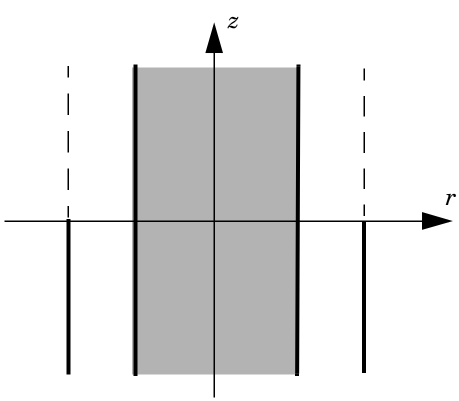
Jet Pipe

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. The flows both inside and outside the duct are uniform mean flows, but because the flow velocities differ, a vortex sheet separates them.



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 The Aeroacoustics Branch chapter in the *Acoustics Module User's Guide* for further details. The field equations are only valid when the 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:

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

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

In these equations, ω is the angular velocity, 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

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

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

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

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 circumferential mode order 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 circumferential mode numbers $m = 17$ and 24 and with radial mode order $n = 1$.

Results and Discussion

The boundary mode analysis made with the circumferential wave number $m = 4$, 17, and 24 gives several eigenmodes corresponding to different radial mode numbers. This example, like Ref. 1, uses the following eigenmodes as incident waves in the duct.

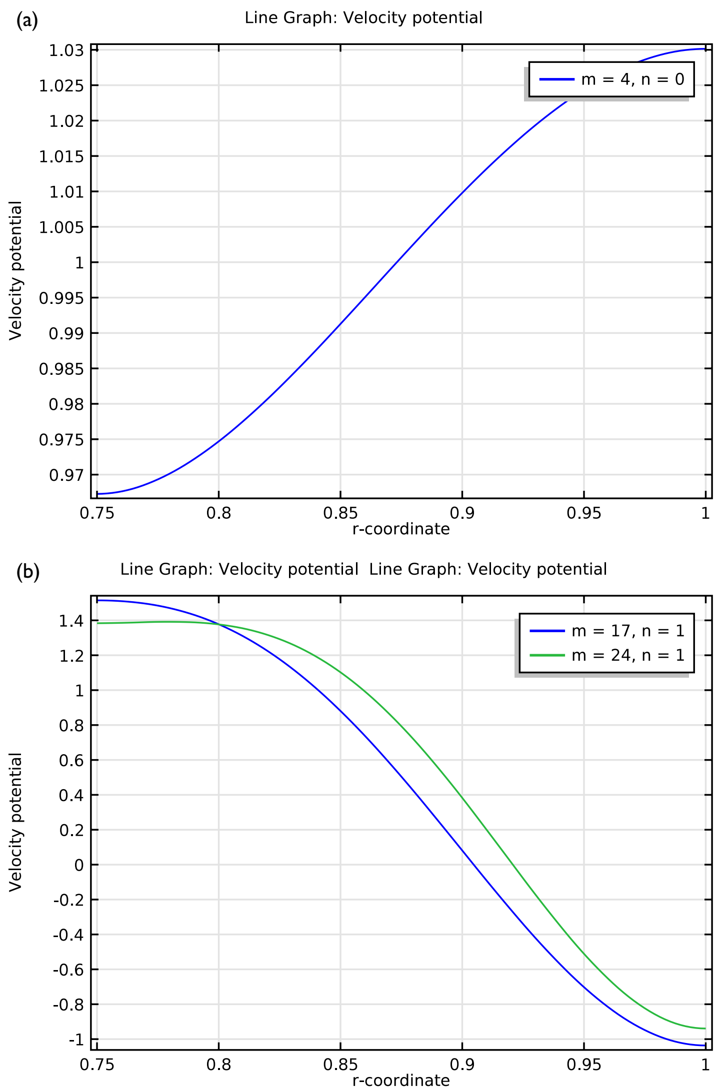


Figure 1: (a) Mode shape for m = 4, n = 0; (b) Mode shapes for m = 17, n = 1 and m = 24, n = 1.

The near field around the duct obtained by COMSOL Multiphysics can be compared to the results for the near field in Ref. 1. Figure 2 to Figure 4 show the near-field solution for a Mach number equal to 0.45 in the pipe and 0.25 on the outside. The figures show the field for the different eigenmodes shown in Figure 1.

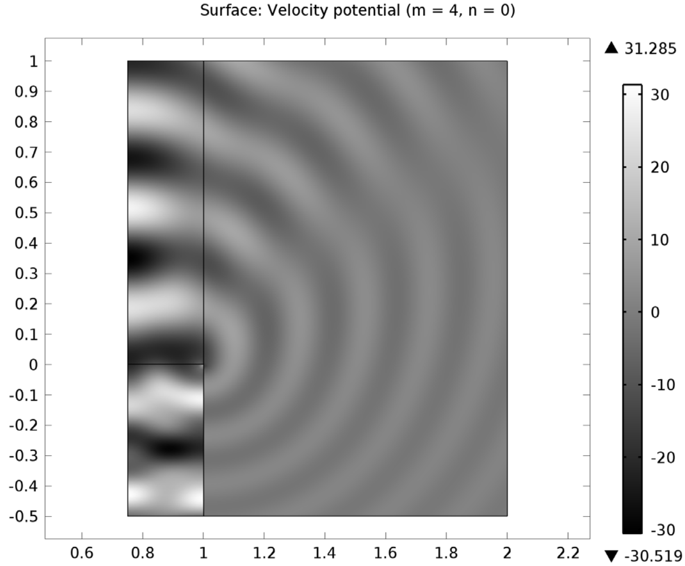


Figure 2: The near-field solution for m = 4 and n = 0.

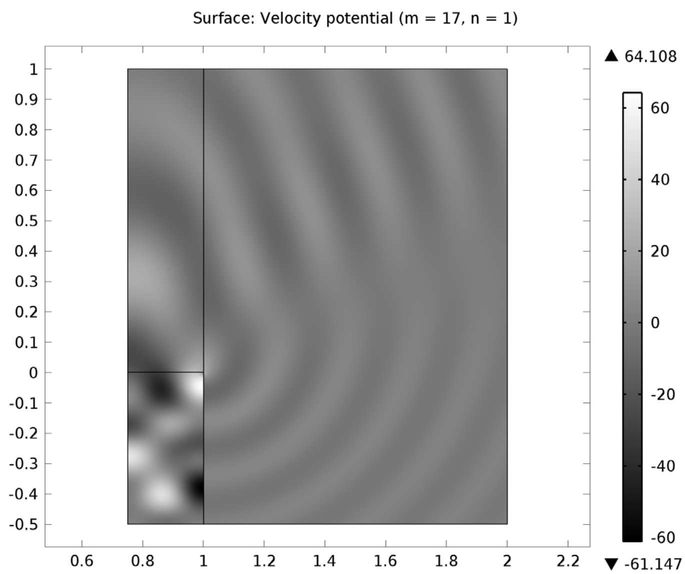


Figure 3: The near-field solution for m = 17 and n = 1.

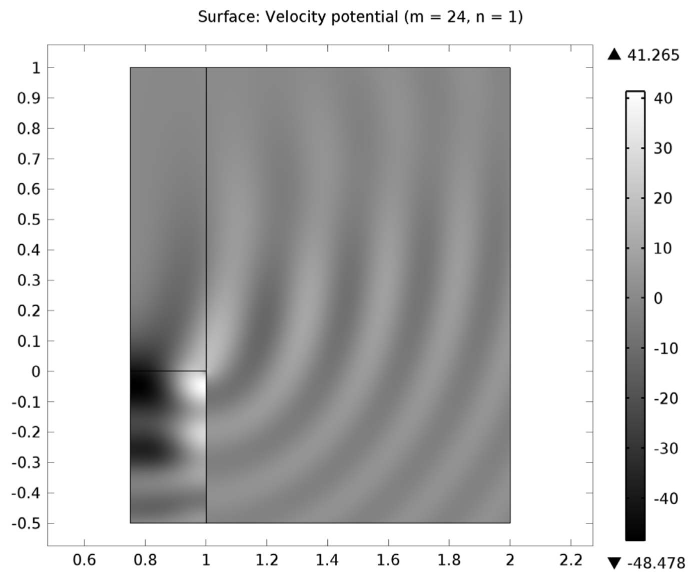


Figure 4: The near-field solution for m = 24 and n = 1.

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.

Model Library path: `Acoustics_Module/Tutorial_Models/jet_pipe`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

1 In the **Model Wizard** window, click the **2D Axisymmetric** button.

2 In the **Select physics** tree, select **Acoustics>Aeroacoustics>Linearized Potential Flow, Boundary Mode (aebm)**.
3 Click the **Add** button.
4 In the **Select physics** tree, select **Acoustics>Aeroacoustics>Linearized Potential Flow, Frequency Domain (ae)**.
5 Click the **Add** button.
6 Click the **Study** button.
7 In the tree, select **Custom Studies>Preset Studies for Some Physics>Mode Analysis**.
8 Click the **Done** button.

ROOT

1 In the **Model Builder** window, click **Untitled.mph (root)**.
2 In the **Root** 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 **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Rectangle**.
2 In the **Rectangle** settings window, locate the **Size** section.
3 In the **Width** edit field, type `0.25`.
4 In the **Height** edit field, type `0.5`.
5 Locate the **Position** section. In the **r** edit field, type `0.75`.
6 In the **z** edit field, type `-0.5`.

Rectangle 2 (r2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.
2 In the **Rectangle** settings window, locate the **Size** section.
3 In the **Width** edit field, type `0.25`.
4 Locate the **Position** section. In the **r** edit field, type `0.75`.

Rectangle 3 (r3)

1 Right-click **Geometry 1** and choose **Rectangle**.
2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.25`.

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

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

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

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

Rectangle 4 (r4)

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

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

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

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

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

Rectangle 5 (r5)

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `1.2`.

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

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

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

Rectangle 6 (r6)

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

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

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

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

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

Rectangle 7 (r7)

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `1.2`.

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

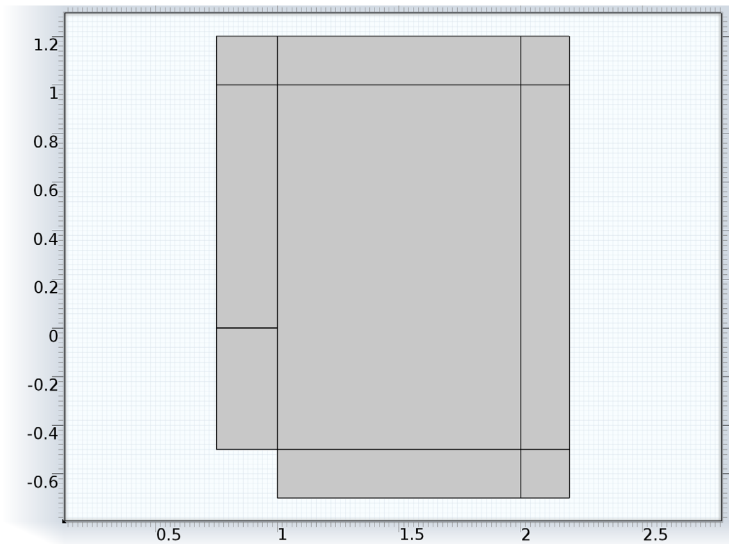
5 Locate the **Position** section. In the **r** edit field, type 1.

6 In the **z** edit field, type 1.

7 Click the **Build All Objects** button.

8 Click the **Zoom Extents** button on 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 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
M0	0.25	0.2500	Mach number outside the duct
M1	0.45	0.4500	Mach number inside the duct

Name	Expression	Value	Description
m	4	4.000	Circumferential wave number
f	30/(2*pi)	4.775	Frequency

DEFINITIONS

Explicit 1

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

2 In the **Explicit** settings window, locate the **Input Entities** section.

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

4 Select Boundary 2 only.

5 Right-click **Component 1 (comp1)>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Duct Cross Section` in the **New name** edit field.

7 Click **OK**.

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 **New Material**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

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

Material 2 (mat2)

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

2 In the **Material** settings window, 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	Name	Value	Unit	Property group
Density	rho	1		Basic
Speed of sound	c	1		Basic

LINEARIZED POTENTIAL FLOW, BOUNDARY MODE (AEBM)

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

2 In the **Linearized Potential Flow, Boundary Mode** settings window, 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 edit field, type `m`.

Linearized Potential Flow Model 1

1 In the **Model Builder** window, expand the **Linearized Potential Flow, Boundary Mode (aebm)** node, then click **Linearized Potential Flow Model 1**.

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

3 Specify the **V** vector as

0	r
M1	z

MESH 1

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

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

3 From the **Element size** list, choose **Extremely fine**.

4 Click the **Build All** button.

STUDY 1

Step 1: Mode Analysis

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

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

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

4 In the **Desired number of modes** edit field, type 10.

5 In the **Search for modes around** edit field, type 0.

6 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics	**Solve for**	**Discretization**
Aeroacoustics, Frequency Domain	×	physics

Parametric Sweep

1 On the **Study** toolbar, click **Extension Steps** and choose **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter names	**Parameter value list**
m	4 17 24

5 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (aebm)

Delete the default 2D and 3D plot groups and replace them by 1D Plot Group nodes.

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

2 Click **Yes** to confirm.

Acoustic Pressure, 3D (aebm)

1 In the **Model Builder** window, under **Results** right-click **Acoustic Pressure, 3D (aebm)** and choose **Delete**.

2 Click **Yes** to confirm.

Similarly, remove the Revolution 2D data set that was added for the revolved surface plot you just removed.

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Revolution 2D 1** and choose **Delete**.

2 Click **Yes** to confirm.

Before adding the 1D Plot Group nodes, add separate solution data sets for the three parameter values m = 4, 17, and 24.

3 On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

4 In the **Solution** settings window, locate the **Solution** section.

5 From the **Solution** list, choose **m=4**.

6 Right-click **Results>Data Sets>Solution 3** and choose **Rename**.

7 Go to the **Rename Solution** dialog box and type `aebm: m = 4` in the **New name** edit field.

8 Click **OK**.

9 On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

10 In the **Solution** settings window, locate the **Solution** section.

11 From the **Solution** list, choose **m=17**.

12 Right-click **Results>Data Sets>Solution 4** and choose **Rename**.

13 Go to the **Rename Solution** dialog box and type `aebm: m = 17` in the **New name** edit field.

14 Click **OK**.

15 On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

16 In the **Solution** settings window, locate the **Solution** section.

17 From the **Solution** list, choose **m=24**.

18 Right-click **Results>Data Sets>Solution 5** and choose **Rename**.

19 Go to the **Rename Solution** dialog box and type `aebm: m = 24` in the **New name** edit field.

20 Click **OK**.

Start with a separate 1D Plot Group for the mode m = 4, n = 0.

1D Plot Group 1

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

2 On the **1D plot group** toolbar, click **Line Graph**.

3 In the **Line Graph** settings window, locate the **Data** section.

4 From the **Data set** list, choose **aebm: m = 4**.

5 From the **Out-of-plane wave number selection** list, choose **From list**.

6 In the **Out-of-plane wave number** list, select **54.19438+1.189336e-5i**.

This is the mode with the highest real part.

7 Select Boundary 2 only.

8 Locate the **x-axis data** section. Click **r-coordinate (r)** in the upper-right corner of the section. Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
m = 4, n = 0

11 On the **1D plot group** toolbar, click **Plot**.

12 Right-click **Results>1D Plot Group 1>Line Graph 1** and choose **Copy**.

You will soon re-use this plot

13 Right-click **1D Plot Group 1** and choose **Rename**.

14 Go to the **Rename 1D Plot Group** dialog box and type `aebm: m = 4` in the **New name** edit field.

15 Click **OK**.

The graph that appears should be the same as that in Figure 1 (a). This corresponds to the lowest radial mode (n = 0).

Next, use this plot to reproduce Figure 1 (b) for the modes m = 17, n = 1 and m = 24, n = 1.

1D Plot Group 2

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

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

3 In the **Line Graph** settings window, locate the **Data** section.

4 From the **Data set** list, choose **aebm: m = 17**.

5 In the **Out-of-plane wave number** list, select the value with the second-highest real part.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
m = 17, n = 1

7 On the **1D plot group** toolbar, click **Plot**.

8 Right-click **Results>1D Plot Group 2>Line Graph 1** and choose **Duplicate**.

9 In the **Line Graph** settings window, locate the **Data** section.

10 From the **Data set** list, choose **aebm: m = 24**.

11 In the **Out-of-plane wave number** list, select the value with the second-highest real part.

12 Locate the **Legends** section. In the table, enter the following settings:

Legends
`m = 24, n = 1`

13 On the **1D plot group** toolbar, click **Plot**.

14 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Rename**.

15 Go to the **Rename 1D Plot Group** dialog box and type `aebm: m = 17, 24` in the **New name** edit field.

16 Click **OK**.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 3, 4, and 6–9 only.

3 In the **Perfectly Matched Layer** settings window, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

5 Locate the **Scaling** section. From the **Typical wavelength from** list, choose **User defined**.

6 In the **Typical wavelength** edit field, type `1/f`.

Here, 1 is the normalized speed of sound.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (AE)

Linearized Potential Flow Model 1

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

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

3 Specify the **V** vector as

0	r
M0	z

Vortex Sheet 1

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

2 Select Boundaries 12 and 13 only.

Interior Sound Hard Boundary (Wall) 1

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

2 Select Boundary 10 only.

3 In the **Model Builder** window, click **Linearized Potential Flow, Frequency Domain (ae)**.

4 In the **Linearized Potential Flow, Frequency Domain** settings window, click to expand the **Equation** section.

5 Locate the **Linearized Potential Flow Equation Settings** section. In the m edit field, type `m`.

Linearized Potential Flow Model 2

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

2 Select Domains 1–3 only.

3 In the **Linearized Potential Flow Model** settings window, locate the **Linearized Potential Flow Model** section.

4 Specify the **V** vector as

0	r
M1	z

Velocity Potential 1

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

2 In the **Velocity Potential** settings window, locate the **Velocity Potential** section.

3 In the ϕ_0 edit field, type `phi`.

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

ROOT

On the **Home** toolbar, click **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the tree, select **Custom Studies>Preset Studies for Some Physics>Frequency Domain**.

3 In the **Add study** window, click **Add Study**.

4 On the **Home** toolbar, click **Add Study**.

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** edit field, type `f`.

4 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics	**Solve for**	**Discretization**
Boundary Mode Aeroacoustics	×	physics

Solver 6

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solver 6** node, then click **Dependent Variables 1**.

4 In the **Dependent Variables** settings window, locate the **General** section.

5 From the **Defined by study step** list, choose **User defined**.

6 Locate the **Values of Variables Not Solved For** section. From the **Method** list, choose **Solution**.

7 From the **Solution** list, choose **Parametric 2**.

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

9 From the **Out-of-plane wave number** list, choose **54.19438+1.189336e-5i**.

10 Right-click **Study 2>Solver Configurations>Solver 6>Dependent Variables 1** and choose **Compute**.

RESULTS

STUDY 2

Solver 6

1 In the **Model Builder** window, under **Study 2>Solver Configurations** right-click **Solver 6** and choose **Rename**.
2 Go to the **Rename Solver** dialog box and type `m = 4` in the **New name** edit field.
3 Click **OK**.
4 Right-click **Study 2>Solver Configurations>Solver 6** and choose **Compute**.

RESULTS

Acoustic Pressure (ae)

Follow these steps to reproduce the plot in Figure 2.

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** click **Solution 6**.
2 In the **Solution** settings window, locate the **Solution** section.
3 In the **Solution at angle (phase)** edit field, type `180`.
4 Right-click **Results>Data Sets>Solution 6** and choose **Add Selection**.
5 In the **Selection** settings window, locate the **Geometric Entity Selection** section.
6 From the **Geometric entity level** list, choose **Domain**.
7 Select Domains 1, 2, and 5 only.
8 Right-click **Solution 6** and choose **Rename**.
9 Go to the **Rename Solution** dialog box and type `ae: m = 4, n = 0` in the **New name** edit field.
10 Click **OK**.

Acoustic Pressure (ae)

1 In the **2D Plot Group** settings window, click to expand the **Title** section.
2 From the **Title type** list, choose **Manual**.
3 In the **Title** text area, type `Surface: Velocity potential (m = 4, n = 0)`.
4 In the **Model Builder** window, expand the **Acoustic Pressure (ae)** node, then click **Surface 1**.
5 In the **Surface** settings window, locate the **Coloring and Style** section.

6 From the **Color table** list, choose **GrayScale**.

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

8 In the **Model Builder** window, right-click **Acoustic Pressure (ae)** and choose **Rename**.

9 Go to the **Rename 2D Plot Group** dialog box and type `phi (m = 4, n = 0)` in the **New name** edit field.

10 Click **OK**.

You have now solved the example for the eigenmode m = 4 and n = 0. To generate Figure 3 you need to solve the example with m = 17 and n = 1 and to generate Figure 4 you need to use m = 24 and n = 1. Begin with the mode m = 17, n = 1. First, however, delete the default plot groups that you did not use.

Sound Pressure Level (ae)

1 In the **Model Builder** window, under **Results** right-click **Sound Pressure Level (ae)** and choose **Delete**.

2 Click **Yes** to confirm.

Repeat this procedure for the last two plot groups.

GLOBAL DEFINITIONS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
m	17	17.00	Circumferential wave number

STUDY 2

m = 4

In the **Model Builder** window, under **Study 2>Solver Configurations** right-click **m = 4** and choose **Disable**.

Solver 7

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 7** node, then click **Dependent Variables 1**.

3 In the **Dependent Variables** settings window, locate the **General** section.
4 From the **Defined by study step** list, choose **User defined**.
5 Locate the **Values of Variables Not Solved For** section. From the **Method** list, choose **Solution**.
6 From the **Solution** list, choose **Parametric 2**.
7 From the **Use** list, choose **m=17**.
8 From the **Out-of-plane wave number** list, choose the value with the second-highest real part.
9 In the **Model Builder** window, right-click **Solver 7** and choose **Rename**.
10 Go to the **Rename Solver** dialog box and type `m = 17` in the **New name** edit field.
11 Click **OK**.
12 Right-click **Solver 7** and choose **Compute**.

RESULTS

Acoustic Pressure (ae)

Reproduce the plot in Figure 3 as follows.

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** click **Solution 7**.
2 In the **Solution** settings window, locate the **Solution** section.
3 In the **Solution at angle (phase)** edit field, type `180`.
4 Right-click **Results>Data Sets>Solution 7** and choose **Add Selection**.
5 In the **Selection** settings window, locate the **Geometric Entity Selection** section.
6 From the **Geometric entity level** list, choose **Domain**.
7 Select Domains 1, 2, and 5 only.
8 Right-click **Solution 7** and choose **Rename**.
9 Go to the **Rename Solution** dialog box and type `ae: m = 17, n = 1` in the **New name** edit field.
10 Click **OK**.

Acoustic Pressure (ae)

1 In the **2D Plot Group** settings window, locate the **Title** section.
2 From the **Title type** list, choose **Manual**.
3 In the **Title** text area, type `Surface: Velocity potential (m = 17, n = 1)` .

4 In the **Model Builder** window, expand the **Acoustic Pressure (ae)** node, then click **Surface 1**.
5 In the **Surface** settings window, locate the **Coloring and Style** section.
6 From the **Color table** list, choose **GrayScale**.
7 Click the **Zoom Extents** button on the Graphics toolbar.
8 In the **Model Builder** window, right-click **Acoustic Pressure (ae)** and choose **Rename**.
9 Go to the **Rename 2D Plot Group** dialog box and type `phi (m = 17, n = 1)` in the **New name** edit field.
10 Click **OK**.

Again, remove the three unused default plot groups before proceeding.

Sound Pressure Level (ae)

1 In the **Model Builder** window, under **Results** right-click **Sound Pressure Level (ae)** and choose **Delete**.
2 Click **Yes** to confirm.

Repeat the procedure for 3D Plot Group 6 and 3D Plot Group 7.

Finally, consider the mode m = 24, n = 1.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2 In the **Parameters** settings window, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
m	24	24.00	Circumferential wave number

STUDY 2

m = 17

In the **Model Builder** window, under **Study 2>Solver Configurations** right-click **m = 17** and choose **Disable**.

Solver 8

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 8** node, then click **Dependent Variables 1**.
3 In the **Dependent Variables** settings window, locate the **General** section.
4 From the **Defined by study step** list, choose **User defined**.
5 Locate the **Values of Variables Not Solved For** section. From the **Method** list, choose **Solution**.
6 From the **Solution** list, choose **Parametric 2**.
7 From the **Use** list, choose **m=24**.
8 From the **Out-of-plane wave number** list, choose the value with the second highest real part.
9 In the **Model Builder** window, right-click **Solver 8** and choose **Rename**.
10 Go to the **Rename Solver** dialog box and type `m = 24` in the **New name** edit field.
11 Click **OK**.
12 Right-click **Solver 8** and choose **Compute**.

RESULTS

Acoustic Pressure (ae)

These steps reproduce the plot in Figure 4.

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** click **Solution 8**.
2 In the **Solution** settings window, locate the **Solution** section.
3 In the **Solution at angle (phase)** edit field, type `180`.
4 Right-click **Results>Data Sets>Solution 8** and choose **Add Selection**.
5 In the **Selection** settings window, locate the **Geometric Entity Selection** section.
6 From the **Geometric entity level** list, choose **Domain**.
7 Select Domains 1, 2, and 5 only.
8 Right-click **Solution 8** and choose **Rename**.
9 Go to the **Rename Solution** dialog box and type `ae: m = 24, n = 1` in the **New name** edit field.
10 Click **OK**.

Acoustic Pressure (ae)

1 In the **2D Plot Group** settings window, locate the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Surface: Velocity potential (m = 24, n = 1)`.

4 In the **Model Builder** window, expand the **Acoustic Pressure (ae)** node, then click **Surface 1**.

5 In the **Surface** settings window, locate the **Coloring and Style** section.

6 From the **Color table** list, choose **GrayScale**.

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

8 In the **Model Builder** window, right-click **Acoustic Pressure (ae)** and choose **Rename**.

9 Go to the **Rename 2D Plot Group** dialog box and type `phi (m = 24, n = 1)` in the **New name** edit field.

10 Click **OK**.

Finally, the image you see when opening this model from the Acoustics Module Model Library is produced from the last obtained solution (m = 24, n = 1) with the following steps:

Sound Pressure Level (ae)

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

2 In the **2D Plot Group** settings window, locate the **Plot Settings** section.

3 Clear the **Plot data set edges** check box.

4 In the **Model Builder** window, expand the **Sound Pressure Level (ae)** node.

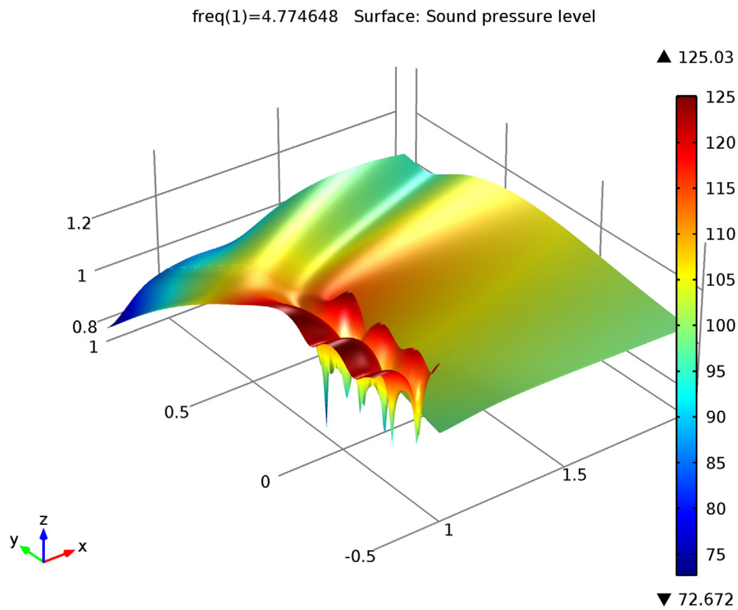
5 Right-click **Surface 1** and choose **Height Expression**.

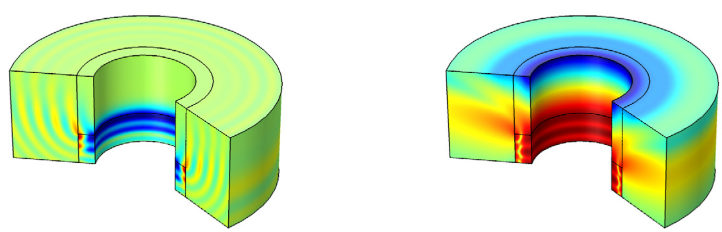
6 Right-click **Sound Pressure Level (ae)** and choose **Rename**.

7 Go to the **Rename 2D Plot Group** dialog box and type `SPL (m = 24, n = 1)` in the **New name** edit field.

8 Click **OK**.

The remaining 3D default plots visualize the acoustic pressure and the sound pressure level as revolved surface plots..





Pressure (left) and sound pressure level (right) for the mode m = 24, n = 1.