Vibrating Micromirror with Viscous and Thermal Damping

Introduction

Micromirror systems are part of small MEMS (micro-electro-mechanical systems) systems for integrated scanning micromirror systems in telecommunications and consumer applications. The mirrors are low power consuming and are low cost to manufacture. Their application range from microscopy applications, medical imaging, scanners, and heads up displays, to some fiber optics applications.

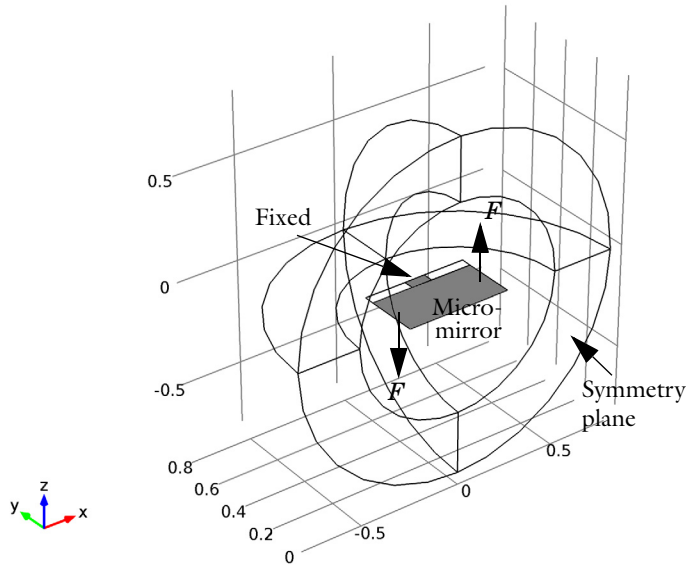


Figure 1: Geometry of the micromirror system including the fixed constraint, the torquing force components F, and the symmetry plane.

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 predict the performance of the system, can save both time and money.

Model Definition

A sketch of the idealized micromirror system modeled here is depicted 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 the thermoacoustics. The combined multiphysics interface Thermoacoustic-Shell Interaction, Frequency Domain is used to set up the model.

One of the goals with the model is to get a corrector assessment of the damping the mirror experiences. Therefore, the detailed thermoacoustics 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.

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

$$\delta_v = \sqrt{\frac{2\mu}{\omega\rho_0}} \tag{1}$$

At 100 Hz this is 0.22 mm; hence, the definition of the parameter `dvisc` given as `0.22[mm]*sqrt(100[Hz]/f0)`. The boundary layer thickness may also be evaluated in post processing 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, both given in the parameters list (the expected values). The Q-factor is defined as

$$Q = \frac{f_0}{\Delta f} \tag{2}$$

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 Q-factor is related to the attenuation rate α through

 ©2013 COMSOL

$$\alpha = \frac{\omega_0}{2Q} \qquad \omega_0 = 2\pi f_0 \tag{3}$$

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

$$f_0 = \mathrm{Re}(f_c) \qquad \alpha = 2\pi \mathrm{Im}(f_c) \tag{4}$$

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

$$\frac{v}{F} = \frac{\omega_0^2}{(i\omega)^2 + 2\alpha i\omega + \omega_0^2} \tag{5}$$

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 4).

Results and Discussion

The displacement of the micromirror with the torquing actuation is depicted in Figure 2 at a frequency of 10 kHz. From the figure it is evident to see where the micromirror is fixed at the two side arms. The acoustic temperature variations and the acoustic pressure distribution at 11 kHz is depicted in Figure 3.

The displacement response of the system is plotted in Figure 4 (top). This plot shows the absolute value of the z-component of the displacement field $|w_{\mathrm{shell}}|$ (plotting `abs(z_shell)`) evaluated at the tip of the micromirror. The bottom part of this figure shows the (scaled) power response, on the dB scale, by plotting

$$20\log(|i\omega w_{\mathrm{shell}}|) \tag{6}$$

and comparing to the model expression given in Equation 5. A manually set scaling factor, accounting for the magnitude of the actuation force, is subtracted to give the same magnitudes. The parameters used in the model expression stem from the eigenfrequency derived in the eigenfrequency study. Note that the two curves do not match perfectly. This is because the value of the eigenfrequency f_c used in the model curve (`f_num` from the parameters list **Definitions>Parameters**), was derived using a finer mesh than in the default setup of this model. The size of the mesh in the model is set to `1.2*dvisc` which is not fine enough - a value of `0.5*dvisc` is more suited in

order to well resolve the acoustic boundary layer. The coarse mesh is selected here to reduce the required memory, to solve, to about 4 GB of RAM.

Some of the resulting modes from the eigenfrequency study are depicted in Figure 5. The torquing mode studied in the frequency sweep is the top figure where f_c = 10560+145i, while a symmetric vibrating mode is found at about 13 kHz.

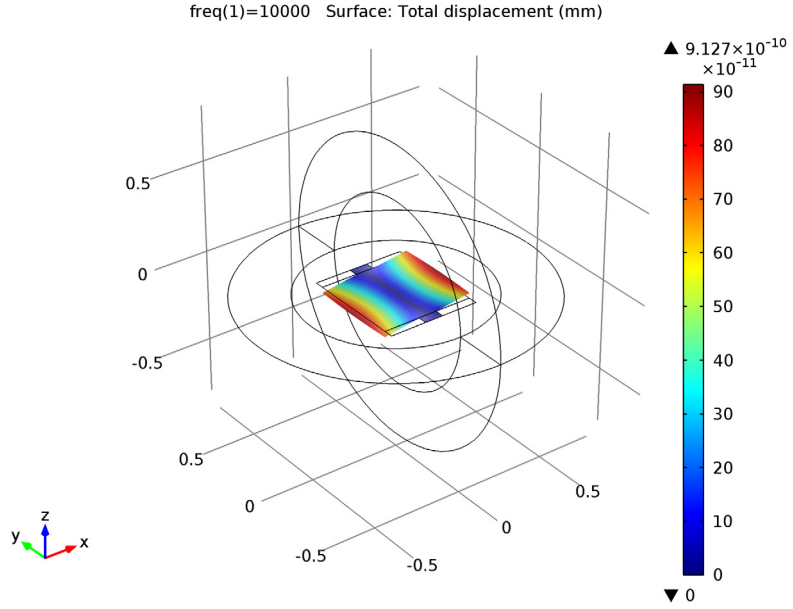


Figure 2: Displacement of the micromirror at 10 kHz subject to the torquing force.

 ©2013 COMSOL

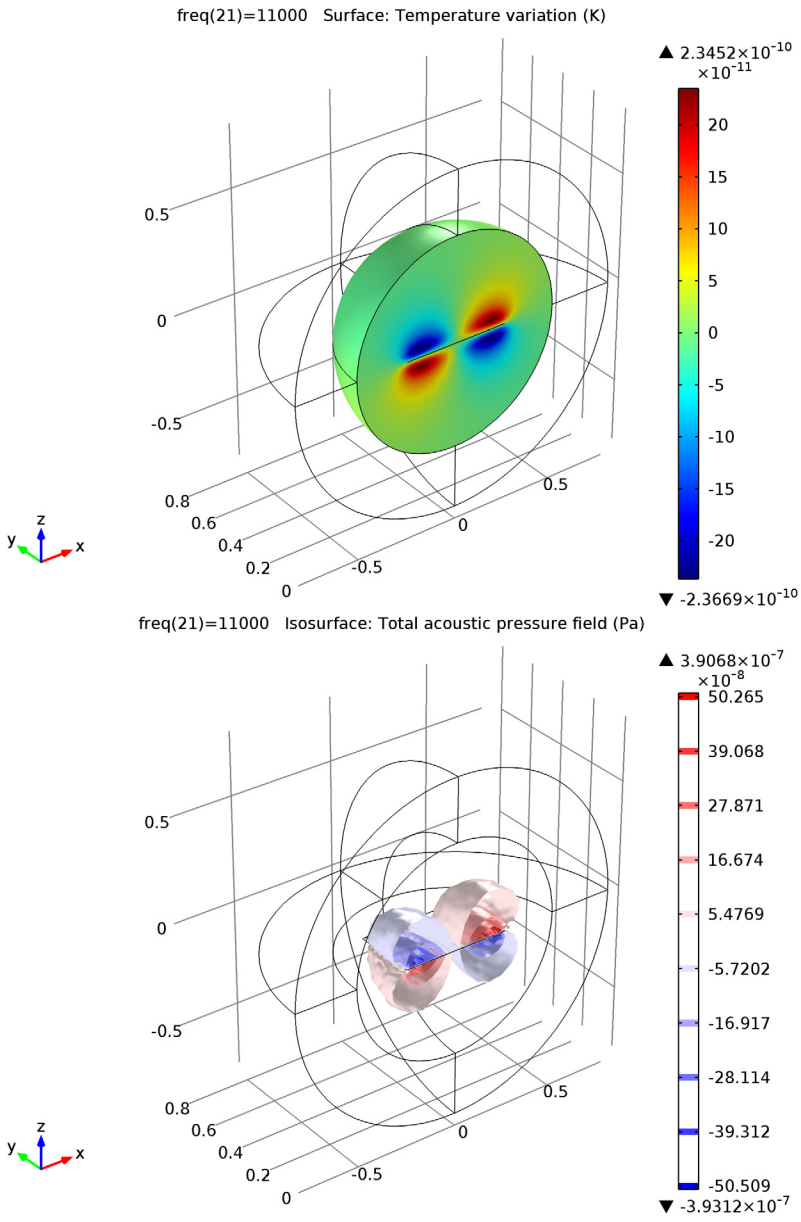


Figure 3: (top) Temperature field in the thermoacoustic domain around the micromirror, and (bottom) pressure iso-surfaces showing the anti-symmetric pressure distribution.

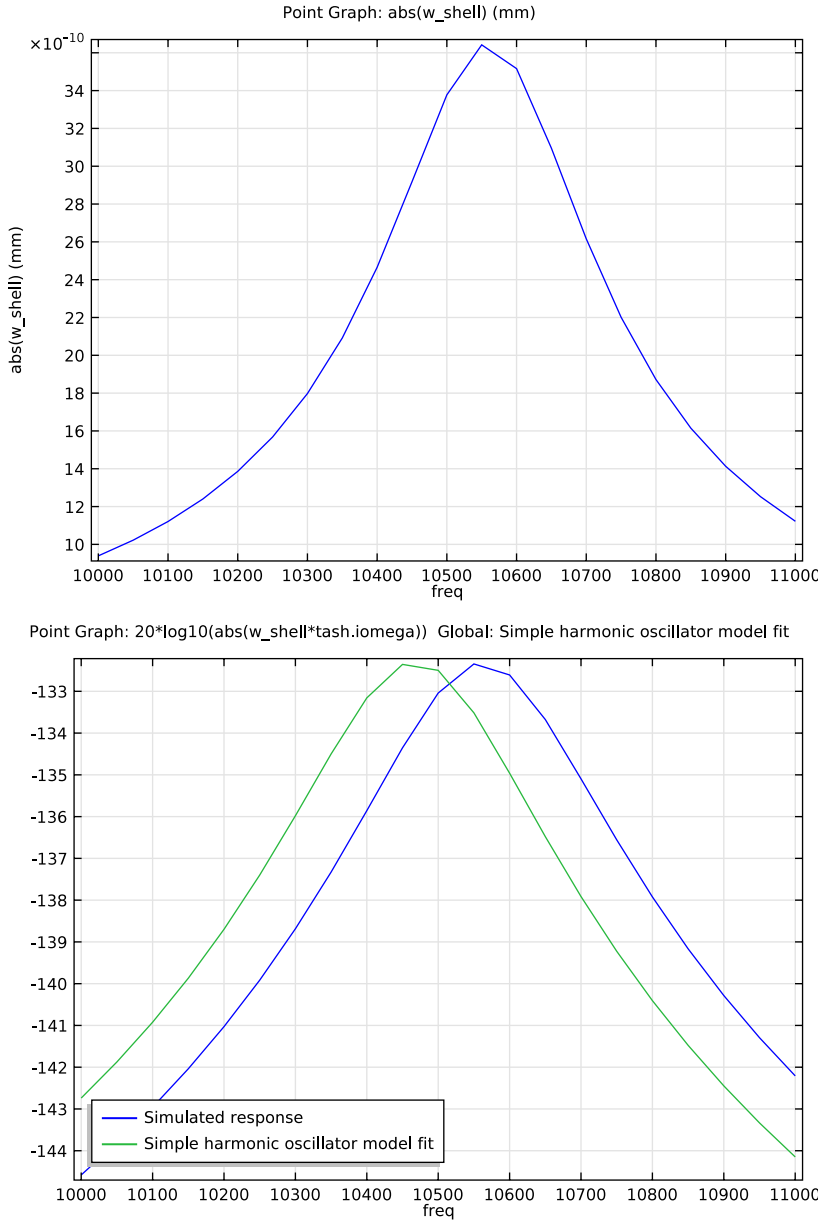


Figure 4: (top) Displacement response, and (bottom) power response with model curve.

 ©2013 COMSOL

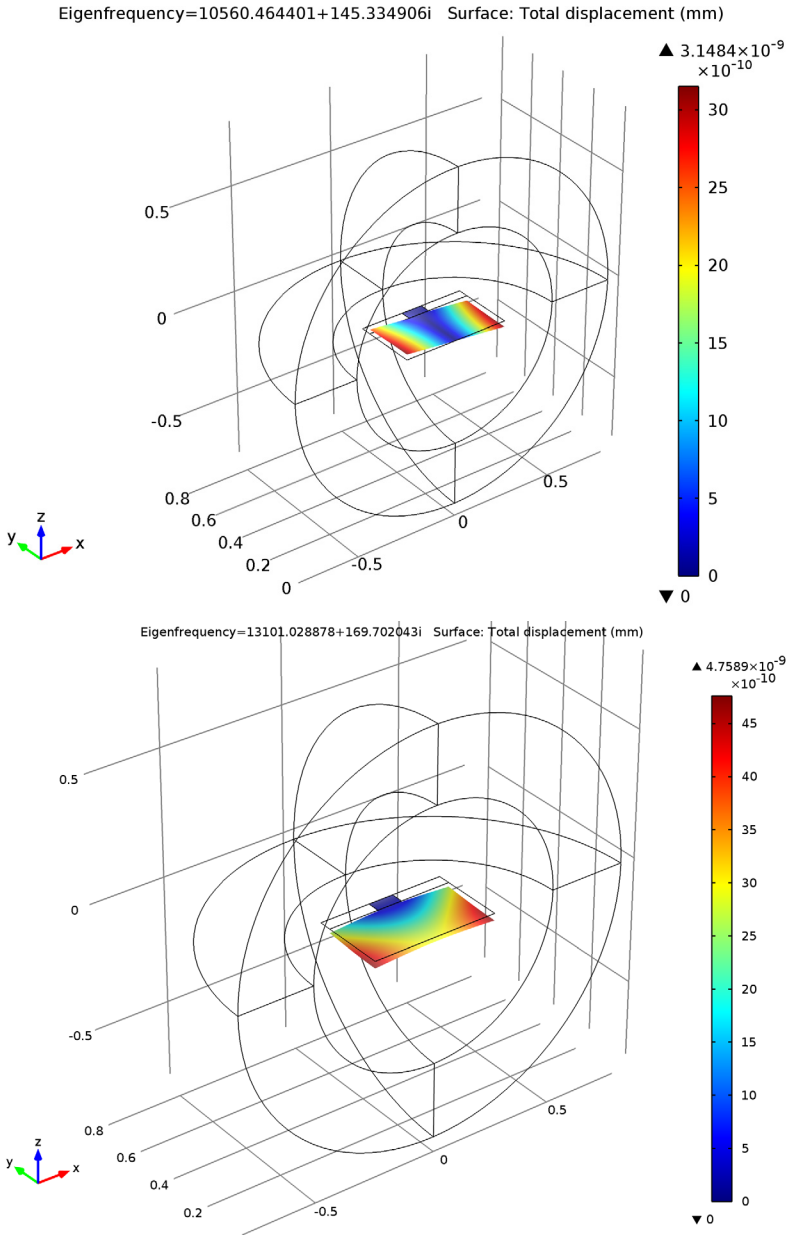


Figure 5: Torquing (top) and symmetric (bottom) vibrating modes of the micromirror.

Model Library path: `Acoustics_Module/Tutorial_Models/vibrating_micromirror`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Acoustics>Thermoacoustics>Thermoacoustic-Shell Interaction, Frequency Domain (tash)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

6 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

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

2 In the **Parameters** settings window, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `vibrating_micromirror_parameters.txt`.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

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

4 Right-click **Model 1>Geometry 1** and choose **Work Plane**.

Rectangle 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

 ©2013 COMSOL

3 In the **Width** edit field, type `0.5`.

4 In the **Height** edit field, type `0.5`.

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

Rectangle 2

1 Right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.1`.

4 In the **Height** edit field, type `0.05`.

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

6 In the **yw** edit field, type `0.25`.

Rectangle 3

1 Right-click **Model 1>Geometry 1>Work Plane 1>Plane Geometry>Rectangle 2** and choose **Duplicate**.

2 In the **Rectangle** settings window, locate the **Position** section.

3 In the **yw** edit field, type `-0.3`.

Rectangle 4

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.5`.

4 In the **Height** edit field, type `0.6`.

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

6 Click the **Build All** button.

Sphere 1

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

2 In the **Sphere** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit 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

1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.

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

3 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.

4 Click the **Build Selected** button.

Block 1

1 Right-click **Geometry 1** and choose **Block**.

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

3 In the **Width** edit field, type `2`.

4 In the **Depth** edit field, type `2`.

5 In the **Height** edit field, type `2`.

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

7 In the **y** edit field, type `-1`.

8 Click the **Build Selected** button.

Difference 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.

2 In the **Difference** settings window, locate the **Difference** section.

3 In the **Relative repair tolerance** edit field, type `1e-6`.

4 Under **Objects to add**, click **Activate Selection**.

5 Select the object **uni1** only.

6 In the **Difference** settings window, locate the **Difference** section.

7 Under **Objects to subtract**, click **Activate Selection**.

8 Select the object **blk1** only.

9 Click the **Build All** button.

10 Click the **Go to Default 3D View** button on the Graphics toolbar.

11 Click the **Wireframe Rendering** button on the Graphics toolbar.

The geometry should look like the one depicted in Figure 1.

 ©2013 COMSOL

DEFINITIONS

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

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

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

4 Select Boundaries 9 and 11 only.

5 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Shell` in the **New name** edit field.

7 Click **OK**.

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click **Add Material to Model**.

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Bulk viscosity	muB	0

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Silicon**.

3 Click **Add Material to Model**.

Silicon

1 In the **Model Builder** window, under **Model 1>Materials** click **Silicon**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

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

THERMOACOUSTIC-SHELL INTERACTION, FREQUENCY DOMAIN

1 In the **Model Builder** window, under **Model 1** click **Thermoacoustic-Shell Interaction, Frequency Domain**.

2 In the **Thermoacoustic-Shell Interaction, Frequency Domain** settings window, locate the **Thickness** section.

3 In the d edit field, type `h_mirror`.

Thermoacoustics Model 1

1 In the **Model Builder** window, under **Model 1>Thermoacoustic-Shell Interaction, Frequency Domain** click **Thermoacoustics Model 1**.

2 In the **Thermoacoustics Model** settings window, locate the **Thermoacoustics Model** section.

3 From the $\rho_0(p_0,T_0)$ list, choose **From material**.

Pressure Acoustics Model 1

1 In the **Model Builder** window, under **Model 1>Thermoacoustic-Shell Interaction, Frequency Domain** click **Pressure Acoustics Model 1**.

2 Select Domains 1, 2, 4, and 5 only.

Interior Shell 1

1 In the **Model Builder** window, right-click **Thermoacoustic-Shell Interaction, Frequency Domain** and choose **Interior Shell**.

2 In the **Interior Shell** settings window, locate the **Boundary Selection** section.

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

Fixed Constraint 1

1 Right-click **Thermoacoustic-Shell Interaction, Frequency Domain** and choose **Edges>Fixed Constraint**.

2 Select Edge 15 only.

Body Load 1

1 Right-click **Thermoacoustic-Shell Interaction, Frequency Domain** and choose **Shell>Body Load**.

2 Select Boundary 9 only.

3 In the **Body Load** settings window, locate the **Force** section.

 ©2013 COMSOL

4 In the $\mathbf{F}_V$ table, enter the following settings:

0	x
0	y
x/(0.25[mm])*1[N/m^3]	z

This corresponds to a torquing force proportional to the x coordinate acting in the z-direction. A load of 0 N/m^3 at in the middle (at x = 0) rising to +/- 1 N/m^3 at the mirror edge.

Symmetry 11

1 Right-click **Thermoacoustic-Shell Interaction, Frequency Domain** and choose **Edges>Symmetry**.

2 Select Edge 9 only.

3 In the **Symmetry** settings window, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Global coordinate system**.

Symmetry 12

1 Right-click **Thermoacoustic-Shell Interaction, Frequency Domain** and choose **Symmetry**.

2 Select Boundaries 1, 2, 6, 13, and 17 only.

Spherical Wave Radiation 1

1 Right-click **Thermoacoustic-Shell Interaction, Frequency Domain** and choose the boundary condition **Pressure Acoustics, Frequency Domain>Spherical Wave Radiation**.

2 Select Boundaries 4, 5, 14, and 19 only.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Free Triangular** settings window, locate the **Boundary Selection** section.

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

Size 1

1 Right-click **Model 1>Mesh 1>Free Triangular 1** and choose **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated edit field, type `1.2*dvisc`.

This is an important mesh setting as it determines the resolution of the acoustic boundary layer. A coarse setting is of the order `1.2*dvisc` and will solve with about 4GB of RAM. A well resolved setting, of maybe `0.5*dvisc`, will result in a model that requires about 12 GB of RAM to solve.

6 Select the **Minimum element size** check box.

7 In the associated edit field, type `dvisc/10`.

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `343[m/s]/f0`.

5 In the **Resolution of narrow regions** edit field, type `1`.

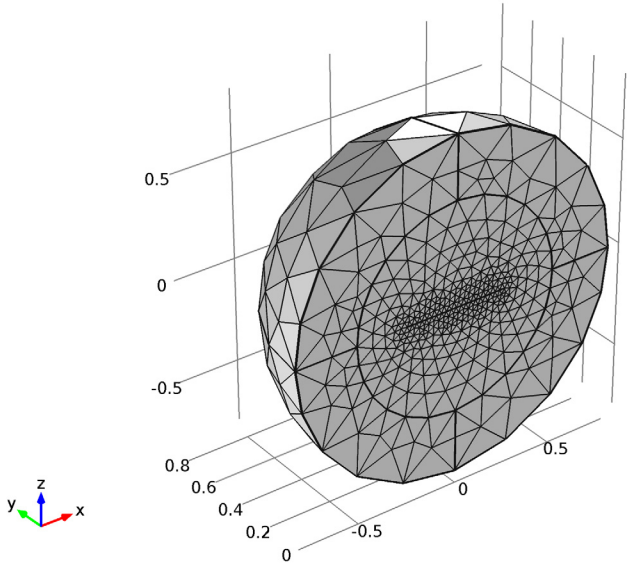
Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

 ©2013 COMSOL

2 In the **Settings** window, click **Build All**.

The geometry should look like the figure below.



STUDY 1

Step 1: Frequency Domain

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

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

3 In the **Frequencies** edit field, type `range(f0-deltaf,50,f0+deltaf)`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Data Sets

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

2 Right-click **Data Sets** and choose **More Data Sets>Mirror 3D**.

3 In the **Mirror 3D** settings window, locate the **Plane Data** section.

4 From the **Plane** list, choose **xz-planes**.

Displacement (tash)

1 In the **Model Builder** window, under **Results** click **Displacement (tash)**.

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

3 From the **Data set** list, choose **Mirror 3D 1**.

4 From the **Parameter value (freq)** list, choose **10000**.

5 Click the **Plot** button.

The plot should look like the one in Figure 2.

Temperature (tash)

1 In the **Model Builder** window, under **Results** right-click **Temperature (tash)** and choose **Plot**.

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

Acoustic Pressure (tash)

1 In the **Model Builder** window, under **Results** right-click **Acoustic Pressure (tash)** and choose **Plot**.

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

Proceed to define 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 amplitude). The second plot depicts the power (proportional to the velocity squared) on the dB scale and compares it to a simple harmonic oscillator model tuned using parameters extracted from an eigenfrequency study of the system (instructions follow below). The parameters are readily given in the parameters list. Note that they are based on a fine mesh resolution of `0.5*dvisc`.

1D Plot Group 4

1 Right-click **Results** and choose **1D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Point Graph**.

3 Select Point 16 only.

4 In the **Point Graph** settings window, locate the **y-Axis Data** section.

5 In the **Expression** edit field, type `abs(w_shell)`.

6 Click the **Plot** button.

The plot should look like the one in Figure 4 (top).

7 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Rename**.

 ©2013 COMSOL

8 Go to the **Rename 1D Plot Group** dialog box and type `Displacement response` in the **New name** edit field.

9 Click **OK**.

1D Plot Group 5

1 Right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower left**.

4 Right-click **Results>1D Plot Group 5** and choose **Point Graph**.

5 Select Point 16 only.

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

7 In the **Expression** edit field, type `20*log10(abs(w_shell*tash.iomega))`.

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

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

10 In the table, enter the following settings:

Legends
`Simulated response`

11 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Global**.

12 In the **Global** settings window, locate the **y-Axis Data** section.

13 In the table, enter the following settings:

Expression	Description
`20*log10(abs(omega_r^2/((tash.iomega)^2+2*alpha_num*tash.iomega+omega_r^2)))-163.5`	`Simple harmonic oscillator model fit`

14 Click the **Plot** button.

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

15 Right-click **1D Plot Group 5** and choose **Rename**.

16 Go to the **Rename 1D Plot Group** dialog box and type `Power response` in the **New name** edit field.

17 Click **OK**.

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 parameters list corresponds to a finer mesh than the default used here.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Eigenfrequency**.

3 Click **Finish**.

STUDY 2

Step 1: Eigenfrequency

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

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

3 In the **Desired number of eigenfrequencies** edit field, type `6`.

4 In the **Search for eigenfrequencies around** edit field, type `10500`.

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

To decide which eigenfrequencies are physical vibrating models (and not spurious modes) you need to investigate the plots. It is here important to note that the exact value of the calculated eigenfrequency will converge for better mesh resolutions. That is, changing the boundary resolution from the selected `1.2*dvisc` to, for example, `0.5*dvisc`. The latter finer resolution produces the value of `f_num` found in the parameters list.

The first default selected eigenvalue is a spurious mode.

RESULTS

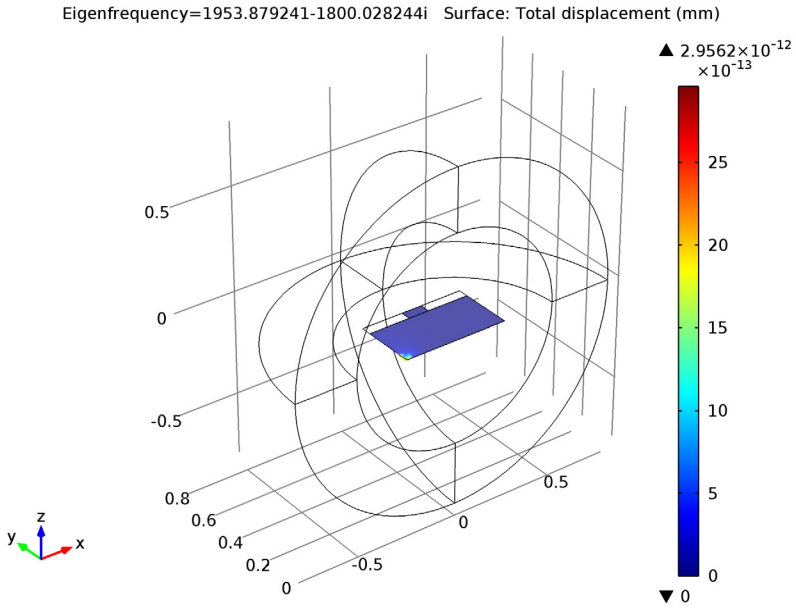
Displacement (tash) 1

1 In the **3D Plot Group** settings window, locate the **Data** section.

 ©2013 COMSOL

2 From the **Eigenfrequency** list, choose **1953.879241-1800.028244i**.

The first (spurious) mode will look something like the figure below.



Now select the eigenfrequency 10560+145i and plot it (look at the eigenvalue when selecting the solution).This represents the vibrating torqued mode that was studied in the frequency domain study. The figure should look like the one in Figure 5 (top).

3 In the **3D Plot Group** settings window, locate the **Data** section.

4 From the **Eigenfrequency** list, choose **10560+145i**.

5 Click the **Plot** button.

Next select the eigenfrequency 13101+169i and plot it (look at the eigenvalue when selecting the solution).This is a symmetric flexing mode. The figure should look like the one in Figure 5 (bottom).

6 In the **3D Plot Group** settings window, locate the **Data** section.

7 From the **Eigenfrequency** list, choose **13101+169i**.

8 Click the **Plot** button.

 ©2013 COMSOL