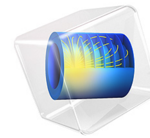
Created in COMSOL Multiphysics 5.6



Hollow Cylinder

Introduction

Fluid acoustics coupled to structural objects (vibroacoustics), such as membranes or plates, represents an important application area in many engineering fields. Some examples are loudspeakers, acoustic sensors, nondestructive impedance testing, and medical ultrasound diagnostics of the human body. Through the use of the *Acoustic - Solid Interaction, Frequenccy Domain* multiphysics interface, this model demonstrates how fluid acoustics can be easily coupled to structures.

Model Definition

This model provides a general demonstration of an acoustic fluid phenomenon in 3D coupled to a solid object. In this study, the solid object is a capped, hollow aluminum cylinder filled with and immersed in water. The acoustic waves created by a source inside the cylinder impact the cylinder walls.

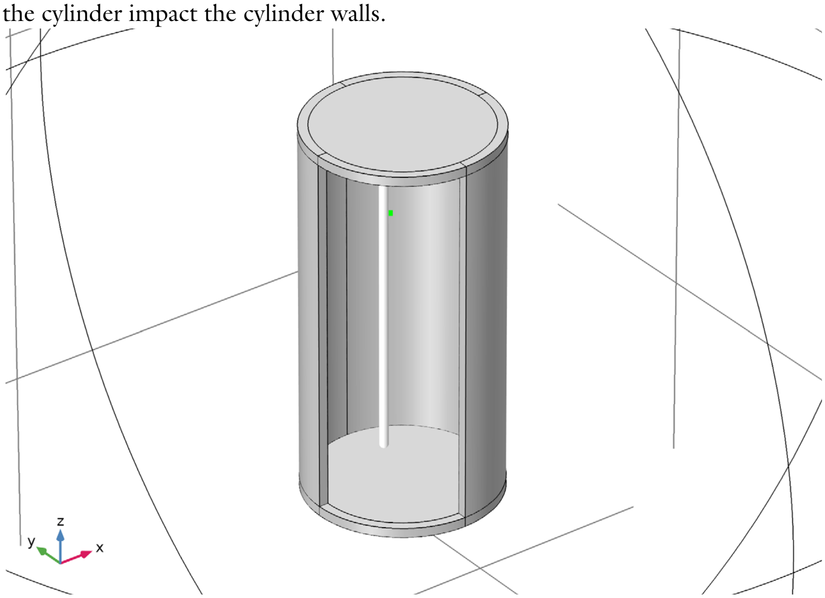


Figure 1: A hollow aluminum cylinder is immersed in water. The white line inside the cylinder indicates the line source, and the tiny sphere next to the line shows the position of the point source. The simulation domain is bounded by a large sphere.

Figure 1 illustrates the aluminum cylinder immersed in water. The cylinder is 2 cm in height and has an outer diameter of 1 cm. The thickness of its walls is 0.5 mm.

The water-filled acoustic domain outside the cylinder is truncated to a sphere with a reasonably large diameter. In two different versions of the model, the system is driven either by a line source coinciding with the axis of the cylinder and located entirely within the cylinder or by a point source in the interior of the cylinder. The frequency studied is 60 kHz, that is, in the ultrasound region. The fluid and solid domains are coupled using the built-in **Acoustic-Structure Boundary** multiphysics coupling. The coupling ensures a fully two-way coupled analysis of the problem.

DOMAIN AND BOUNDARY CONDITIONS

Water Domain

For harmonic sound waves, in the frequency domain, the model solves the Helmholtz equation defined in the Pressure Acoustics, Frequency Domain interface.

Solid Domain

You calculate the harmonic stresses and strains inside the solid cylinder walls using the same equation formulation as a frequency response analysis in the 3D Solid Mechanics interface. The material data comes from the built-in database for Aluminum 3003-H18.

Outer Perimeter

On the outer spherical perimeter of the water domain (Figure 1), use the predefined **Spherical Wave Radiation** condition. This boundary condition allows a spherical wave to travel out of the system, giving only slight reflections for the non-spherical components of the wave. For mathematical details on the radiation boundary condition, see the *Acoustics Module User's Guide*.

Cylinder-Water Interface

The model uses the **Acoustic-Solid Interaction, Frequency Domain** multiphysics interface, in which the **Acoustic-Structure Boundary** multiphysics coupling condition automatically apply to all acoustic-structure boundaries. The condition ensures that the acoustic pressure is added as a load on the solid and that the solid normal acceleration is added as a source on the acoustic domain.

EDGE AND POINT SETTINGS

In the two cases considered, the sound waves are generated by either a point monopole source or a line source. A line source along the z-axis defines the following right-hand-side source term to the Helmholtz equation:

$$\frac{4\pi}{\rho}\frac{1}{L_{\mathrm{edge}}}\sqrt{\frac{P_{\mathrm{rms}}c\rho}{2\pi}}\delta^{(2)}(\mathbf{r})$$

Here P_{rms} is the free space reference rms power per unit length of a line source of length L_{edge} placed in a homogeneous medium extending to infinity. Furthermore, $\delta^{(2)}(\mathbf{r})$ is the

Dirac delta function in two dimensions, $\mathbf{r}$ denoting the projection of the position vector onto the *xy*-plane.

A monopole point source of rms power P_{rms} located at the point $\mathbf{R} = \mathbf{R}_0$ in an infinite homogeneous space defined the following source term

$$\frac{4\pi}{\rho}\sqrt{\frac{P_{\mathrm{rms}}\,c\rho}{2\pi}}\,\delta^{(3)}(\mathbf{R}-\mathbf{R}_0)$$

where $\delta^{(3)}(\mathbf{R})$ is the Dirac delta function in three dimensions. Any type of confinement results in higher power usage.

Results

Figure 2 shows the resulting deformations and local sound pressure level from a computation with a line source.

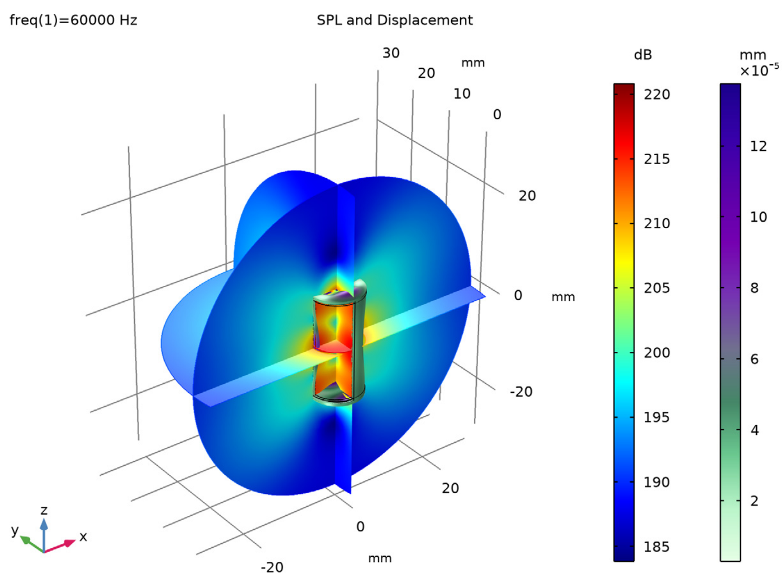


Figure 2: Sound-pressure level plot (dB) of the acoustic waves in the coupled problem, using a line source inside the cylinder. The surface of the cylinder shows its deformation (mm).

In Figure 3, you see the solution from an off-centered point. Note that the deformation color scale is different, implying that this source gives deformations that at least locally are greater than those from the line source.

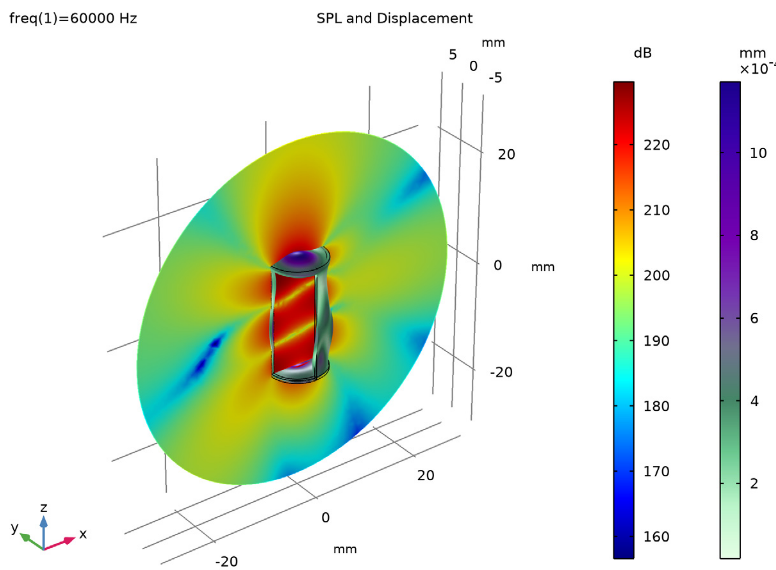


Figure 3: Sound-pressure level (dB) and structural deformations (mm) with a point source inside the cylinder. Some of the surfaces are hidden to reveal the pressure distribution inside the cylinder.

Application Library path: `Acoustics_Module/Vibrations_and_FSI/hollow_cylinder`

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>Acoustic-Structure Interaction>Acoustic-Solid Interaction, Frequency Domain**.
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 In the table, enter the following settings:

Name	Expression	Value	Description
edgeL	1.7[cm]	0.017 m	Length of line source
cylh	20[mm]	0.02 m	Height of the cylinder
cylr	5[mm]	0.005 m	Radius of the cylinder
cylth	0.5[mm]	5E-4 m	Thickness of the cylinder
f0	60[kHz]	60000 Hz	Frequency of the analysis
c0	1482[m/s]	1482 m/s	Speed of sound in water
lam0	c0/f0	0.0247 m	Wavelength at f0

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

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 `cylr`.
4 In the **Height** text field, type `cylh`.
5 Locate the **Position** section. In the **z** text field, type `-cylh/2`.

6 Click to expand the **Layers** section. Select the **Layers on bottom** check box.

7 Select the **Layers on top** check box.

8 In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`cylth`

9 Click **Build All Objects**.

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

With wireframe rendering, you now see the contours of both cylinders.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **y** text field, type `(cylr-cylth)/3`.

6 In the **z** text field, type `-edgeL/2`.

7 Locate the **Endpoint** section. In the **y** text field, type `(cylr-cylth)/3`.

8 In the **z** text field, type `edgeL/2`.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Point**.

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

3 In the **x** text field, type `(cylr-cylth)/3`.

4 In the **y** text field, type `(cylr-cylth)*2/3`.

5 In the **z** text field, type `(cylh/2-cylth)/3`.

Sphere 1 (sph1)

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

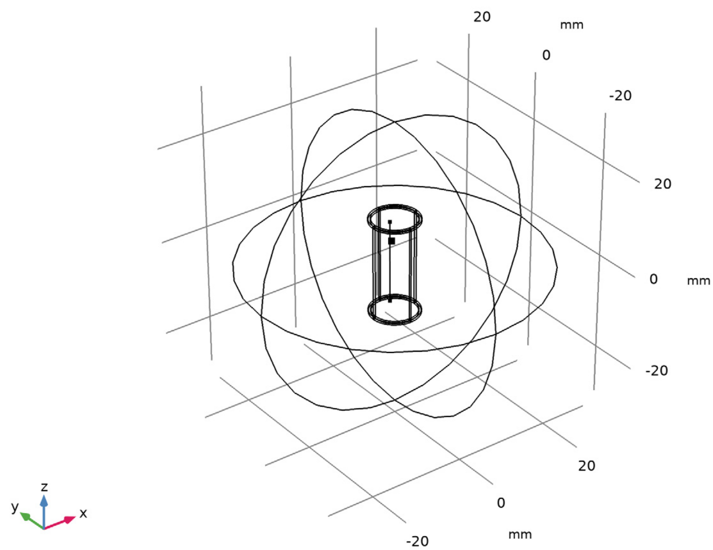
2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `1.5*cylh`.

4 Click **Build All Objects**.

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

The geometry should look like this.



DEFINITIONS

Solid

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

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

3 Select Domains 2–8 and 10–16 only.

Water

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

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

3 Select Domains 1 and 9 only.

Line Source

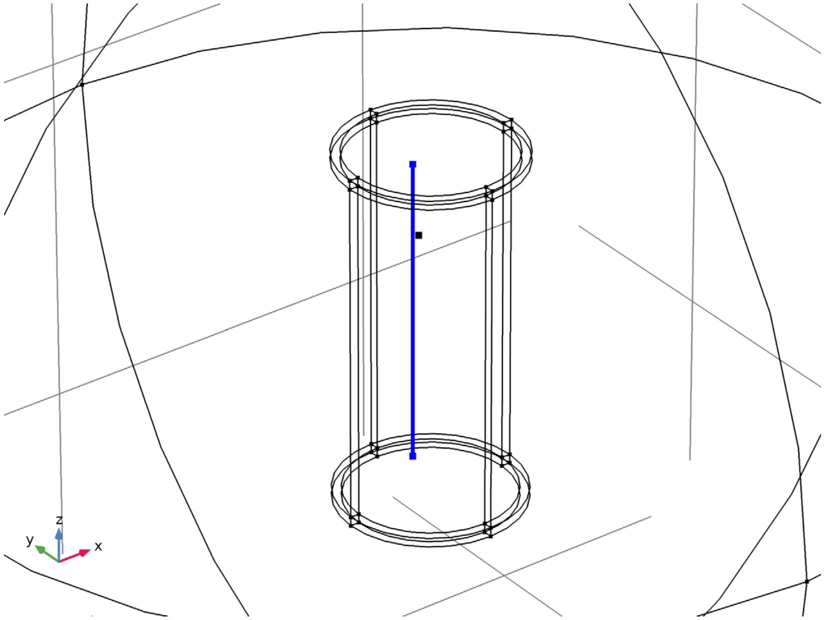
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Edge 56 only.

This is the line that will act as a source. The selection should look like this.



Point Source

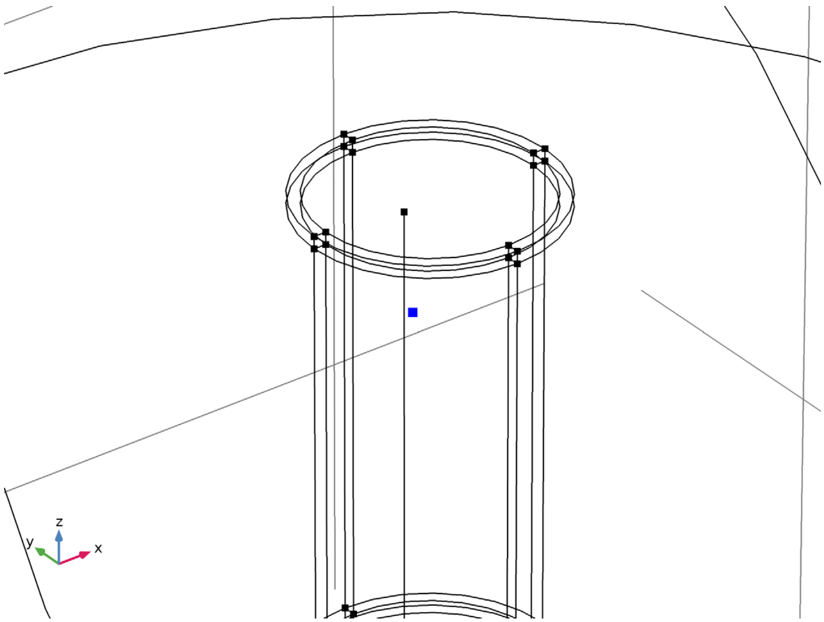
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Point 32 only.

Point 32 lies inside the cylinder and is the only point that is not connected to any edges. The selection should look like this.



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>Water, liquid**.

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

5 In the tree, select **Built-in>Aluminum 3003-H18**.

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

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

MATERIALS

Water, liquid (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat1)**.

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

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

Aluminum 3003-H18 (mat2)

1 In the **Model Builder** window, click **Aluminum 3003-H18 (mat2)**.

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

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

By default, all domains are using the Pressure Acoustics model, which holds for fluids. Use the Water selection to specify where Pressure Acoustics is applied.

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 **Water**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

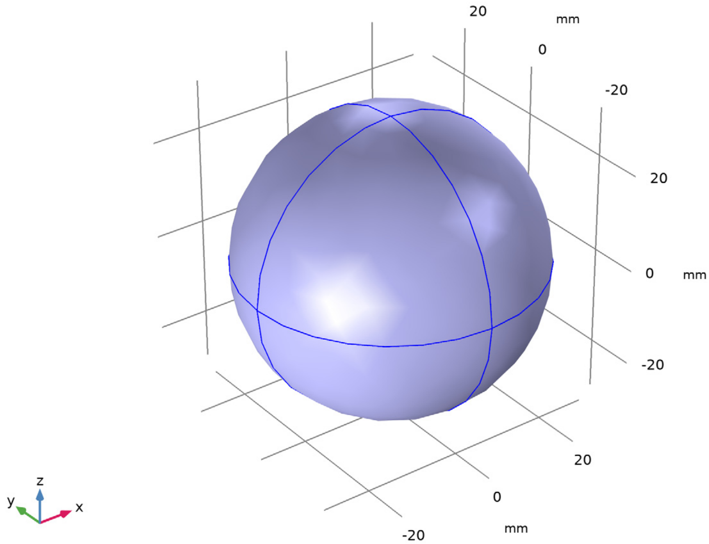
The water reference pressure is 1 μPa. This does not affect the physics, but only serves as a reference in the definition of the sound pressure level.

Spherical Wave Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spherical Wave Radiation**.

2 Select Boundaries 1–4, 32, 33, 47, and 48 only.

These are the boundaries that will have a radiation condition on them. The selection should look like this.



Line Source 1

1. In the **Physics** toolbar, click **Edges** and choose **Line Source**.
2. In the **Settings** window for **Line Source**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Line Source**.
4. Locate the **Line Source** section. From the **Type** list, choose **Power**.
5. In the P_{rms} text field, type `0.5`.

Monopole Point Source 1

1. In the **Physics** toolbar, click **Points** and choose **Monopole Point Source**.
2. In the **Settings** window for **Monopole Point Source**, locate the **Point Selection** section.
3. From the **Selection** list, choose **Point Source**.
4. Locate the **Monopole Point Source** section. From the **Type** list, choose **Power**.
5. In the P_{rms} text field, type `0.5`.

   You will select either the line source or the point source under the study by enabling **Modify physics tree and variables for study step**.

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 **Solid**.

MULTIPHYSICS

Now is a good time to inspect the **Multiphysics** node. Notice that the Acoustic-Structure Boundary feature is automatically added to all boundaries, but only applied to the boundaries that have water on one side and the solid on the other.

MESH 1

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

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

3 From the **Sequence type** list, choose **User-controlled mesh**.

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 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lam0/5`.

This value, combined with the default choice of second-order elements, makes sure that the rule-of-thumb minimum of 10-12 degrees of freedom per wavelength for the solution to be reliable is satisfied.

5 In the **Minimum element size** text field, type `lam0/10`.

6 Click **Build Selected**.

Free Quad 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Quad**.

2 Select Boundaries 25–27, 30, 45, and 53 only.

The model uses some free quad mesh to control what will be the sweep direction in the solid.

Size 1

1 Right-click **Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

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

5 In the associated text field, type `cylth*3`.

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

7 In the associated text field, type `cylth`.

8 Click **Build Selected**.

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 From the **Selection** list, choose **Solid**.

Distribution 1

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

With the sweep operation, the 2D mesh previously created will be sweep through the domains. In general it is recommended to have at least two quadratic elements through the thickness.

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

3 In the **Number of elements** text field, type `2`.

4 Click **Build All**.

STUDY 1 - LINE SOURCE

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

2 In the **Settings** window for **Study**, type `Study 1 - Line Source` in the **Label** text field.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - Line Source** 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`.

Now, disable the point source by modifying the physics tree.

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

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)>Monopole Point Source 1**.

6 Click **Disable**.

7 In the **Home** toolbar, click **Compute**.

The default plots include all the surfaces of the model, obscuring the cylinder. A selection added to the dataset will make sure that the plots shows the requested variables on the selection.

RESULTS

Study 1 - Line Source/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1 - Line Source/Solution 1 (sol1)**.

Selection

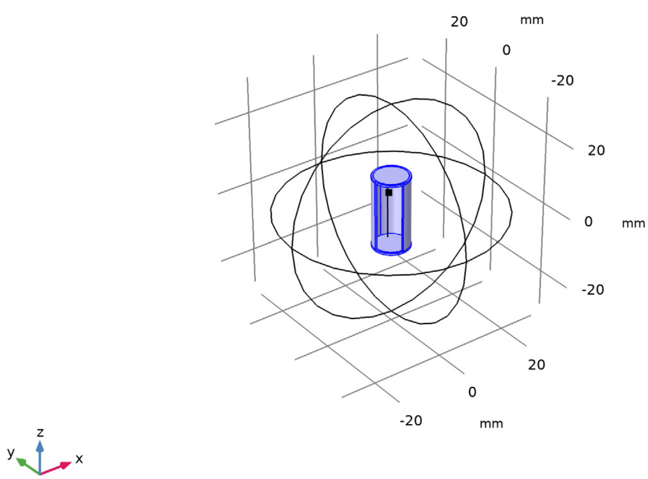
1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Boundaries 5–10, 12–24, 26–31, 34–46, and 49–64 only.

The selection should look like this.



Four default plots are created after running this model. In the following steps a additional plots will be added to show the space distribution of the acoustic field.

Slice 1

1 In the **Model Builder** window, right-click **Acoustic Pressure (acpr)** and choose **Slice**.

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

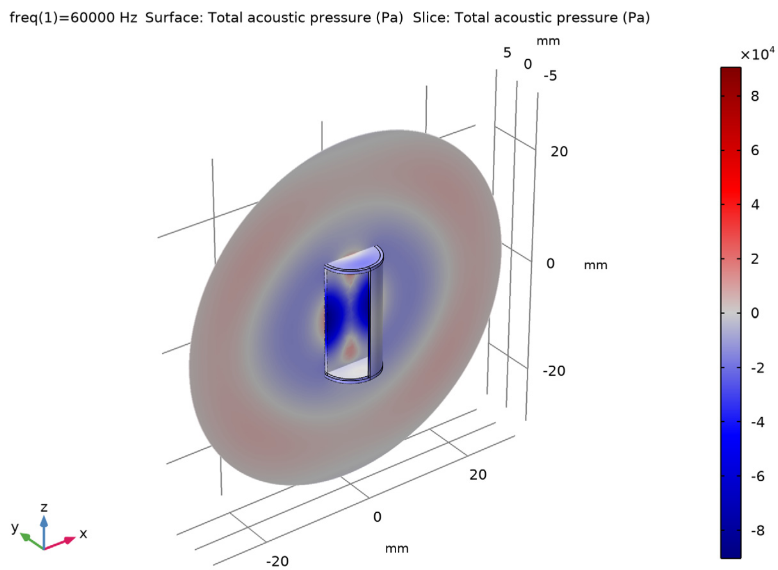
3 From the **Plane** list, choose **ZX-planes**.

4 In the **Planes** text field, type `1`.

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

6 In the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

The plot should look like this.



Slice 1

1 In the **Model Builder** window, right-click **Sound Pressure Level (acpr)** and choose **Slice**.

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

3 In the **Expression** text field, type `acpr.Lp_t`.

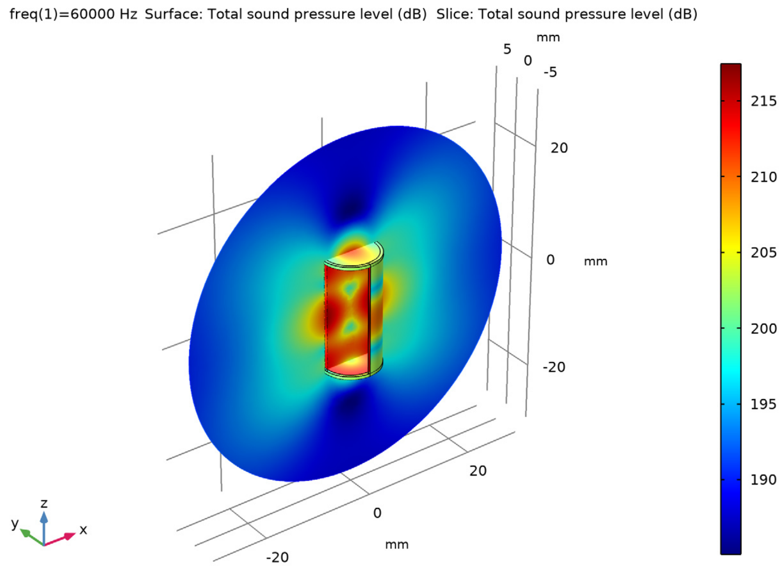
4 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

5 In the **Planes** text field, type `1`.

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 In the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

The plot should look like this.



Filter 1

1 In the **Model Builder** window, expand the **Results>Acoustic Pressure, Isosurfaces (acpr)** node.

2 Right-click **Isosurface 1** and choose **Filter**.

3 In the **Settings** window for **Filter**, locate the **Element Selection** section.

4 In the **Logical expression for inclusion** text field, type `x>0`.

Surface 1

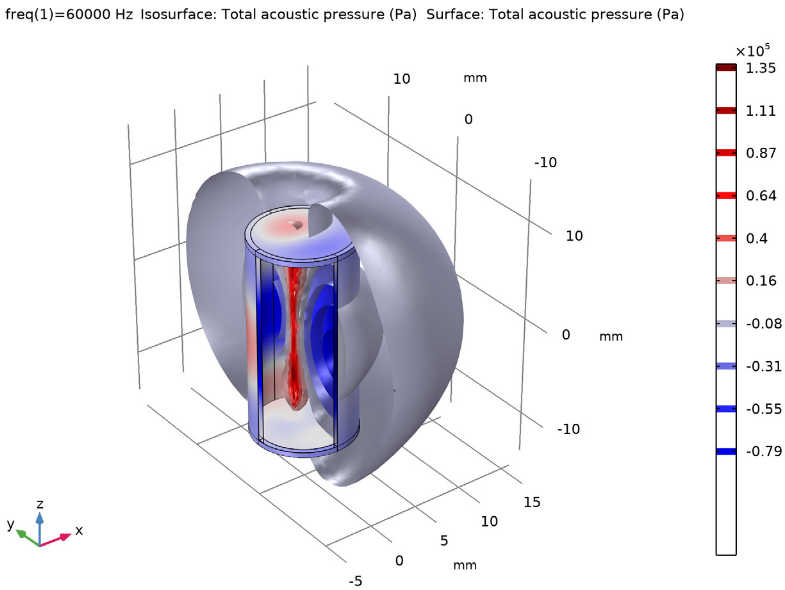
1 In the **Model Builder** window, right-click **Acoustic Pressure, Isosurfaces (acpr)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Isosurface 1**.

4 In the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.

The plot should look like this.



Create a new plot of the deformation and sound pressure level. Then proceed to solving the system again with a monopole point source.

Displacement/SPL - Line Source

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

2 In the **Settings** window for **3D Plot Group**, type `Displacement/SPL - Line Source` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `SPL and Displacement`.

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

Multislice 1

1 In the **Displacement/SPL - Line Source** toolbar, click **More Plots** and choose **Multislice**.

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

3 In the **Expression** text field, type `acpr.Lp_t`.

Add a filter to only show the SPL in the water domain outside of the hollow cylinder.

Filter 1

1 Right-click **Multislice 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `y>-(cylr-cylth)`.

Surface 1

1 In the **Model Builder** window, right-click **Displacement/SPL - Line Source** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.disp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraBorealis**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Displacement/SPL - Line Source** toolbar, click **Plot**.

Your plot should now look like Figure 2. The remaining instructions show you how to set up a second study and replace the line source with a point source, while disabling the line source.

Create a group with all plots to facilitate the navigation through the different results.

Acoustic Pressure (acpr), Acoustic Pressure, Isosurfaces (acpr), Displacement/SPL - Line Source, Sound Pressure Level (acpr), Stress (solid)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr)**, **Sound Pressure Level (acpr)**, **Acoustic Pressure, Isosurfaces (acpr)**, **Stress (solid)**, and **Displacement/SPL - Line Source**.

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

Plots - Line Source

In the **Settings** window for **Group**, type `Plots - Line Source` in the **Label** text field.

ADD STUDY

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

STUDY 2 - POINT SOURCE

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

2 In the **Settings** window for **Study**, type `Study 2 - Point Source` in the **Label** text field.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Point Source** 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 **Physics and variables selection** tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)>Line Source 1**.

6 Click **Disable**.

7 In the **Home** toolbar, click **Compute**.

The results for the second study will also include a selection.

RESULTS

Study 2 - Point Source/Solution 2 (sol2)

In the **Model Builder** window, under **Results>Datasets** click **Study 2 - Point Source/Solution 2 (sol2)**.

Selection

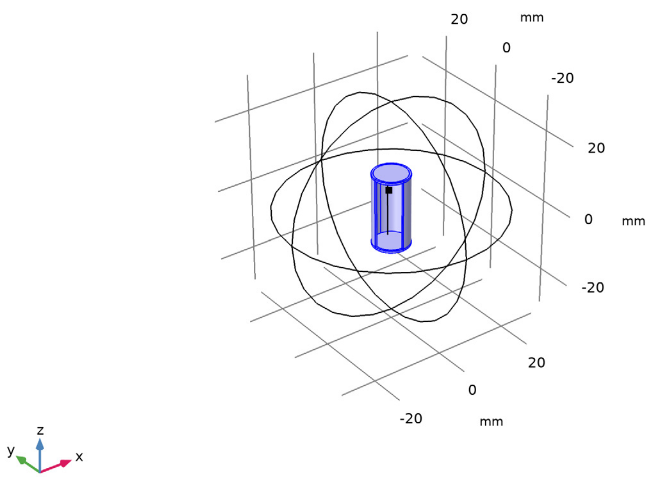
1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Boundaries 5–10, 12–24, 26–31, 34–46, and 49–64 only.

The selection should look like this.



Displacement/SPL - Point Source

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement/SPL - Point Source` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `SPL and Displacement`.

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

6 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Point Source/ Solution 2 (sol2)**.

Slice 1

1 Right-click **Displacement/SPL - Point Source** and choose **Slice**.

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

3 From the **Plane** list, choose **ZX-planes**.

4 In the **Planes** text field, type `1`.

5 Locate the **Expression** section. In the **Expression** text field, type `acpr.Lp_t`.

Surface 1

1 In the **Model Builder** window, right-click **Displacement/SPL - Point Source** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.disp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraBorealis**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Displacement/SPL - Point Source** toolbar, click **Plot**.

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

You can see the resulting plot in Figure 3.

Create a group with all plots to facilitate the navigation through the different results.

Acoustic Pressure (acpr) 1, Acoustic Pressure, Isosurfaces (acpr) 1, Displacement/SPL - Point Source, Sound Pressure Level (acpr) 1, Stress (solid) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr) 1**, **Sound Pressure Level (acpr) 1**, **Acoustic Pressure, Isosurfaces (acpr) 1**, **Stress (solid) 1**, and **Displacement/SPL - Point Source**.

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

Plots - Point Source

In the **Settings** window for **Group**, type `Plots - Point Source` in the **Label** text field.