Probe Tube Microphone

Introduction

It is often not possible to insert a normal microphone directly into the sound field being measured. The microphone may be too big to fit inside the measured system, such as for in-the-ear measurements for hearing aid fitting. The size of the microphone may also be too large compared to the wavelength, so that it disturbs the acoustic field. In these cases, a probe tube may be attached to the microphone case (see the sketch in Figure 1) in order to distance the microphone from the measurement point. In this model, the effect on the microphone's sensitivity due to the addition of this small probe will be investigated, see Ref. 1 (Chapter 4, pp. 160–162) for further details.

This is a time-dependent model of a generic probe tube microphone setup consisting of an external acoustic domain, an elastic probe tube, and the cavity in front of the microphone diaphragm; see the sketch in Figure 1. The probe tube, modeled using the Pipe Acoustics, Transient physics interface, is connected to two separate 3D pressure acoustics domains, leading to a fully coupled acoustics simulation.

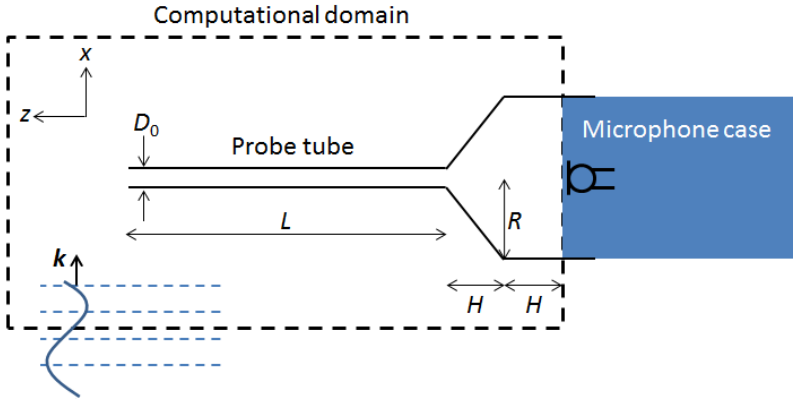


Figure 1: Sketch of the probe tube microphone setup, consisting of a probe tube of length L and outer diameter D_0. The tube is connected to a cavity (with the indicated dimensions) in front of the microphone diaphragm. An external sound field hits the probe tube with wave vector **k**.

Note: This model requires the Acoustics Module and the Pipe Flow Module.

Model Definition

The probe tube microphone geometry and the computational domain considered in this model are depicted schematically in Figure 1. From right to left, the model consists of the cavity in front of the microphone, which takes the form of a cylinder of radius R and height H connected to a cone of bottom radius R and top radius D_0. D_0 is also the outer diameter of the probe tube of length L. The probe is made of an elastic material with inner diameter $2d_r$ = 2 mm and wall thickness d_w = 0.3 mm. The tube material is set to have a Young's modulus of 0.1 GPa and a Poisson's ratio of 0.4.

Table 1 lists the model parameters,

TABLE 1: MODEL PARAMETERS

VARIABLE	VALUE	DESCRIPTION
f	500 Hz	Incident wave frequency
T	$1/f$	Incident wave period
ω	$2\pi f$	Incident wave angular frequency
c_0	343 m/s	Speed of sound at 20 ^{0}C and 1 atm
λ_0	c_0/f	Wave length of incident wave
k	$2\pi/\lambda_0$	Wave number of incident wave
L	20 mm	Probe tube length
R	5 mm	Microphone casing radius
H	5 mm	Cavity height
d_r	1 mm	Pipe inner radius
d_w	0.3 mm	Pipe wall thickness
D_0	$2d_r + 2d_w = 2.6$ mm	Probe tube outer diameter
kD_0	$k \cdot D_0 = 0.02$	Wave number times tube diameter
d_{visc}	0.1 mm	Viscous boundary layer thickness at 500 Hz
dt	$0.2 \cdot (L/10)/c_0$	CFL time step for quadratic elements

A sinusoidal wave moving in the positive x direction is incident on the probe tube microphone. The wave with an amplitude of 1 Pa is given by

 ©2012 COMSOL

$$p_{\text{in}} = 1\,\text{Pa} \cdot \sin(\omega t - kx) \quad (1)$$

where the angular frequency, ω, and the wave number, k, are given in Table 1.

Modeled using the Pipe Acoustics, Transient interface, the long probe tube is treated as 1D structure. This assumption is valid as long as the interaction between the probe tube and the incoming sound field can be neglected. This, in turn, requires that $k \cdot D_0 << 1$, which is true for the current choice of parameters because $k \cdot D_0 = 0.02$; see Table 1.

Another important assumption is that there are no significant thermal and viscous boundary losses inside the probe tube. Also this assumption holds for the current setup, where the incident field is a monochromatic wave. At a driving frequency of 500 Hz the viscous boundary layer thickness inside the probe tube is of the order 0.1 mm, which is 10 times smaller than the internal radius d_{r}. The thickness of the boundary layer is proportional to $1/\sqrt{f}$.

Because the diaphragm is not a fully rigid structure, assume it to have a resistive loss given by the impedance $Z = 100 \cdot 10^6$ Ns/m^5. This value is of the order of magnitude observed in common condenser microphones.

Results and Discussion

An important parameter of a probe tube extension to a microphone is the relation between the pressure at the tip and the pressure at the diaphragm (as measured by the microphone). This transfer function is necessary to calibrate the measurement system. The pressure at the probe's tip and the pressure at the diaphragm are shown in Figure 2. After an initial transient, the solution becomes periodic after about 4 ms. Hereafter, the system has a gain of about 1.4 and a phase shift equal to $\phi = \Delta t \cdot \omega = 0.1\ \text{ms} \cdot 2\pi f = 9°$. The gain and phase shift depend on the frequency content of the applied signal—in this case a pure harmonic tone of 500 Hz.

A full transfer function could be obtained, for example, by using a Gaussian pulse as the incident signal and then performing a Fourier transform of the signal. This requires resolving the frequency content with an appropriate mesh that would become very dense. Moreover, the model assumptions might not hold for the full range of frequencies.

Figure 3 depicts the pressure distribution in the xz-plane at the end of the time interval shown in Figure 2.

©2012 COMSOL

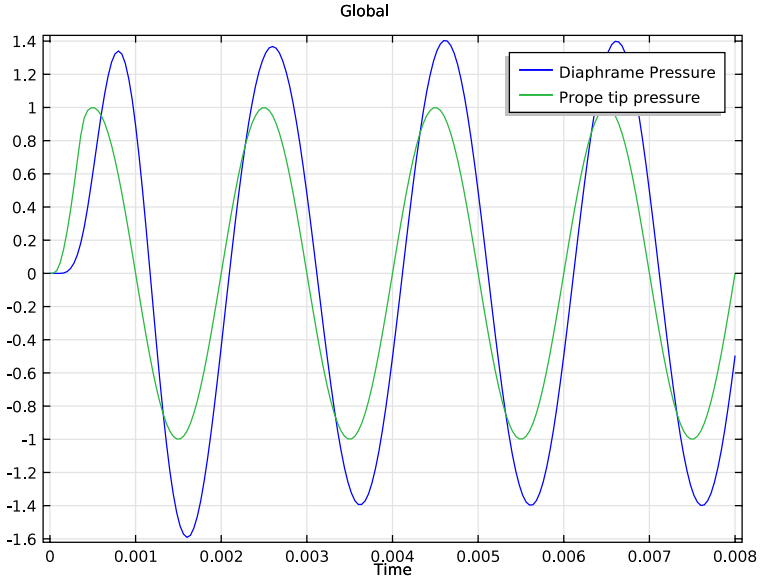


Figure 2: Pressure as function of time at the probe's tip (green line) and at the microphone diaphragm (blue line).

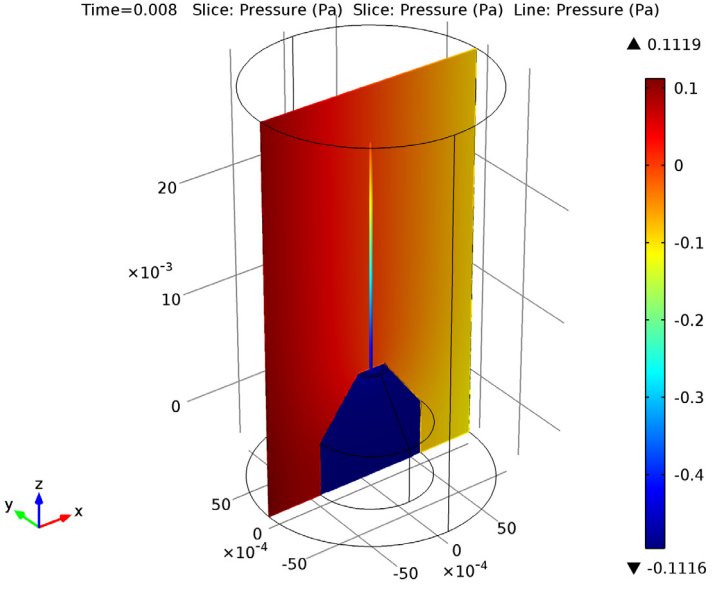


Figure 3: Pressure distribution in the xz-plane at time t = 8 ms.

©2012 COMSOL

Notes About the COMSOL Implementation

COUPLING BETWEEN 1D AND 3D DOMAINS

To couple the 1D pipe acoustic domain with the two 3D pressure acoustic domains requires some manual steps. First of all, note that the dependent variables are p and u in the pipe domain, p_2 in the external air domain, and p_3 in the microphone cap.

At the tip of the probe tube, set the pressure inside the tube equal to the pressure at the outer domain

$$p = p_2 \tag{2}$$

COMSOL Multiphysics treats this type of Dirichlet boundary condition as a bidirectional constraint. This means that the pressure fields are coupled in both directions, so that the constraint only needs to be stated on one of the physics. To see this, choose **Show>Advanced Physics Options** and then expand the **Constraint Settings** panel. The default is **Bidirectional, symmetric**.

Continuity also needs to be imposed at the interface between the probe tube and the cone that tops the volume covering the microphone. This amounts to setting the pressure in the tube equal to the mean pressure on the connecting boundary as well as setting the normal acceleration on the boundary equal to the time derivative of the tangential velocity in the tube:

$$\begin{aligned} p &= \frac{1}{A}\int p_3 dS \qquad \text{in tube} \\ a_n &= \mathbf{a}\cdot\mathbf{n} = -\frac{du}{dt} \qquad \text{on boundary} \end{aligned} \tag{3}$$

The first condition needs to be a (unidirectional) Dirichlet condition while the other is a Neumann condition on the pressure domain.

TRANSIENT SOLVER ADJUSTMENTS

Because the model is transient, use the time-dependent solver. For the solution to converge correctly and to help the solving procedure, take the following steps:

1. Use a ramp function to slowly step up the external actuation of the model. This is done in the model by defining the ramp function `rm1` that smoothly grows from 0 to 1 over the first one sixth of a period.

©2012 COMSOL

2 The maximal time step taken by the solver, $dt_{\max}$, needs to be such that any fluctuations in the solution do not travel more than 20% of the mesh in one time step (for quadratic elements). This condition is referred to as the CFL condition:

$$dt_{\max} = \frac{0.2 \cdot h_{\min}}{c} \tag{4}$$

Finally, note that solving a transient model can be quite time consuming. This model solves in around 5 to 15 minutes depending on the hardware.

Reference

1. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

Model Library path: `Acoustics_Module/Tutorial_Models/probe_tube_microphone`

Modeling Instructions

MODEL WIZARD

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

2 Click **Next**.

3 In the **Add physics** tree, select **Acoustics>Acoustic-Structure Interaction>Pipe Acoustics, Transient (patd)**.

4 Click **Add Selected**.

5 In the **Add physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.

6 Click **Add Selected**.

7 In the **Add physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.

8 Click **Add Selected**.

9 Click **Next**.

10 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Time Dependent**.

11 Click **Finish**.

 ©2012 COMSOL

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 `probe_tube_microphone_parameters.txt`.

GEOMETRY 1

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

Use the wire frame rendering to more easily see and access faces and lines inside the geometry.

Cone 1

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

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

3 In the **Bottom radius** edit field, type `R`.

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

5 From the **Specify top size using** list, choose **Radius**.

6 In the **Top radius** edit field, type `dr+dw`.

Cylinder 1

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

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

3 In the **Radius** edit field, type `R`.

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

5 Locate the **Position** section. In the **z** edit field, type `-H`.

Polygon 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **More Primitives>Polygon**.

2 In the **Polygon** settings window, locate the **Coordinates** section.

3 In the **x** edit field, type `0 0`.

4 In the **y** edit field, type `0 0`.

5 In the **z** edit field, type `H H+L`.

Cylinder 2

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

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

3 In the **Radius** edit field, type `10[mm]`.

4 In the **Height** edit field, type `35[mm]`.

5 Locate the **Position** section. In the **z** edit field, type `-H`.

6 Click the **Build Selected** button.

Now, create a work plane and draw a circle on top of the cone representing the inner part of the probe tube.

Work Plane 1

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

2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **cone1**, select Boundary 4 only.

Circle 1

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

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

3 In the **Radius** edit field, type `dr`.

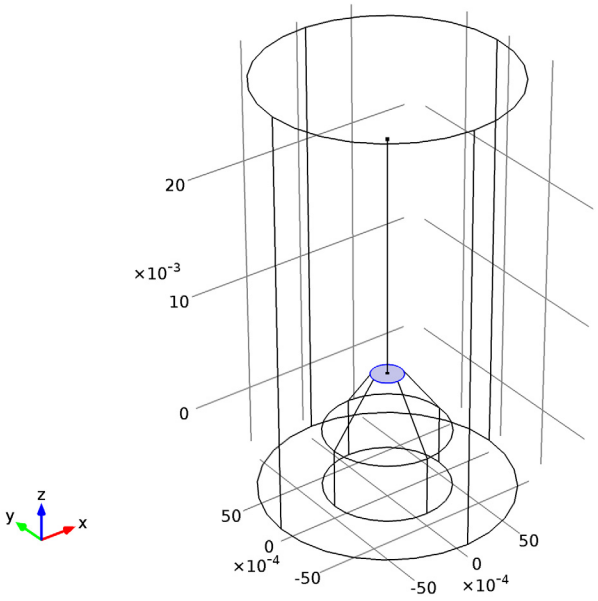
4 Click the **Build All** button.

5 Click twice the **Zoom Extents** button on the Graphics toolbar.

 ©2012 COMSOL

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

The geometry should look like the figure below.



MATERIALS

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

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Air**.

4 Right-click and choose **Add Material to Model** from the menu.

Air

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

2 In the **Material Browser** window, locate the **Materials** section.

3 Clear the selection in the tree.

4 In the **Material Browser** window, locate the **Materials** section.

5 In the tree, select **Built-In>Air**.

6 Right-click and choose **Add Material to Model** from the menu.

Air (2)

1 In the **Model Builder** window, under **Model 1>Materials** click **Air (2)**.

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

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

4 Select Edge 25 only.

DEFINITIONS

Ramp 1

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

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

3 In the **Location** edit field, type `0.2`.

4 Select the **Cutoff** check box.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** edit field, type `0.4`.

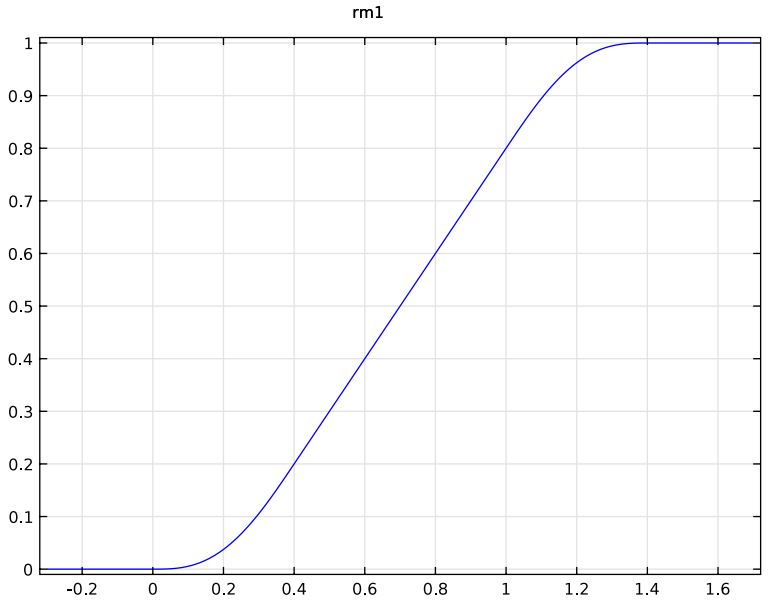
6 Select the **Smooth at start** check box.

7 Select the **Smooth at cutoff** check box.

 ©2012 COMSOL

8 Click the **Plot** button.

The ramp function should look like the figure below.



Average 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Average**.

2 In the **Average** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `aveop`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 12 only.

Average 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Average**.

2 In the **Average** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `aveop_mic`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 7 only.

Integration 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `intop`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Select Point 13 only.

Integration 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `intop_tip`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Select Point 14 only.

Proceed to set up the physics of the problem. Select the domains where the different physics are applied and add the appropriate boundary conditions to couple these.

PIPE ACOUSTICS, TRANSIENT

1 In the **Model Builder** window, under **Model 1** click **Pipe Acoustics, Transient**.

2 In the **Pipe Acoustics, Transient** settings window, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Select Edge 25 only.

Pipe Properties 1

1 In the **Model Builder** window, under **Model 1>Pipe Acoustics, Transient** click **Pipe Properties 1**.

2 In the **Pipe Properties** settings window, locate the **Pipe Shape** section.

3 From the associated list, choose **Round**.

4 In the d_i edit field, type `2*dr`.

5 Locate the **Pipe Model** section. From the **Pipe model** list, choose **Anchored at one end**.

6 From the E list, choose **User defined**. In the associated edit field, type `0.1[GPa]`.

 ©2012 COMSOL

7 From the v list, choose **User defined**. In the associated edit field, type `0.4`.

8 From the Δw list, choose **User defined**. In the associated edit field, type `dw`.

Pressure 1

1 In the **Model Builder** window, right-click **Pipe Acoustics, Transient** and choose **Pressure**.

2 Select Point 14 only.

3 In the **Pressure** settings window, locate the **Pressure** section.

4 In the p edit field, type `p2`.

This is the boundary condition given in Equation 2.

Pressure 2

1 In the **Model Builder** window, right-click **Pipe Acoustics, Transient** and choose **Pressure**.

2 Select Point 13 only.

3 In the **Pressure** settings window, locate the **Pressure** section.

4 In the p edit field, type `aveop(p3)`.

This is the first boundary condition given in Equation 3.

PRESSURE ACOUSTICS, TRANSIENT

1 In the **Model Builder** window, under **Model 1** click **Pressure Acoustics, Transient**.

2 In the **Pressure Acoustics, Transient** settings window, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

5 Right-click **Model 1>Pressure Acoustics, Transient** and choose the boundary condition **Cylindrical Wave Radiation**.

Cylindrical Wave Radiation 1

1 Select Boundaries 1, 2, 13, and 18 only.

2 In the **Cylindrical Wave Radiation** settings window, locate the **Cylindrical Wave Radiation** section.

3 Specify the r_{axis} vector as

0	x

0	y
1	z

4 Right-click **Model 1>Pressure Acoustics, Transient>Cylindrical Wave Radiation 1** and choose the boundary condition **Incident Pressure Field**.

Incident Pressure Field 1

1 In the **Incident Pressure Field** settings window, locate the **Incident Pressure Field** section.

2 In the p_i edit field, type `sin(omega*t-k*x)*rm1(omega*t-k*x)`.

This is the definition of the incident plane wave given in Equation 1 multiplied with the smoothing ramp function you defined earlier.

PRESSURE ACOUSTICS, TRANSIENT 2

1 In the **Model Builder** window, under **Model 1** click **Pressure Acoustics, Transient 2**.

2 In the **Pressure Acoustics, Transient** settings window, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 2 and 3 only.

5 Right-click **Model 1>Pressure Acoustics, Transient 2** and choose the boundary condition **Impedance**.

Impedance 1

1 Select Boundary 7 only.

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

3 In the Z_i edit field, type `100e6[N*s/m^5]*R^2*pi`.

Normal Acceleration 1

1 In the **Model Builder** window, right-click **Pressure Acoustics, Transient 2** and choose the boundary condition **Normal Acceleration**.

2 Select Boundary 12 only.

3 In the **Normal Acceleration** settings window, locate the **Normal Acceleration** section.

4 In the a_n edit field, type `-intop(d(u,t))`.

This is the second boundary condition given in Equation 3.

 ©2012 COMSOL

MESH 1

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

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 `lambda0/10`.
5 In the **Resolution of narrow regions** edit field, type `2`.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Free Tetrahedral 1**.
2 In the **Free Tetrahedral** settings window, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 2 and 3 only.

Size 1

1 Right-click **Model 1>Mesh 1>Free Tetrahedral 1** and choose **Size**.
2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 12 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7 In the associated edit field, type `dr/3`.
8 Click the **Build Selected** button.

Free Tetrahedral 2

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

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Tetrahedral 2** and choose **Size**.
2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 25 only.

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

6 Click the **Custom** button.

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

8 In the associated edit field, type `L/10`.

9 Click the **Build All** button.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,T/50,4*T)`.

The data will be recorded for 4 periods (4T) with a sampling of 50 points per period.

Solver 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 1** node, then click **Time-Dependent Solver 1**.

3 In the **Time-Dependent Solver** settings window, click to expand the **Time Stepping** section.

4 Select the **Maximum step** check box.

5 In the associated edit field, type `2*dt`.

In order to speed up the calculation the CFL constraint on the time step is relaxed slightly by allowing steps of size `2*dt`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1** right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

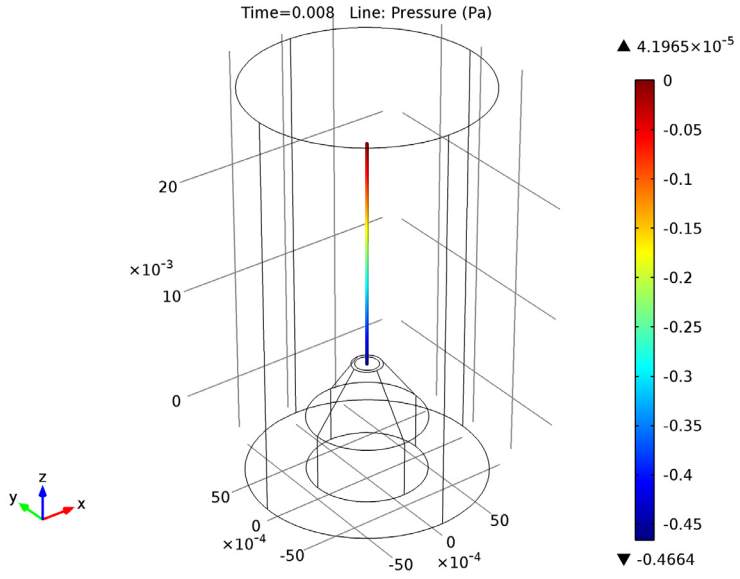
Now, solve the model and proceed to look at the results. First, look at the default plots and then create two custom plot groups. Note that, solving the model may take between 5 and 15 minutes.

7 In the **Settings** window, click **Compute All**.

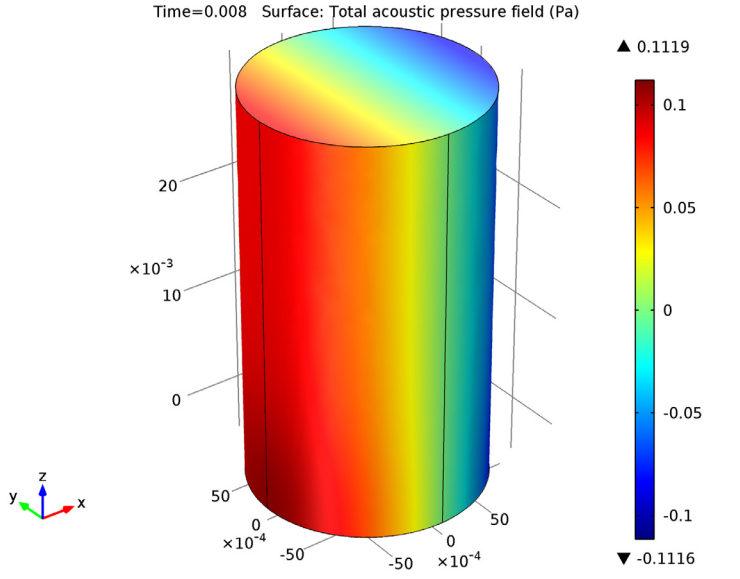
 ©2012 COMSOL

RESULTS

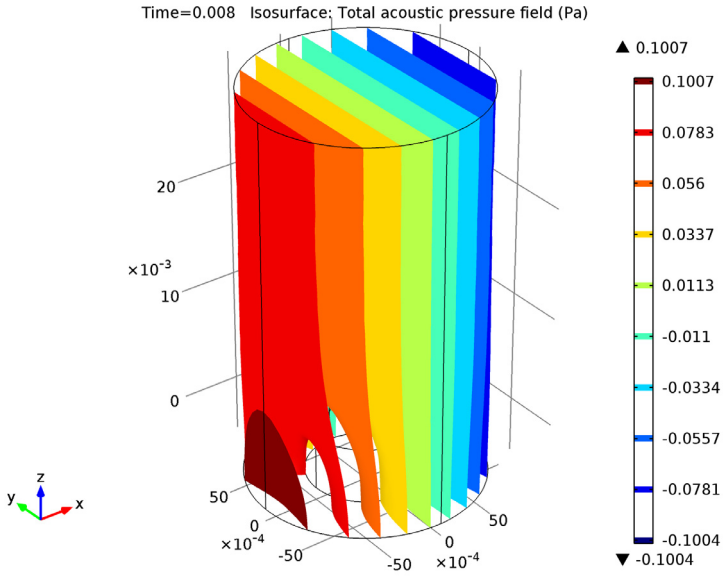
Pressure



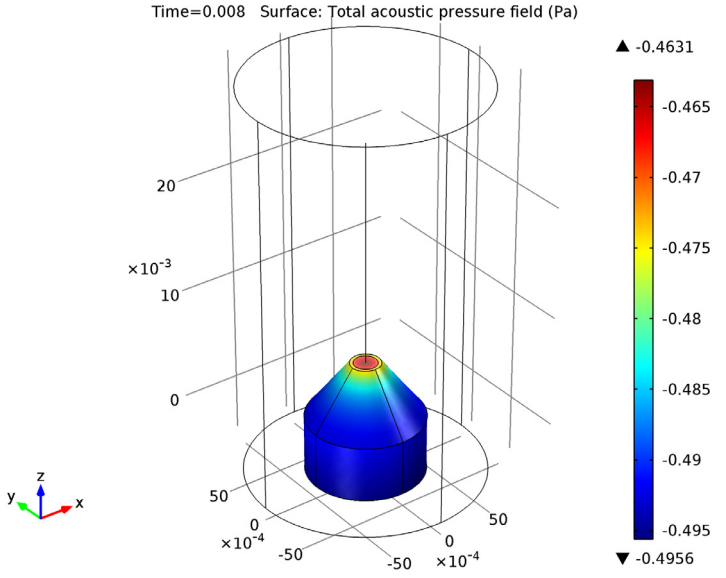
Acoustic Pressure (actd)



Acoustic Pressure, Isosurfaces (actd)

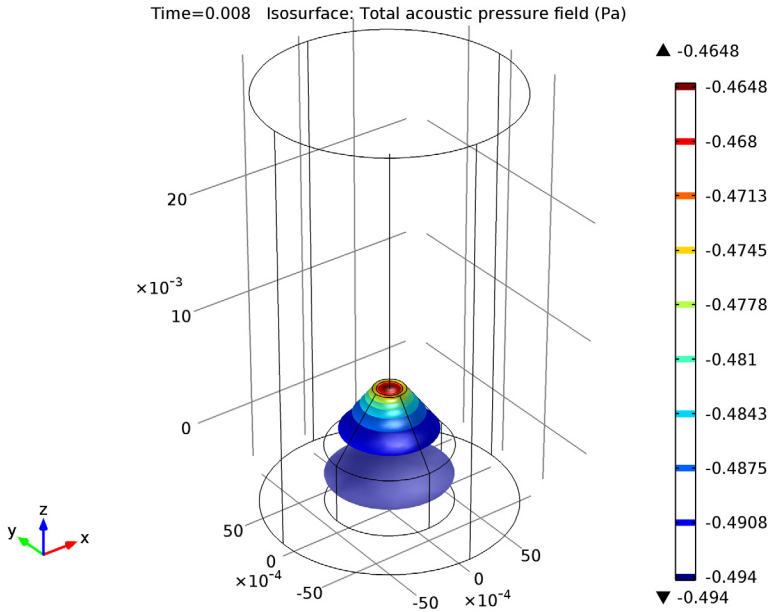


Acoustic Pressure (actd2)



©2012 COMSOL

Acoustic Pressure, Isosurfaces (actd2)



Follow the steps below to reproduce the plot in Figure 3.

3D Plot Group 7

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 7** and choose **Slice**.

3 In the **Slice** settings window, locate the **Expression** section.

4 In the **Expression** edit field, type `p2`.

5 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

6 In the **Planes** edit field, type `1`.

7 In the **Model Builder** window, right-click **3D Plot Group 7** and choose **Slice**.

8 In the **Slice** settings window, locate the **Expression** section.

9 In the **Expression** edit field, type `p3`.

10 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

11 In the **Planes** edit field, type `1`.

12 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

13 In the **Model Builder** window, right-click **3D Plot Group 7** and choose **Line**.

14 In the **Line** settings window, locate the **Coloring and Style** section.

15 From the **Line type** list, choose **Tube**.

16 In the **Tube radius expression** edit field, type `2*patd.dh`.

17 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

18 Click the **Plot** button.

19 In the **Model Builder** window, right-click **3D Plot Group 7** and choose **Rename**.

20 Go to the **Rename 3D Plot Group** dialog box and type `Cross Sections` in the **New name** edit field.

21 Click **OK**.

The figure should look like the one depicted in Figure 3.

1D Plot Group 8

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

2 In the **Model Builder** window, under **Results** right-click **1D Plot Group 8** and choose **Global**.

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

4 In the table, enter the following settings:

EXPRESSION	UNIT	DESCRIPTION
`aveop_mic(p3)`	`Pa`	`Diaphragm Pressure`
`intop_tip(p)`	`Pa`	`Probe Tip Pressure`

5 In the **Model Builder** window, click **1D Plot Group 8**.

6 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

7 Select the **x-axis label** check box.

8 In the associated edit field, type `Time (s)`.

9 Select the **y-axis label** check box.

10 In the associated edit field, type `Pressure (Pa)`.

11 Click the **Plot** button.

12 In the **Model Builder** window, right-click **1D Plot Group 8** and choose **Rename**.

13 Go to the **Rename 1D Plot Group** dialog box and type `Pressure Profiles` in the **New name** edit field.

14 Click **OK**.

The figure should look like the one in Figure 2.

 ©2012 COMSOL