Created in COMSOL Multiphysics 6.1

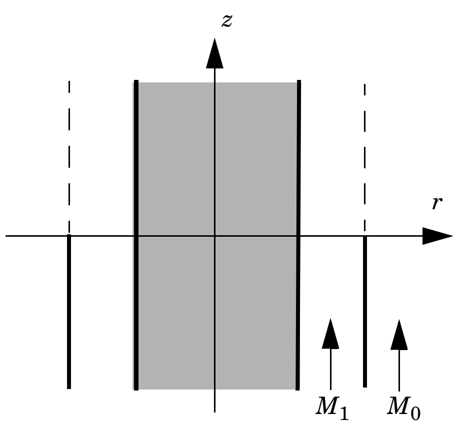
Jet Pipe

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 boundary conditions on the two sides of the vortex sheet are defined as follows:

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

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

In these equations, ω is the angular velocity, $\mathbf{V}$ 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 $M = V/c_0$ 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 duct has a hard wall, which you also model using an interior boundary condition.

The acoustic field inside the duct can be described as a sum of eigenmodes 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 produced by a single eigenmode at a time. First you calculate the eigenmodes with the azimuthal mode number 4 on the inlet boundary. From these eigenmodes, the one with radial mode order 0 is used as incident wave. You then calculate the velocity fields with azimuthal mode numbers m = 17 and 24 and with radial mode order n = 1.

Results and Discussion

The inlet sources are found using a boundary mode analysis. The analysis is made with the circumferential wave number m = 4, 17, and 24 and gives several eigenmodes corresponding to different radial mode numbers. This example, like Ref. 1, uses the eigenmodes $(m,n) = (4,0)$, $(17,1)$, and $(24,1)$ as incident waves in the duct. The modes are depicted in Figure 1. The radial mode $n = 0$ corresponds to the largest eigenvalue for a given m, while $n = 1$ corresponds to the smallest eigenvalue.

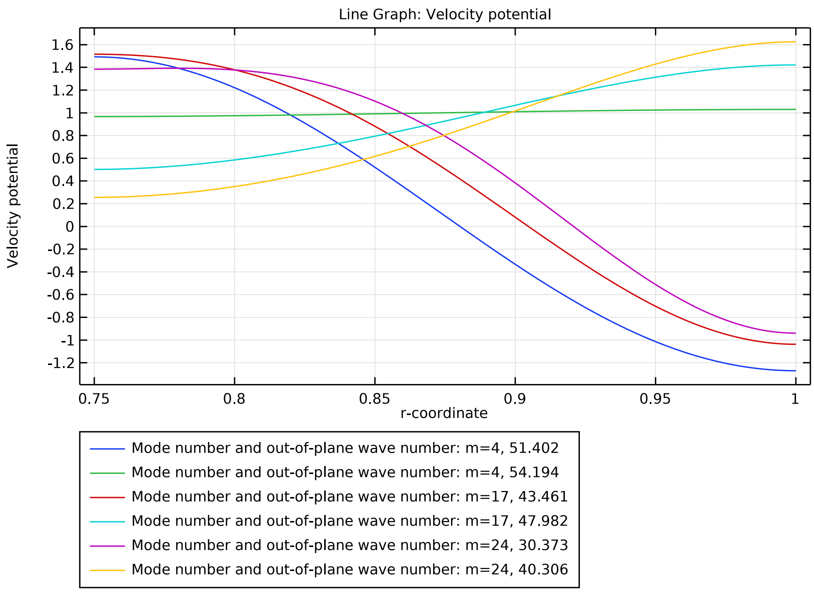


Figure 1: Mode shape for $m = 4$, 17, and 24 for both $n = 0$ and $n=1$

In Figure 2 the source velocity potential is depicted in the revolved geometry including the azimuthal mode contribution, given as

$$\phi \cdot e^{-im\varphi}$$

where m is the mode number and φ is the azimuthal angle.

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 3 through Figure 7 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 different source eigenmodes shown in Figure 1.

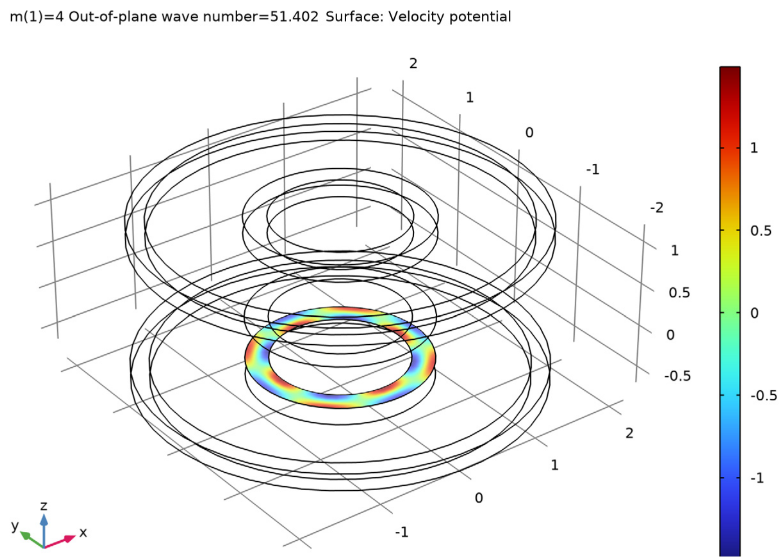


Figure 2: The boundary mode (m,n) = (4,0) depicted in the revolved geometry including the azimuthal wave number contribution.

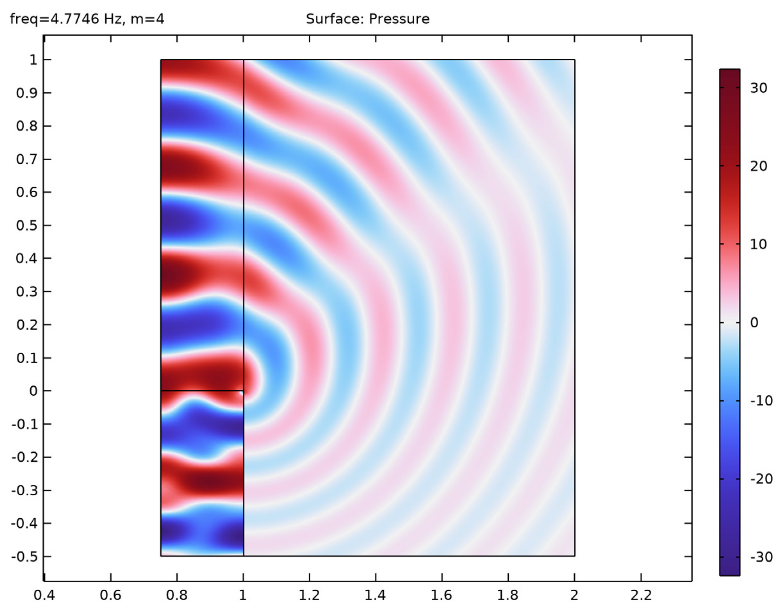


Figure 3: The near-field solution for m = 4 and n = 0.

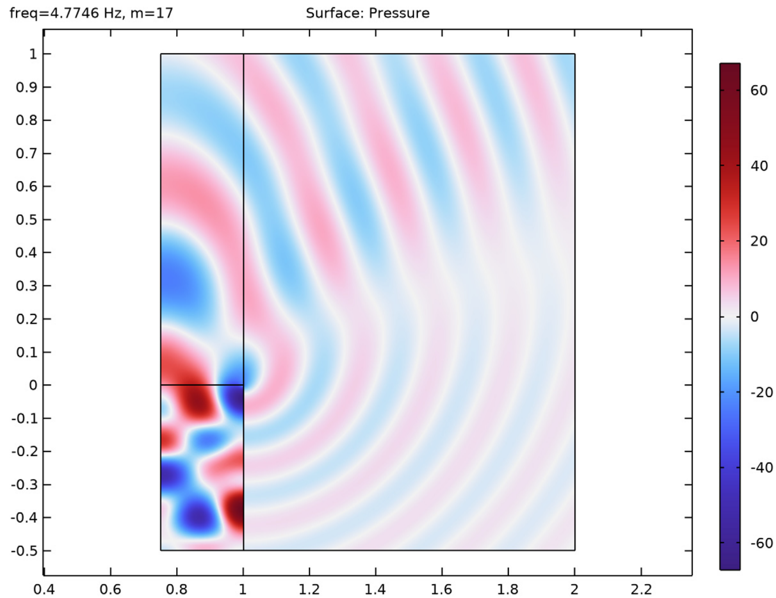


Figure 4: The near-field solution for m = 17 and n = 1.

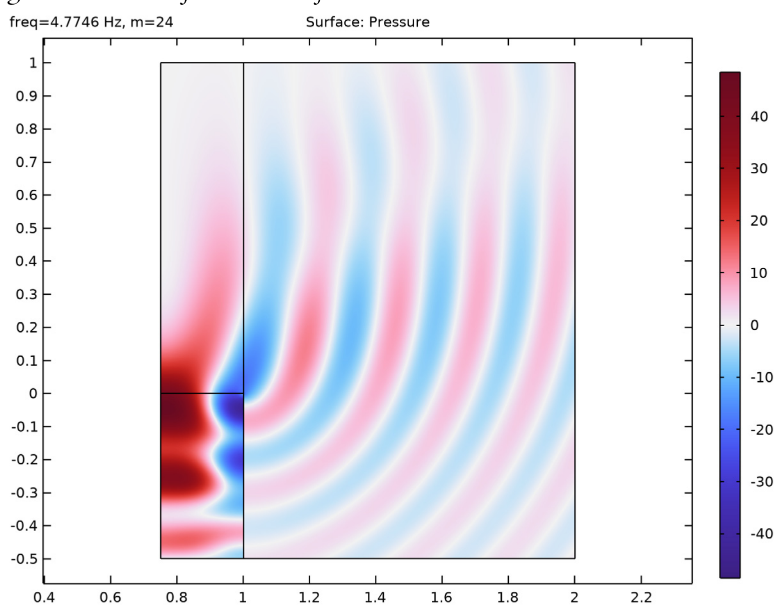


Figure 5: The near-field solution for m = 24 and n = 1.

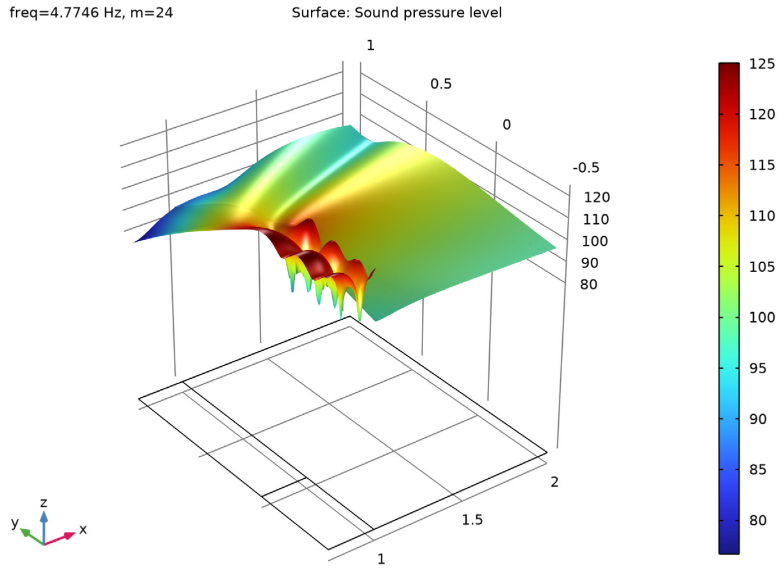


Figure 6: The near-field sound pressure level for m = 24 and n = 1.

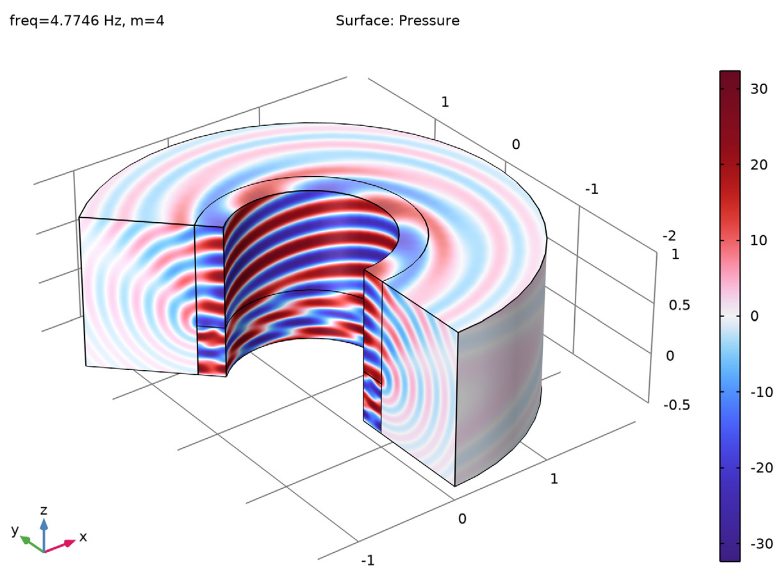


Figure 7: The near-field pressure plotted in the revolved geometry for m = 4 and n = 0.

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, Boundary Mode (lpfbm)**.

3 Click **Add**.

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

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Mode Analysis**.

8 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 `0.25`.

4 In the **Height** text field, type `0.5`.

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

6 In the **z** text field, type `-0.5`.

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

Rectangle 3 (r3)

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 `0.2`.

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

6 In the **z** text field, type `1`.

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

Rectangle 4 (r4)

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

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

3 In the **Height** text field, type `1.5`.

4 Locate the **Position** section. In the **r** text field, type `1`.

5 In the **z** text field, type `-0.5`.

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

Rectangle 5 (r5)

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 `1.2`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **r** text field, type `1`.

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

Rectangle 6 (r6)

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 `0.2`.

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

5 Locate the **Position** section. In the **r** text field, type `2`.

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

Rectangle 7 (r7)

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 `1.2`.

4 In the **Height** text field, type `0.2`.

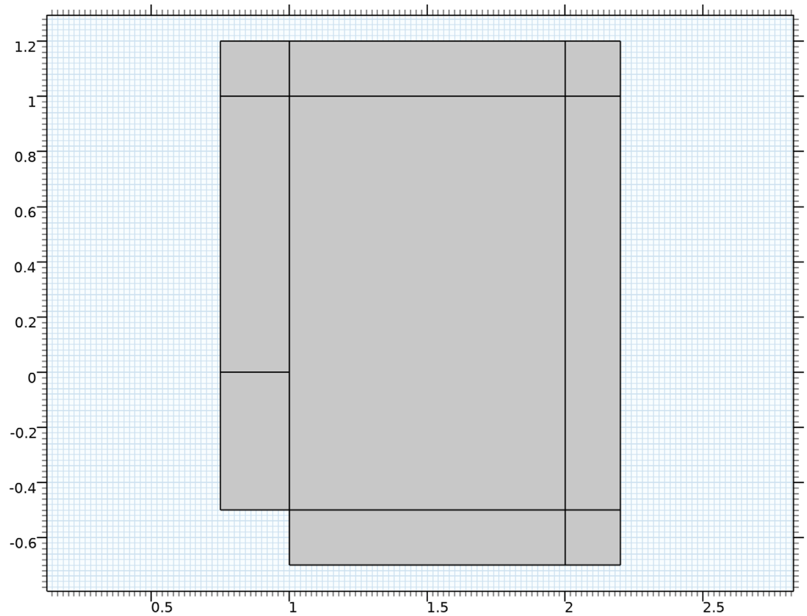
5 Locate the **Position** section. In the **r** text field, type `1`.

6 In the **z** text field, type `1`.

7 Click **Build All Objects**.

8 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.



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
m	4	4	Circumferential wave number
f	30/(2*pi)	4.7746	Frequency
k0	2*pi*f/(1-M1)	54.545	Largest wave number

DEFINITIONS

Duct Cross Section

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

2 In the **Settings** window for **Explicit**, type `Duct Cross Section` 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. You need to add a separate material node for the duct cross section because it is a boundary and not a domain.

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

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

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

4 From the **Selection** list, choose **Duct Cross Section**.

5 Locate the **Material Contents** section. 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, BOUNDARY MODE (LPFBM)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Boundary Mode (lpfbm)**.

2 In the **Settings** window for **Linearized Potential Flow, Boundary Mode**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Duct Cross Section**.

4 Click to expand the **Equation** section. Locate the **Linearized Potential Flow Equation Settings** section. In the m text field, type `m`.

Linearized Potential Flow Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Linearized Potential Flow, Boundary Mode (lpfbm)** click **Linearized Potential Flow Model 1**.

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

3 Specify the **V** vector as

0	r
M1	z

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`.

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 **Linearized Potential Flow Model** section.

3 Specify the **V** 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 **Linearized Potential Flow Model** section.
4 Specify the **V** vector as

0	r
M1	z

Velocity Potential 1

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

   The value of the velocity potential is given by the boundary mode solutions. The dependent variable is `phi`.
2 In the **Settings** window for **Velocity Potential**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Duct Cross Section**.
4 Locate the **Velocity Potential** section. In the ϕ_0 text field, type `phi`.

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**.

Start by solving the boundary mode problem to find the source potentials that will be applied in the full model.

STUDY 1 - MODE ANALYSIS

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Study 1 - Mode Analysis` in the **Label** text field.
3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	4 17 24	

Step 1: Mode Analysis

1 In the **Model Builder** window, click **Step 1: Mode Analysis**.

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

3 In the **Mode analysis frequency** text field, type `f`.

4 From the **Mode search method around shift** list, choose **Smaller real part**.

5 Select the **Desired number of modes** check box. In the associated text field, type `2`.

6 Select the **Search for modes around** check box. In the associated text field, type `k0`.

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

RESULTS

Boundary Modes

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

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

3 In the **Settings** window for **1D Plot Group**, type `Boundary Modes` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Mode Analysis/ Parametric Solutions 1 (sol2)**.

5 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

6 From the **Position** list, choose **Bottom**.

7 In the **Number of rows** text field, type `6`.

Line Graph 1

1 Right-click **Boundary Modes** and choose **Line Graph**.

2 Select Boundary 2 only.

3 In the **Boundary Modes** toolbar, click **Plot**.

4 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

5 From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `r`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Mode number and out-of-plane wave number: .`

9 In the **Boundary Modes** toolbar, click **Plot**.

The graph that appears should be the same as that in Figure 1. This shows the two radial modes (`n=0` and `1`) for `m=4`, `17`, and `24`.

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 - Mode Analysis/Parametric Solutions 1 (sol2)**.

5 Click to expand the **Advanced** section. In the **Azimuthal mode number** text field, type `m`.

Boundary Modes 2D Revolved

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

2 In the **Settings** window for **3D Plot Group**, type `Boundary Modes 2D Revolved` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (m)** list, choose **4**.

Surface 1

1 Right-click **Boundary Modes 2D Revolved** and choose **Surface**.

The circumferential behavior is automatically represented as you defined the **Azimuthal mode number** in the revolved solution dataset.

2 In the **Boundary Modes 2D Revolved** toolbar, click **Plot**.

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

The graph that appears should be the same as that in Figure 2.

Now, add three studies to solve three cases with different sources: `(m=4,n=0)`, `(m=17, n=1)`, and `(m=24,n=1)`, respectively. After setting up three studies and solving then plot the results.

First, solve the case with the circumferential wave number `m=4` and the first radial mode (`n=0`), that is the mode with the highest eigenvalue.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Linearized Potential Flow, Boundary Mode (lpfbm)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

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

STUDY 2 - M=4 AND N=0

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

2 In the **Settings** window for **Study**, type `Study 2 - m=4 and n=0` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - m=4 and n=0** 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`.

4 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Mode Analysis, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol2)**.

8 From the **Use** list, choose **m=4 (sol3)**.

9 From the **Out-of-plane wave number** list, choose **54.194**.

The value of `m` is not stored in the mode analysis dataset so you need to set it manually as follows.

10 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	4	

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

Secondly, solve the case with the circumferential wave number `m=17` and the second radial mode (`n=1`). That is the mode with the second highest eigenvalue. The steps are the same as for Study 2.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Linearized Potential Flow, Boundary Mode (lpfbm)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

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

STUDY 3 - M=17 AND N=1

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3 - m=17 and n=1` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3 - m=17 and n=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`.

4 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Mode Analysis, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol2)**.

8 From the **Use** list, choose **m=17 (sol4)**.

9 From the **Out-of-plane wave number** list, choose **43.461**.

The value of `m` is not stored in the mode analysis dataset so you need to set it manually as follows.

10 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	17	

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

Finally, solve the case with the circumferential wave number `m=27` again with the second radial mode (`n=1`). That is the mode with the second highest eigenvalue. The steps are the same as for Study 2 and Study 3.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Linearized Potential Flow, Boundary Mode (lpfbm)**.
4. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.
5. Click **Add Study** in the window toolbar.
6. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4 - M=24 AND N=1

1. In the **Model Builder** window, click **Study 4**.
2. In the **Settings** window for **Study**, type `Study 4 - m=24 and n=1` in the **Label** text field.
3. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 4 - m=24 and n=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`.
4. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
5. From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Mode Analysis, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol2)**.

8 From the **Use** list, choose **m=24 (sol5)**.

9 From the **Out-of-plane wave number** list, choose **30.373**.

The value of m is not stored in the mode analysis dataset so you need to set it manually as follows.

10 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	24	

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

Proceed to plotting the pressure near field solution shown in Figure 3, Figure 4, and Figure 5.

First, add selections to the datasets such that the results are not shown in the PML domains. In here the solutions is not physical.

RESULTS

Study 2 - m=4 and n=0/Solution 6 (sol6)

In the **Model Builder** window, under **Results>Datasets** click **Study 2 - m=4 and n=0/ Solution 6 (sol6)**.

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, 2, and 5 only.

Study 3 - m=17 and n=1/Solution 7 (sol7)

In the **Model Builder** window, under **Results>Datasets** click **Study 3 - m=17 and n=1/ Solution 7 (sol7)**.

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, 2, and 5 only.

Study 4 - m=24 and n=1/Solution 8 (sol8)

In the **Model Builder** window, under **Results>Datasets** click **Study 4 - m=24 and n=1/ Solution 8 (sol8)**.

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, 2, and 5 only.

Now, create the plots.

5 In the **Results** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 4 - m=24 and n=1/Solution 8 (sol8)>Linearized Potential Flow, Frequency Domain>Acoustic Pressure (lpff)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Results** toolbar, click **Add Predefined Plot**.

RESULTS

Near Field Pressure

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (lpff)**.

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

3 In the **Near Field Pressure** toolbar, click **Plot**.

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

The graph that appears should be the same as that in Figure 3. Change the evaluation dataset to generate the graphics in Figure 4 and Figure 5.

5 In the **Model Builder** window, click **Near Field Pressure**.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - m=17 and n=1/ Solution 7 (sol7)**.

7 In the **Near Field Pressure** toolbar, click **Plot**.

The graph that appears should be the same as that in Figure 4.

8 From the **Dataset** list, choose **Study 4 - m=24 and n=1/Solution 8 (sol8)**.

9 In the **Near Field Pressure** toolbar, click **Plot**.

The graph that appears should be the same as that in Figure 5.

Create a plot of the sound pressure level as the one shown in Figure 6. Just as for the pressure plot you can change the evaluation dataset to see the other solutions.

10 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 4 - m=24 and n=1/Solution 8 (sol8)>Linearized Potential Flow, Frequency Domain>Sound Pressure Level (lpff)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Near Field SPL

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

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

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**.

Finally, plot the pressure in the revolved geometry including the circumferential wave number and create Figure 7.

Revolution 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

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

3 From the **Dataset** list, choose **Study 2 - m=4 and n=0/Solution 6 (sol6)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `-90`.

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

6 Locate the **Advanced** section. In the **Azimuthal mode number** text field, type `m`.

7 In the **Results** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 2 - m=4 and n=0/Solution 6 (sol6)>Linearized Potential Flow, Frequency Domain>Acoustic Pressure, 3D (lpff)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Results** toolbar, click **Add Predefined Plot**.

RESULTS

Near Field Pressure 2D Revolved

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

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

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

The circumferential behavior is automatically represented as you defined the **Azimuthal mode number** in the revolved solution dataset.

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