Created in COMSOL Multiphysics 6.2

MEMS Microphone with Slip Wall

Introduction

The development in the area of MEMS devices and microphones is fast, and MEMS microphones are becoming a standard part of various products from laptops to earbuds. This tutorial demonstrates how to set up a model of a MEMS microphone consisting of a micro-perforated plate (MPP) and a vibrating membrane, see Figure 1. The design with an MPP and a diaphragm is used in Ref. 1, the geometry in this model is inspired by Ref. 1 but is simplified and uses different parameters.

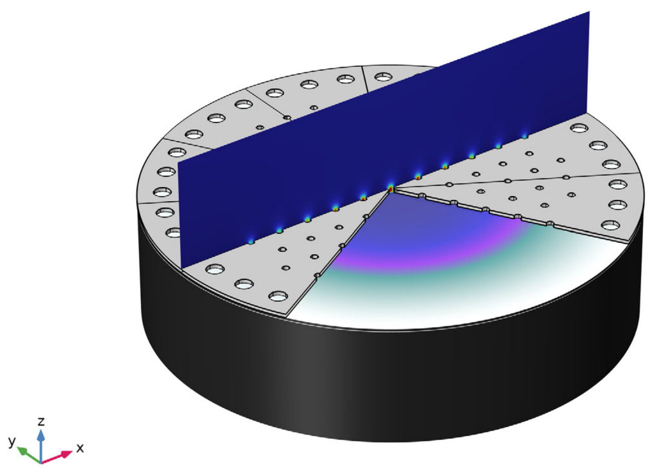


Figure 1: MEMS microphone consisting of a micro-perforated plate (MPP) and a vibrating membrane. The harmonic displacement of the membrane is shown together with the acoustic velocity through the MPP.

The development in manufacturing processes allows to make MEMS devices with smaller length scales. The small length scales lead to high Knudsen numbers and therefore a domain where noncontinuum effects can become important. This model shows how to include a slip velocity boundary condition and thereby increase the range of Knudsen numbers where the finite element model is valid. The slip velocity boundary condition is typically used for Knudsen numbers in the range from 0.001 to 0.01.

Note, the perforation ratio of the MPP in the model is quite low, this is to reduce the number of holes and thus the size of the numerical model.

Model Definition

The MEMS microphone consists of an MPP and a vibrating diaphragm, and a closed backing volume. The MEMS microphone is cylindrical with the holes in the MPP placed in a hexagonal lattice structure. The symmetry allows for modeling a 30-degree section of the cylinder.

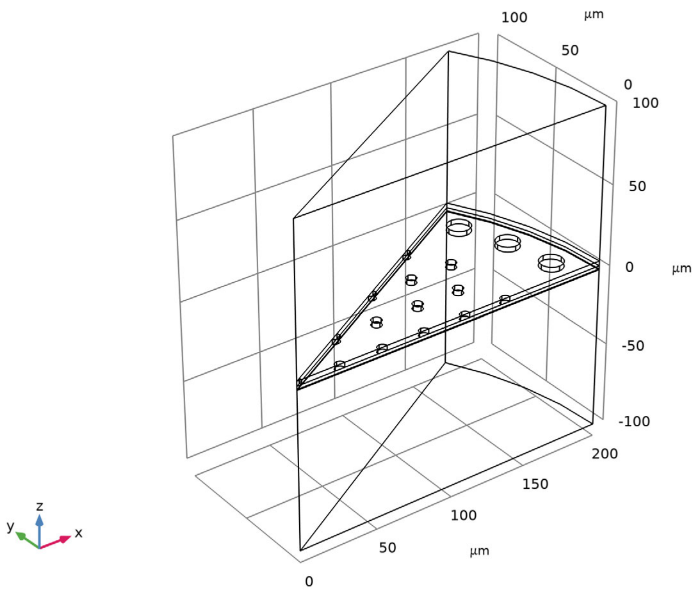


Figure 2: Geometry of the MEMS microphone. A 30-degree section is modeled with symmetry planes.

The MPP consists of holes with 3 µm radius in a hexagonal structure and vent holes with 7 µm radius at the edge of the plate. The MPP has a thickness of 3 µm, and the diaphragm a thickness of 0.5 µm. The distance between the MPP and diaphragm is 2 µm. Below the diaphragm is a closed backing volume modeled with Pressure Acoustics**,** and the domain above the MPP; the holes in the MPP; and the gap between the MPP and diaphragm are modeled with Thermoviscous Acoustics.

The **Slip Wall** boundary condition is applied to the surface of the diaphragm and MPP in the Thermoviscous Acoustics domain. When setting a **Slip Wall** boundary condition the tangential velocity at the wall will depend on the stress in the fluid at the boundary, thus creating a discontinuity between the velocity of the solid and the fluid.

The diaphragm is prestressed by an electric field which gives a stationary deformation of the diaphragm. The diaphragm is connected to ground while a charge is set on the MPP. This results in an electric field between the MPP and diaphragm and a deformation which is modeled with **Moving Mesh**.

In the frequency domain study, a pressure is applied on the surface above the MPP in the **Thermoviscous Acoustics** domain. The diaphragm will vibrate due to the pressure field, and this will cause a varying electrical signal due to the variations in the distance between the MPP and diaphragm.

To model the correct electric response of the MEMS microphone it is important to model the correct damping in the flow through the holes in the MPP and in the squeezing flow between the MPP and the diaphragm. For small length scales it can be necessary to include the noncontinuum effects with the **Slip Wall** boundary condition.

Results and Discussion

The resulting acoustic pressure in all domains is shown in Figure 3.

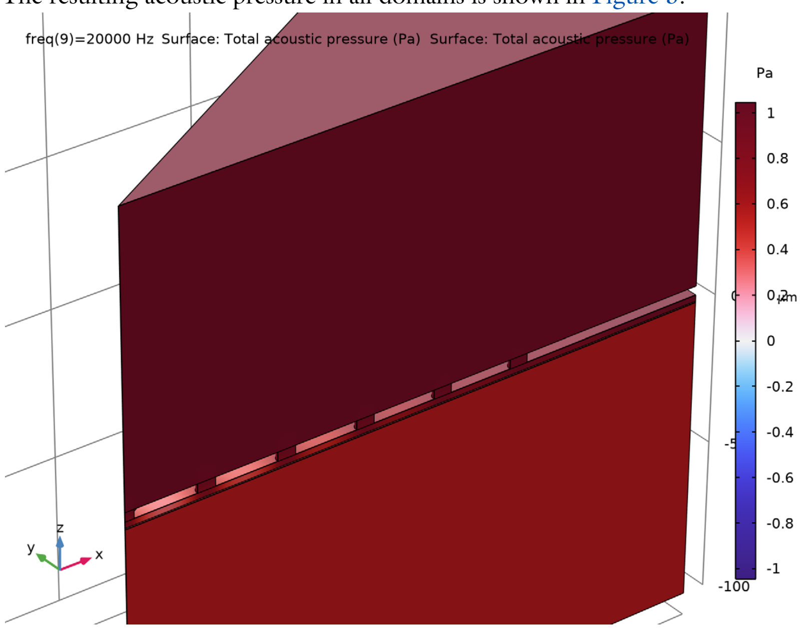


Figure 3: Acoustic pressure at 20 kHz.

The acoustic velocity is shown in Figure 4. Note the high velocities both through the holes in the MPP and in the squeezing flow between the MPP and the diaphragm. These areas are the sources for the viscous damping in the system.

.

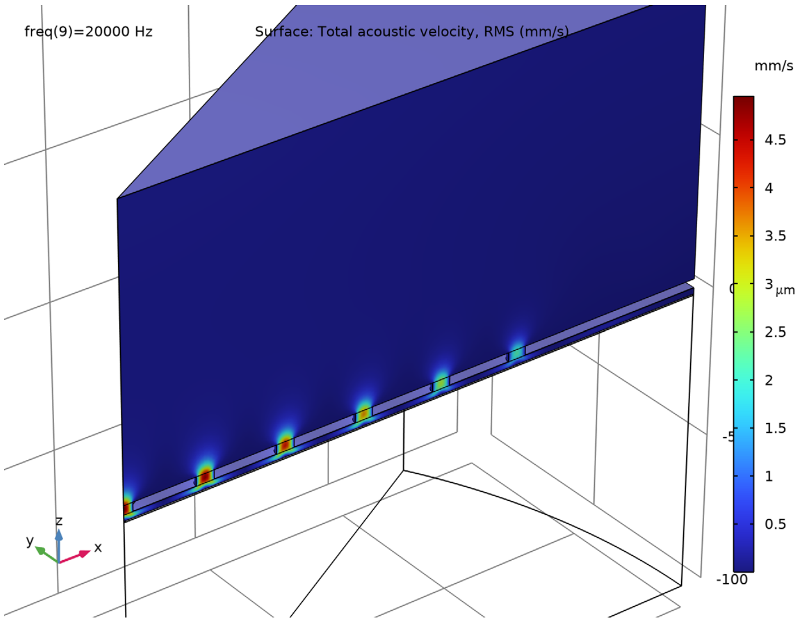


Figure 4: Acoustic pressure.

Figure 5 shows the frequency response of the MEMS microphone from 200 Hz to 20 kHz. At he lower frequencies the spectrum is flat before it drops at higher frequencies. Because of the small length scale of the model, the resonances are located at higher frequencies and thus the spectrum is flat in the audio range.

.

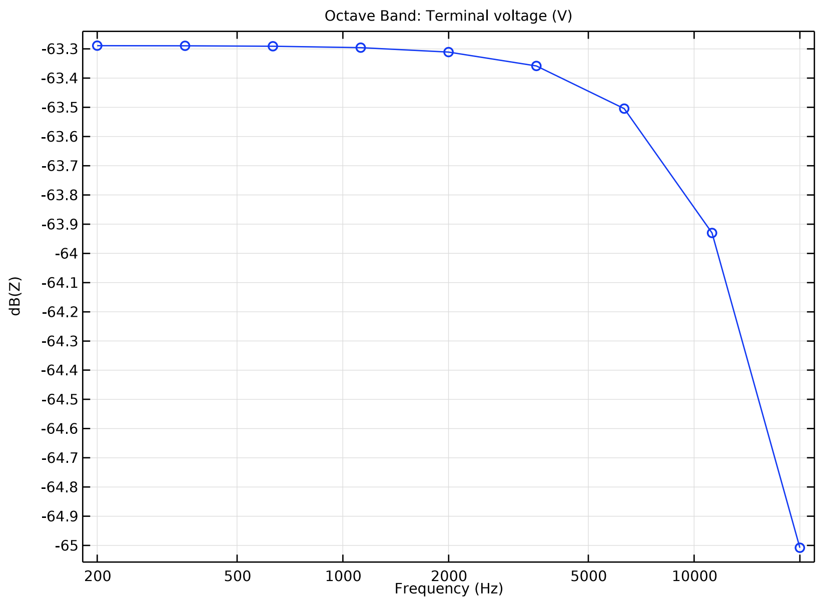


Figure 5: Frequency response of the MEMS microphone.

Notes About the COMSOL Implementation

The multiphysics coupling **Thermoviscous Acoustics-Structure Boundary** does not include the option to set a slip velocity boundary condition in the fluid. Thus, the model uses a manual coupling between the **Thermoviscous Acoustics, Frequency Domain** and **Solid Mechanics** physics interfaces. In the **Slip Wall** node, the **Mechanical condition** is set to **Moving Wall** and the velocity input is chosen to be **Velocity (solid/lemm1)**. In the **Solid Mechanics** interface, a **Boundary Load** is added on the diaphragm, where the **Force per unit area** is set to **Acoustic slip wall traction per unit area (ta/slw2)** picked up from the **Slip Wall** boundary in the **Thermoviscous Acoustics, Frequency Domain** interface.

The size of the model makes it advantageous to use an iterative solver, but it is necessary to manually set up the preconditioners for the iterative solver because of the manual coupling used in the model. The variables are split into four preconditioners, the first contains the acoustic pressure and velocity from Thermoviscous Acoustics, the Lagrange multipliers related to the **Slip Wall** boundary conditions, and the pressure from Pressure Acoustics. The second preconditioner contains the acoustic temperature, and the third preconditioner the mechanical and electrical degrees of freedom. Lastly, the fourth preconditioner contains the variables related to the moving mesh.

Reference

1. P. Loeppert and S. Lee, "Sisonic–the first commercialized MEMS microphone," *Solid-state sensors, actuators and microsystems workshop, Hilton Head Island, South Carolina*, pp. 27–30, 2006.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/mems_microphone_slip_wall`

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. In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
5. Click **Add**.
6. In the **Select Physics** tree, select **Acoustics>Elastic Waves>Solid Mechanics (Elastic Waves) (solid)**.
7. Click **Add**.
8. In the **Select Physics** tree, select **AC/DC>Electric Fields and Currents>Electrostatics (es)**.
9. Click **Add**.
10. In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh>Free Deformation**.

11 Click **Add**.

12 In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh**.

13 Click **Study**.

14 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.

15 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 `mems_microphone_slip_wall_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**.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `dia_r*1`.

4 In the **Height** text field, type `200[µm]`.

5 Locate the **Position** section. In the **z** text field, type `-100[µm]`.

6 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

Diaphragm

1 In the **Model Builder** window, click **Cylinder 2 (cyl2)**.

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

3 In the **Height** text field, type `dia_t`.

4 Locate the **Position** section. In the **z** text field, type `0`.

5 In the **Label** text field, type `Diaphragm`.

6 Right-click **Diaphragm** and choose **Duplicate**.

Backplate

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Diaphragm 1 (cyl3)**.

2 In the **Settings** window for **Cylinder**, type `Backplate` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `back_t`.

4 Locate the **Position** section. In the **z** text field, type `gap+dia_t`.

5 Right-click **Backplate** and choose **Duplicate**.

Initial hole

1 In the **Model Builder** window, click **Backplate 1 (cyl4)**.

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

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

4 In the **Label** text field, type `Initial hole`.

Create an array of holes in a hexagonal lattice structure.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

3 Select the object **cyl4** only.

4 In the **Settings** window for **Array**, locate the **Size** section.

5 From the **Array type** list, choose **Linear**.

6 In the **Size** text field, type `2`.

7 Locate the **Displacement** section. In the **x** text field, type `hole_dist`.

Array 2 (arr2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the objects **arr1(1)** and **arr1(2)** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `2`.

6 Locate the **Displacement** section. In the **x** text field, type `hole_dist/2`.

7 In the **y** text field, type `sin(pi/3)*hole_dist`.

Array 3 (arr3)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the objects **arr2(1,1)**, **arr2(1,2)**, **arr2(2,1)**, and **arr2(2,2)** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `8`.

5 In the **y size** text field, type `8`.

6 Locate the **Displacement** section. In the **x** text field, type `2*hole_dist`.

7 In the **y** text field, type `2*sin(pi/3)*hole_dist`.

8 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

9 In the **New Cumulative Selection** dialog box, type `Holes` in the **Name** text field.

10 Click **OK**.

Cylinder Selection 1 (cylsel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Cylinder Selection**.

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

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

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog box, select **Holes** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Cylinder Selection**, locate the **Size and Shape** section.

9 In the **Outer radius** text field, type `4*dia_r`.

10 In the **Inner radius** text field, type `0.75*dia_r`.

11 Locate the **Output Entities** section. From the **Include entity if** list, choose **Some vertex inside cylinder**.

12 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.

13 Find the **Cumulative selection** subsection. Click **New**.

14 In the **New Cumulative Selection** dialog box, type `Holes tmp` in the **Name** text field.

15 Click **OK**.

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 **Selection** list, choose **Holes tmp**.

Array 1 (arr1), Array 2 (arr2), Array 3 (arr3), Cylinder Selection 1 (cylsel1), Delete Entities 1 (del1), Initial hole (cyl4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Initial hole (cyl4)**, **Array 1 (arr1)**, **Array 2 (arr2)**, **Array 3 (arr3)**, **Cylinder Selection 1 (cylsel1)**, and **Delete Entities 1 (del1)**.

2 Right-click and choose **Group**.

Holes

In the **Settings** window for **Group**, type `Holes` in the **Label** text field.

Vent 1

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

2 In the **Settings** window for **Cylinder**, type `Vent 1` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `vent_r`.

4 In the **Height** text field, type `back_t`.

5 Locate the **Position** section. In the **x** text field, type `cos(5*pi/180)*vent_d`.

6 In the **y** text field, type `sin(5*pi/180)*vent_d`.

7 In the **z** text field, type `gap+dia_t`.

8 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

9 In the **New Cumulative Selection** dialog box, type `Vents` in the **Name** text field.

10 Click **OK**.

11 Right-click **Vent 1** and choose **Duplicate**.

Vent 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Vent 1.1 (cyl6)**.

2 In the **Settings** window for **Cylinder**, type `Vent 2` in the **Label** text field.

3 Locate the **Position** section. In the **x** text field, type `cos(15*pi/180)*vent_d`.

4 In the **y** text field, type `sin(15*pi/180)*vent_d`.

5 Right-click **Vent 2** and choose **Duplicate**.

Vent 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Vent 2.1 (cyl7)**.

2 In the **Settings** window for **Cylinder**, type `Vent 3` in the **Label** text field.

3 Locate the **Position** section. In the **x** text field, type `cos(25*pi/180)*vent_d`.

4 In the **y** text field, type `sin(25*pi/180)*vent_d`.

Vent 1 (cyl5), Vent 2 (cyl6), Vent 3 (cyl7)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Vent 1 (cyl5)**, **Vent 2 (cyl6)**, and **Vent 3 (cyl7)**.

2 Right-click and choose **Group**.

Vents

In the **Settings** window for **Group**, type `Vents` in the **Label** text field.

Cylinder Selection 2 (cylsel2)

1 In the **Geometry** toolbar, click **Selections** and choose **Cylinder Selection**.

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

3 In the **Outer radius** text field, type `1[mm]`.

4 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

5 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `AllCyl` in the **Name** text field.

7 Click **OK**.

8 In the **Settings** window for **Cylinder Selection**, click **Build Selected**.

Union 1 (uni1)

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

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

3 From the **Input objects** list, choose **AllCyl**.

Use a **Work Plane** and **Intersection** node to create a 30 degree section of the geometry.

Work Plane 1 (wp1)

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

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

3 In the **z-coordinate** text field, type `-100[µm]`.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>View 2** node, then click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>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 `dia_r`.
4 In the **Sector angle** text field, type `30`.
5 In the **Work Plane** toolbar, click **Build All**.

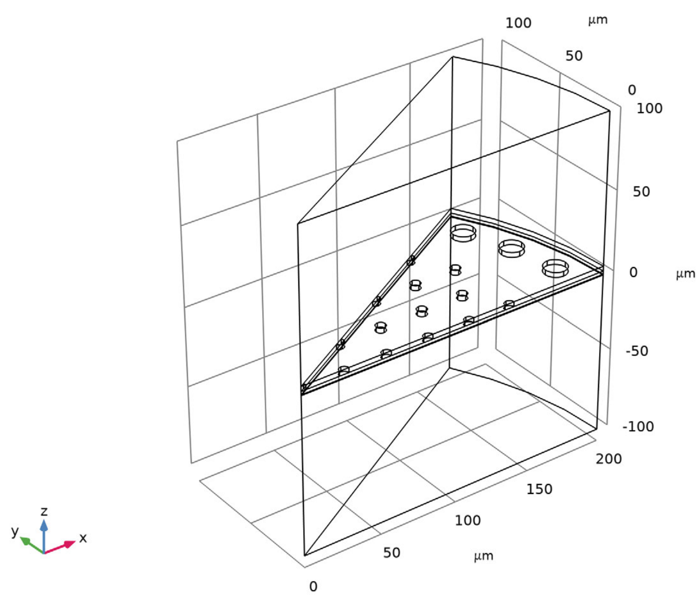
Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

Distances (µm)
`0.5[mm]`

Intersection 1 (int1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.
2 Select the objects **ext1** and **uni1** only.
3 In the **Settings** window for **Intersection**, click **Build All Objects**.



DEFINITIONS

Create selections to make it easier to select domains and surfaces.

Diaphragm

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

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

3 Select Domain 2 only.

Diaphragm boundaries

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

2 In the **Settings** window for **Adjacent**, type `Diaphragm boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Diaphragm** in the **Input selections** list.

5 Click **OK**.

Slip wall boundaries

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

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

3 Select Domains 3 and 6 only.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

5 Select the **Interior boundaries** check box.

Large air domain

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

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

3 Select Domains 3 and 5 only.

Air - TA

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

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

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Large air domain**, **Holes**, and **Vents**.

5 Click **OK**.

Symmetry

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

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

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

4 Select Boundaries 13 and 14 only.

5 Select the **Group by continuous tangent** check box.

6 In the **Label** text field, type `Symmetry`.

All domains - Exterior boundaries

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

2 In the **Settings** window for **Explicit**, locate the **Output Entities** section.

3 From the **Output entities** list, choose **Adjacent boundaries**.

4 Locate the **Input Entities** section. Select the **All domains** check box.

5 In the **Label** text field, type `All domains - Exterior boundaries`.

Top of MPP

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

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

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

4 Select Boundary 12 only.

5 Select the **Group by continuous tangent** check box.

Air - ACPR

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

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

3 Select Domain 1 only.

Backplate boundaries

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

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

3 Select Domain 6 only.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

Electrostatics domains

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

2 In the **Settings** window for **Union**, type `Electrostatics domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Diaphragm**, **Air - TA**, and **Air - ACPR**.

5 Click **OK**.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in>Air** and **Built-in>Silicon**.

3 Click **Add to Component** in the window toolbar.

4 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 **Selection** list, choose **Diaphragm**.

Air (mat1)

1 In the **Model Builder** window, click **Air (mat1)**.

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

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

MOVING MESH

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Moving Mesh** click **Deforming Domain 1**.

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

3 Click **Clear Selection**.

4 Select Domains 1 and 3 only.

Set **Mesh smoothing type** to **Laplace** which is more efficient for small perturbations to the mesh.

5 Locate the **Smoothing** section. From the **Mesh smoothing type** list, choose **Laplace**.

COMPONENT 1 (COMP1)

Symmetry/Roller 1

1 In the **Moving Mesh** toolbar, click **Symmetry/Roller**.
2 In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Symmetry**.

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 From the **Selection** list, choose **Air - TA**.

Slip Wall 1

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

Wall 1

1 In the **Model Builder** window, click **Wall 1**.
2 In the **Settings** window for **Wall**, locate the **Mechanical** section.
3 From the **Mechanical condition** list, choose **Slip (perfect)**.
4 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

Slip Wall - MPP

1 In the **Model Builder** window, under **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)** click **Slip Wall 1**.
2 In the **Settings** window for **Slip Wall**, type `Slip Wall - MPP` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Slip wall boundaries**.

Slip Wall - Diaphragm

1 In the **Physics** toolbar, click **Boundaries** and choose **Slip Wall**.
2 In the **Settings** window for **Slip Wall**, type `Slip Wall - Diaphragm` in the **Label** text field.
3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Diaphragm boundaries**.
4 Locate the **Mechanical** section. From the **Mechanical condition** list, choose **Moving wall**.

5 From the $\mathbf{u}_w$ list, choose **Velocity (solid/lemm1)**.

Manually couple to the velocity of the diaphragm.

Symmetry 1

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 (Adiabatic) 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 Select Boundary 16 only.

3 In the **Settings** window for **Pressure (Adiabatic)**, locate the **Pressure** section.

4 In the p_{bnd} text field, type `linper(1)`.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

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

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

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

Symmetry 1

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**.

SOLID MECHANICS (SOLID)

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

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

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

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **All domains - Exterior boundaries**.

Symmetry 1

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**.

Add a **Boundary Load** to make the manual coupling to the *Thermoviscous Acoustics*.

Boundary Load 1

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

2 Select Boundary 9 only.

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

4 From the $\mathbf{F}_A$ list, choose **Acoustic slip wall traction per unit area (ta/slw2)**.

ELECTROSTATICS (ES)

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

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

3 From the **Selection** list, choose **Electrostatics domains**.

Charge Conservation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** click **Charge Conservation 1**.

2 In the **Settings** window for **Charge Conservation**, locate the **Material Type** section.

3 From the **Material type** list, choose **Solid**.

Ground 1

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

2 Select Boundary 6 only.

Terminal 1

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

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

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

4 Locate the **Terminal** section. In the Q_0 text field, type `1e-13[C]`.

Symmetry Plane 1

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

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

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

MULTIPHYSICS

Electromechanical Forces 1 (eme1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Electromechanical Forces**.

Acoustic–Structure Boundary 1 (asb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic–Structure Boundary**.
2 In the **Settings** window for **Acoustic–Structure Boundary**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.

MESH 1

Size 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.

Size

1 In the **Settings** window for **Size**, locate the **Element Size** section.
2 From the **Predefined** list, choose **Finer**.

Size - Holes

1 In the **Model Builder** window, click **Size 1**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Holes**.
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 `hole_r/1.5`.
8 In the **Label** text field, type `Size - Holes`.
9 Right-click **Size - Holes** and choose **Duplicate**.

Size - Vents

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size - Holes 1**.
2 In the **Settings** window for **Size**, type `Size - Vents` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Vents**.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `vent_r/1.5`.

Free Triangular 1

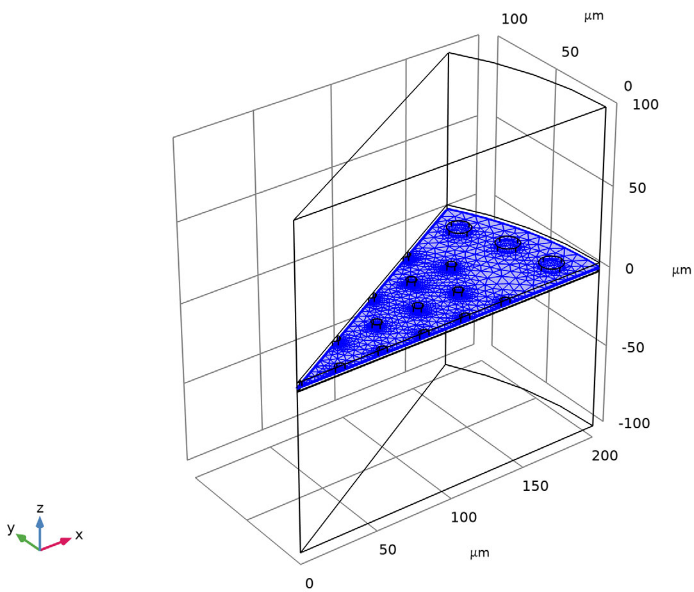
1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

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

3 From the **Selection** list, choose **Top of MPP**.

4 Click **Build Selected**.

The triangular mesh should be as the following figure.



Swept - Gap and diaphragm

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

2 In the **Settings** window for **Swept**, type `Swept - Gap and diaphragm` in the **Label** text field.

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

4 Select Domains 2 and 3 only.

Distribution 1

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

2. In the **Settings** window for **Distribution**, locate the **Domain Selection** section.
3. In the list, select **2**.
4. Select Domain 3 only.
5. Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.
6. In the **Element ratio** text field, type `5`.
7. Select the **Symmetric distribution** check box.
8. Right-click **Distribution 1** and choose **Duplicate**.

Distribution 2

1. In the **Model Builder** window, click **Distribution 2**.
2. In the **Settings** window for **Distribution**, locate the **Distribution** section.
3. From the **Distribution type** list, choose **Fixed number of elements**.
4. Select Domain 2 only.
5. In the **Number of elements** text field, type `2`.

Swept - Holes

1. In the **Mesh** toolbar, click **Swept**.
2. In the **Settings** window for **Swept**, type `Swept - Holes` in the **Label** text field.
3. Locate the **Domain Selection** section. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Holes**.

Distribution 1

1. Right-click **Swept - Holes** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Distribution** section.
3. From the **Distribution type** list, choose **Predefined**.
4. Select the **Symmetric distribution** check box.
5. In the **Element ratio** text field, type `5`.

Swept - Holes

In the **Model Builder** window, right-click **Swept - Holes** and choose **Duplicate**.

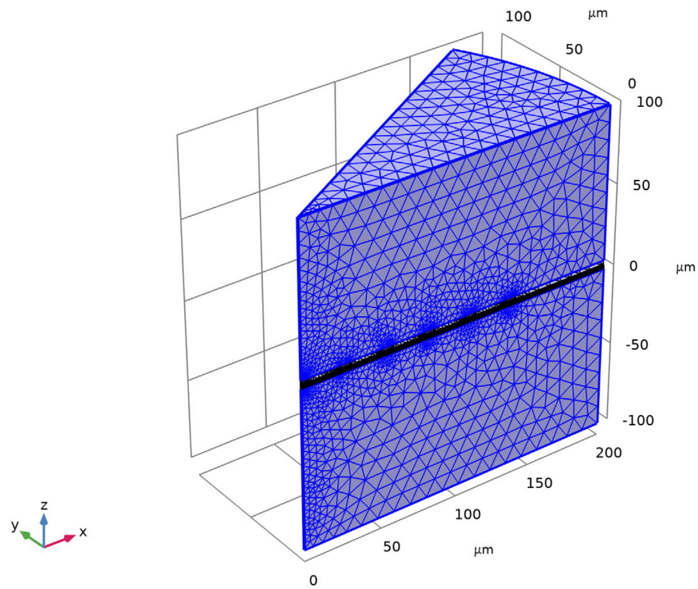
Swept - Vents

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Swept - Holes 1**.
2. In the **Settings** window for **Swept**, type `Swept - Vents` in the **Label** text field.
3. Locate the **Domain Selection** section. From the **Selection** list, choose **Vents**.
4. Click **Build Selected**.

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 Domains 1 and 5 only.
5 Click **Build All**.

The finalized mesh should be as the following figure.



STUDY 1

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, expand the **Study 1** node.
2 Right-click **Study 1** and choose **Study Steps>Frequency Domain>Frequency-Domain Perturbation**.
3 In the **Settings** window for **Frequency-Domain Perturbation**, locate the **Study Settings** section.
4 In the **Frequencies** text field, type `10^{range(log10(200),1/4,log10(20000))}`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Stationary Solver 2**.

4 In the **Settings** window for **Stationary Solver**, locate the **General** section.

5 In the **Relative tolerance** text field, type `1.0E-3`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** right-click **Suggested Iterative Solver (GMRES with Direct Precon.) ()** and choose **Enable**.

Adjust the predefined iterative solver to include the variables for *Solid Mechanics*, *Pressure Acoustics*, *Electrostatics*, and *Moving Mesh*.

7 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2>Suggested Iterative Solver (GMRES with Direct Precon.) ()** node, then click **Direct Preconditioner 1**.

8 In the **Settings** window for **Direct Preconditioner**, click to expand the **Hybridization** section.

9 Under **Preconditioner variables**, click **Add**.

10 In the **Add** dialog box, select **Acoustic pressure (comp1.p2)** in the **Preconditioner variables** list.

11 Click **OK**.

12 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** right-click **Suggested Iterative Solver (GMRES with Direct Precon.) ()** and choose **Direct Preconditioner**.

13 In the **Settings** window for **Direct Preconditioner**, locate the **General** section.

14 From the **Solver** list, choose **PARDISO**.

15 Locate the **Hybridization** section. In the **Preconditioner variables** list, choose **Acoustic pressure (comp1.p)**, **Acoustic pressure (comp1.p2)**, **Spatial mesh displacement (comp1.spatial.disp)**, **Temperature variation (comp1.T)**, **Lagrange multiplier (spatial frame) (comp1.ta.slw1.lm_tau)**, **Lagrange multiplier (spatial frame) (comp1.ta.slw2.lm_tau)**, and **Acoustic velocity (spatial frame) (comp1.u)**.

16 Under **Preconditioner variables**, click **Delete**.

17 Right-click **Suggested Iterative Solver (GMRES with Direct Precon.) ()** and choose **Direct Preconditioner**.

18 In the **Settings** window for **Direct Preconditioner**, locate the **General** section.

19 From the **Solver** list, choose **PARDISO**.

20 Locate the **Hybridization** section. In the **Preconditioner variables** list, choose **Acoustic pressure (comp1.p)**, **Acoustic pressure (comp1.p2)**, **Temperature variation (comp1.T)**, **Lagrange multiplier (spatial frame) (comp1.ta.slw1.lm_tau)**, **Lagrange multiplier (spatial frame) (comp1.ta.slw2.lm_tau)**, **Acoustic velocity (spatial frame) (comp1.u)**, **Displacement field (comp1.u2)**, **Electric potential (comp1.V)**, and **Terminal voltage (comp1.es.term1.V0_ode)**.

21 Under **Preconditioner variables**, click **Delete**.

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

RESULTS

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

Sector 3D 1

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

2 Right-click **Results>Datasets** and choose **More 3D Datasets>Sector 3D**.

3 In the **Settings** window for **Sector 3D**, locate the **Symmetry** section.

4 In the **Number of sectors** text field, type 12.

5 From the **Sectors to include** list, choose **Manual**.

6 In the **Number of sectors to include** text field, type 12.

7 From the **Transformation** list, choose **Rotation and reflection**.

8 Click **Plot**.

9 Right-click **Sector 3D 1** and choose **Duplicate**.

Sector 3D 2

1 In the **Model Builder** window, click **Sector 3D 2**.

2 In the **Settings** window for **Sector 3D**, locate the **Symmetry** section.

3 In the **Number of sectors** text field, type 12.

4 In the **Number of sectors to include** text field, type 9.

5 In the **Start sector** text field, type -2.

6 Click **Plot**.

Frequency Response

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

2 In the **Settings** window for **1D Plot Group**, type `Frequency Response` in the **Label** text field.

Octave Band 1

1 In the **Frequency Response** toolbar, click **More Plots** and choose **Octave Band**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `es.V0_1`.

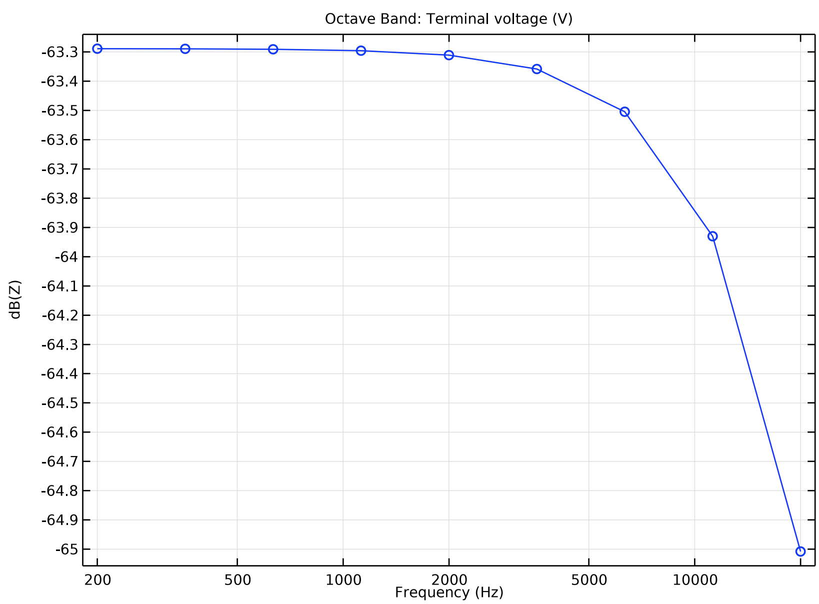
5 Click **Replace Expression**.

6 In the **Amplitude reference** text field, type `1/sqrt(2)`.

7 Locate the **Plot** section. From the **Quantity** list, choose **Continuous power spectral density**.

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 In the **Frequency Response** toolbar, click **Plot**.



Acoustic Pressure (ta.p_t+acpr.p_t)

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 (ta.p_t+acpr.p_t)` in the **Label** text field.

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

TA

1 Right-click **Acoustic Pressure (ta.p_t+acpr.p_t)** and choose **Surface**.

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

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

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

5 Click **OK**.

6 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

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

ACPR

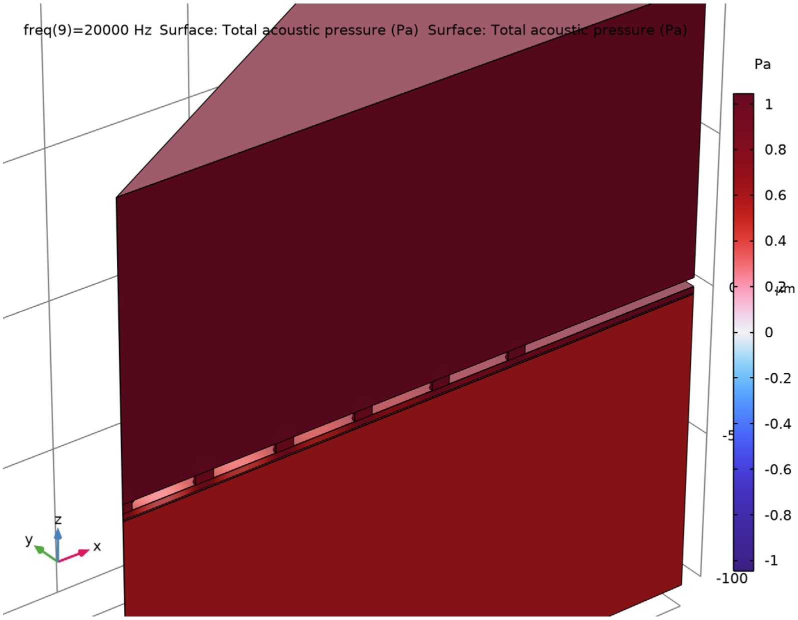
1 In the **Model Builder** window, right-click **Acoustic Pressure (ta.p_t+acpr.p_t)** and choose **Surface**.

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

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

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **TA**.

5 In the **Acoustic Pressure (ta.p_t+acpr.p_t)** toolbar, click **Plot**.



Acoustic Velocity (ta.v_rms)

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 (ta.v_rms)` in the **Label** text field.

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

Surface 1

1 Right-click **Acoustic Velocity (ta.v_rms)** and choose **Surface**.

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

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

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

5 In the **Acoustic Velocity (ta.v_rms)** toolbar, click **Plot**.