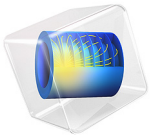


Model created in COMSOL Multiphysics 6.4

Sound Radiation from a Circular Duct with Flow

Introduction

Model of the sound radiation from a circular duct with uniform flow. The convected acoustic problem is described using the linearized potential flow equations solved in the frequency domain. The acoustic inlet is treated including higher order modes using the **Port** boundary condition.

Model Definition

The mode consists of a duct of radius a placed in a uniform axial flow with magnitude U_0. The duct is open at one end radiating sound into an infinite domain; see Figure 1. The setup mimics a very simplified and conceptual engine intake configuration. The single-mode radiation from the duct and the spatial radiation characteristics are computed. The modal content and the nonreflecting condition at the duct inlet are set up using **Port** boundary conditions. One mode is excited (incident) while all propagating outgoing modes are included at the port boundary. The model parameters and model setup is inspired by Ref. 1.

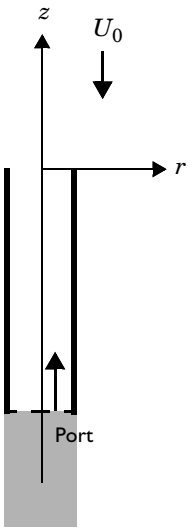


Figure 1: Sketch of the problem setup, showing a duct open at one end and excited at the port.

The model reproduces the pressure radiation pattern from Figure 8 (b) in Ref. 1, by using the duct radius 0.085 m, Mach number 0.12, and flow velocity parameters from Table 1 in Ref. 1. The excited mode is the $(m,n) = (1,0)$ mode.

Results and Discussion

The radiation pattern evaluated at 2 m from the duct entrance is depicted in Figure 2. Note that the SPL value evaluated in COMSOL is corrected to represent the peak value to match the result from Ref. 1. The pressure field in the (r,z)-plane (0 azimuthal angle) is depicted in Figure 3. The pressure field including the azimuthal component is depicted in Figure 4 and Figure 5. The sound pressure level (RMS based) is depicted in Figure 6 and Figure 7.

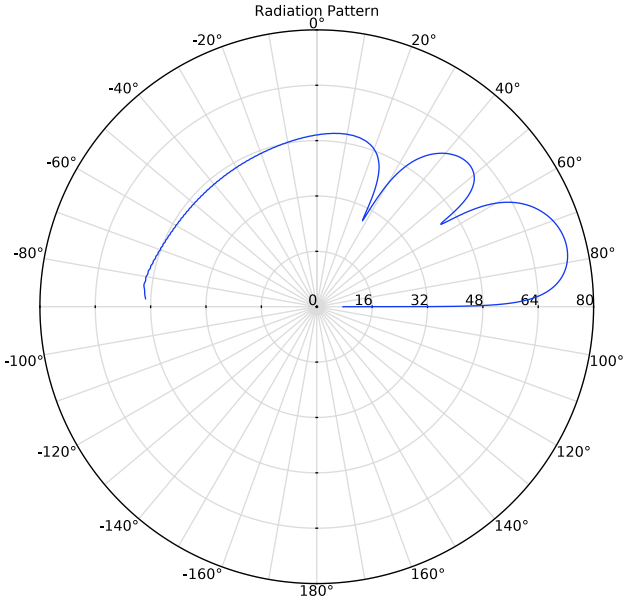


Figure 2: Radiation pattern evaluated at 2 m.

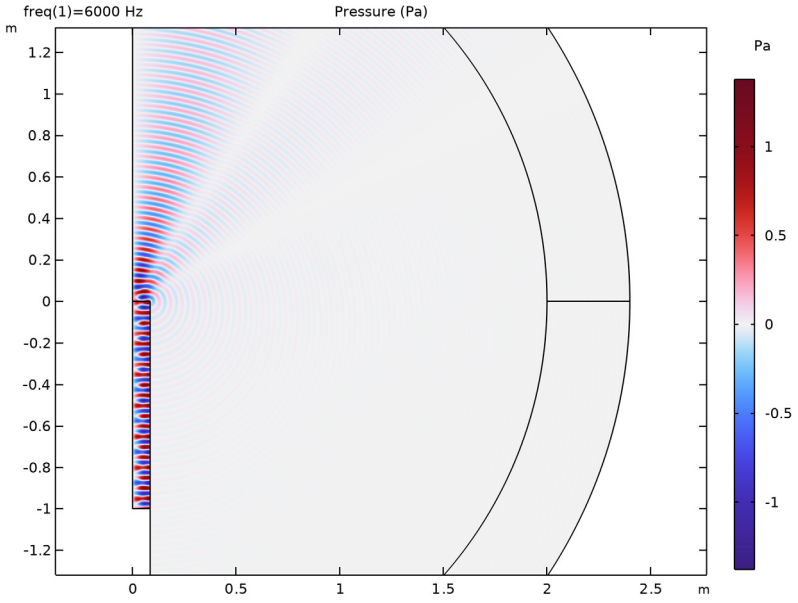


Figure 3: Acoustic pressure in the (r,z)-plane (0 azimuthal angle).

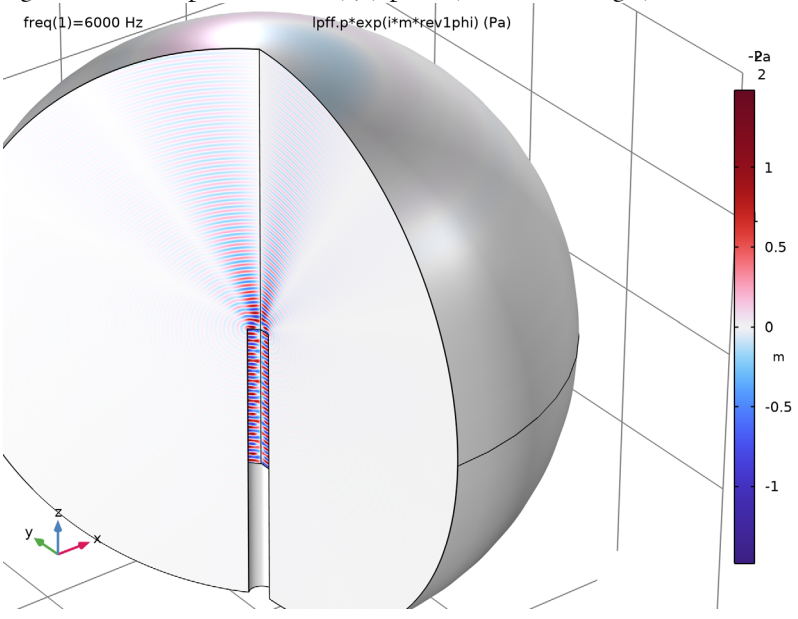


Figure 4: Acoustic pressure represented in 3D including the azimuthal component.

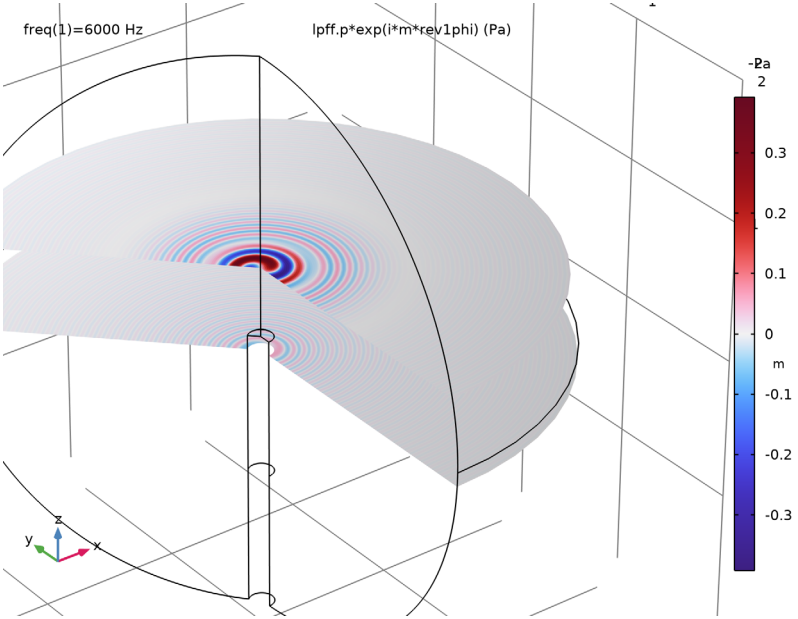


Figure 5: Acoustic pressure represented in two z-cut planes including the azimuthal component of the acoustic field.

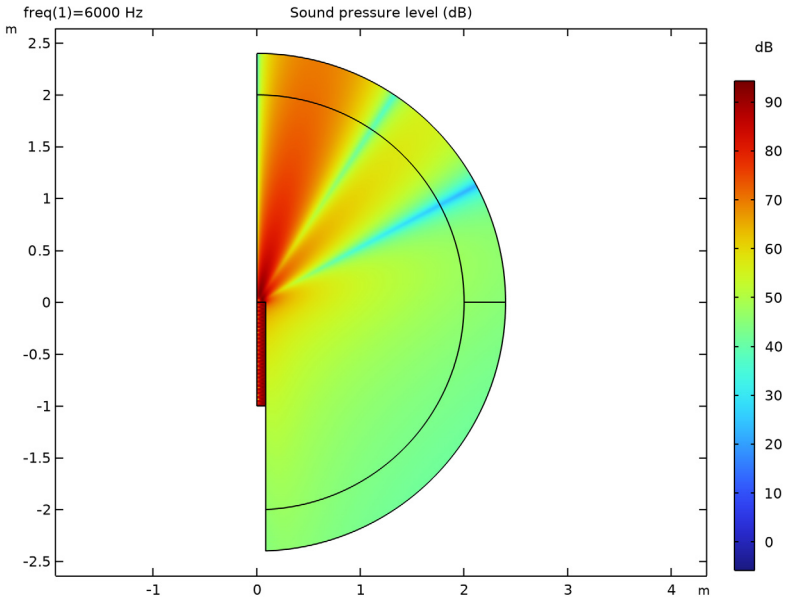


Figure 6: The sound pressure level evaluated in the (r,z)-plane.

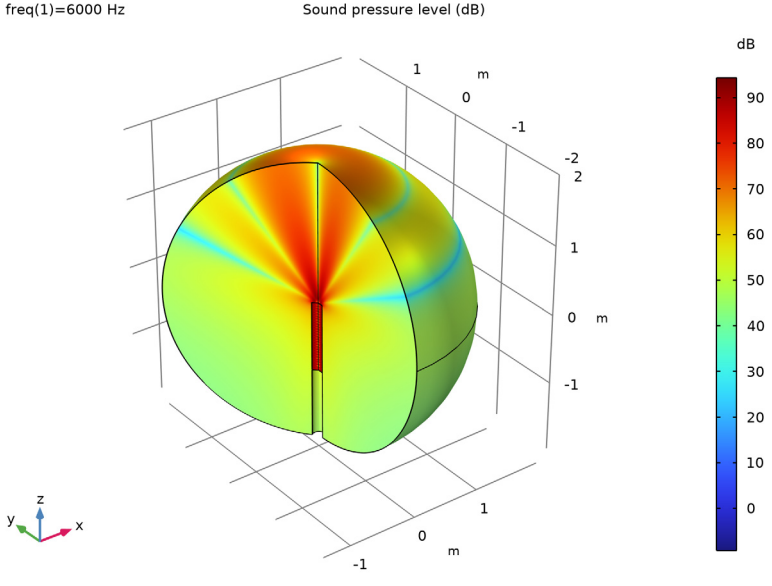


Figure 7: The sound pressure level evaluated in 3D.

Reference

1. C. Ford, A. Pereira, and C. Bailly, "Radiation of higher order modes from circular ducts with flow," *Acta Acustica*, vol. 7, 2023.
Article available at: acta-acustica.edpsciences.org/articles/aacus/full_html/2023/01/aacus220052/aacus220052.html. The model *Sound Radiation from a Circular Duct with Flow* reproduces the pressure radiation pattern from Figure 8 (b), by using the duct radius, Mach number, and flow velocity parameters from Table 1.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/sound_radiation_circular_duct`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Acoustics** > **Aeroacoustics** > **Linearized Potential Flow, Frequency Domain (lpff)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `sound_radiation_circular_duct_parameters.txt`.

GEOMETRY 1

Circle 1 (c1)

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

2 Right-click **Geometry 1** and choose **Circle**.

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

4 In the **Radius** text field, type `1.4*R0`.

5 In the **Sector angle** text field, type `180`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.2*R0
Layer 2	0.2*R0

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `a`.

4 In the **Height** text field, type `1.5*R0`.

5 Locate the **Position** section. In the **z** text field, type `-1.5*R0`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	L

7 Clear the **Layers on bottom** checkbox.

8 Select the **Layers on top** checkbox.

Union 1 (uni1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

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

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

4 On the object **uni1**, select Domains 1–4 only.

Form Union (fin)

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

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Basic Properties** > **Density**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties** > **Speed of Sound**.

6 Click **Add to Material**.

7 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho0	kg/m³	Basic
Speed of sound	c	c0	m/s	Basic

MESH 1

In this model, the mesh is set up manually. Proceed by directly adding the desired mesh component. The mesh includes boundary layers which are there to control and create a fine mesh near the edge of the pipe. The well resolved pipe corner as well as a good resolution of the wavelength in the model is necessary to get an accurate radiation pattern.

Mapped 1

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

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

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

4 Select Domain 1 only.

Distribution 1

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

2 Select Boundary 2 only.

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

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

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.

In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details, see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide* .

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `(c0-U0)/f0/6`.

5 In the **Minimum element size** text field, type `(c0-U0)/f0/10`.

Boundary Layers 1

1 In the **Mesh** toolbar, click  **Boundary Layers**.

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

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

4 Select Domain 1 only.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** checkbox.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

4 In the **Number of layers** text field, type `2`.

5 In the **Thickness adjustment factor** text field, type `3`.

Boundary Layers 2

1 In the **Mesh** toolbar, click  **Boundary Layers**.

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

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

4 Select Domain 1 only.

5 Locate the **Transition** section. Clear the **Smooth transition to interior mesh** checkbox.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundary 10 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

4 In the **Number of layers** text field, type `2`.

5 In the **Thickness adjustment factor** text field, type `3`.

Free Triangular 1

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

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

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

4 Select Domains 2, 3, and 6 only.

Size 1

1 Right-click **Free Triangular 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.

5 Select the **Maximum element size** checkbox. In the associated text field, type `(c0-U0)/f0/6`.

Mapped 2

In the **Mesh** toolbar, click  **Mapped**.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 6, 7, and 12 only.

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

4 In the **Number of elements** text field, type `20`.

5 Click  **Build All**.

Proceed to set up the physics. The **Port** conditions are used at the source to excite the desired mode (m,n)=(1,0), as well as absorb reflected acoustic modes. The reflected modes

are generated due to the impedance jump at the pipe opening. It is necessary to include the radial modes n=1 and n=2 also.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (LPFF)

In a 2D axisymmetric model the azimuthal mode number (m) is set at the physics interface level. Also choose the desired mode scaling. The power scaling will ensure that the absolute levels seen in the reference paper are achieved.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Frequency Domain (lpff)**.

2 In the **Settings** window for **Linearized Potential Flow, Frequency Domain**, locate the **Linearized Potential Flow Equation Settings** section.

3 In the *m* text field, type `m`.

4 Locate the **Global Port Settings** section. From the **Mode shape normalization** list, choose **Power normalization**.

Linearized Potential Flow Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** click **Linearized Potential Flow Model 1**.

2 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Model Input** section.

3 Specify the $\mathbf{u}_0$ vector as

-U0	z

Interior Sound Hard Boundary (Wall) 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 Select Boundary 10 only.

Port 1

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

2 Select Boundary 2 only.

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

4 From the **Type of port** list, choose **Circular**.

5 Locate the **Port Incident Mode Settings** section. From the **Incident wave excitation at this port** list, choose **On**.

6 From the **Define incident wave** list, choose **Mode scale**.

7 In the S^{in} text field, type 1.

Port 2

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

2 Select Boundary 2 only.

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

4 From the **Type of port** list, choose **Circular**.

5 Locate the **Port Mode Settings** section. In the n text field, type 1.

Port 3

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

2 Select Boundary 2 only.

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

4 From the **Type of port** list, choose **Circular**.

5 Locate the **Port Mode Settings** section. In the n text field, type 2.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 4 and 5 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 In the **PML scaling curvature parameter** text field, type 3.

STUDY 1

Step 1: Frequency Domain

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

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

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

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

5 In the **Settings** window for **Study**, type `Study 1 - Frequency Domain` in the **Label** text field.

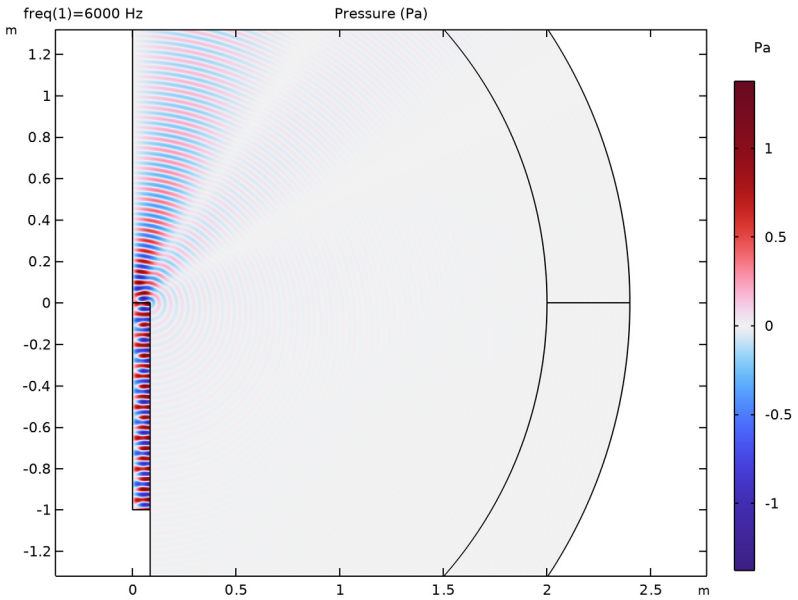
6 In the **Study** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (lpff)

Now, look at the default plots and make some modifications, like adding selections to the physical domain (excluding the PML).

1. In the **Settings** window for **2D Plot Group**, click to expand the **Selection** section.
2. From the **Geometric entity level** list, choose **Domain**.
3. Select Domains 1–3 and 6 only.
4. Select the **Apply to dataset edges** checkbox.
5. In the **Acoustic Pressure (lpff)** toolbar, click **Plot**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

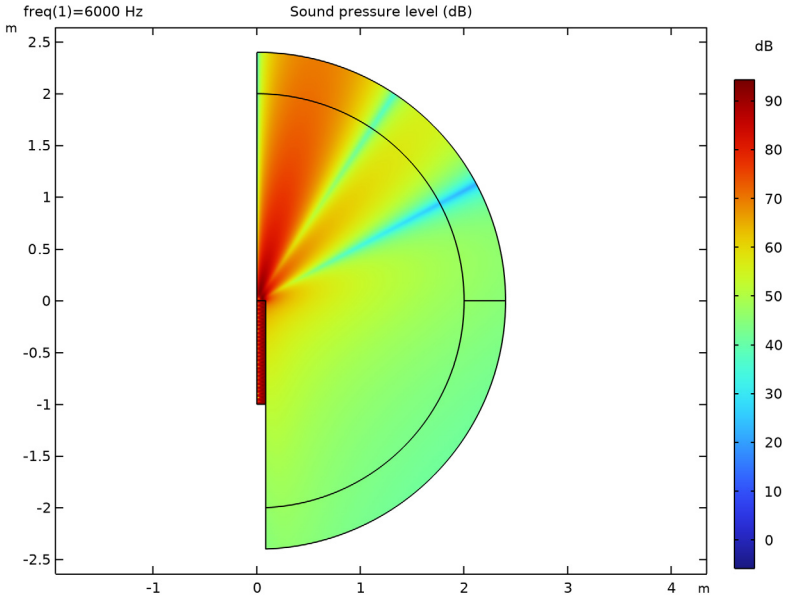


Sound Pressure Level (lpff)

1. In the **Model Builder** window, click **Sound Pressure Level (lpff)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1–3 and 6 only.
5. Select the **Apply to dataset edges** checkbox.

6 In the **Sound Pressure Level (lpff)** toolbar, click **Plot**.

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



Revolution 2D

In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Revolution 2D**.

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 **Domain**.

4 Select Domains 1 and 2 only.

Surface

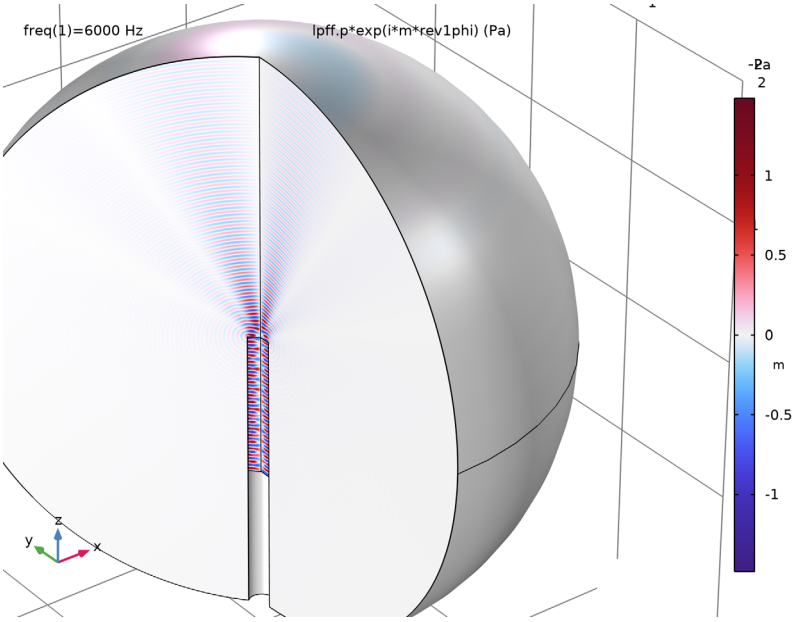
1 In the **Model Builder** window, expand the **Acoustic Pressure, 3D (lpff)** node, then click **Surface**.

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

3 In the **Expression** text field, type `lpff.p*exp(i*m*rev1phi)`.

4 In the **Acoustic Pressure, 3D (lpff)** toolbar, click **Plot**.

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

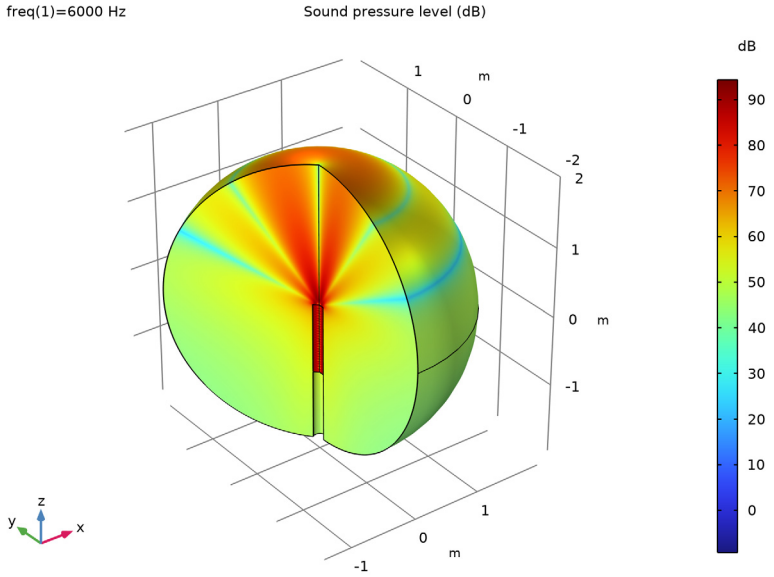


Sound Pressure Level, 3D (lpff)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level, 3D (lpff)**.

2 In the **Sound Pressure Level, 3D (lpff)** toolbar, click **Plot**.

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



Radiation Pattern

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

2 In the **Settings** window for **Polar Plot Group**, type `Radiation Pattern` in the **Label** text field.

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

4 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

5 In the **r maximum** text field, type 80.

6 Select the **Symmetric angle range** checkbox.

7 From the **Zero angle** list, choose **Up**.

8 From the **Rotation direction** list, choose **Clockwise**.

9 Locate the **Grid** section. Select the **Manual spacing** checkbox.

10 In the θ **spacing** text field, type 10.

11 In the **r spacing** text field, type 16.

Line Graph 1

1 In the **Radiation Pattern** toolbar, click **Line Graph**.

2 Select Boundaries 13 and 18 only.

Here, the extra term transforms form the RMS level to the peak level, this is the typical 3 dB correction.

3 In the **Settings** window for **Line Graph**, locate the **r-Axis Data** section.

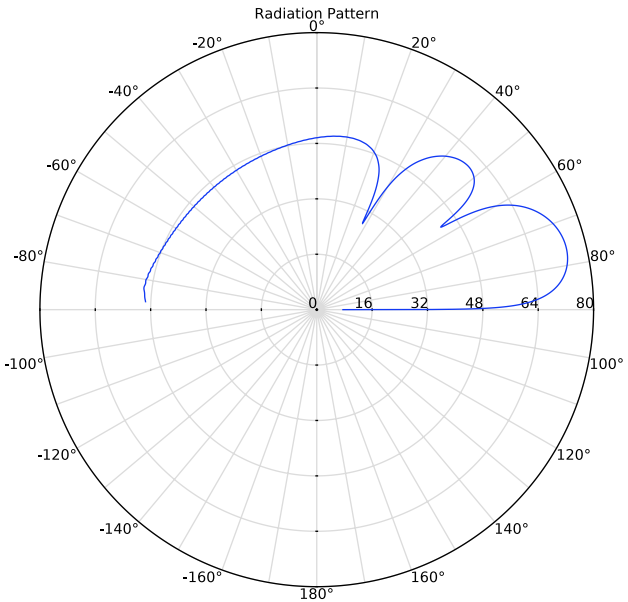
4 In the **Expression** text field, type `lpff.Lp+20*log10(sqrt(2))[dB]`.

Define the polar angle expression.

5 Locate the θ **Angle Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `atan2(z,r)`.

7 In the **Radiation Pattern** toolbar, click **Plot**.



Now, also create a 3D plot of the pressure just below and above the pipe entrance. For this create a new dataset where the selection is restricted to the outside air domain.

Revolution 2D 2

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

Selection

1 In the **Model Builder** window, expand the **Revolution 2D 2** node, then click **Selection**.

2 Select Domain 2 only.

Acoustic Pressure, 3D Slices

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, 3D Slices` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D 2**.

4 Locate the **Color Legend** section. Select the **Show units** checkbox.

Slice 1

1 Right-click **Acoustic Pressure, 3D Slices** and choose **Slice**.

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

3 In the **Expression** text field, type `lpff.p*exp(i*m*rev1phi)`.

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

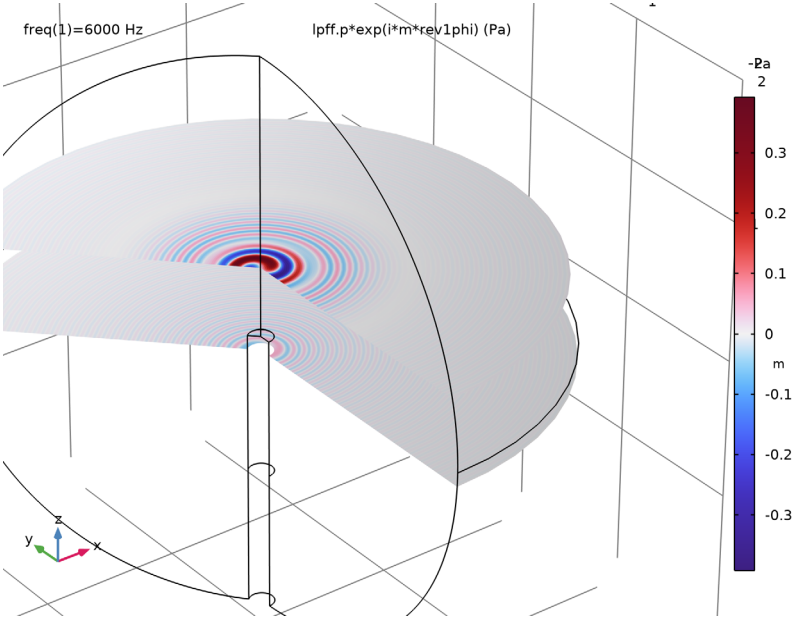
5 From the **Entry method** list, choose **Coordinates**.

6 In the **Z-coordinates** text field, type `-0.1 0.5`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

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

9 In the **Acoustic Pressure, 3D Slices** toolbar, click **Plot**.



Finally, add an evaluation group computing the power of the incident and reflected (outgoing) port modes.

Evaluation Group 1 - Port Mode Powers

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

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group 1 - Port Mode Powers` in the **Label** text field.

Global Evaluation 1

1 Right-click **Evaluation Group 1 - Port Mode Powers** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain** > **Ports** > **Port 1** > **lpff.port1.P_in - Power of incident mode - W**.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`lpff.port1.P_out`	`W`	`Power of outgoing mode`
`lpff.port2.P_out`	`W`	`Power of outgoing mode`
`lpff.port3.P_out`	`W`	`Power of outgoing mode`

4 In the **Evaluation Group 1 - Port Mode Powers** toolbar, click **Evaluate**.