

# 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 F, and the symmetry plane.

The dynamic vibrating behavior of the micromirror 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  $\mu$ m.

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  $\delta_v$  is given by the expression

$$\delta_{\rm v} = \sqrt{\frac{2\mu}{\omega_0 \rho_0}} = 0.22 [\rm mm] \sqrt{\frac{100[\rm Hz]}{f_0}} \tag{1}$$

since  $\delta_v = 0.22 \text{ mm}$  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 given in the parameters list. The Q-factor is defined as

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

where  $f_0$  is the resonance frequency and  $\Delta f$  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 5 (bottom). The Q-factor is related to the attenuation rate  $\alpha$  through

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

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

$$f_0 = \operatorname{Re}(f_c) \qquad \alpha = 2\pi \operatorname{Im}(f_c) \tag{2}$$

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

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

where 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 displacement of the micromirror with the torquing actuation is shown in Figure 2 at a frequency of 10 kHz. The acoustic temperature variations and the acoustic pressure distribution at 11 kHz is shown in Figure 3. The instantaneous local velocity (acoustic variations in the velocity) is shown in Figure 4. 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 3 (top).

The displacement response of the system is plotted in Figure 5 (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  $f_c$  used in the analytical model curve is stored in the parameter f\_num (see the parameters list **Global Definitions>Parameters**). Using the computed value leads to good agreement between the two.

Some of the resulting modes from the eigenfrequency study are shown in Figure 6. The torquing mode studied in the frequency sweep is the top figure where  $f_c$  is about 10.6 kHz, while a symmetric vibrating mode is found at about 13 kHz.



Figure 2: Displacement of the micromirror at 10 kHz subject to the torquing force.



Figure 3: (top) Temperature field in the thermoviscous acoustics domain around the micromirror, and (bottom) pressure isosurfaces showing the antisymmetric pressure distribution.



Figure 4: 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 5: (top) Displacement response, and (bottom) power response with half-power bandwidth and analytical model curve.



Eigenfrequency=10551+148.49i Hz Surface: Displacement magnitude (µm)

Eigenfrequency=13090+172.96i Hz Surface: Displacement magnitude (µm)



Figure 6: Torquing (top) and symmetric (bottom) vibrating modes of the micromirror.

# 9 | VIBRATING MICROMIRROR WITH VISCOUS AND THERMAL DAMPING

**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

- I In the Model Wizard window, click 间 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>Frequency Domain.
- IO Click 🗹 Done.

# GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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:

- I 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 **J Go to Default View** button in the **Graphics** toolbar.
- 5 Click the 🔁 Wireframe Rendering button in the Graphics toolbar.

#### DEFINITIONS

#### Shell

- I 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 Boundaries 9 and 11 only.

#### Symmetry

- I In the **Definitions** toolbar, click **herefore 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, 13, and 17 only.

# Symmetry Edge

- I In the **Definitions** toolbar, click **here 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

- I 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)

- I 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)

- I In the Model Builder window, under Component I (compl) 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 I

- I In the Model Builder window, under Component I (compl)>Shell (shell) click Thickness and Offset I.
- 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 I

- I In the Physics toolbar, click 🔚 Edges and choose Fixed Constraint.
- 2 Select Edge 15 only.

#### Symmetry 1

- I 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

- I 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  $\mathbf{F}_{\mathbf{V}}$  vector as

0	x
0	у
x/(0.25[mm])*1e5[N/m^3]	z

This corresponds to a torquing force proportional to the *x*-coordinate acting in the *z* direction. A load of  $0 \text{ N/m}^3$  act in the middle (at x = 0) rising to 1e5 N/m<sup>3</sup> at the mirror edge.

#### THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

- I In the Model Builder window, under Component I (compl) 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

- I 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)

- I In the Model Builder window, under Component I (compl) 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.

#### Symmetry I

- I In the Settings window for Symmetry, locate the Boundary Selection section.
- 2 From the Selection list, choose Symmetry.

#### Spherical Wave Radiation 1

- I In the Physics toolbar, click 🔚 Boundaries and choose Spherical Wave Radiation.
- 2 Select Boundaries 4, 5, 14, and 19 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 1 (atb1)

- I In the Physics toolbar, click Automatic 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 1 (tsb1)

- I In the Physics toolbar, click An 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 1

- I In the Mesh toolbar, click  $\bigwedge$  Boundary 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

- I 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 1.5\* 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 dvisc/10.

# Size

- I In the Model Builder window, under Component I (compl)>Mesh I 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]/f0/6.
- 5 In the Resolution of narrow regions text field, type 2.

# Free Tetrahedral I

In the Mesh toolbar, click \land Free Tetrahedral.

Boundary Layers 1

- I In the Mesh toolbar, click Boundary Layers.
- 2 In the Settings window for Boundary Layers, click to expand the Transition section.
- **3** Clear the **Smooth transition to interior mesh** check box.

#### Boundary Layer Properties

- I 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\*dvisc.

# 7 Click 📗 Build All.

The geometry with mesh should look like the figure below.



# STUDY I

#### Step 1: Frequency Domain

- I In the Model Builder window, under Study I click Step I: Frequency Domain.
- 2 In the Settings window for Frequency Domain, locate the Study Settings section.
- 3 In the Frequencies text field, type range(f0-deltaf, 50, f0+deltaf).

In the following instructions an iterative solver is selected. In the frequency domain the optimal performance is achieved when the **Reuse solution from previous step** is set to **No**, as follows.

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

Turn off the generation of default plots and only create the ones you need. 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.

- 5 In the Model Builder window, click Study I.
- 6 In the Settings window for Study, locate the Study Settings section.

7 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 1 (soll)

- I In the Study toolbar, click **The Show Default Solver**.
- 2 In the Model Builder window, expand the Solution I (soll) 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 I>Solver Configurations> Solution I (soll)>Stationary Solver I node.
- 4 Right-click Study I>Solver Configurations>Solution I (soll)>Stationary Solver I> Suggested Iterative Solver (GMRES with Direct Precon.) (tsbl\_atbl) and choose Enable.
- **5** In the **Study** toolbar, click **= Compute**.

# RESULTS

In the Model Builder window, expand the Results node.

## Mirror 3D I

- I 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.

#### Displacement

- I In the Results toolbar, click 间 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 1

- I 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

- I 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 **I** Plot.

The plot should look like the one in Figure 2.

#### Temperature

- I In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, type Temperature in the Label text field.
- **3** Locate the **Color Legend** section. Select the **Show units** check box.

#### Slice 1

- I Right-click Temperature 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. 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.
- **IO** From the Scale list, choose Linear symmetric.
- II In the **Temperature** toolbar, click **I** Plot.

The plot should look like the one in Figure 3 top.

# Pressure

- I In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, type Pressure in the Label text field.
- **3** Locate the **Color Legend** section. Select the **Show units** check box.

#### Isosurface 1

I Right-click 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 I (compl)>Multiphysics> atbl.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 **Pressure** toolbar, click **I** Plot.

The plot should look like the one in Figure 3 bottom.

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.

# Velocity

- I In the Model Builder window, right-click Temperature and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Velocity in the Label text field.
- 3 Locate the Color Legend section. Select the Show units check box.

# Slice 1

- I In the Model Builder window, expand the Velocity node, then click Slice I.
- 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.

# **IO** In the **Velocity** toolbar, click **O Plot**.

The plot should look like the one in Figure 4.

#### Displacement Response

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Displacement Response in the Label text field.
- 3 Click to expand the Title section. From the Title type list, choose Label.

## Point Graph 1

- I 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 5 top.

#### Power Response

- I 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** Click to expand the **Title** section. From the **Title type** list, choose **Labe**l.
- 4 Locate the Legend section. From the Position list, choose Lower left.

#### Point Graph 1

- I 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\*log10(abs(w\*ta.iomega))-with(12,20\*log10(abs(w\*ta.iomega))).
- 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

- I 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
20*log10(abs(omega_r^2/((ta.iomega)^2+2*alpha_num* ta.iomega+omega_r^2)))-20*log10(abs(omega_r/(2*i*	Simple harmonic oscillator model
<pre>ta.iomega+omega_r^2)))-20*log10(abs(omega_r/(2*i* alpha_num)))</pre>	oscillator mode fit

Graph Marker I

- I 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 Select the **Relative to peak** check box.
- 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 **O** Plot.

The plot should look like the one in Figure 5 bottom.

Proceed to set up an eigenfrequency study. The eigenfrequency including the complex part yields information about the attenuation in the system. The value of f\_num as given in the **Definitions>Parameters** list corresponds to a finer mesh than the default used here.

# ADD STUDY

- I In the Home toolbar, click  $\sim\sim$  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> Eigenfrequency.
- 4 Click Add Study in the window toolbar.
- 5 In the Home toolbar, click  $\stackrel{\text{res}}{\longrightarrow}$  Add Study to close the Add Study window.

#### STUDY 2

Step 1: Eigenfrequency

- I In the Settings window for Eigenfrequency, locate the Study Settings section.
- 2 In the Search for eigenfrequencies around text field, type 10000.

- 3 From the Eigenfrequency search method around shift list, choose Larger real part.
- 4 Select the Desired number of eigenfrequencies check box. In the associated text field, type3.

These selections will search for the first three eigenmodes above 10 kHz. As we know from the frequency domain analysis, the first eigenfrequency is at about 10500 Hz. The eigenfrequency search condition will make the solution converge faster. Selecting the **Closest in absolute value** will search for modes with both lower 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.

- 5 In the Model Builder window, click Study 2.
- 6 In the Settings window for Study, locate the Study Settings section.
- 7 Clear the Generate default plots check box.

#### Solution 2 (sol2)

- I In the Study toolbar, click The Show Default Solver.
- 2 In the Model Builder window, expand the Solution 2 (sol2) node.

As in the previous study, 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 2>Solver Configurations> Solution 2 (sol2)>Eigenvalue Solver I node.
- 4 Right-click Study 2>Solver Configurations>Solution 2 (sol2)>Eigenvalue Solver I> Suggested Iterative Solver (GMRES with Direct Precon.) (tsb1\_atb1) and choose Enable.
- **5** In the **Study** toolbar, click **= Compute**.

# RESULTS

Displacement Modes

- I In the Model Builder window, right-click Displacement and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Displacement Modes in the Label text field.
- 3 Locate the Color Legend section. Clear the Show units check box.
- 4 Locate the Data section. From the Dataset list, choose Study 2/Solution 2 (sol2).

**5** In the **Displacement Modes** toolbar, click **O Plot**.

The exact value of the calculated eigenfrequency converges for a well resolved mesh (resolving the acoustic boundary layers). The value of the first computed eigenfrequency is entered as the value for f\_num, found in the **Global Definitions>Parameters** list.

The default (first) selected eigenvalue is the vibrating torque mode at about 10500 Hz that was studied in the frequency domain study.

The first mode should look like the one in Figure 6 top.

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

- 6 In the Model Builder window, click Displacement Modes.
- 7 From the Eigenfrequency (Hz) list, choose 13090+172.96i.
- 8 In the Displacement Modes toolbar, click **I** Plot.

The second mode should look like the one in Figure 6 bottom.

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

# Appendix — Geometry Modeling Instructions

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

# GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

# Work Plane I (wp1)

- I In the Geometry toolbar, click 📥 Work Plane.
- 2 In the Settings window for Work Plane, click 📥 Show Work Plane.

# Work Plane I (wpI)>Plane Geometry

In the Model Builder window, click Plane Geometry.

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

- I 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 I (wp1)>Rectangle 2 (r2)

- I 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.

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

- I Right-click Component I (comp1)>Geometry I>Work Plane I (wp1)>Plane Geometry> Rectangle 2 (r2) and choose Duplicate.
- 2 In the Settings window for Rectangle, locate the Position section.
- **3** In the **yw** text field, type -0.3.

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

- I 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.

Sphere I (sphI)

- I In the Model Builder window, right-click Geometry I 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 I (uniI)

I In the Geometry toolbar, click i 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 틤 Build Selected.

#### Block I (blkI)

- I 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 (dif1)

- I In the Geometry toolbar, click i Booleans and Partitions and choose Difference.
- 2 Select the object unil only.
- 3 In the Settings window for Difference, locate the Difference section.
- **4** Find the **Objects to subtract** subsection. Click to select the **Selection** toggle button.
- 5 Select the object **blk1** only.
- 6 Click 🟢 Build All Objects.

# 26 | VIBRATING MICROMIRROR WITH VISCOUS AND THERMAL DAMPING