Created in COMSOL Multiphysics 6.2

Viscous Damping of a Microperforated Plate in the Slip Flow Regime

Introduction

The advances in the fabrication of micromechanical devices and transducers has reduced the dimensions of thermoviscous acoustic (microacoustic) systems to require the handling of high Knudsen's numbers. The Knudsen number is high when the characteristic dimensions of a system becomes comparable to the mean free path of the gas molecules. This either occurs at low pressure where the mean free path becomes large or in systems with small geometric dimensions. In the slip-flow regime $0.001 < \mathrm{Kn} < 0.1$ the effects can be included by using the so-called slip-velocity boundary condition instead of the traditional no-slip boundary condition. At even higher Knudsen numbers it is necessary to use the *Transitional Flow* or *Free Molecular Flow* interface available in the Molecular Flow Module. However, these are not formulated for acoustic problems.

This tutorial model is of a perforated membrane/plate (also known as a micro perforated plate or MPP) backed by a vibrating structure. This is a typical configuration in, for example, a MEMS microphone. The vibrating structure is not modeled explicitly, but just assigned a vibration velocity. The vibrating structure creates a pressure field that squeezes air through the perforated membrane. In the system there are two main sources for viscous losses: viscous losses in the squeezing flow between the gap between the vibrating structure and perforated membrane, and the viscous losses from the flow through the holes in the perforates. In this model the optimal hole size, to achieve minimal resistive losses, is analyzed for a given gap height and a fixed porosity (area ratio) for the perforates. The results are compared to the analytical results by Homentcovschi and Miles in Ref. 1.

The model also investigates the importance of using a slip-velocity boundary condition instead of a no-slip boundary condition. A boundary condition with a slip velocity will lead to smaller gradients in the velocity field near boundaries and thus less damping. The optimal geometry is roughly the same for the two boundary conditions, however the magnitude of the resistive loss changes significantly.

Model Definition

The model represents a hexagonal unit cell using symmetry planes, to only model one twelfth of the unit cell (see Figure 1). The model consists solely of an air domain described by the Thermoviscous Acoustics, Frequency Domain interface. The model consists of a vibrating surface placed $d = 1.5$ μm from the perforated membrane/plate, and on the other side the air domain ends in a Port using a Plane Wave. The port is placed at a distance equal to twice the thermal boundary layer thickness away from the perforated membrane. This is to ensure that generated entropy waves vanish and do not interact with the Port. The area of the hole is set to be 20% of the unit cell area. The geometry is defined by the distance between the vibrating surface and the perforated membrane d = 1.5 µm, the thickness of the perforated membrane h = 1.0 µm, the area factor AR = 0.2, and the radius of the holes R. See Figure 1 for the geometry of the hexagonal unit cell. A parametric sweep is run over the radius R to find the optimal radius. The optimal radius is chosen to be the radius with lowest resistive loss (real part of the input impedance). The optimal radius is compared to the analytical results of Ref. 1. Here the optimal amount of holes per area for an incompressible gas is given as

$$N_{\mathrm{opt}} = \sqrt{\frac{3}{2hd^3}\left(\frac{\mathrm{AR}}{2} - \frac{\mathrm{AR}^2}{8} - \frac{\ln\mathrm{AR}}{4} - \frac{3}{8}\right)}\frac{\mathrm{AR}}{\pi} \qquad (1)$$

The optimal radius is given by the hole density N_{opt} and the area factor AR as

$$R_{\mathrm{opt}} = \sqrt{\frac{\mathrm{AR}}{\pi N_{\mathrm{opt}}}} \qquad (2)$$

For the parameters h = 1.0 µm, d = 1.5 µm, and AR = 0.2 the optimal radius is analytically given as R_{opt} = 2.3 µm. The model is run with both the Slip Wall boundary condition and the classical No Slip boundary condition (the No slip option of the Wall condition) to see the importance of including the slip velocity formulation at high Knudsen number.

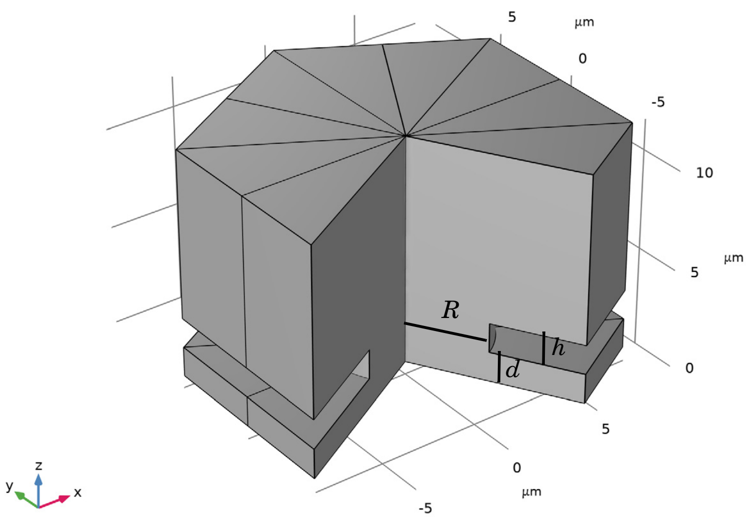


Figure 1: Geometry of unit cell. There are three vertical symmetry planes to create a hexagonal unit cell. At acoustics is actuated by a vibration of the boundary at z = 0, and a port i placed at the top boundary. The perforated plate is assumed to be fixed.

Results and Discussion

The model is run from radius $R = 1.0$ µm to $R = 4.0$ µm. The acoustic velocity with the Slip Wall boundary condition is shown in Figure 2 for three radii R. Since a fixed area factor AR is used the size of the unit cell increases with the radius R. At the large radius of $R = 4.0$ µm the largest velocity is between the membrane and perforated plate in the xy-plane, while for the smallest radius of $R = 1$ µm the highest velocity is located in the perforated holes and is in the z direction.

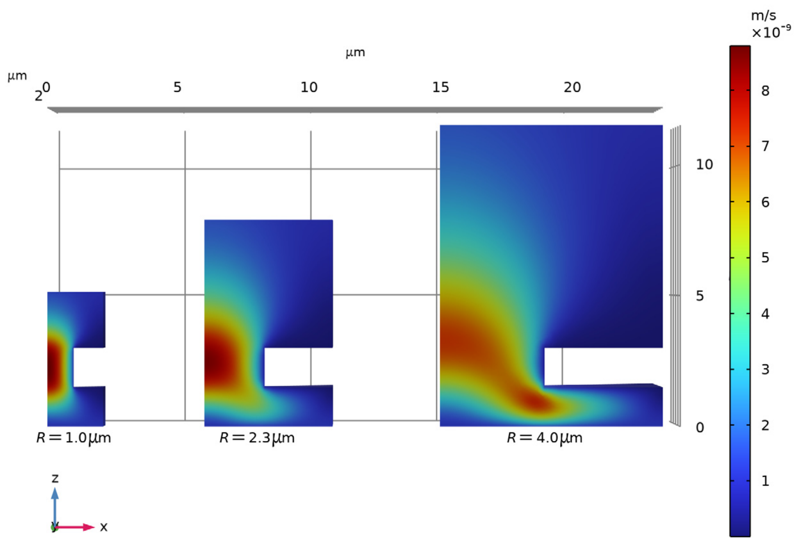


Figure 2: Acoustic velocity for R = 1, 2.3, 4 µm the velocity profile consists of a horizontal squeezing flow between the two plates and a vertical flow through the holes in the perforated plate.

The input (specific) impedance is evaluated on the vibrating surface as

$$Z = \frac{1}{A}\int_{\Omega} \frac{p_t}{w\rho_0 c} d\Omega \tag{3}$$

Where the integration area is the vibrating membrane at $z = 0$, A is the area of the integration surface, and w is the actuation velocity set to 1 nm/s. The impedance as a function of R can be seen in Figure 3 for both the Slip Wall boundary condition and the No Slip boundary condition. The location of the minimum impedance is in good agreement with the analytical prediction of $R_{\mathrm{opt}} = 2.3$ µm.

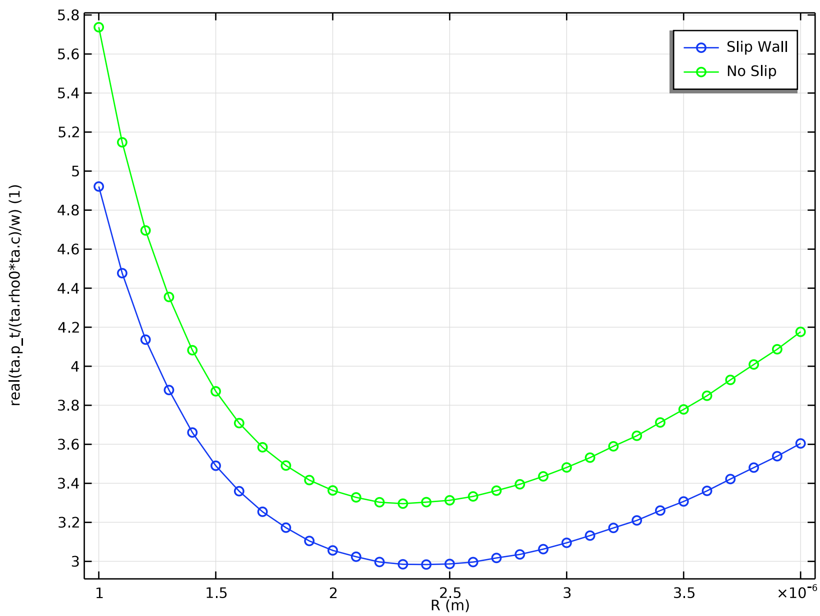


Figure 3: The real part of the impedance as a function of the hole radius R. In blue the model with Slip Wall boundary conditions and in green the model with No Slip boundary conditions.

The dissipation is lowest around R = 2.3 µm. At smaller radii the dissipation is dominated by the viscous losses from the vertical flow through the holes of the perforated membrane and for larger radii the viscous losses is dominated by the horizontal squeezing flow between the two plates. In Figure 4 the total dissipation of the acoustic field is shown for the three different radii, this shows how the dissipation occurs at different locations in the model at large and small radii. The dissipation for R = 1.0 µm is located near the vertical wall in the perforated holes, while the dissipation for R = 4.0 µm is largest between the two plates at the edge of the perforated holes. For R = 2.3 µm the losses is shared between the horizontal squeezing flow and the vertical flow through the holes, but also including losses beneath the hole where the flow changes from a horizontal to vertical flow.

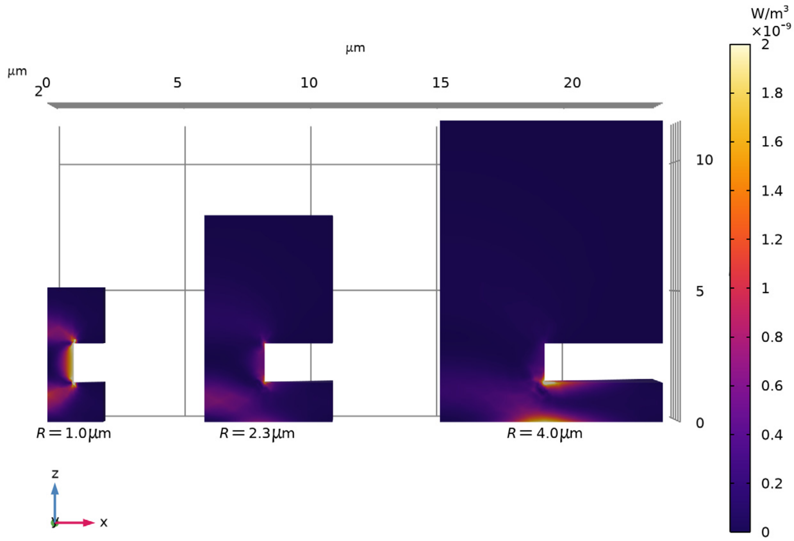


Figure 4: Acoustic dissipation for R = 1, 2.3, 4 µm. The dissipation is either dominated by the horizontal squeezing flow or the vertical flow through the holes.

Notes About the COMSOL Implementation

The implementation uses the boundary condition **Slip Wall**, available in the thermoviscous acoustics interfaces, to model the slip velocity for high Knudsen numbers. The theoretical background for the boundary condition can be found in the documentation of the Slip Wall boundary condition found in the *Acoustics Module User's Guide*.

Reference

1. D. Homentcovschi and R.N. Miles, "Viscous damping of perforated planar micromechanical structures," *Sensors and Actuators A: Physical*, vol. 119, no. 2, pp. 544–552, 2005.

Application Library path: `Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/viscous_damping_mpp`

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 **Acoustics>Thermoviscous Acoustics>Thermoviscous Acoustics, Frequency Domain (ta)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

6 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 `viscous_damping_mpp_parameters.txt`.

GEOMETRY 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node, then click **Geometry 1**.

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

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

Work Plane 1 (wp1)

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

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>Circle 1 (c1)

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

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

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

4 In the **Sector angle** text field, type 30.

Work Plane 1 (wp1)>Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

xw (µm)	yw (µm)
0	0
W	0
W	tan(pi/6)*W

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

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

3 In the table, enter the following settings:

Distances (µm)
Hslit
Hslit+H
Hslit+H+W
Hslit+H+2*delta_t

The Port needs to be placed twice the thermal boundary layer thickness away from the membrane for the entropy wave to vanish before interacting with the Port.

4 Click **Build All Objects**.

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

Delete Entities 1 (del1)

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

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

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

4 On the object **ext1**, select Domain 6 only.

Form Composite Domains 1 (cmd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

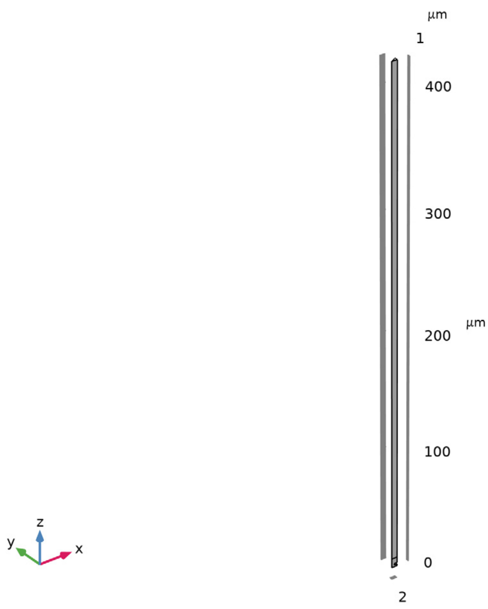
2 On the object **fin**, select Domains 1–3, 5, and 6 only.

Form Composite Domains 2 (cmd2)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 On the object **cmd1**, select Domains 2 and 3 only.

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



Having built the geometry, it can easily be modified as it is parameterized. Simply change the value of a dimension in the parameter list; this will update the geometry automatically.

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 **Home** toolbar, click **Add Material** to close the **Add Material** window.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Symmetry 1

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

2 Select Boundaries 1, 2, 4, 5, and 11–13 only.

Three symmetry planes are used, creating a hexagonal unit cell.

Port 1

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

2 Select Boundary 7 only.

3 In the **Settings** window for **Port**, locate the **Incident Mode Settings** section.

4 From the **Incident wave excitation at this port** list, choose **Off**.

5 Locate the **Port Properties** section. From the **Type of port** list, choose **Plane wave**.

6 Locate the **Port Geometry** section. From the A_{scale} list, choose **User defined**.

7 In the text field, type `12`.

The Multiplication Factor is set to 12 as the geometry is one twelfth of a hexagonal unit cell.

Isothermal 1

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

2 Select Boundary 3 only.

Velocity 1

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

2 Select Boundary 3 only.

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

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

6 Select the **Prescribed in z direction** check box.

7 In the u_{0z} text field, type `1[nm/s]`.

Slip Wall - Fixed

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

2 In the **Settings** window for **Slip Wall**, type `Slip Wall - Fixed` in the **Label** text field.

3 Select Boundaries 8–10 only.

Slip Wall - Vibrating

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

2 In the **Settings** window for **Slip Wall**, type `Slip Wall - Vibrating` in the **Label** text field.

3 Locate the **Mechanical** section. From the **Mechanical condition** list, choose **Moving wall**.

4 Specify the $\mathbf{u}_{\mathrm{w}}$ vector as

`1[nm/s]`	z

5 Select Boundary 3 only.

The Isothermal and Velocity boundary conditions are overwritten by the Slip Wall boundary condition. They are used in the No Slip study step where the Slip Wall boundary condition is disabled.

MESH 1

Free Tetrahedral 1

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

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

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

4 Select Domain 1 only.

Size 1

1 Right-click **Free Tetrahedral 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 **Boundary**.

4 Select Boundaries 3, 9, and 11 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 `Hslit/3`.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 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 **Boundary**.

4 Select Boundary 8 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 `R/8`.

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 From the **Predefined** list, choose **Extra fine**.

Swept 1

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

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

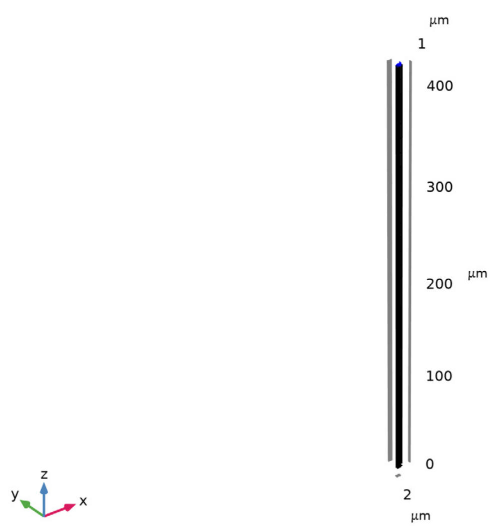
3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Click to expand the **Source Faces** section. Select Boundary 6 only.

6 Click to expand the **Destination Faces** section. Select Boundary 7 only.

7 Click **Build All**.



STUDY - SLIP WALL

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Study - Slip Wall` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, click to select the cell at row number 1 and column number 2.
5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
R (Hole radius)		m

6 Click **Range**.
7 In the **Range** dialog box, type `1[µm]` in the **Start** text field.
8 In the **Step** text field, type `0.1[µm]`.
9 In the **Stop** text field, type `4[µm]`.
10 Click **Replace**.

Step 1: Frequency Domain

1 In the **Model Builder** window, click **Step 1: Frequency Domain**.
2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3 In the **Frequencies** text field, type `f0`.
4 In the **Study** toolbar, click **Compute**.

ADD STUDY

1 In the **Study** 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 **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY - NO SLIP

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

2 In the **Settings** window for **Study**, type `Study - No Slip` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, click to select the cell at row number 1 and column number 2.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
R (Hole radius)		m

6 Click **Range**.

7 In the **Range** dialog box, type `1[µm]` in the **Start** text field.

8 In the **Step** text field, type `0.1[µm]`.

9 In the **Stop** text field, type `4[µm]`.

10 Click **Replace**.

Step 1: Frequency Domain

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

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

3 In the **Frequencies** text field, type `f0`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)>Slip Wall - Fixed**.

6 Right-click and choose **Disable**.

7 In the tree, select **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)>Slip Wall - Vibrating**.

8 Right-click and choose **Disable**.

The two Slip Wall boundary conditions are disabled in the study step.

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

RESULTS

Selection

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

2 Right-click **Study - Slip Wall/Parametric Solutions 1 (sol2)** and choose **Selection**.

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

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

5 Select Domain 1 only.

Selection

1 In the **Model Builder** window, right-click **Study - No Slip/Parametric Solutions 2 (sol35)** and choose **Selection**.

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

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

4 Select Domain 1 only.

Evaluation Group - Slip wall

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

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

3 From the **Dataset** list, choose **Study - Slip Wall/Parametric Solutions 1 (sol2)**.

4 In the **Label** text field, type `Evaluation Group - Slip wall`.

Surface Average 1

1 Right-click **Evaluation Group - Slip wall** and choose **Average>Surface Average**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
real(ta.p_t/(ta.rho0*ta.c)/w)	1	

5 In the **Evaluation Group - Slip wall** toolbar, click **Evaluate**.

Evaluation Group - No slip

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

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group - No slip` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study - No Slip/ Parametric Solutions 2 (sol35)**.

Surface Average 1

1 Right-click **Evaluation Group - No slip** and choose **Average>Surface Average**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`real(ta.p_t/(ta.rho0*ta.c)/w)`	1	

5 In the **Evaluation Group - No slip** toolbar, click **Evaluate**.

Impedance vs R

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

2 In the **Settings** window for **1D Plot Group**, type `Impedance vs R` in the **Label** text field.

Table Graph 1

1 Right-click **Impedance vs R** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

4 From the **x-axis data** list, choose **R (m)**.

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

6 In the **Columns** list, select **real(ta.p_t/(ta.rho0*ta.c)/w) (1)**.

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

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
`Slip Wall`

Table Graph 2

1 In the **Model Builder** window, right-click **Impedance vs R** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

4 From the **Evaluation group** list, choose **Evaluation Group - No slip**.

5 From the **x-axis data** list, choose **R (m)**.

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

7 In the **Columns** list, select **real(ta.p_t/(ta.rho0*ta.c)/w) (1)**.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

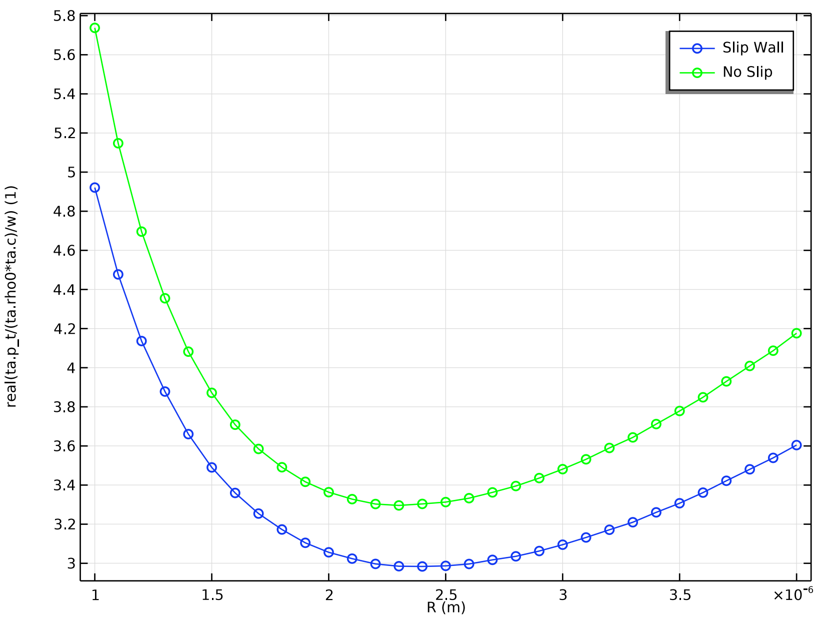
10 Click to expand the **Legends** section. Select the **Show legends** check box.

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

12 In the table, enter the following settings:

Legends
`No Slip`

13 In the **Impedance vs R** toolbar, click **Plot**.



Acoustic Velocity - Slip Wall

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

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Velocity - Slip Wall` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study - Slip Wall/Parametric Solutions 1 (sol2)**.

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

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

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Surface 1

1 Right-click **Acoustic Velocity - Slip Wall** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study - Slip Wall/Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (R (m))** list, choose **1E-6**.

5 Locate the **Expression** section. In the **Expression** text field, type `ta.v_inst`.

Surface 2

1 In the **Model Builder** window, right-click **Acoustic Velocity - Slip Wall** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study - Slip Wall/Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (R (m))** list, choose **2.3E-6**.

5 Locate the **Expression** section. In the **Expression** text field, type `ta.v_inst`.

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

Translation 1

1 Right-click **Surface 2** and choose **Translation**.

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

3 In the **x** text field, type `6`.

Surface 2

In the **Model Builder** window, right-click **Surface 2** and choose **Duplicate**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

3 From the **Parameter value (R (m))** list, choose **4E-6**.

Translation 1

1. In the **Model Builder** window, expand the **Surface 3** node, then click **Translation 1**.
2. In the **Settings** window for **Translation**, locate the **Translation** section.
3. In the **x** text field, type `15`.

Annotation R=1.0

1. In the **Model Builder** window, right-click **Acoustic Velocity - Slip Wall** and choose **Annotation**.
2. In the **Settings** window for **Annotation**, type `Annotation R=1.0` in the **Label** text field.
3. Locate the **Annotation** section. Select the **LaTeX markup** check box.
4. In the **Text** text field, type `$R=1.0\; \mathrm{\mu m}$`.
5. Locate the **Position** section. In the **X** text field, type `1`.
6. Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Upper middle**.
7. Clear the **Show point** check box.
8. Right-click **Annotation R=1.0** and choose **Duplicate**.

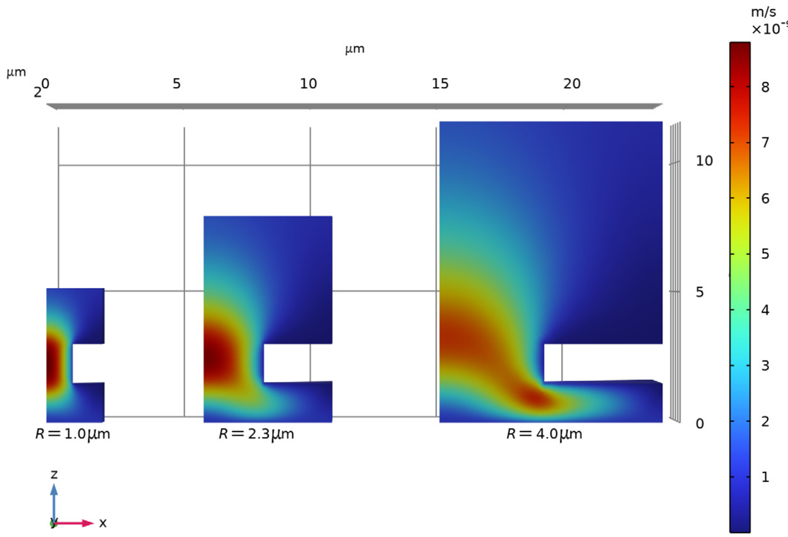
Annotation R=2.3

1. In the **Model Builder** window, under **Results>Acoustic Velocity - Slip Wall** click **Annotation R=1.0.1**.
2. In the **Settings** window for **Annotation**, type `Annotation R=2.3` in the **Label** text field.
3. Locate the **Annotation** section. In the **Text** text field, type `$R=2.3\; \mathrm{\mu m}$`.
4. Locate the **Position** section. In the **X** text field, type `8`.
5. Right-click **Annotation R=2.3** and choose **Duplicate**.

Annotation R=4.0

1. In the **Model Builder** window, under **Results>Acoustic Velocity - Slip Wall** click **Annotation R=2.3.1**.
2. In the **Settings** window for **Annotation**, type `Annotation R=4.0` in the **Label** text field.
3. Locate the **Annotation** section. In the **Text** text field, type `$R=4.0\; \mathrm{\mu m}$`.
4. Locate the **Position** section. In the **X** text field, type `19`.
5. In the **Acoustic Velocity - Slip Wall** toolbar, click **Plot**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

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



Acoustic Velocity - Slip Wall

In the **Model Builder** window, right-click **Acoustic Velocity - Slip Wall** and choose **Duplicate**.

Dissipation - Slip Wall

1 In the **Model Builder** window, under **Results** click **Acoustic Velocity - Slip Wall 1**.

2 In the **Settings** window for **3D Plot Group**, type `Dissipation - Slip Wall` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Dissipation - Slip Wall** node, then click **Surface 1**.

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

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

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `2E-9`.

6 In the **Minimum** text field, type `0`.

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

8 In the **Color Table** dialog box, select **Thermal>HeatCamera** in the tree.

9 Click **OK**.

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

Surface 2

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

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

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

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

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

4 In the **Dissipation - Slip Wall** toolbar, click  **Plot**.