Model created in COMSOL Multiphysics 6.4

Flow Duct

Introduction

The modeling of aircraft-engine noise attenuation is a central problem in the field of computational aeroacoustics (CAA). In this example you simulate the harmonically time-varying acoustic field from a turbofan engine, under various background flow conditions (a convected acoustic simulation), and analyze the modal sound transmission loss made possible by introducing a layer of lining inside the engine duct. The source is generated by a single mode excitation at a boundary, the source plane, see Figure 2. Sources and nonreflecting conditions are applied using Port boundary conditions, with the built-in Annular and Circular port type options. These port types are based on analytical and semi-analytical solutions to the uniform flow and hard walled configuration. The model analysis is performed in two steps, first computing the background mean flow (compressible irrotational potential flow) and then solving the acoustic field in the flow duct with the linearized potential flow equations. Results are presented for situations with and without a background flow and for the cases of hard and lined duct walls.

Two other variants of the model exist:

- Flow Duct — With Boundary Mode Analysis, where the hard walled port modes are computed using the boundary mode physics interface.
- Flow Duct — Modes with Impedance Condition, where the mode computation includes the impedance conditions.

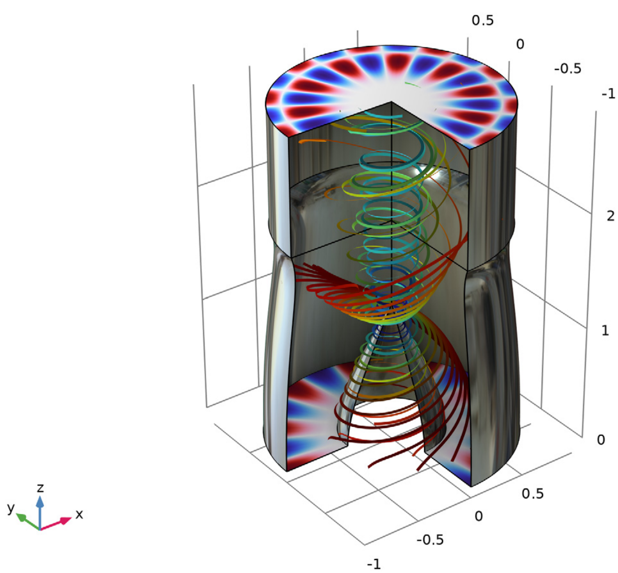


Figure 1: The flow duct configuration showing the intensity lines and selected port modes.

Model Definition

The 2D axisymmetric duct geometry representation used in this model, shown in Figure 2, is taken from Ref. 1. It is an approximate model of the inlet section of a turbofan engine in the very common CFM56 series.

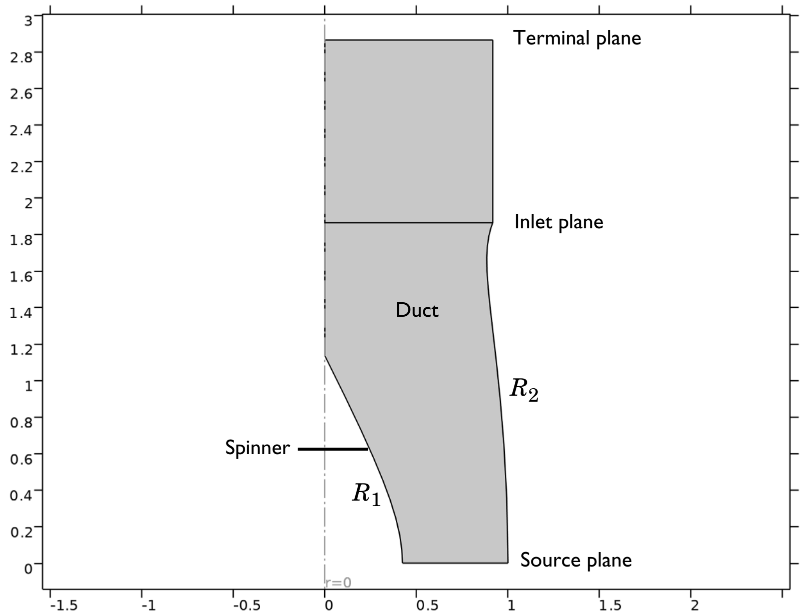


Figure 2: The duct geometry including reference planes used in the model.

The spinner and duct-wall profiles are given, respectively, by the equations

$$R_1(z') = \max[0, 0{,}64212 - (0{,}04777 + 0{,}98234 z'^2)^{1/2}]$$

$$R_2(z') = 1 - 0{,}18453 z'^2 + 0{,}10158 \frac{e^{-11(1-z')} - e^{-11}}{1 - e^{-11}}$$

where $0 \le z' = z/L \le 1$, and L = 1.86393 is the duct length. A noise source is imposed at $z' = 0$, henceforth referred to as the *source plane*. This is where the fan would be located in the actual engine geometry. The plane $z = L$ corresponds to the fore end of the engine and is referred to as the *inlet plane*. The attenuation of the liner for specific flow conditions is computed from the source plane to the inlet plane. A cylindrical domain, adjoined at the inlet plane and extending to the *terminal plane*, extends the modeling domain into a region where you can consider the mean flow as being uniform. This allows you to impose the simple boundary condition of a constant velocity potential and a vanishing tangential velocity for the background flow. For the acoustic problem, port boundary conditions are used at the source plane and the terminal plane to set up ideal nonreflecting conditions as well as imposing the source.

The model will analyze so-called modal sound transmission, where a single propagating mode is used as source. In this particular example the first radial mode is used as the source, see Ref. 1 for details. All propagating modes are used when setting up the ports to ensure good nonreflecting performance. The sound transmission loss is computed from the source plane to the inlet plane. The power of the incident mode is defined through a predefined variable and the power of the transmitted sound at the inlet plane is computed as the integral of the axial intensity.

MODEL CONDITIONS

Assume that the flow in the axisymmetric duct is compressible, inviscid, perfectly isentropic, and irrotational. This is an assumption often used for the study of duct or engine acoustics. In this case the background mean flow is well described by the *Compressible Potential Flow* interface and the acoustic field is well described by the *Linearized Potential Flow, Frequency Domain* interface.

For more theory information on the governing equations, see the aeroacoustics theory chapter in the *Acoustics Module User's Guide*.

Compressible Potential Flow

This study examines two cases for the mean-flow normal velocity component at the source plane V_z, which (owing to the choice of reference speed) alternatively can be referred to as the source-plane axial Mach number $M = -0.5$, approximately representative of a passenger aircraft at cruising speed, and $M = 0$.

The governing equations are nondimensionalized in the present study. For the reference quantities in this model, choose the duct radius, the mean-flow speed of sound, and the mean-flow density at the source plane. Hence, all three of these quantities take the value 1.

The remaining boundary conditions for the mean flow consist of a natural boundary condition specifying the mass-flow rate through the source plane via the normal velocity and the density; slip conditions (vanishing tangential velocity) at the duct wall and at the spinner; and axial symmetry at $r = 0$.

Linearized Potential Flow

For the aeroacoustic field, the model considers two different boundary conditions at the duct wall:

- *Sound hard* — the normal component of the acoustic particle velocity vanishes at the boundary.
- *Impedance* — the normal component of the acoustic particle velocity is related to the particle displacement through the equation

$$i\omega(\mathbf{u}\cdot\mathbf{n}) = [i\omega + \mathbf{u}_0\cdot\nabla - (\mathbf{n}\cdot(\mathbf{n}\cdot\nabla\mathbf{u}_0))]\frac{p}{Z}$$

  where Z is the impedance, $\mathbf{u}_0$ is the mean background flow, p is the acoustic pressure, and $\mathbf{u}$ is the acoustic velocity. This condition is often referred to as the *Ingard–Myers* impedance condition. This boundary condition, first derived by Myers (Ref. 2), was later recast in a weak form by Eversman (Ref. 3); it is this weak version, which is directly suitable for finite element modeling, that is implemented in the Acoustics Module's Linearized Potential Flow, Frequency Domain interface. The impedance boundary condition represents a lined duct wall. In this model, following Ref. 1, the impedance is taken to be $Z = 2 - i$.

The spinner, in contrast, is always assumed to be acoustically hard.

One of the configurations from Ref. 1 is studied in this model. This is the case where the dimensionless angular frequency (nondimensionalized through division by R_∞/c_∞) is $\omega = 16$, and the azimuthal mode number is $m = 10$. If you want to obtain a deeper understanding of the duct's aeroacoustic characteristics, you can, of course, perform a systematic exploration of parameter space by varying these quantities independently. Several more cases are examined in the reference paper.

Results and Discussion

THE MEAN-FLOW FIELD

For the nontrivial case of a source-plane axial Mach number of $M = -0.5$, the resulting mean-flow field appears in Figure 3. Note that the velocity potential is uniform well beyond the terminal plane, thus justifying the boundary condition imposed there. Furthermore, as could be expected, deviations from the mean density value appear primarily near the nonuniformities of the duct geometry, such as at the tip of the spinner.

As a complement, a more quantitative picture of the variations of the mean-flow velocity and density profiles along the axial direction (for $r = 0.8$) appear in the cross-section plots in Figure 4.

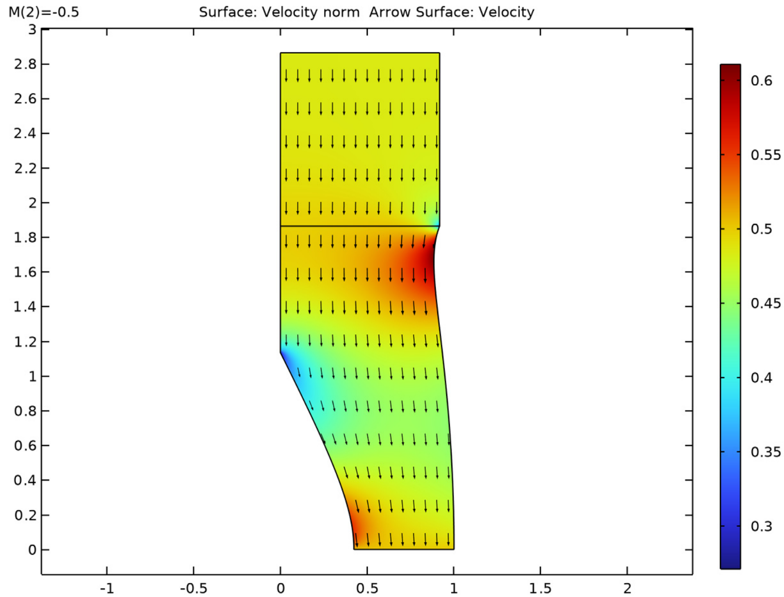


Figure 3: Mean-flow velocity potential and density for source-plane Mach number $M = -0.5$.

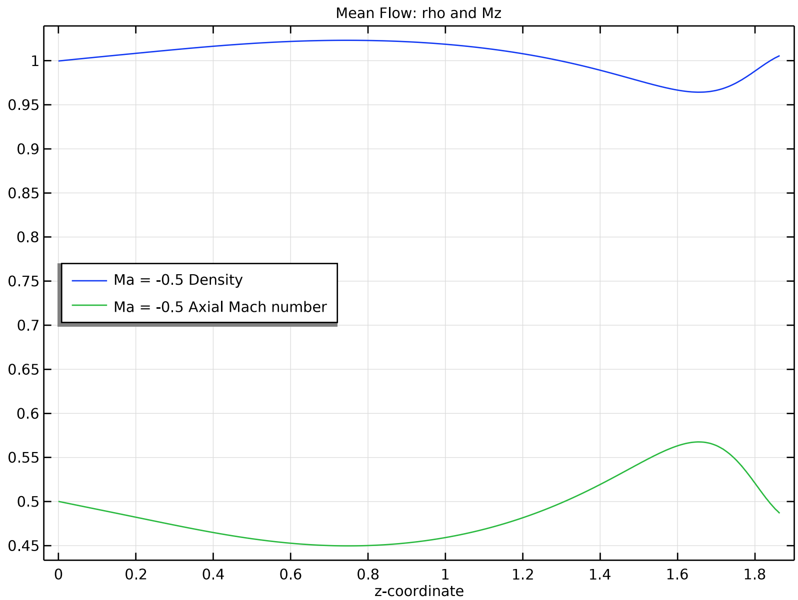


Figure 4: Mean-flow cross section plot at a sample radius of 0.8.

The Aeroacoustic Field

The normalized pressure fields for the case without a no background mean flow ($M = 0$), shown in Figure 5, very closely match those for the corresponding finite element model (FEM) solutions presented in Figure 6 of Ref. 1. Similarly, the results for the attenuation between the source and inlet planes in the lined-wall case are in good agreement: 51.0 dB for the COMSOL Multiphysics solution versus 51.6 dB for the FEM solution, as shown in Table. 1 in Ref. 1.

Turning to the case with a mean flow ($M = -0.5$), the pressure field for the hard-wall as well as the lined wall (soft wall) cases in Figure 6 closely resembles the FEM solution obtained by Rienstra and Eversman in Ref. 1. This observation extends to the attenuation, for which the calculated value of 28.3 dB is in good agreement with the value of 27.2 dB obtained in Ref. 1.

Note that the port modes (including the source) in the COMSOL Multiphysics calculation was derived for the case of a hard duct wall with uniform flow, whereas Rienstra and Eversman used a noise source adapted to the acoustic lining. However this fact does not seem to have a large influence on the solution for this particular problem. The propagating mode for the lined wall is actually a linear combination of the two hard-wall propagating modes. In the model Flow Duct — Modes with Impedance Condition the modes that match the impedance condition at the lined walls are computed and used when setting up the port conditions. In another variant of the Flow Duct model, the Flow Duct — With Boundary Mode Analysis model, the hard walled modes are computed using a boundary mode analysis. This has a small effect at the source plane, where the flow is not uniform.

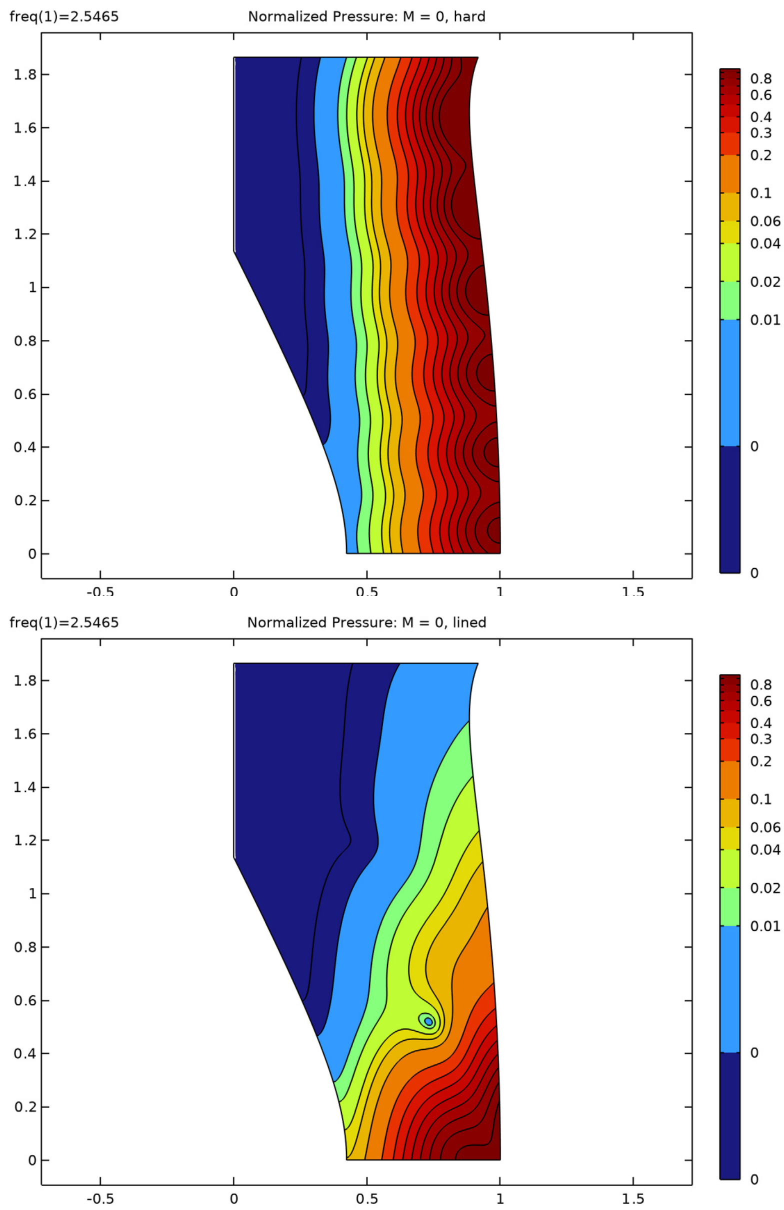


Figure 5: Acoustic pressure field for the cases of hard (top) and lined (bottom) duct wall with no mean flow ($M = 0$); azimuthal mode number m = 10 and angular frequency ω = 16.

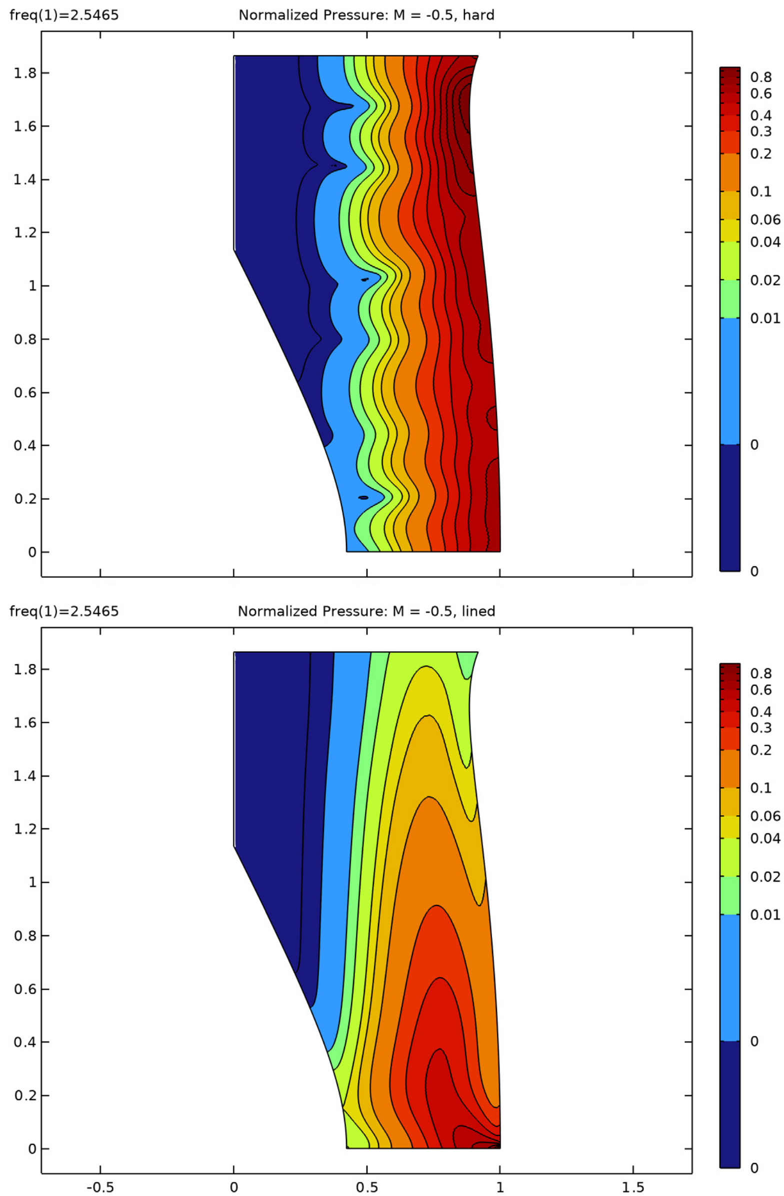


Figure 6: Acoustic pressure distribution for the cases of hard (top) and lined (bottom) duct wall with mean flow ($M = -0.5$); azimuthal mode number $m = 10$ and angular frequency $\omega = 16$.

PHYSICS INTERFACES

- *Compressible Potential Flow* (cpf) — for modeling the background mean-flow velocity field as a potential flow (a lossless and irrotational flow).
- *Linearized Potential Flow, Frequency Domain* (lpff) — for modeling the time-harmonic acoustic field in the duct for the various excitation and flow condition. The Port condition is used at the source and terminal planes using the built-in Annular and Circular port type options.

MODES USED AT THE PORTS

At the ports (the source and terminal planes) only the modes based on the hard walled and uniform flow configuration are used (the built-in semi-analytical options). In the lined case this results in a small numerical error near the wall. This can be visualized by plotting the intensity magnitude (`lpff.I_mag`) for the lined solutions, as seen in Figure 7. The intensity field for the hard walled configuration is seen in Figure 8.

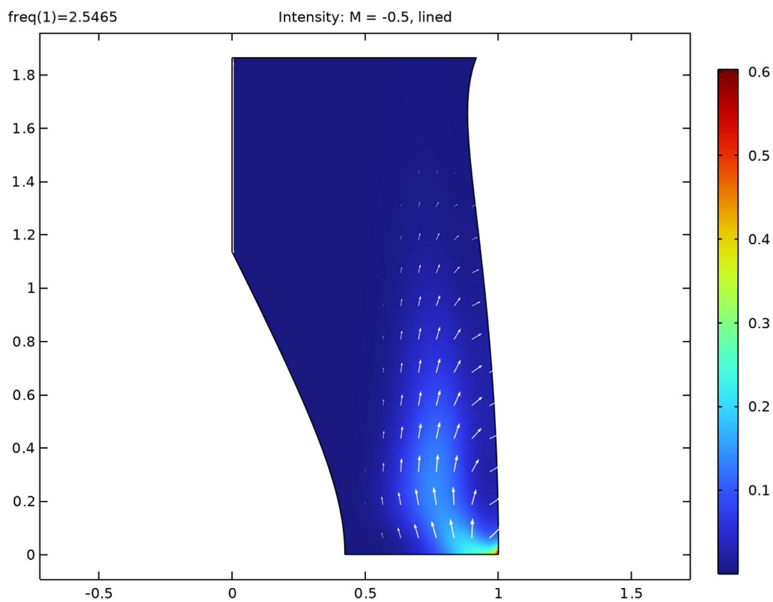


Figure 7: Intensity field in the lined case.

Computing and using the correct modes, including the impedance (lining) is possible. An **Impedance** condition can be added to the Boundary Mode interfaces, and the numerically computed modes can be used at the ports. However, identifying and sorting the modes is more involved in this case. For an extension of the current model using impedance conditions see the Flow Duct — Modes with Impedance Condition library model.

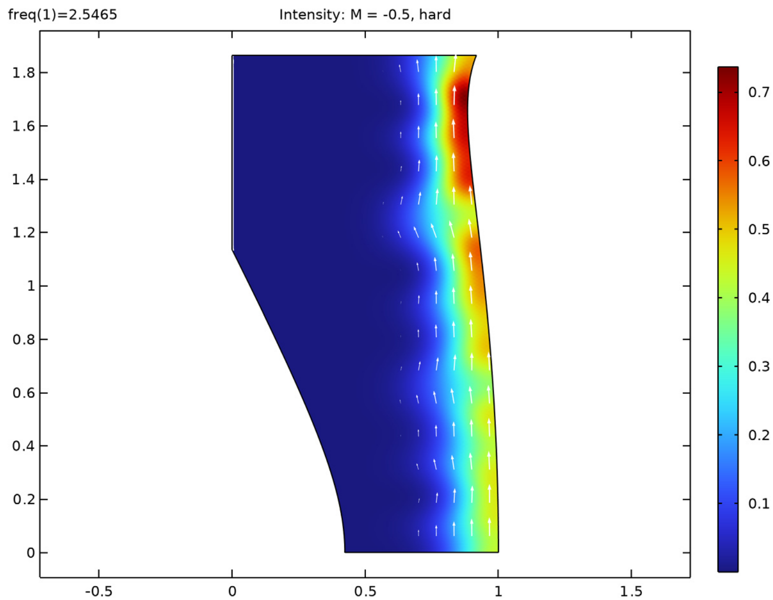


Figure 8: Intensity field in the hard walled case.

References

1. S.W. Rienstra and W. Eversman, "A Numerical Comparison Between the Multiple-Scales and Finite-Element Solution for Sound Propagation in Lined Flow Ducts," *J. Fluid Mech.*, vol. 437, pp. 367–384, 2001.

2. M.K. Myers, "On the Acoustic Boundary Condition in the Presence of Flow," *J. Sound Vib.*, vol. 71, pp. 429–434, 1980.

3. W. Eversman, "The Boundary Condition at an Impedance Wall in a Non-Uniform Duct with Potential Mean Flow," *J. Sound Vib.*, vol. 246, pp. 63–69, 2001. Errata: *ibid*, vol. 258, pp. 791–792, 2002.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/flow_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** > **Compressible Potential Flow (cpf)**.

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** > **Stationary**.

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.

GLOBAL DEFINITIONS

Parameters 1

Load the parameters from a file. They define model, geometry, and physical properties including the liner impedance. Then proceed and create the geometry of the duct.

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 `flow_duct_parameters.txt`.

Proceed and draw the geometry of the engine duct. Use the **Parametric Curve** features to draw the shapes defined by the functions described in the main document.

GEOMETRY 1

Parametric Curve 1 (pc1)

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

2 In the **Settings** window for **Parametric Curve**, locate the **Expressions** section.

3 In the **r** text field, type `1-0.18453*s^2+0.10158*(exp(-11*(1-s))-exp(-11))/(1-exp(-11))`.

4 In the **z** text field, type `s*zi`.

Parametric Curve 2 (pc2)

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

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `0.7`.

4 Locate the **Expressions** section. In the **r** text field, type `0.64212-sqrt(0.04777+0.98234*s^2)`.

5 In the **z** text field, type `s*zi`.

Line Segment 1 (ls1)

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

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

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

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

5 In the **z** text field, type `zi`.

Union 1 (uni1)

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

2 Select the objects **ls1** and **pc2** only.

Line Segment 2 (ls2)

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

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

3 Click to select the **Activate Selection** toggle button for **Start vertex**.

4 On the object **uni1**, select Point 5 only.

5 Locate the **Endpoint** section. Click to select the **Activate Selection** toggle button for **End vertex**.

6 On the object **pc1**, select Point 1 only.

Line Segment 3 (ls3)

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

2 On the object **uni1**, select Point 4 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **pc1**, select Point 2 only.

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

Delete Entities 1 (del1)

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

2 On the object **uni1**, select Boundaries 1 and 3 only.

3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Click in the **Graphics** window and then press Ctrl+D to clear all objects.

3 Click the **Select All** button in the **Graphics** toolbar.

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 `ri`.

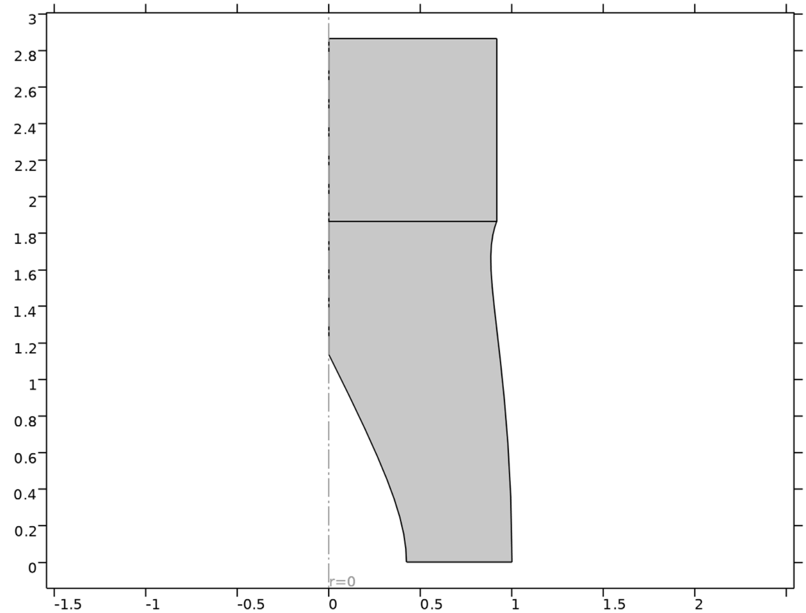
4 Locate the **Position** section. In the **z** text field, type `zi`.

Form Union (fin)

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

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

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



Proceed and set up variables used for the results analysis. One is a normalized absolute pressure which uses a maximum operator over the domain. Define selections for the source, inlet and terminal planes. Finally, define an integration operator used to compute the power through the inlet plane.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

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

4 Select Domain 1 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
Mz	-cpf.Vz		Axial Mach number
pabsn	abs(lpff.p)/ comp1.maxop1(lpff.p)		Normalized pressure

Source Plane

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Source Plane` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 5 only.

Inlet Plane

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Inlet Plane` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 3 only.

Terminal Plane

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Terminal Plane` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 4 only.

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.
2 Select Domain 1 only.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2 In the **Settings** window for **Integration**, type `intop_ip` in the **Operator name** text field.
3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Inlet Plane**.

Now proceed and set up the physics for the Compressible Potential Flow.

COMPRESSIBLE POTENTIAL FLOW (CPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Compressible Potential Flow (cpf)**.
2 In the **Settings** window for **Compressible Potential Flow**, locate the **Reference Values** section.
3 In the p_{ref} text field, type `cpf.rhoref^gamma/gamma`.

4 In the ρ_{ref} text field, type `rho0`.

5 In the v_{ref} text field, type `M`.

Compressible Potential Flow Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Compressible Potential Flow (cpf)** click **Compressible Potential Flow Model 1**.

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

3 From the γ list, choose **User defined**. In the associated text field, type `gamma`.

Normal Flow 1

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

2 In the **Settings** window for **Normal Flow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Terminal Plane**.

Mass Flow 1

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

2 In the **Settings** window for **Mass Flow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Source Plane**.

Set up a fully user defined mesh for the computational domain.

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 Domain 1 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 **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Select Points 4 and 7 only.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.005`.

Size

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

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

3 Click the **Custom** button.

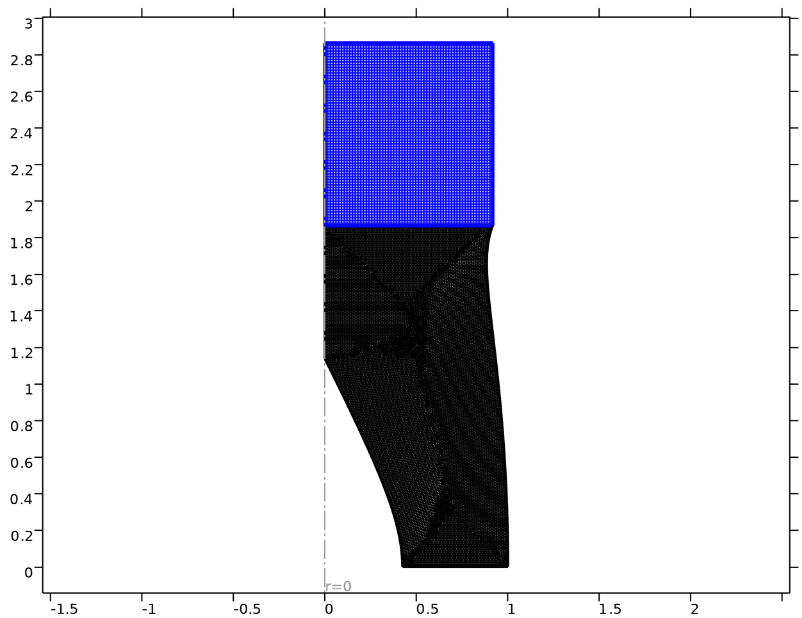
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.015`.

5 In the **Minimum element size** text field, type `0.001`.

Mapped 1

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

2 In the **Settings** window for **Mapped**, click **Build All**.



Now, first solve the background flow and look at the results. The flow is solved for two Mach numbers using a Parametric Sweep.

STUDY 1 - BACKGROUND FLOW

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

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

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 (Mean flow Mach number)	`0 -0.5`	

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

RESULTS

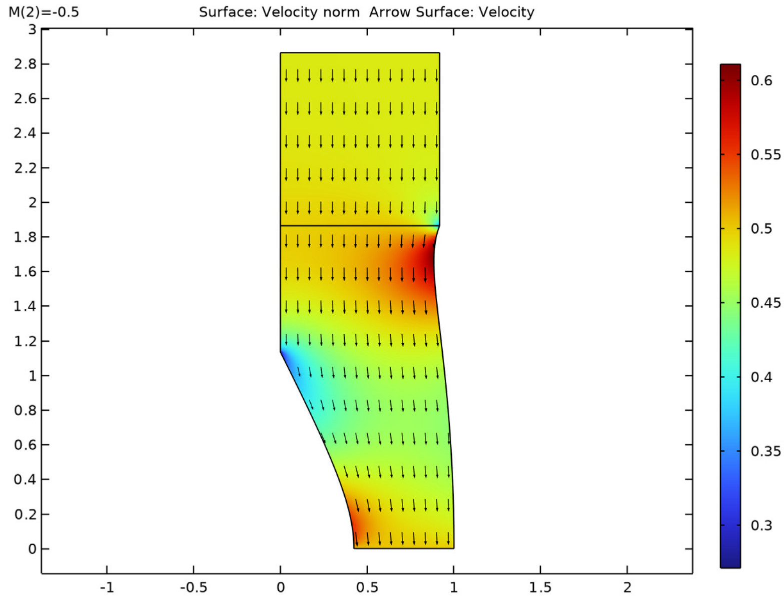
Arrow Surface 1

1 Right-click **Mean Flow Velocity (cpf)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Black**.

4 In the **Mean Flow Velocity (cpf)** toolbar, click **Plot**.



Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

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

3 In row **Point 1**, set **R** to `0.8`.

4 In row **Point 2**, set **R** to `0.8`.

5 In row **Point 2**, set **Z** to `zi`.

Mean Flow: rho and Mz

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

2 In the **Settings** window for **1D Plot Group**, type `Mean Flow: rho and Mz` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

4 From the **Parameter selection (M)** list, choose **Last**.

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

6 Locate the **Legend** section. From the **Position** list, choose **Middle left**.

Line Graph 1

1 Right-click **Mean Flow: rho and Mz** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `rho`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `z`.

6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

7 Find the **Include** subsection. Select the **Description** checkbox.

8 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Ma = `.

9 In the **Mean Flow: rho and Mz** toolbar, click **Plot**.

Line Graph 2

1 In the **Model Builder** window, right-click **Mean Flow: rho and Mz** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `Mz`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

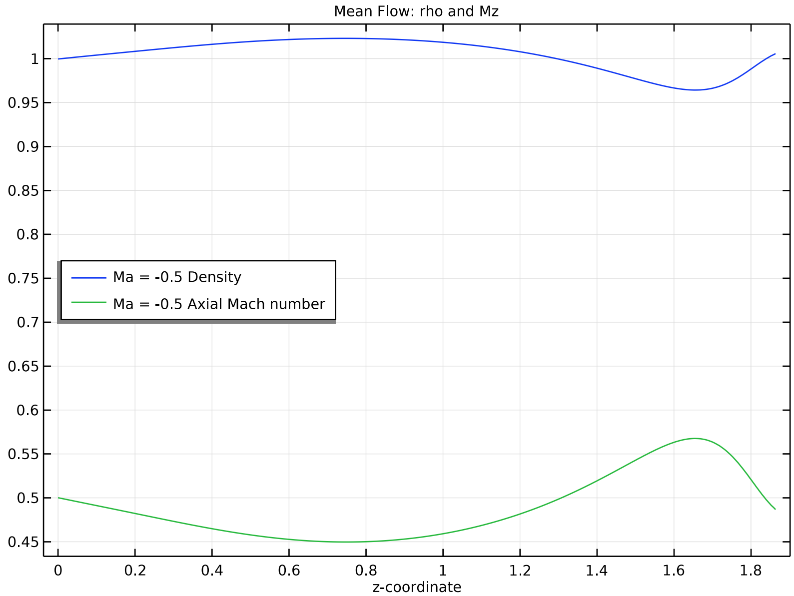
5 In the **Expression** text field, type `z`.

6 Locate the **Legends** section. Select the **Show legends** checkbox.

7 Find the **Include** subsection. Select the **Description** checkbox.

8 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Ma = `.

9 In the **Mean Flow: rho and Mz** toolbar, click **Plot**.



Finally, set up the physics and boundary conditions for the Linearized Potential Flow, Frequency Domain physics interface. Of particular importance is the setup of the Port conditions. The ports are divided into those applied at the source and inlet planes.

At the source plane the **Annular** port option is used while at the terminal plane the **Circular** port option is used. These port type options automatically compute the mode shapes, based on analytical and semi-analytical expressions. Note that it is assumed that the walls are sound hard and that the flow is uniform. The flow is uniform at the terminal plane while being only nearly uniform at the source plane. Note also that the modes will not fulfill the impedance conditions in the lined configurations. This assumptions is often used in real systems where the engine source has been projected onto the hard-walled modes.

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 **Power normalization**.

MULTIPHYSICS

Background Potential Flow Coupling 1 (pfc1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global** > **Background Potential Flow Coupling**.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (LPFF)

Impedance 1

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

2 Select Boundaries 6 and 8 only.

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

4 In the Z_{n} text field, type `Zw`.

Source Plane

1 In the **Model Builder** window, right-click **Linearized Potential Flow, Frequency Domain (lpff)** and choose **Node Group**.

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

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 **Source Plane**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Annular**.

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`.

Only the first radial mode is exciting the system in this modal transmission loss analysis.

Port 2

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 **Source Plane**.

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 `1`.

Terminal Plane

1 Right-click **Linearized Potential Flow, Frequency Domain (lpff)** and choose **Node Group**.

2 In the **Settings** window for **Group**, type `Terminal Plane` in the **Label** text field.

Port 3

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 **Terminal Plane**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Circular**.

Port 4

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 **Terminal Plane**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Circular**.

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

Solve the frequency domain model for the no-flow (M = 0) and flow (M = -0.5) cases as well as having a liner (finite impedance) and a sound hard configuration. Then analyze the results.

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** checkbox for **Compressible Potential Flow (cpf)**.

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

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

STUDY 2 - FREQUENCY DOMAIN (M = 0, LINED)

In the **Settings** window for **Study**, type `Study 2 - Frequency Domain (M = 0, lined)` in the **Label** text field.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Frequency Domain (M = 0, lined)** 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 - Background Flow, Stationary**.

7 From the **Parameter value (M)** list, choose **0**.

8 In the **Model Builder** window, click **Study 2 - Frequency Domain (M = 0, lined)**.

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

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

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

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** checkbox for **Compressible Potential Flow (cpf)**.

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

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

STUDY 3 - FREQUENCY DOMAIN (M = 0, HARD)

In the **Settings** window for **Study**, type `Study 3 - Frequency Domain (M = 0, hard)` in the **Label** text field.

1 In the **Model Builder** window, under **Study 3 - Frequency Domain (M = 0, hard)** 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 **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Impedance 1**.

6 Click **Disable**.

7 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**.

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

9 From the **Study** list, choose **Study 1 - Background Flow, Stationary**.

10 From the **Parameter value (M)** list, choose **0**.

11 In the **Model Builder** window, click **Study 3 - Frequency Domain (M = 0, hard)**.

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

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

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

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** checkbox for **Compressible Potential Flow (cpf)**.

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

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

STUDY 4 - FREQUENCY DOMAIN (M = -0.5, LINED)

In the **Settings** window for **Study**, type `Study 4 - Frequency Domain (M = -0.5, lined)` in the **Label** text field.

1 In the **Model Builder** window, under **Study 4 - Frequency Domain (M = -0.5, lined)** 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 - Background Flow, Stationary**.

7 From the **Parameter value (M)** list, choose **-0.5**.

8 In the **Model Builder** window, click **Study 4 - Frequency Domain (M = -0.5, lined)**.

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

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

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

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** checkbox for **Compressible Potential Flow (cpf)**.

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

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

STUDY 5 - FREQUENCY DOMAIN (M = -0.5, HARD)

In the **Settings** window for **Study**, type `Study 5 - Frequency Domain (M = -0.5, hard)` in the **Label** text field.

1 In the **Model Builder** window, under **Study 5 - Frequency Domain (M = -0.5, hard)** 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 **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Impedance 1**.

6 Click **Disable**.

7 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**.

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

9 From the **Study** list, choose **Study 1 - Background Flow, Stationary**.

10 From the **Parameter value (M)** list, choose **-0.5**.

11 In the **Model Builder** window, click **Study 5 - Frequency Domain (M = -0.5, hard)**.

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

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

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

RESULTS

Normalized Pressure: M = 0, lined

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Normalized Pressure: M = 0, lined` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Domain (M = 0, lined)/Solution 2 (sol2)**.

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

5 Select Domain 1 only.

6 Select the **Apply to dataset edges** checkbox.

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

Contour 1

1 Right-click **Normalized Pressure: M = 0, lined** and choose **Contour**.

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

3 In the **Expression** text field, type `pabsn`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.0001 0.001 0.01 0.02 0.04 0.06 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9`.

6 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

7 From the **Scale** list, choose **Logarithmic**.

8 In the **Normalized Pressure: M = 0, lined** toolbar, click **Plot**.

Contour 2

1 Right-click **Contour 1** and choose **Duplicate**.

2 In the **Settings** window for **Contour**, locate the **Coloring and Style** section.

3 From the **Contour type** list, choose **Line**.

4 From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

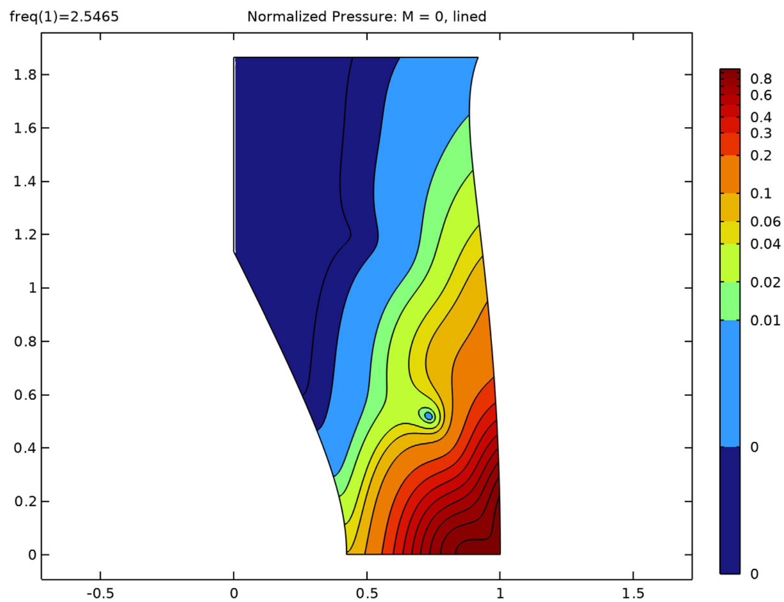
6 Clear the **Color legend** checkbox.

Normalized Pressure: M = 0, lined

1 In the **Model Builder** window, click **Normalized Pressure: M = 0, lined**.

2 In the **Normalized Pressure: M = 0, lined** toolbar, click **Plot**.

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



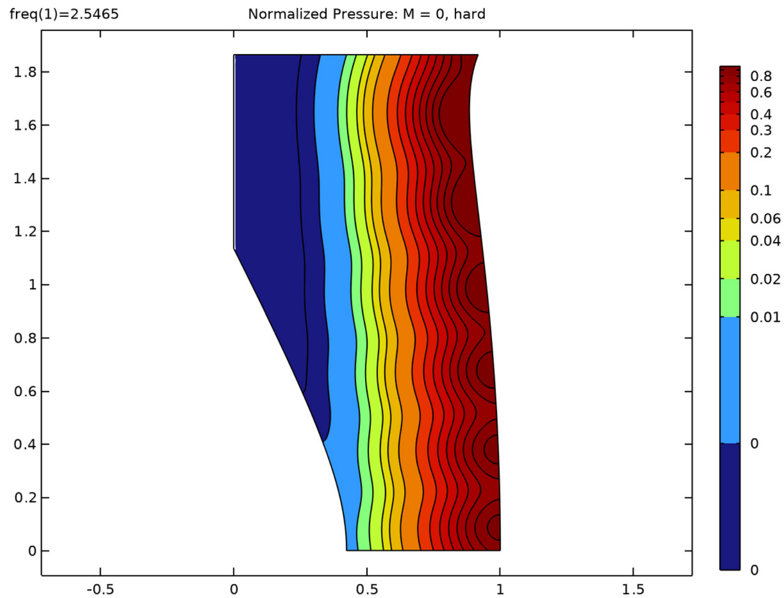
Normalized Pressure: M = 0, hard

1 Right-click **Normalized Pressure: M = 0, lined** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Normalized Pressure: M = 0, hard` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Frequency Domain (M = 0, hard)/Solution 3 (sol3)**.

4 In the **Normalized Pressure: M = 0, hard** toolbar, click **Plot**.



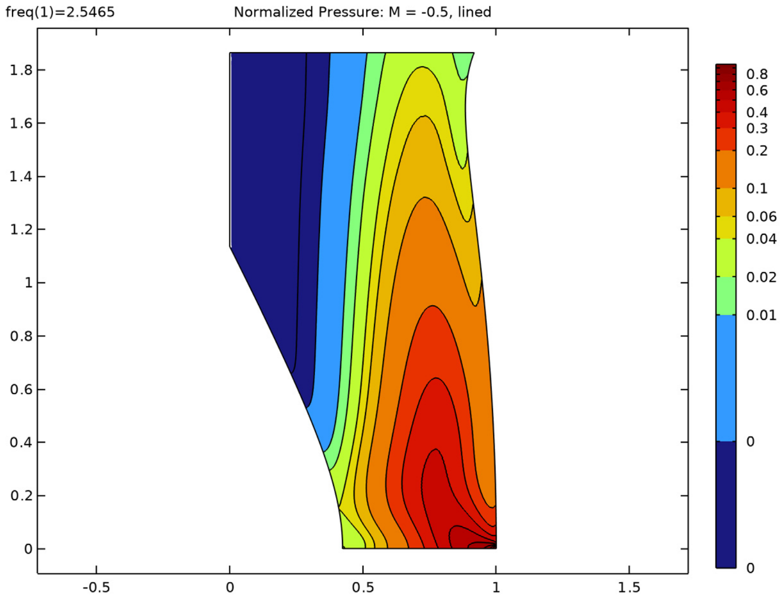
Normalized Pressure: M = -0.5, lined

1 Right-click **Normalized Pressure: M = 0, hard** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Normalized Pressure: M = -0.5, lined` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4 - Frequency Domain (M = -0.5, lined)/Solution 4 (sol4)**.

4 In the **Normalized Pressure: M = -0.5, lined** toolbar, click **Plot**.



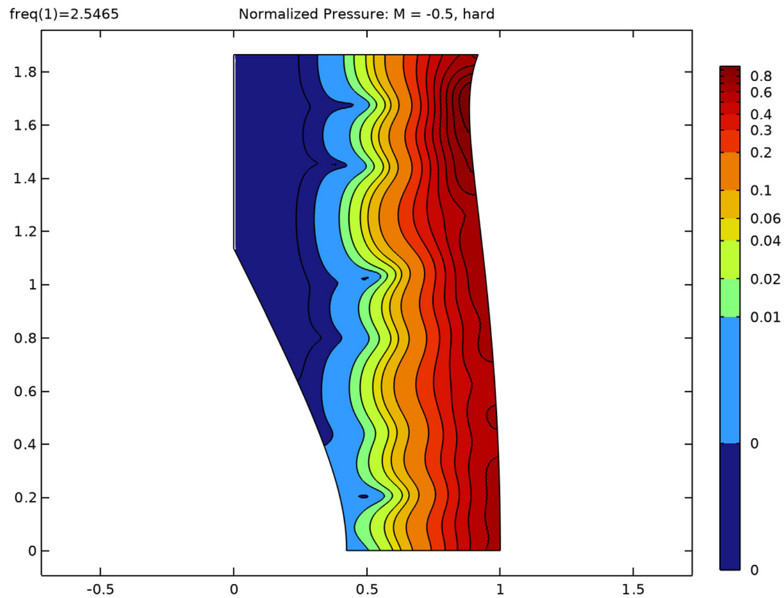
Normalized Pressure: M = -0.5, hard

1 Right-click **Normalized Pressure: M = -0.5, lined** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Normalized Pressure: M = -0.5, hard` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 5 - Frequency Domain (M = -0.5, hard)/Solution 5 (sol5)**.

4 In the **Normalized Pressure: M = -0.5, hard** toolbar, click **Plot**.



Create two plots of the intensity magnitude and the intensity field. The plots are here for the flow case. The plots illustrate how the sound-hard modes introduce a small error at the inlet, as they do not match the lined condition.

Intensity: M = -0.5, lined

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Intensity: M = -0.5, lined` in the **Label** text field.

3 Click to expand the **Selection** section. Locate the **Data** section. From the **Dataset** list, choose **Study 4 - Frequency Domain (M = -0.5, lined)/Solution 4 (sol4)**.

4 Locate the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 1 only.

6 Select the **Apply to dataset edges** checkbox.

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

Surface 1

1 Right-click **Intensity: M = -0.5, lined** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain** > **Intensity** > **lpff.I_mag - Intensity magnitude - kg/s³**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Intensity: M = -0.5, lined** and choose **Arrow Surface**.

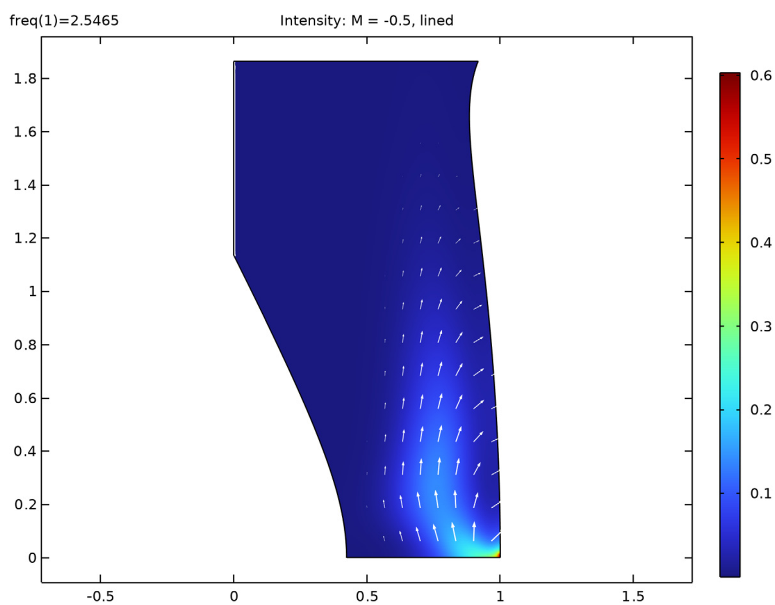
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain** > **Intensity** > **lpff.Ir,lpff.Iz - Intensity**.

3 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

4 From the **Color** list, choose **White**.

5 In the **Intensity: M = -0.5, lined** toolbar, click **Plot**.

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



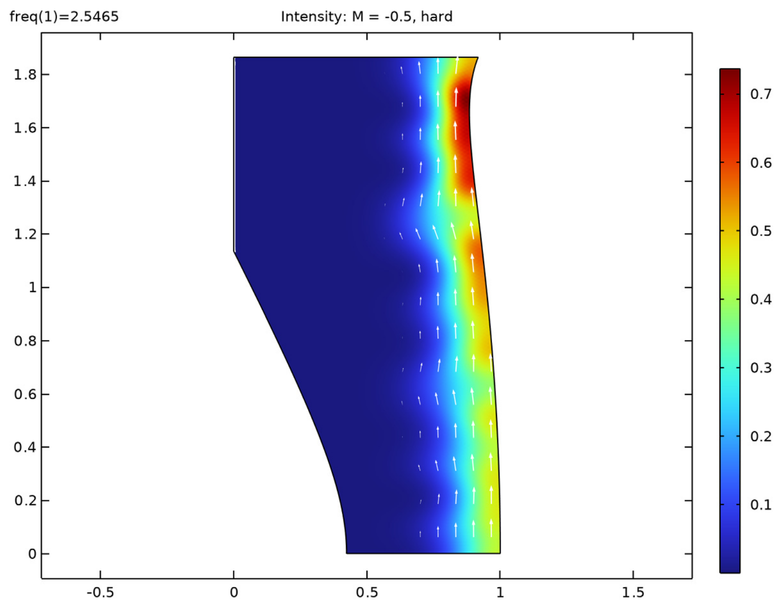
Intensity: M = -0.5, hard

1 Right-click **Intensity: M = -0.5, lined** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Intensity: M = -0.5, hard` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 5 - Frequency Domain (M = -0.5, hard)/Solution 5 (sol5)**.

4 In the **Intensity: M = -0.5, hard** toolbar, click **Plot**.



Create an evaluation group for computing the attenuation of the propagating mode when the liner is present in the model.

Evaluation Group: Attenuation

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

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group: Attenuation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

Global Evaluation 1

1 Right-click **Evaluation Group: Attenuation** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2 - Frequency Domain (M = 0, lined)/ Solution 2 (sol2)**.

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

Expression	Unit	Description
`10*log10(lpff.port1.P_in/intop_ip(lpff.Iz))`		`M = 0, lined`

Global Evaluation 2

1 In the **Model Builder** window, right-click **Evaluation Group: Attenuation** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 4 - Frequency Domain (M = -0.5, lined)/ Solution 4 (sol4)**.

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

Expression	Unit	Description
`10*log10(lpff.port1.P_in/ intop_ip(lpff.Iz))`		`M = -0.5, lined`

5 In the **Evaluation Group: Attenuation** toolbar, click **Evaluate**.

The final plot generates a nice thumbnail image, look at the plot for the details of the setup. Three additional datasets are created for setting up the plot.