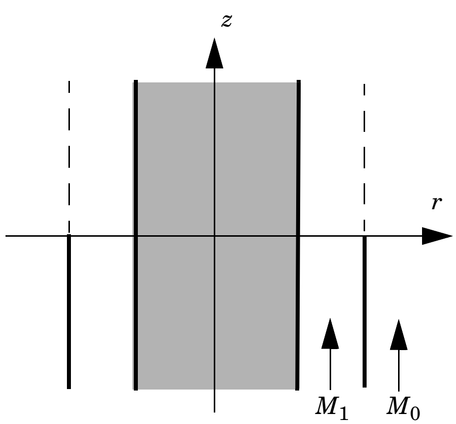
Jet Pipe

Introduction

This example models the radiation of fan noise from the annular duct of a turbofan aeroengine. When the jet stream excites the duct, a vortex sheet appears along the extension of the duct wall. In the model you calculate the near field on both sides of the vortex sheet. The background mean-flow is assumed to be well described by a potential flow, in this model a uniform flow. This means that, the acoustic field can be modeled by solving the linearized potentiality flow equations in the frequency domain.

Model Definition

The model is axisymmetric with the symmetry axis coinciding with the engine's centerline (gray area in the figure below). The flows both inside and outside the duct are uniform mean flows, they have a magnitude of M_1 and M_0, respectively. Because the flow velocities differ, a vortex sheet separates them (dashed line in the figure below).



Sketch of the turbofan motor.

The Linearized Potential Flow, Frequency Domain interface in the Acoustics Module describes acoustic waves in a moving fluid with the potential ϕ, for the local particle velocity as the basic dependent variable; see the chapter about aeroacoustics in the *Acoustics Module User's Guide* for details. The field equations are only valid when the background velocity field is irrotational, a condition that is not satisfied across a vortex sheet. As a consequence, the velocity potential is discontinuous across this sheet. To model this discontinuity, you use the **Vortex Sheet** boundary condition which is available on interior boundaries. The model is excited using the **Port** boundary condition.

THE VORTEX SHEET CONDITION

The **Vortex Sheet** boundary conditions on the two sides of the vortex sheet are defined as follows:

$$\left[\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi+\mathbf{u}_0\cdot\nabla\phi)\right)\right]_i = [\rho_0(i\omega+\mathbf{u}_0\cdot\nabla)w]_i \qquad i = \text{up, down}$$

$$p_{\text{up}} = p_{\text{down}} \qquad\qquad w_{\text{up}} = -w_{\text{down}}$$

In these equations, ω is the angular velocity, $\mathbf{u}_0$ is the mean-flow velocity, w is the outward normal displacement, ϕ is the velocity potential, and p is the pressure. The subscripts "up" and "down" refer to the two sides of the boundary.

The velocity normal to the vortex sheet is zero, which implies that the last two terms on the left-hand side of the condition vanishes. In the model the variables are made dimensionless. The velocities are divided by the speed of sound in air and the densities are divided by the density for air. For example, the model uses the Mach number $M = u_0/c_0$ as the mean flow velocity. This leads to the boundary conditions

$$(i\omega+M_{\text{up}}\nabla_{\text{T}})w = \frac{\partial\phi_{\text{up}}}{\partial n}$$

$$(i\omega+M_{\text{down}}\nabla_{\text{T}})w = \frac{\partial\phi_{\text{down}}}{\partial n}$$

$$p_{\text{up}} = p_{\text{down}} \qquad w_{\text{up}} = -w_{\text{down}}$$

where M denotes the transverse Mach number.

THE PORT CONDITION

The acoustic field inside the duct can be described as a sum of modes propagating in the duct and then radiating in the free space. This is discussed in section 2.1 in Ref. 1. In this example you study the radiated acoustic waves by exciting the system with a single mode source at a time. The source is introduced using the **Port** boundary condition with the built-in **Annular** port option. The port is nonreflecting for the components of the reflected acoustic field that consists of the same modal content. The acoustic field will be reflected due to the impedance change as the outlet of the duct. To get a full nonreflecting behavior several ports need to be added, one for each propagating mode in the system. The mode cutoff frequencies can be evaluated, as done in the first evaluation group in the model. The scattering coefficients associated with the ports are evaluated in the second evaluation group in the model.

The inlet sources defined using the Port condition. This example, like Ref. 1, uses the eigenmodes $(m,n) = (4,0)$, $(17,1)$, and $(24,1)$ as incident waves to the duct. In Figure 1 the three source mode pressure fields at the port are depicted in the revolved geometry, including the azimuthal mode contribution.

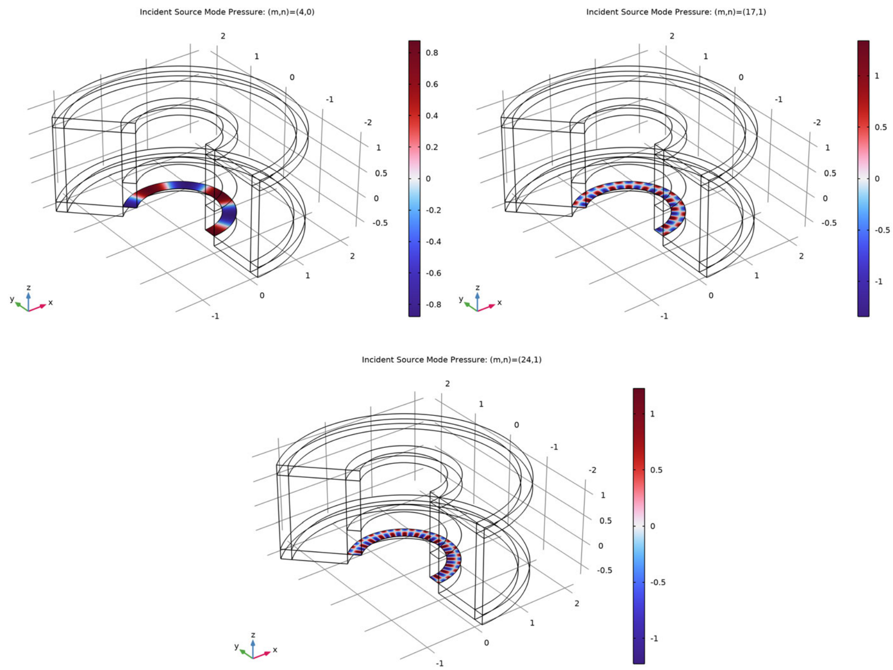


Figure 1: The incident mode pressures (m,n) = (4,0), (m,n) = (17,1), and (m,n) = (24,1), depicted in the revolved geometry including the azimuthal dependency.

The near-field pressure around the duct obtained by COMSOL Multiphysics can be compared to the results for the near field in Ref. 1. Figure 2 through Figure 6 show the near-field solution for a Mach number equal to $M_1 = 0.45$ in the pipe and $M_0 = 0.25$ on the outside. The figures show the pressure field for the three different source modes.

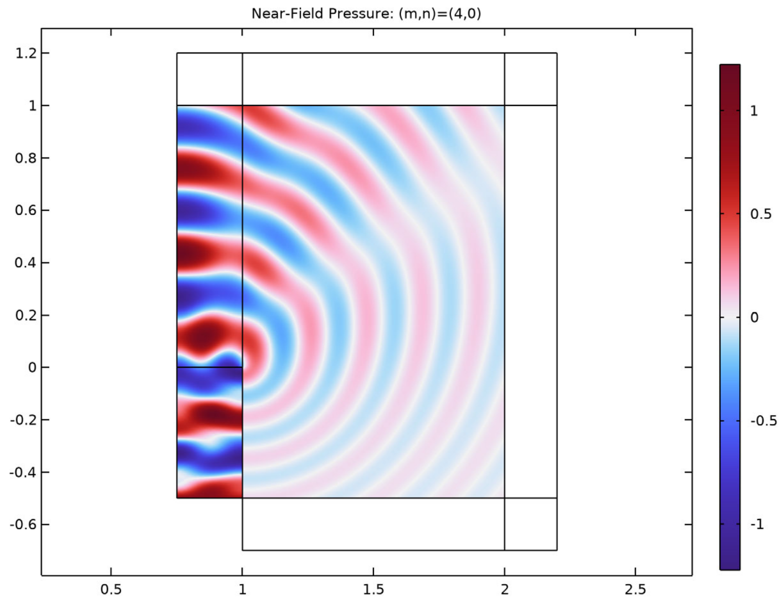


Figure 2: The near-field solution for m = 4 and n = 0.

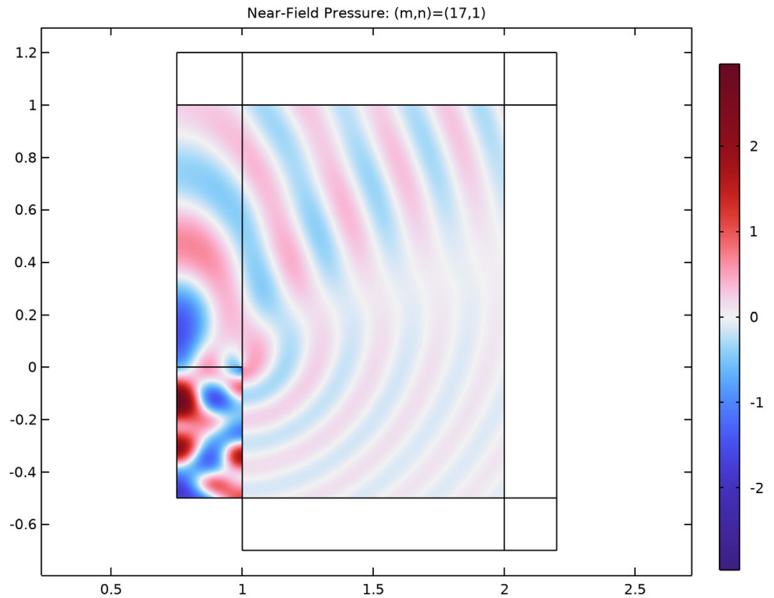


Figure 3: The near-field solution for m = 17 and n = 1.

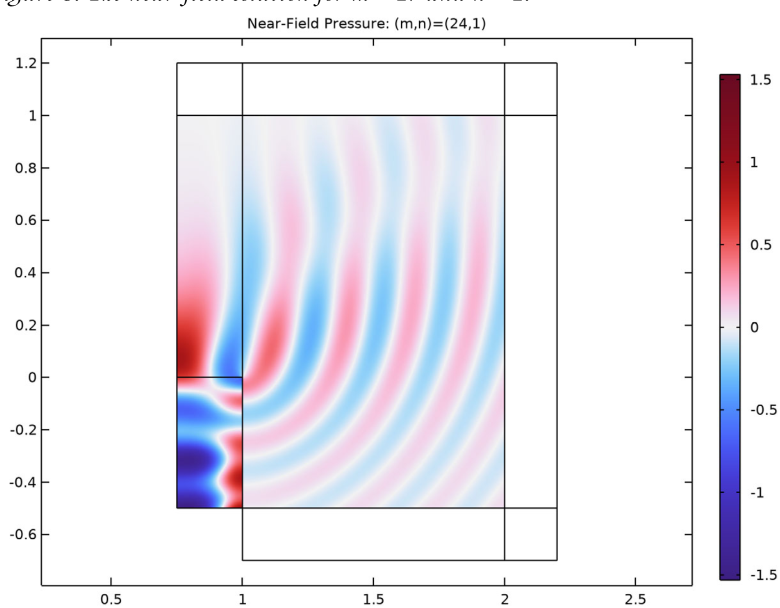


Figure 4: The near-field solution for m = 24 and n = 1.

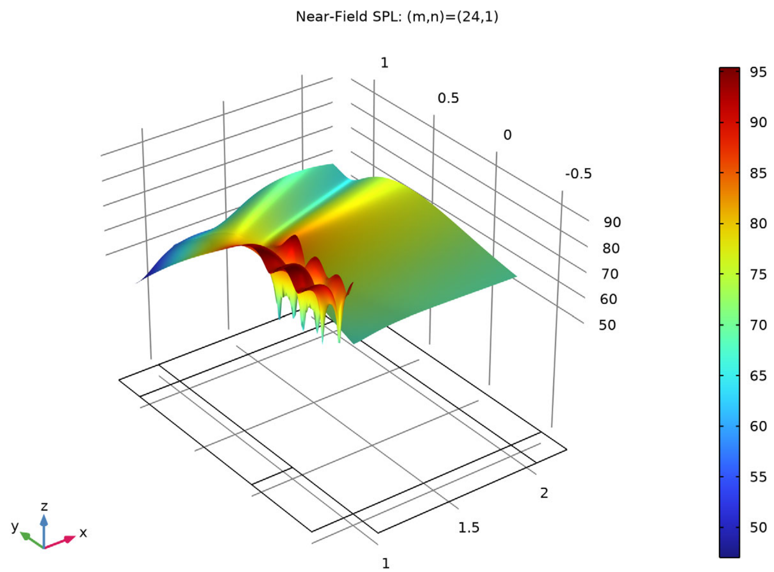


Figure 5: The near-field sound pressure level for m = 24 and n = 1.

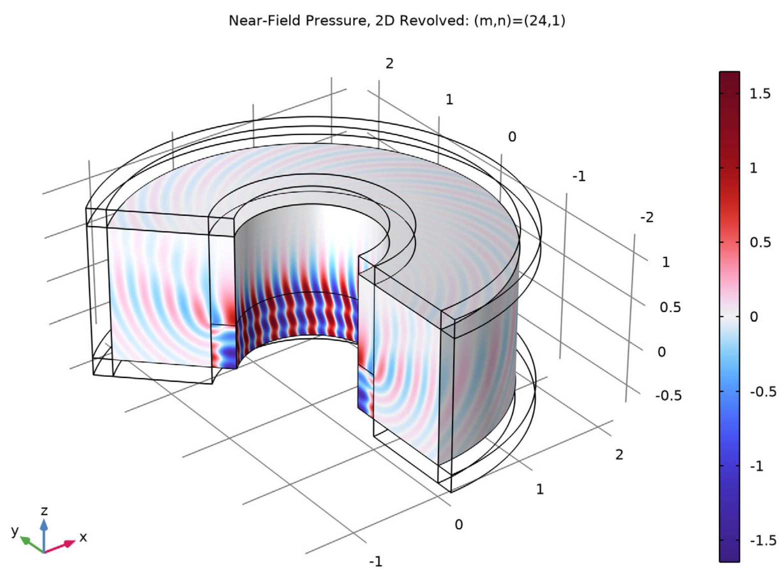


Figure 6: The near-field pressure plotted in the revolved geometry for m = 24 and n = 1.

Reference

1. G. Gabard and R.J. Astley, “Theoretical Model for Sound Radiations from Annular Jet Pipes: Far- and Near-field Solution,” *J. Fluid Mech.*, vol. 549, pp. 315–341, 2006.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/jet_pipe`

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**.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node’s **Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **None**.

This setting turns off all unit support in the model.

GEOMETRY 1

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 `1.45`.

4 In the **Height** text field, type `1.9`.

5 Locate the **Position** section. In the **r** text field, type `0.75`.

6 In the **z** text field, type `-0.7`.

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

Layer name	Thickness
Layer 1	0.2

8 Select the **Layers to the right** checkbox.

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

Rectangle 2 (r2)

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 `0.25`.

4 In the **Height** text field, type `1.9`.

5 Locate the **Position** section. In the **r** text field, type `0.75`.

6 In the **z** text field, type `-0.7`.

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

Layer name	Thickness
Layer 1	0.7

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

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

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

4 On the object **uni1**, select Domain 1 only.

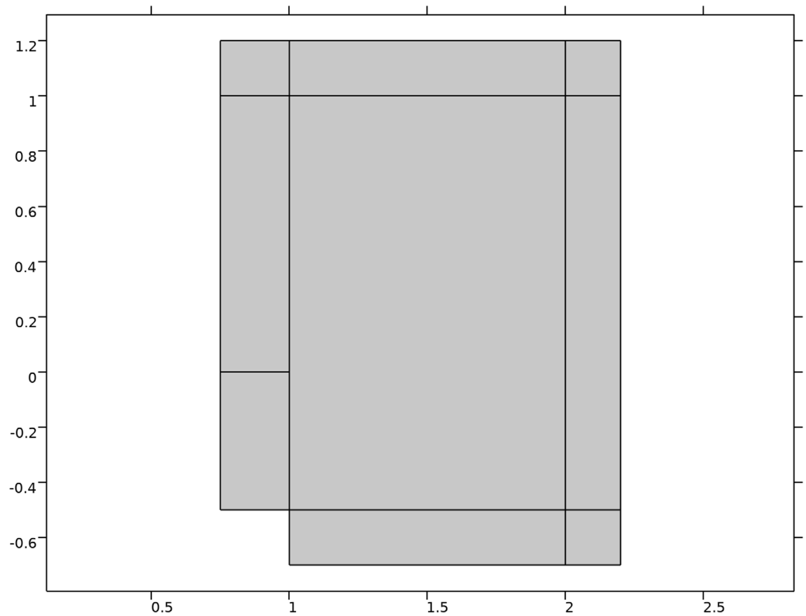
Form Union (fin)

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

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

This completes the geometry-modeling state. The geometry in the **Graphics** window should now look like that in the figure below.

3 In the **Model Builder** window, click **Form Union (fin)**.



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
M0	0.25	0.25	Mach number outside the duct
M1	0.45	0.45	Mach number inside the duct
f	30/(2*pi)	4.7746	Frequency
m	4	4	Azimuthal mode number
n	0	0	Radial mode number

Create three parameter cases to represent the three different source configurations.

4 In the **Home** toolbar, click **Parameter Case**.

5 In the **Home** toolbar, click **Parameter Case**.

6 In the **Settings** window for **Case**, locate the **Parameters** section.

7 In the table, enter the following settings:

Name	Expression	Description
m	17	Azimuthal mode number
n	1	Radial mode number

8 In the **Home** toolbar, click **Parameter Case**.

9 In the **Settings** window for **Case**, locate the **Parameters** section.

10 In the table, enter the following settings:

Name	Expression	Description
m	24	Azimuthal mode number
n	1	Radial mode number

DEFINITIONS

Port

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

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

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

4 Select Boundary 2 only.

MATERIALS

Specify the density and speed of sound, both normalized to 1, as material parameters.

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**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1		Basic
Speed of sound	c	1		Basic

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (LPFF)

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 **Intensity normalization**.

   The intensity normalization of the port modes is typically used in aeroacoustic problems with modal source decomposition.

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

0	r
M0	z

Vortex Sheet 1

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

2 Select Boundaries 12 and 13 only.

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.

Linearized Potential Flow Model 2

1 In the **Physics** toolbar, click **Domains** and choose **Linearized Potential Flow Model**.

2 Select Domains 1–3 only.

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

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

0	r
M1	z

Port 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. In the **Settings** window for **Port**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Port**.
4. Locate the **Port Properties** section. From the **Type of port** list, choose **Annular**.
5. Locate the **Port Mode Settings** section. In the n text field, type n.
6. Locate the **Port Incident Mode Settings** section. From the **Incident wave excitation at this port** list, choose **On**.
7. From the **Define incident wave** list, choose **Mode scale**.
8. In the S^{in} text field, type 1.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1. In the **Definitions** toolbar, click **Perfectly Matched Layer**.
2. Select Domains 3, 4, and 6–9 only.
3. In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.
4. From the **Type** list, choose **Cylindrical**.
5. Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

In this model, the mesh is set up manually. Proceed by directly adding the desired mesh component.

MESH 1

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 1, 2, and 5 only.

Size

1. In the **Model Builder** window, 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 `(1-M1)/f/6`.

5 In the **Minimum element size** text field, type `(1-M1)/f/6`.

Mapped 1

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

Distribution 1

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

2 Select Boundaries 16 and 19–21 only.

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

4 In the **Number of elements** text field, type `8`.

5 Click **Build All**.

STUDY 1

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

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

3 Clear the **Generate default plots** checkbox.

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 `f`.

Parametric Sweep

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

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

3 From the **Sweep type** list, choose **Parameter switch**.

4 Click **Add**.

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

Proceed to plotting the pressure near-field solution shown in Figure 2, Figure 3, and Figure 4.

RESULTS

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

Revolution 2D 1

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

2 Right-click **Results** > **Datasets** and choose **Revolution 2D**.

3 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

4 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

5 Click to expand the **Revolution Layers** section. From the **Number of layers** list, choose **Custom**.

6 In the **Layers** text field, type `225`.

7 In the **Start angle** text field, type `-90`.

8 In the **Revolution angle** text field, type `225`.

9 Click to expand the **Advanced** section. Select the **Define variables** checkbox.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Parametric Solutions 1 (sol2)** > **Linearized Potential Flow, Frequency Domain** > **Acoustic Pressure (lpff)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Near-Field Pressure

1 In the **Settings** window for **2D Plot Group**, type `Near-Field Pressure` in the **Label** text field.

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

3 In the **Title** text area, type `Near-Field Pressure: (m,n)=(eval(m),eval(n))`.

4 Clear the **Parameter indicator** text field.

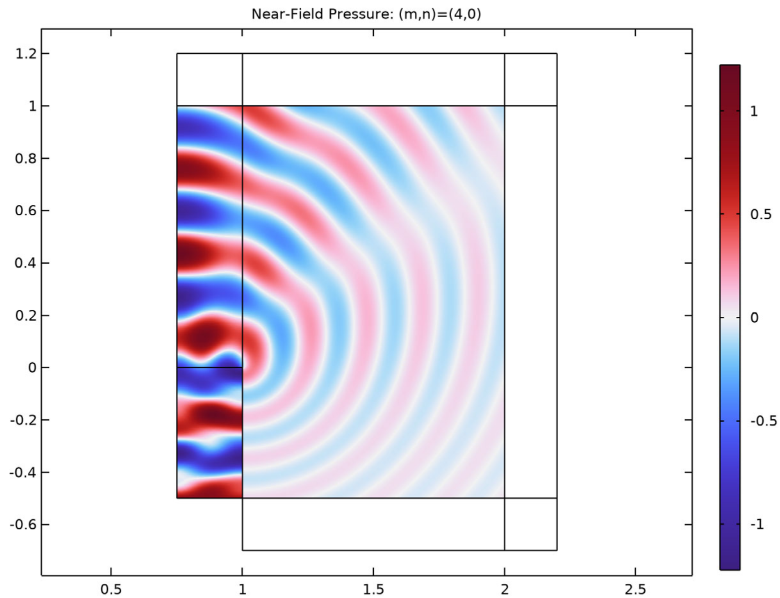
5 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

6 Select Domains 1, 2, and 5 only.

7 Locate the **Data** section. From the **Parameters 1** list, choose **Case 1**.

8 In the **Near-Field Pressure** toolbar, click **Plot**.

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



The graph that appears should be the same as that in Figure 2. Change the parameter cases to generate the graphics in Figure 3 and Figure 4.

10 From the **Parameters 1** list, choose **Case 2**.

11 In the **Near-Field Pressure** toolbar, click **Plot**.

12 From the **Parameters 1** list, choose **Case 3**.

13 In the **Near-Field Pressure** toolbar, click **Plot**.

Create a plot of the sound pressure level as the one shown in Figure 5. Just as for the pressure plot you can change the parameter case to see the other solutions.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Parametric Solutions 1 (sol2)** > **Linearized Potential Flow, Frequency Domain** > **Sound Pressure Level (lpff)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Near-Field SPL

1 In the **Settings** window for **2D Plot Group**, type `Near-Field SPL` in the **Label** text field.

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

3 In the **Title** text area, type `Near-Field SPL: (m,n)=(eval(m),eval(n))`.

4 Clear the **Parameter indicator** text field.

5 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

6 Select Domains 1, 2, and 5 only.

Height Expression 1

1 In the **Model Builder** window, expand the **Near-Field SPL** node.

2 Right-click **Surface** and choose **Height Expression**.

3 In the **Near-Field SPL** toolbar, click **Plot**.

Now, plot the pressure in the revolved geometry including the azimuthal dependency and create Figure 6.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Parametric Solutions 1 (sol2)** > **Linearized Potential Flow, Frequency Domain** > **Acoustic Pressure, 3D (lpff)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Near-Field Pressure, 2D Revolved

1 In the **Settings** window for **3D Plot Group**, type `Near-Field Pressure, 2D Revolved` in the **Label** text field.

Switch to the previously created Revolution 2D dataset. Note that to capture the azimuthal dependency that dataset is set up to use a fine one degree resolution in the circumferential direction.

2 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D 1**.

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

4 In the **Title** text area, type `Near-Field Pressure, 2D Revolved: (m,n)=(eval(m), eval(n))`.

5 Clear the **Parameter indicator** text field.

Surface

1 In the **Model Builder** window, expand the **Near-Field Pressure, 2D Revolved** node, then click **Surface**.

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

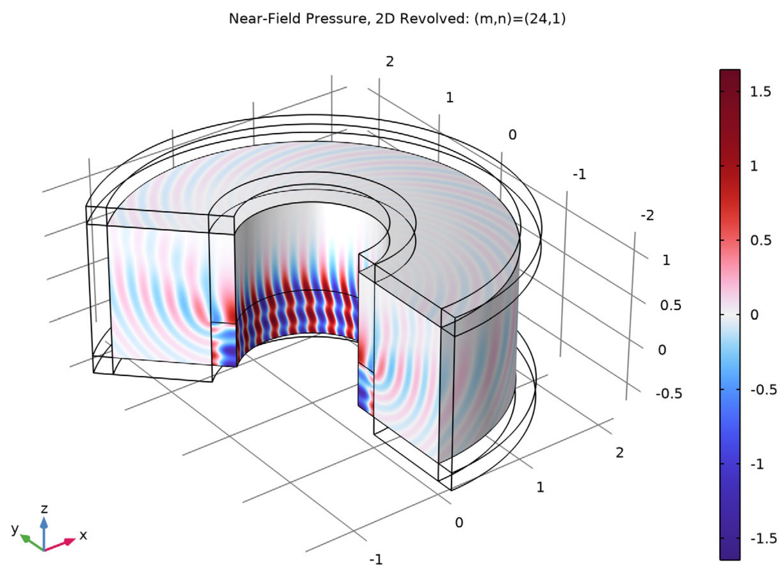
3 In the **Expression** text field, type `lpff.p*exp(-i*m*rev1phi)`.

Selection 1

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

2 Select Domains 1, 2, and 5 only.

3 In the **Near-Field Pressure, 2D Revolved** toolbar, click **Plot**.



Near-Field Pressure, 2D Revolved 1

In the **Model Builder** window, right-click **Near-Field Pressure, 2D Revolved** and choose **Duplicate**.

Surface

1 In the **Model Builder** window, expand the **Near-Field Pressure, 2D Revolved 1** node, then click **Surface**.

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

3 In the **Expression** text field, type `lpff.p_in_1*exp(-i*m*rev1phi)`.

Incident Source Mode Pressure

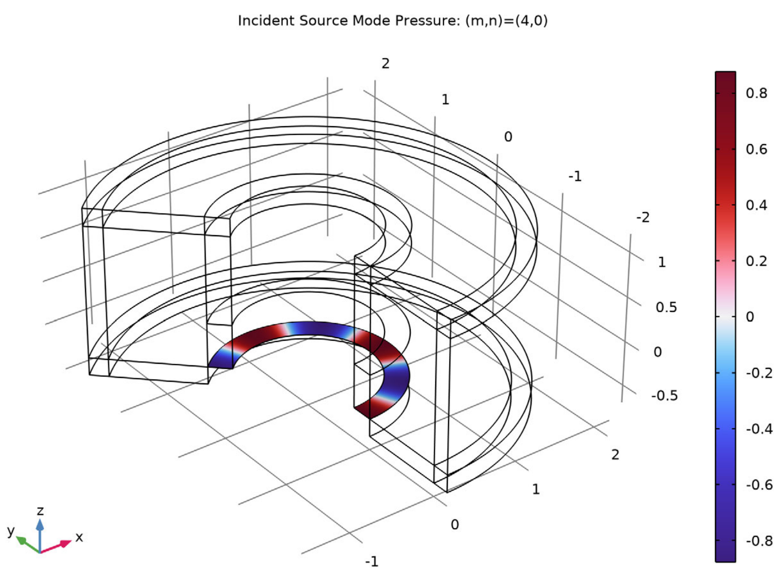
1 In the **Model Builder** window, under **Results** click **Near-Field Pressure, 2D Revolved 1**.

2 In the **Settings** window for **3D Plot Group**, type `Incident Source Mode Pressure` in the **Label** text field.

3 Click to expand the **Title** section. In the **Title** text area, type `Incident Source Mode Pressure: (m,n)=(eval(m),eval(n))`.

4 Locate the **Data** section. From the **Parameters 1** list, choose **Case 1**.

5 In the **Incident Source Mode Pressure** toolbar, click **Plot**.



6 From the **Parameters 1** list, choose **Case 2**.

7 In the **Incident Source Mode Pressure** toolbar, click **Plot**.

8 From the **Parameters 1** list, choose **Case 3**.

9 In the **Incident Source Mode Pressure** toolbar, click **Plot**.

Finally, evaluate some important quantities like the mode cutoff frequency and the scattering coefficients for the port.

Evaluation Group 1 - Mode Cutoff Frequency

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

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

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

Global Evaluation 1

1 Right-click **Evaluation Group 1 - Mode Cutoff Frequency** 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.fc - Mode cutoff frequency - 1/s**.

3 In the **Evaluation Group 1 - Mode Cutoff Frequency** toolbar, click **Evaluate**.

Evaluation Group 2 - Scattering Coefficient

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

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group 2 - Scattering Coefficient` in the **Label** text field.

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

Global Evaluation 1

1 Right-click **Evaluation Group 2 - Scattering Coefficient** 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** > **lpff.S11 - S11**.

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

Expression	**Unit**	**Description**
`abs(lpff.S11)`		

4 In the **Evaluation Group 2 - Scattering Coefficient** toolbar, click **Evaluate**.