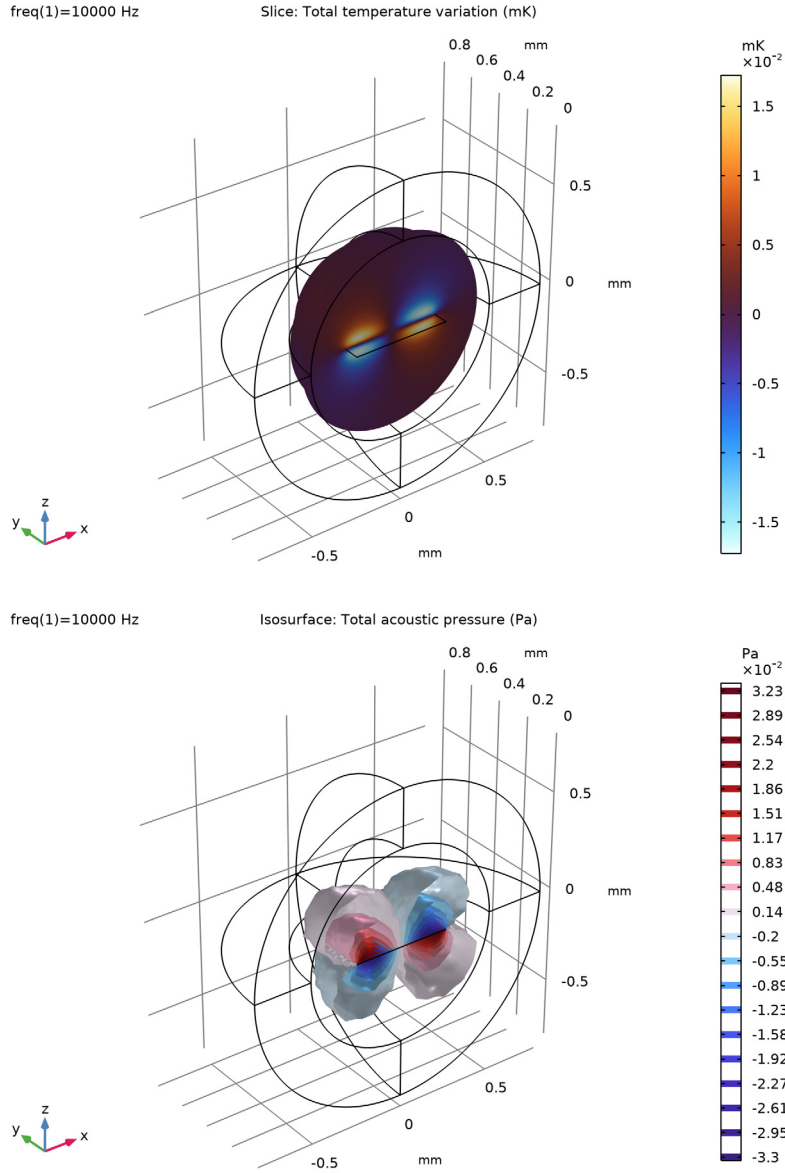


Figure 5: (top) Temperature field in the thermoviscous acoustics domain around the micromirror, and (bottom) pressure isosurfaces showing the antisymmetric pressure distribution.

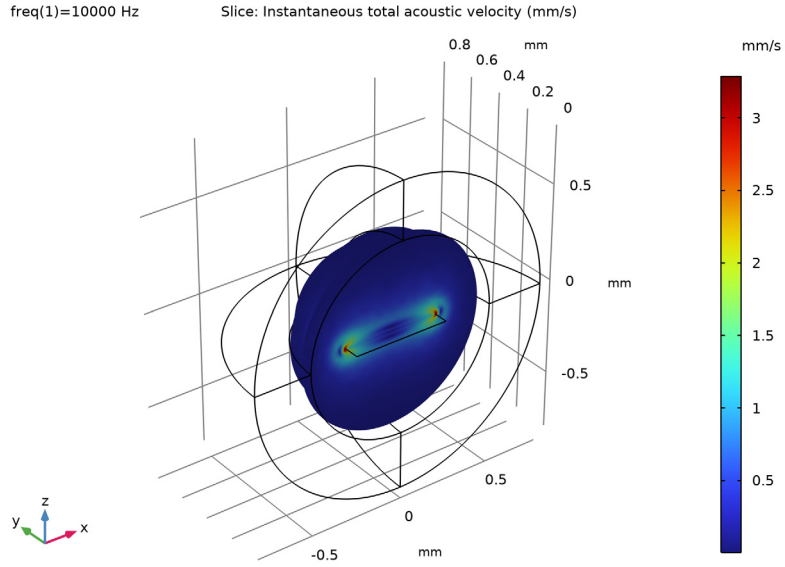


Figure 6: The instantaneous local velocity, with clear high velocity regions near the mirror edges. The extent of the high velocity region is given by the scale of the acoustic viscous boundary layer.

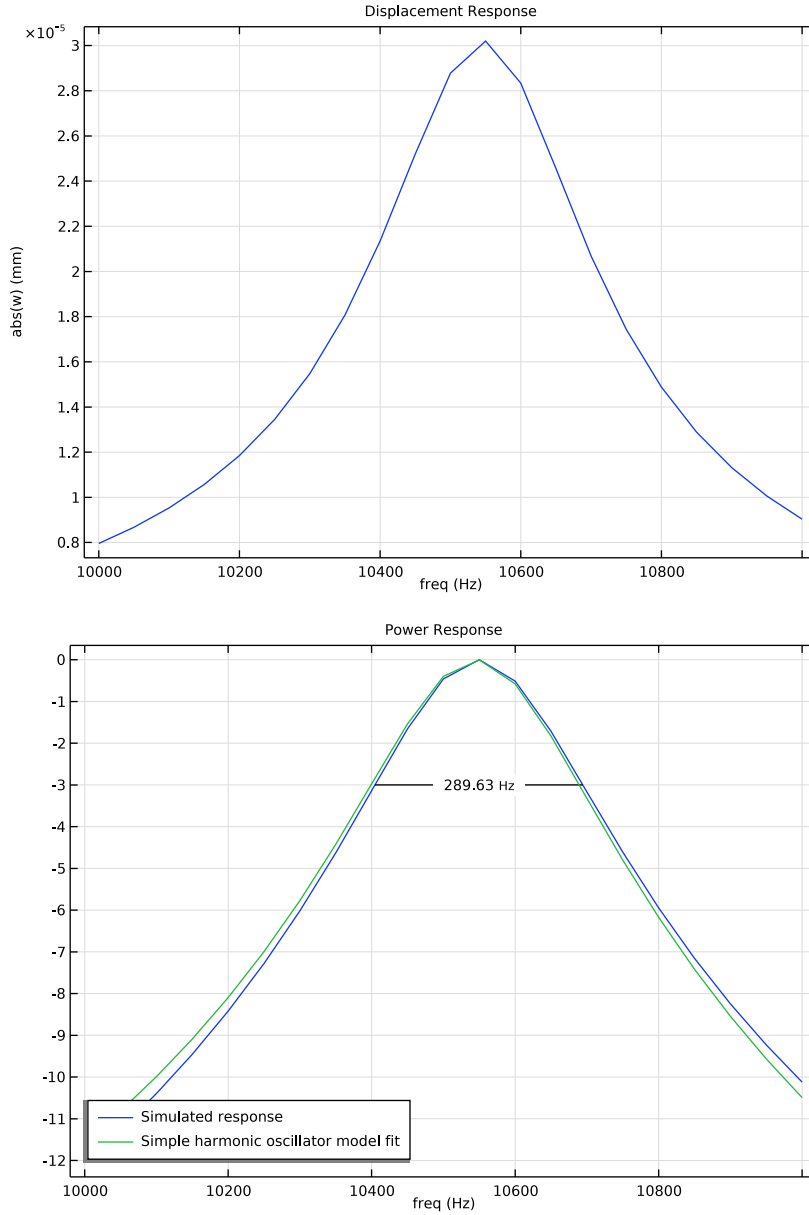



Figure 7: (top) Displacement response, and (bottom) power response with half-power bandwidth and analytical model curve.

Application Library path: Acoustics_Module/Vibrations_and_FSI/
vibrating_micromirror




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>Shell (shell)**.
- 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 **General Studies>Eigenfrequency**.
- 10 Click  **Done**.




GLOBAL DEFINITIONS

Parameters 1

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file vibrating_micromirror_parameters.txt.


GEOMETRY I

The geometry sequence for the model (see [Figure 1](#)) is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the [Appendix — Geometry Modeling Instructions](#) section. Otherwise, insert the geometry sequence as follows:


- 1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
- 2 Browse to the model's Application Libraries folder and double-click the file `vibrating_micromirror_geom_sequence.mph`.
- 3 In the **Geometry** toolbar, click  **Build All**.
- 4 Click the  **Go to Default View** button in the **Graphics** toolbar.
- 5 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.

DEFINITIONS


Variables I

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `vibrating_micromirror_variables.txt`.

Shell

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Shell in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 9 only.

Symmetry



- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Symmetry in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 1, 2, 6, 11, and 15 only.

Symmetry Edge

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Symmetry Edge in the **Label** text field.

- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.
- 4 Select Edge 9 only.

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>Air**.
- 4 Click **Add to Component** in the window toolbar.
- 5 In the tree, select **Built-in>Silicon**.
- 6 Click **Add to Component** in the window toolbar.
- 7 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

Silicon (mat2)

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Geometric entity level** list, choose **Boundary**.
- 3 From the **Selection** list, choose **Shell**.


SHELL (SHELL)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.
- 2 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Shell**.

Thickness and Offset 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.
- 2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
- 3 In the d_0 text field, type h_mirror.

Fixed Constraint 1


- 1 In the **Physics** toolbar, click  **Edges** and choose **Fixed Constraint**.
- 2 Select Edge 13 only.

Symmetry 1

- 1 In the **Physics** toolbar, click  **Edges** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Edge Selection** section.

- 3 From the **Selection** list, choose **Symmetry Edge**.


Body Load I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Body Load**.
- 2 In the **Settings** window for **Body Load**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Shell**.
- 4 Locate the **Force** section. Specify the \mathbf{F}_V vector as


0	x
0	y
$x / (0.25[\text{mm}]) * 1\text{e}5[\text{N}/\text{m}^3]$	z

This corresponds to a torquing force proportional to the x -coordinate acting in the z direction. A load of $0 \text{ N}/\text{m}^3$ act in the middle (at $x = 0$) rising to $1\text{e}5 \text{ N}/\text{m}^3$ at the mirror edge.


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  **Clear Selection**.
- 4 Select Domain 3 only.


Symmetry I

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.
- 2 Select Domains 1, 2, 4, and 5 only.
- 3 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 1 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 2 From the **Selection** list, choose **Symmetry**.


Spherical Wave Radiation I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Spherical Wave Radiation**.
- 2 Select Boundaries 4, 5, 12, and 17 only.


Proceed to set up the Multiphysics couplings that couple Thermoviscous Acoustics to Pressure Acoustics and couple the interior shell to the surrounding thermoviscous acoustics domain.

MULTIPHYSICS

Acoustic–Thermoviscous Acoustic Boundary I (atbI)


- 1 In the **Physics** toolbar, click  **Multiphysics Couplings** and choose **Boundary>Acoustic–Thermoviscous Acoustic Boundary**.
- 2 In the **Settings** window for **Acoustic–Thermoviscous Acoustic Boundary**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.
When you select **All boundaries** COMSOL automatically detects where a thermoviscous acoustics domain and a pressure acoustics domain meet and applies the coupling there.

Thermoviscous Acoustic–Structure Boundary I (tsbI)

- 1 In the **Physics** toolbar, click  **Multiphysics Couplings** and choose **Boundary>Thermoviscous Acoustic–Structure Boundary**.
- 2 In the **Settings** window for **Thermoviscous Acoustic–Structure Boundary**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Shell**.
Selecting **All boundaries** here will result in the same setup as using the Shell selection.

MESH I

Free Triangular I

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Shell**.

Size I

- 1 Right-click **Free Triangular I** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 Click the **Custom** button.

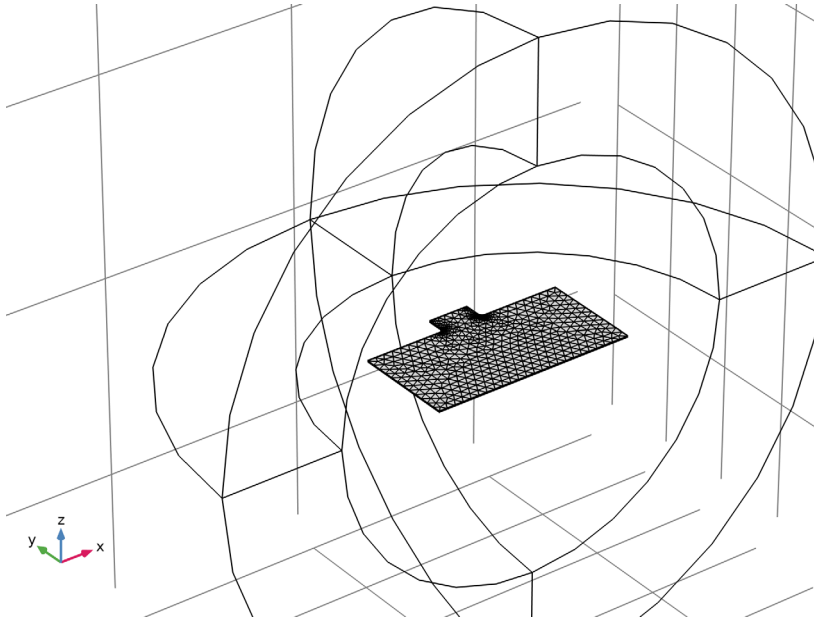
- 4 Locate the **Element Size Parameters** section.
- 5 Select the **Maximum element size** check box. In the associated text field, type dvisc .
To get a fully resolved acoustic boundary layer, it is also necessary to add a boundary layer mesh. Do this as the last mesh step.
- 6 Select the **Minimum element size** check box. In the associated text field, type $0.1 * \text{dvisc}$.
- 7 Select the **Maximum element growth rate** check box. In the associated text field, type 1.4.
- 8 Select the **Curvature factor** check box. In the associated text field, type 0.4.

Size 2

- 1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Point**.
- 4 Select Points 7 and 15 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element size** check box. In the associated text field, type $0.2 * \text{dvisc}$.

8 Click  **Build Selected**.

The surface mesh of the mirror should look like this. Notice the higher mesh resolution regions. As seen in results, these are the regions with higher stress.



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 $343[m/s] / f_0/6$.
- 5 In the **Resolution of narrow regions** text field, type 2.

Free Tetrahedral 1


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

Boundary Layers 1

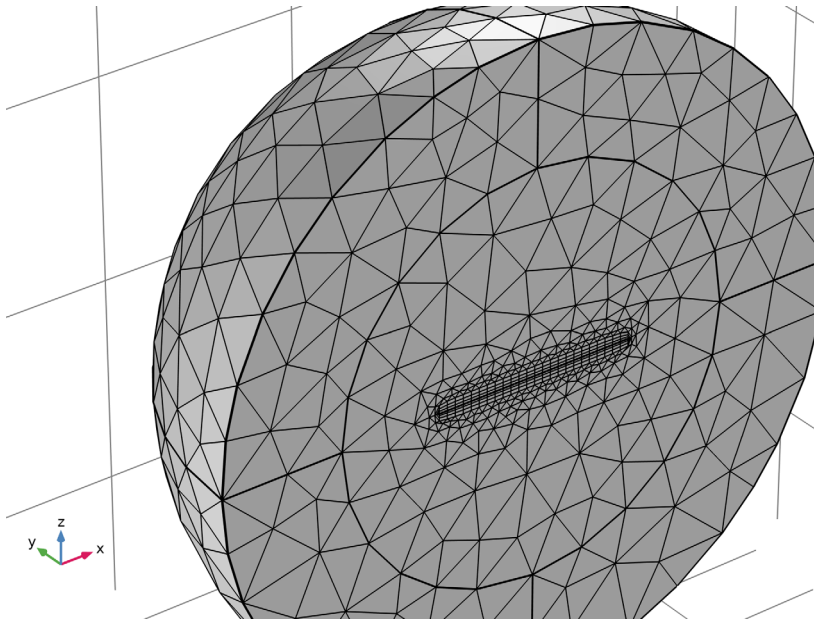
In the **Mesh** toolbar, click  **Boundary Layers**.

Boundary Layer Properties

- 1 In the **Model Builder** window, click **Boundary Layer Properties**.

- 2 In the **Settings** window for **Boundary Layer Properties**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Shell**.
- 4 Locate the **Layers** section. In the **Number of layers** text field, type 3.
- 5 From the **Thickness specification** list, choose **First layer**.
- 6 In the **Thickness** text field, type $0.3 \cdot d_{\text{visc}}$.
- 7 Click  **Build All**.

The geometry with mesh should look like the figure below.



Proceed with the eigenfrequency study. The determined eigenfrequency including the complex part yields information about the attenuation in the system.

STUDY I

Step 1: Eigenfrequency

- 1 In the **Model Builder** window, under **Study I** click **Step 1: Eigenfrequency**.
- 2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
- 3 In the **Search for eigenfrequencies around shift** text field, type 10000.
- 4 From the **Search method around shift** list, choose **Larger real part**.



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

These selections will search for the first three eigenmodes above 10 kHz. It is known that the first eigenfrequency is at about 10500 Hz (this can also be determined using a frequency study looking for the response peaks). The eigenfrequency search condition will make the solution converge faster. Selecting the **Closest in absolute value** will search for modes with both smaller and larger eigenfrequencies.



Again, turn off the generation of the default plots and generate and show the default solver to take a look at the solver suggestions.

- 6 In the **Model Builder** window, click **Study 1**.
- 7 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 8 Clear the **Generate default plots** check box.

Solution 1 (sol1)


- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
Select and enable the first suggested iterative solver. In the eigenfrequency study for this size of problem, the direct solver is faster but it is more memory consuming.
- 3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Eigenvalue Solver 1** node.
- 4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Eigenvalue Solver 1>Suggested Iterative Solver (GMRES with Direct Precon.) (tsb1_atb1)** and choose **Enable**.
- 5 In the **Study** toolbar, click  **Compute**.

ADD PREDEFINED PLOT

- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.
- 3 In the tree, select **Study 1/Solution 1 (sol1)>Shell>Mode Shape (shell)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Mode Shape (shell)

- 1 Click the  **Zoom Extents** button in the **Graphics** toolbar.


The default (first) selected eigenvalue is the vibrating torque mode at about 10,500 Hz. This is the vibration mode that will be studied next in the frequency domain study.

The first mode should look like the one in [Figure 2](#) top.

The next eigenmode represents a symmetric vibration in the transverse direction happening at about 13100 Hz.

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


- 3 From the **Eigenfrequency (Hz)** list, choose **13071+166.83i**.

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

The second mode should look like the one in [Figure 2](#) bottom.

The next mode happens at nearly 40 kHz and can be seen by selecting the last of the three computed eigenfrequencies.

ADD PREDEFINED PLOT

- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.

- 2 Go to the **Add Predefined Plot** window.

- 3 In the tree, select **Study 1/Solution 1 (sol1)>Shell>Modal Stress (shell)**.

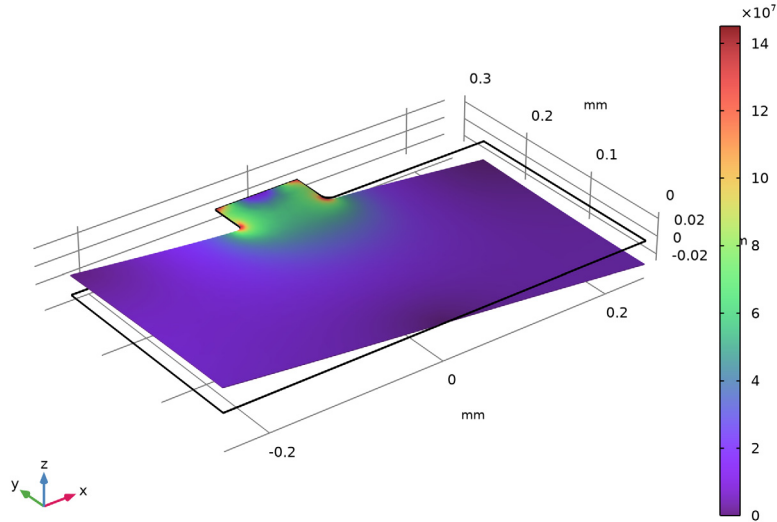
- 4 Click **Add Plot** in the window toolbar.

- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS



Modal Stress (shell)

Eigenfrequency=10547+145.43i Hz Surface: von Mises stress (N/m²)



This second plot shows the von Mises stress distribution in the shell. To further improve the precision of the computed eigenfrequency, the mesh can be refined in the high-stress regions. Note that refining the mesh will rapidly lead to an increased computational time and memory consumption when solving.

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 **General Studies>Frequency Domain**.
- 4 Click **Add Study** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.



STUDY 2

Step 1: Frequency Domain

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

- 2 In the **Frequencies** text field, type `range(f0-deltaf,50,f0+deltaf)`.
Turn off the generation of default plots, only create the ones you need or use the predefined plots. Plot the acoustic variations in temperature, pressure, and velocity as well as response curves. If turned on, the default plots for each physics interface will be generated.
- 3 In the **Model Builder** window, click **Study 2**.
- 4 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 5 Clear the **Generate default plots** check box.
Now, generate and show the default solver to take a look at the solver suggestions automatically generated.

Solution 2 (sol2)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
In this model, which couples pressure acoustics, thermoviscous acoustics, and shell, the default solver is a direct solver. Select and enable the first iterative suggestion. It is faster and more memory efficient than the direct solver.
- 3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.
- 4 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Suggested Iterative Solver (GMRES with Direct Precon.) (tsbl_atb1)** and choose **Enable**.
- 5 In the **Study** toolbar, click  **Compute**.


RESULTS

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

Mirror 3D 1


- 1 In the **Model Builder** window, expand the **Results>Datasets** node.
- 2 Right-click **Results>Datasets** and choose **More 3D Datasets>Mirror 3D**.
- 3 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.
- 4 From the **Plane** list, choose **XZ-planes**.
- 5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Displacement

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Displacement in the **Label** text field.

- 3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D I**.


Surface I

- 1 Right-click **Displacement** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose μm .
- 4 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 5 In the **Color Table** dialog box, select **Rainbow>SpectrumLight** in the tree.
- 6 Click **OK**.

Deformation I


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

Displacement


- 1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 2 From the **Parameter value (freq (Hz))** list, choose **10000**.
- 3 Locate the **Color Legend** section. Select the **Show units** check box.
- 4 In the **Displacement** toolbar, click  **Plot**.


The plot should look like the one in [Figure 4](#).

Temperature Variation

- 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 Variation** in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 4 Locate the **Color Legend** section. Select the **Show units** check box.


Slice I

- 1 Right-click **Temperature Variation** and choose **Slice**.
- 2 In the **Settings** window for **Slice**, locate the **Expression** section.
- 3 In the **Expression** text field, type $t_a.T_t$.
- 4 From the **Unit** list, choose **mK**.
- 5 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.
- 6 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 7 In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.
- 8 Click **OK**.



- 9 In the **Settings** window for **Slice**, locate the **Coloring and Style** section.
- 10 From the **Scale** list, choose **Linear symmetric**.
- 11 In the **Temperature Variation** toolbar, click  **Plot**.

The plot should look like the one in [Figure 5](#) top.

Acoustic Pressure

- 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** in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 4 Locate the **Color Legend** section. Select the **Show units** check box.

Isosurface 1

- 1 Right-click **Acoustic Pressure** and choose **Isosurface**.
- 2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Multiphysics>atb1.p_t - Total acoustic pressure - Pa**.
- 3 Locate the **Levels** section. In the **Total levels** text field, type 20.
- 4 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 5 In the **Color Table** dialog box, select **Wave>Wave** in the tree.
- 6 Click **OK**.
- 7 In the **Settings** window for **Isosurface**, locate the **Coloring and Style** section.
- 8 From the **Scale** list, choose **Linear symmetric**.
- 9 In the **Acoustic Pressure** toolbar, click  **Plot**.

The plot should look like the one in [Figure 5](#) bottom.



Temperature Variation

In the **Model Builder** window, under **Results** right-click **Temperature Variation** and choose **Duplicate**.

Acoustic Velocity

- 1 In the **Model Builder** window, under **Results** click **Temperature Variation 1**.
- 2 In the **Settings** window for **3D Plot Group**, type **Acoustic Velocity** in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 4 Locate the **Color Legend** section. Select the **Show units** check box.


Slice 1

- 1 In the **Model Builder** window, expand the **Acoustic Velocity** node, then click **Slice 1**.
- 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 **Coloring and Style** section. Click  **Change Color Table**.
- 6 In the **Color Table** dialog box, select **Rainbow>Rainbow** 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**.
- 10 In the **Acoustic Velocity** toolbar, click  **Plot**.


The plot should look like the one in [Figure 6](#).

Proceed to setting up two 1D curves that represent the frequency response of the system by probing the displacement at the symmetry plane of the micromirror. The first plot gives the absolute value of the displacement. The second plot depicts the power (proportional to the velocity squared) on the dB scale and compared to a simple harmonic oscillator model tuned using parameters extracted from an eigenfrequency study of the system (instructions follow below). The parameters are given in the parameters list. Both curves are normalized by the maximum value.

Displacement Response


- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **1D Plot Group**.
- 2 In the **Settings** window for **1D Plot Group**, type **Displacement Response** in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Point Graph 1

- 1 Right-click **Displacement Response** and choose **Point Graph**.
- 2 Select Point 16 only.
- 3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type `abs(w)`.
- 5 In the **Displacement Response** toolbar, click  **Plot**.

The plot should look like the one in [Figure 7](#) top. Proceed and create a plot to compare the simple resonator expression with the actual response of the system.

Power Response

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Power Response** in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.
- 5 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Point Graph I

- 1 Right-click **Power Response** and choose **Point Graph**.
- 2 Select Point 16 only.
- 3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type $20 \cdot \log_{10}(\text{abs}(w \cdot t_a \cdot i\omega)) - \text{with}(12, 20 \cdot \log_{10}(\text{abs}(w \cdot t_a \cdot i\omega)))$.
- 5 Click to expand the **Legends** section. Select the **Show legends** check box.
- 6 From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:

Legends
Simulated response


Global I

- 1 In the **Model Builder** window, right-click **Power Response** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Description
P_simple	Simple harmonic oscillator model fit


Graph Marker I

- 1 In the **Model Builder** window, right-click **Point Graph I** and choose **Graph Marker**.
- 2 In the **Settings** window for **Graph Marker**, locate the **Display** section.
- 3 From the **Display mode** list, choose **Bandwidth**.
- 4 From the **Cutoff mode** list, choose **Offset from peak**.
- 5 Locate the **Text Format** section. In the **Display precision** text field, type 5.
- 6 Select the **Include unit** check box.

7 In the **Power Response** toolbar, click  **Plot**.

The plot should look like the one in [Figure 7](#) bottom. Finally, add an evaluation group to evaluate the harmonic oscillator parameters used in the simple analytical model.

Evaluation Group 1 - Harmonic Oscillator Parameters

- 1 In the **Results** toolbar, click  **Evaluation Group**.
- 2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.
- 3 From the **Eigenfrequency selection** list, choose **First**.
- 4 In the **Label** text field, type Evaluation Group 1 - Harmonic Oscillator Parameters.
- 5 Locate the **Transformation** section. Select the **Transpose** check box.

Global Evaluation 1

- 1 Right-click **Evaluation Group 1 - Harmonic Oscillator Parameters** and choose **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
f_r	Hz	Resonance frequency
omega_r	Hz	Resonance angular frequency
alpha_r	Hz	Attenuation coefficient
Q_r	1	Q factor
df_r	Hz	Resonance half power width

- 4 In the **Evaluation Group 1 - Harmonic Oscillator Parameters** toolbar, click  **Evaluate**.

Appendix — Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

GEOMETRY 1

- 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)


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

2 In the **Settings** window for **Work Plane**, click  **Go to Plane Geometry**.

Work Plane 1 (wp1)>Plane Geometry

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

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 0.5.

4 In the **Height** text field, type 0.5.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Work Plane 1 (wp1)>Rectangle 2 (r2)

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 0.1.

4 In the **Height** text field, type 0.05.

5 Locate the **Position** section. In the **xw** text field, type -0.05.

6 In the **yw** text field, type 0.25.

7 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.


Work Plane 1 (wp1)>Rectangle 3 (r3)

1 In the **Model Builder** window, click **Rectangle 3 (r3)**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **yw** text field, type -0.3.

Work Plane 1 (wp1)>Rectangle 4 (r4)

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 0.5.


4 In the **Height** text field, type 0.6.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 Right-click **Rectangle 4 (r4)** and choose **Build All Objects**.

Work Plane 1 (wp1)>Union 1 (uni1)

1 In the **Work Plane** toolbar, click  **Booleans and Partitions** and choose **Union**.

2 Click the  **Select All** button in the **Graphics** toolbar.

3 In the **Settings** window for **Union**, click  **Build Selected**.


Work Plane 1 (wp1)>Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Plane Geometry** and choose **Delete Entities**.

2 On the object **unil**, select Boundaries 1, 2, 5, 7, 10, 12, 15, 19, 20, and 22 only.

3 In the **Settings** window for **Delete Entities**, click  **Build Selected**.

Work Plane 1 (wp1)>Fillet 1 (fil1)

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

2 On the object **del1**, select Points 4, 5, 8, and 9 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 0.01.

5 Click  **Build Selected**.

Sphere 1 (sph1)

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

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

3 In the **Radius** text field, type 0.8.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.3

Union 1 (unil)

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

To better see the entire geometry use the **Zoom Extents** tool.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Union**, click  **Build Selected**.

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 2.

4 In the **Depth** text field, type 2.

5 In the **Height** text field, type 2.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **y** text field, type -1.

8 Click  **Build Selected**.

Difference I (difI)

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

2 Select the object **uniI** only.

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

4 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **blkI** only.

6 Click  **Build All Objects**.