Model created in COMSOL Multiphysics 6.4

Flow Duct — With Boundary Mode Analysis

Introduction

The acoustic field in a model of an axially symmetric lined aero-engine duct, based on modal sound transmission, is analyzed. The source is generated by a single mode excitation at a boundary. Sources and nonreflecting conditions are applied using port boundary conditions. The model analysis is performed in three steps: first computing the background mean flow (compressible irrotational potential flow), then analyzing the propagating modes with a boundary mode analysis, and finally solving the acoustic field in the lined flow-duct with the linearized potential flow equations.

This model represents an extension of the Flow Duct model where the modes used at the ports are computed numerically using the *Linearized Potential Flow, Boundary Mode* interface. Here assuming the hard wall conditions for the modes. The model illustrates how to reference the computed modes that result from a *Mode Analysis* study, and use them in the *Port* boundary conditions. Yet another variant of this tutorial exists, the Flow Duct — Modes with Impedance Condition, where the modes used at the ports are computed including the wall lining (impedance condition).

Model Definition

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

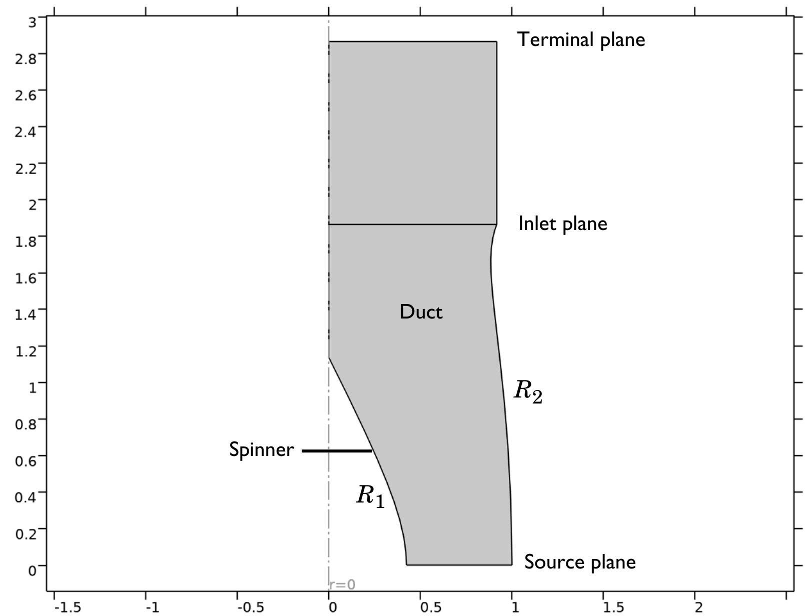


Figure 1: The duct geometry including reference planes used in the model.

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 2. 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 3.

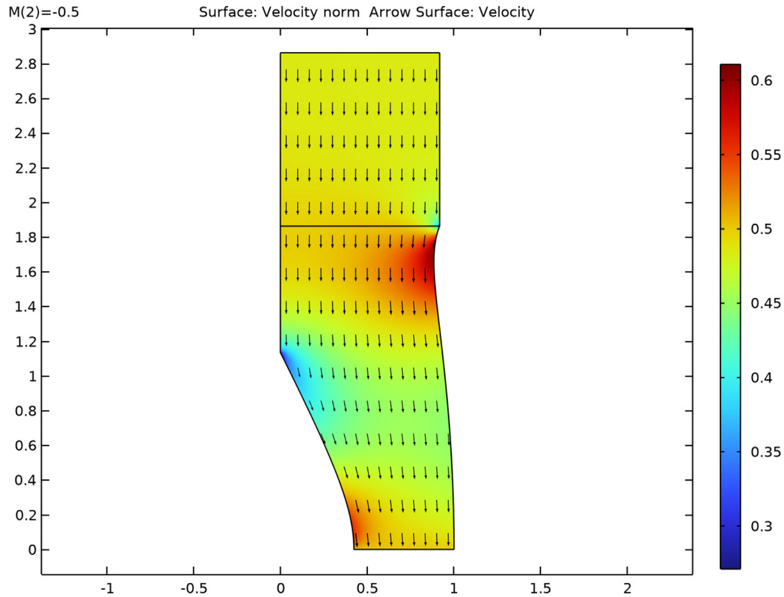


Figure 2: Mean-flow velocity potential and density for source-plane Mach number $M = -0.5$.

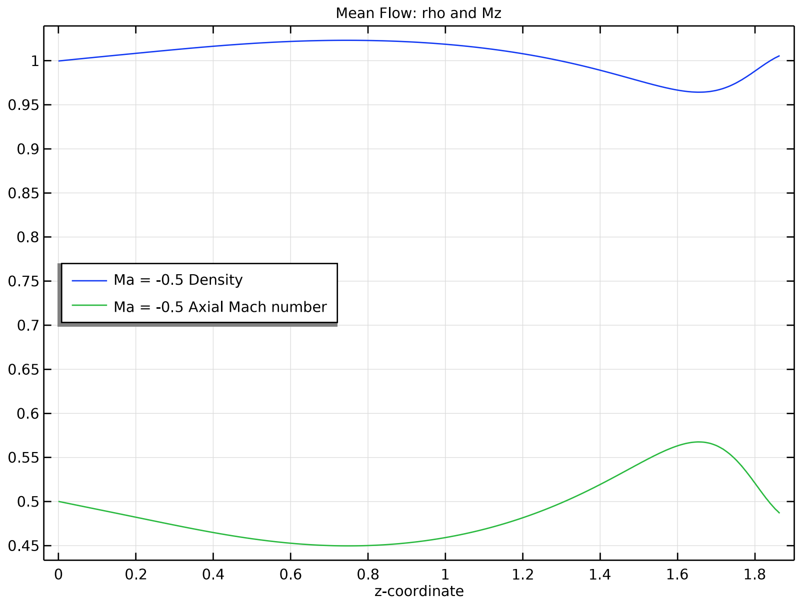


Figure 3: Mean-flow cross section plot at a sample radius of 0.8.

SOURCE-PLANE MODES

With the solution for the mean-flow field at hand, it is possible to calculate the corresponding eigenmodes for the acoustic field at the ports. To use the modes at the ports it is necessary to know if they are incident or outgoing. In the no-flow case this is simply done by looking at the sign of the wave vector. The outgoing mode has a positive wave number and the incident is negative. However, in the flow case this is not as straightforward because the presence of the mean background flow shifts the eigenvalues. Here it can be advantageous to look at the sign of the intensity vector at the port to identify the propagation direction.

A plot of the modes at the source plane ($z = 0$) including the orientation of the axial intensity vector I_z is depicted in Figure 4 for the flow case ($M = -0.5$). The source plane is located below the duct, such that an incident mode has a positive intensity and vice versa. Note that the ordering of the modes is done based on the settings in the **Filtering and Sorting** section of the **Mode Analysis** study step. In this case the modes are sorted by ascending real part of the wave number. Filtering of the modes is also possible as done for the analysis of the terminal plane.

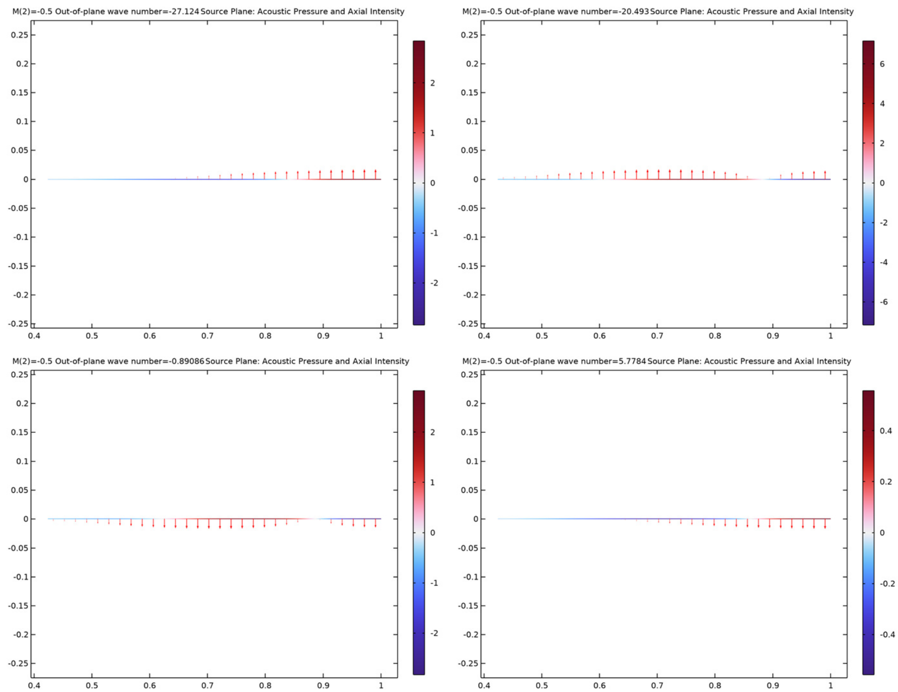


Figure 4: Mode pressures and axial intensity at the source plane (z = 0) for the case of a background flow with Mach number M = −0.5.

Figure 5 shows the resulting velocity-potential profile for all propagating modes at the source plane in the no-flow case ($M = 0$); and Figure 6 shows all the propagating modes in the flow case ($M = -0.5$). Notice that the mode profiles are not identical for the incident and outgoing modes (zoom in the graph in the model to see this clearly), indicating that the mean background flow is not uniform at this boundary.

Combining the information from all the plots, at the source plane, the mode identification necessary for the ports conditions can be done. In all cases the source is the first radial mode as defined in Ref. 1. See also the last two Evaluation Groups in the model where the solution index is shown for the various modes and flow conditions.

- At the source plane in the no-flow case ($M = 0$), the outgoing mode has the wave number $k_n = 10.8$. This is the second eigenvalue in solution list and thus as index 2 (necessary for referencing at the port). The incident mode (the source) has wave number k_n = –10.8 (index 1).

- At the source plane in the flow case (M = –0.5), the outgoing modes have wave numbers are k_n = –0.9 (index 3) and k_n = 5.8 (index 4). The first incident radial mode (the source) has wave number k_n = –27.1 (index 1). In the port condition it is advantageous for clarity (but not necessary) that modes corresponding to the same mode shape should be defined together. This means that the first radial modes (index 1 and 4) are referenced in one port, and the second outgoing radial mode (index 3) is referenced in the second port (where no source is added).

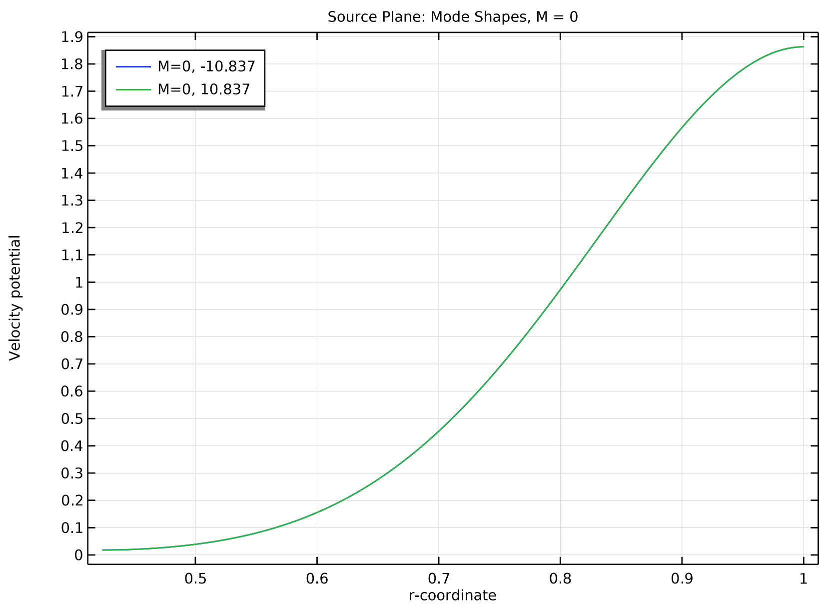


Figure 5: Propagating modes at the source plane for the no-flow case (M = 0).

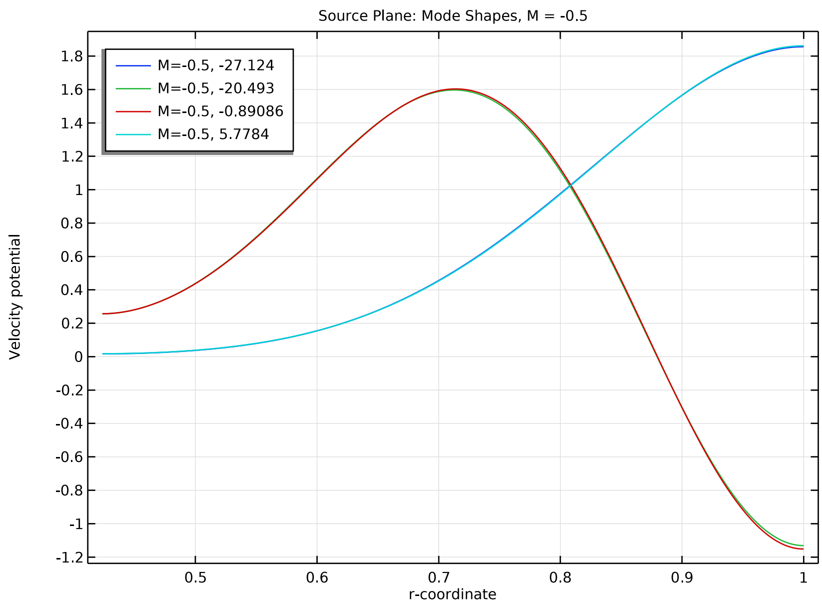


Figure 6: Propagating modes at the source plane for the flow case ($M = -0.5$).

TERMINAL-PLANE MODES

A plot of the outgoing modes at the terminal plane ($z = 1.86393$) including the orientation of the axial intensity vector I_z is depicted in Figure 7 for the flow case ($M = -0.5$). The terminal plane is located above the duct, such that an incident mode has a negative intensity and vice versa. For this analysis filtering of the modes is defined such that only outgoing modes are stored in the output. The logic expression `comp1.intop_tp(comp1.lpfbm2.Iz)` is used in the **Filtering and Sorting** section of the second **Mode Analysis** study step. That is, only modes where the integral of the intensity is larger than 0 are stored.

Figure 8 shows the resulting velocity-potential profile for all propagating modes at the terminal plane in the no-flow case ($M = 0$); and Figure 9 shows all the propagating modes in the flow case ($M = -0.5$). Notice that the mode profiles are identical for the incident and outgoing mode, indicating that the mean background flow is uniform at this boundary (as expected).

Combining the information from all the plots, at the terminal plane, the mode identification necessary for the ports conditions can be done. See also the last two

Evaluation Groups in the model where the solution index is shown for the various modes and flow conditions. No source is defined at the terminal plane.

- At the terminal plane in the no-flow case ($M = 0$), the outgoing mode has the wave number $k_n = 9.6$. This is the first and only eigenvalue in solution list and has solution index 1 (necessary for referencing at the port). No incident mode is necessary as no source is defined here.
- At the terminal plane in the flow case ($M = -0.5$), the outgoing modes have wave numbers $k_n = 13.9$ (index 1) and $k_n = 24.9$ (index 2). No incident mode is necessary as no source is defined here.

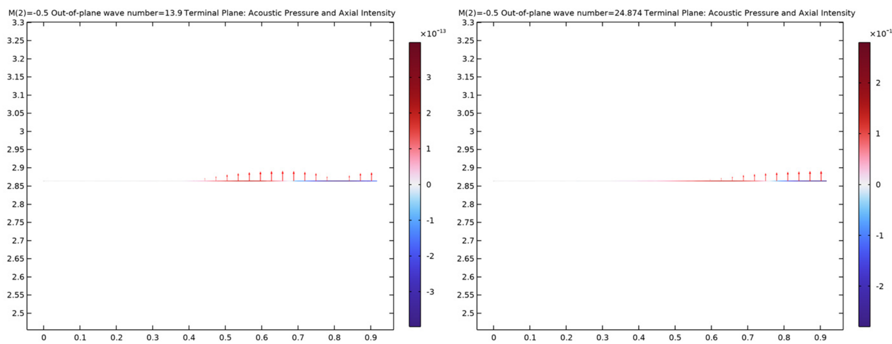


Figure 7: Outgoing mode pressures and axial intensity at the terminal plane for the case of a background flow with Mach number $M = -0.5$.

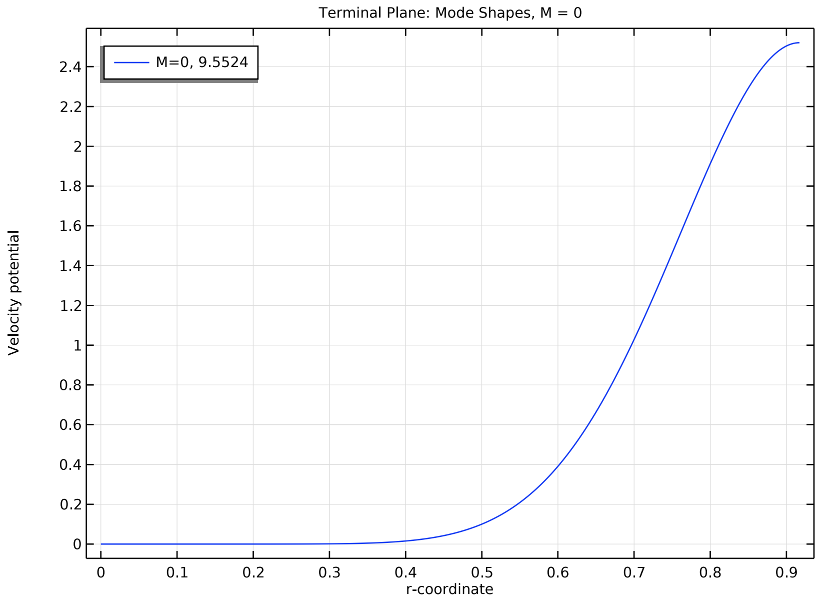


Figure 8: Propagating modes at the terminal plane for the no-flow case (M = 0).

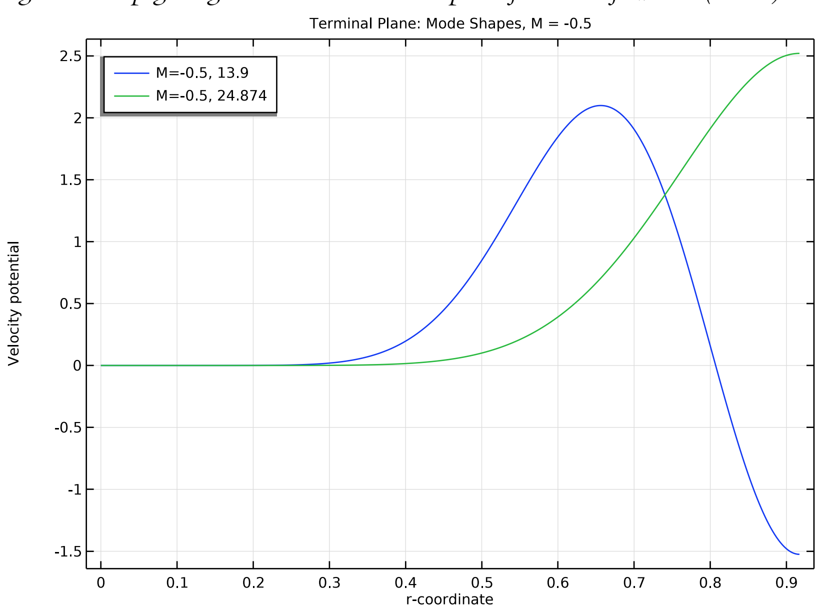


Figure 9: Propagating modes at the terminal plane for the flow case ($M = -0.5$).

THE AEROACOUSTIC FIELD

The normalized pressure fields for the case without a no background mean flow ($M = 0$), shown in Figure 10, 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: 50.67 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 11 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 28.0 dB is in good agreement with the value of 27.2 dB obtained in Ref. 1.

Note that the source mode in the COMSOL Multiphysics calculation was derived for the case of a hard duct wall, 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.

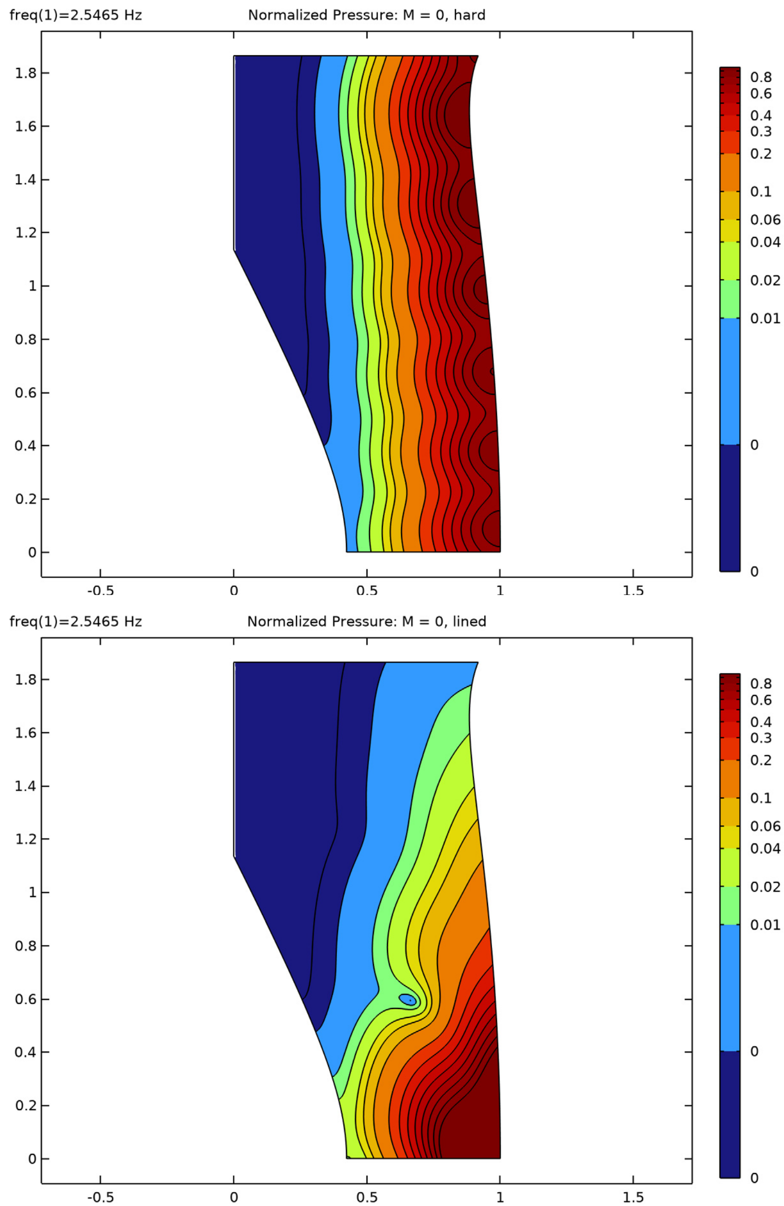


Figure 10: Acoustic pressure field for the cases of hard (top) and lined (bottom) duct wall with no mean flow ($M = 0$); azimuthal mode number $m = 10$ and angular frequency $\omega = 16$.

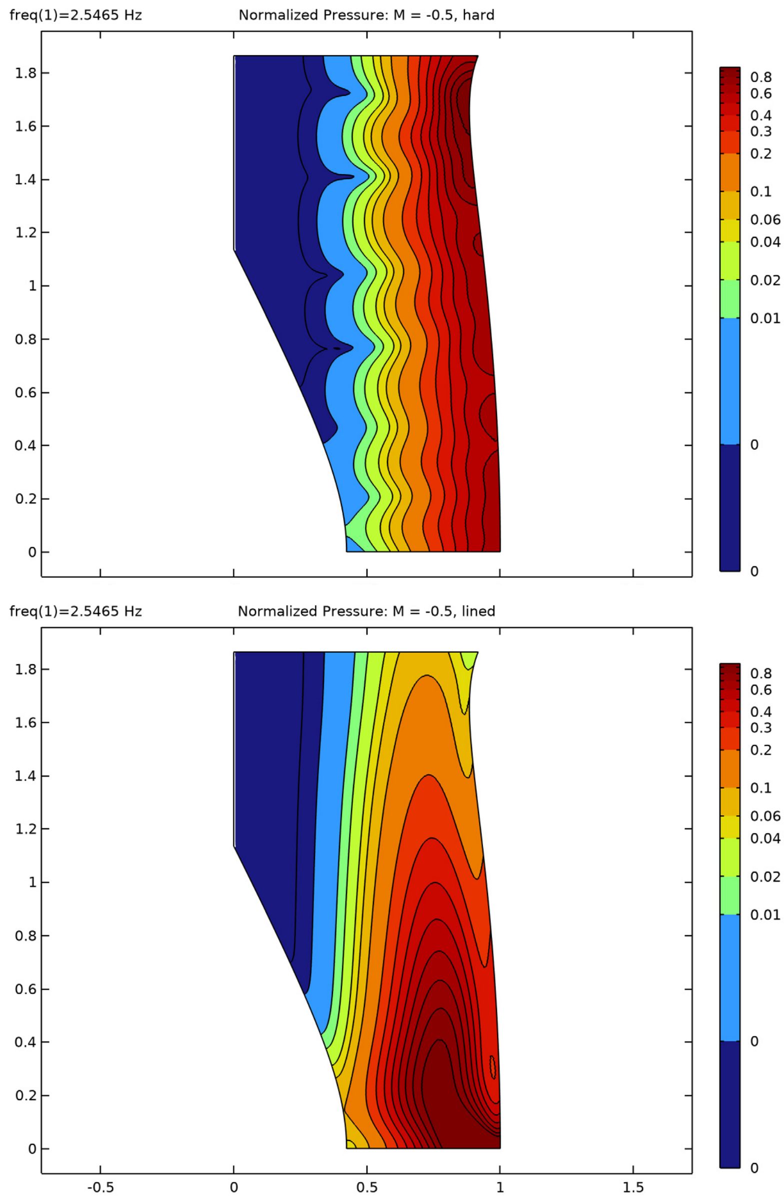


Figure 11: Acoustic pressure distribution for the cases of hard (top) and lined (bottom) duct wall with mean flow ($M = -0.5$); azimuthal mode number $m = 10$ and angular frequency $\omega = 16$.

Notes About the COMSOL Implementation

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, Boundary Mode* (lpfbm) — for calculating the boundary eigenmode to be used by the port boundary conditions, defining outgoing and incident (the source) propagating acoustic field.
- *Linearized Potential Flow, Frequency Domain* (lpff) — for modeling the time-harmonic acoustic field in the duct for the various excitation and flow condition.

REFERENCING THE ACOUSTIC MODES AT THE PORTS

The modes computed at the source plane and the terminal plane with the boundary mode interfaces need to be referenced and used by the port conditions. This is achieved by using the `withsol()` operator. The operator is called with a solution tag, referencing which solution it should look at, here the tag used depends on if the model is solved at the source (`'sol3'`) or the terminal planes (`'sol7'`). The input is the variable needed, for example, `phi_sp` for the source plane potential or `lpfbm.kn` for the source plane mode wave-number. Finally, the operator is called with two arguments using `setval()` to reference if the Mach number `M` is `0` or `-0.5`; and using `setind()` to set the solution index of the mode (of the eigenvalue `lambda`). For the index it is the number of the eigenvalue in the solution object. Note that `lambda` is always used as the internal eigenvalue variable in COMSOL Multiphysics.

FILTERING AND SORTING OF EIGENMODES

It can be advantageous to control the sorting of the results of a mode analysis study. It can also be advantageous to control what modes are stored (filtered) in the output. Both can be achieved with the settings of the **Filtering and Sorting** section of the **Mode Analysis** study step.

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_boundary_mode`

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, Boundary Mode (lpfbm)**.
5. Click **Add**.
6. In the **Velocity potential (m²/s)** text field, type `phi_sp`.

   Here `_sp` stands for source plane.
7. In the **Select Physics** tree, select **Acoustics** > **Aeroacoustics** > **Linearized Potential Flow, Boundary Mode (lpfbm)**.
8. Click **Add**.
9. In the **Velocity potential (m²/s)** text field, type `phi_tp`.

   Here `_tp` stands for terminal plane.
10. In the **Select Physics** tree, select **Acoustics** > **Aeroacoustics** > **Linearized Potential Flow, Frequency Domain (lpff)**.
11. Click **Add**.
12. Click **Study**.

13 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces** > **Stationary**.

14 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_boundary_mode_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`.

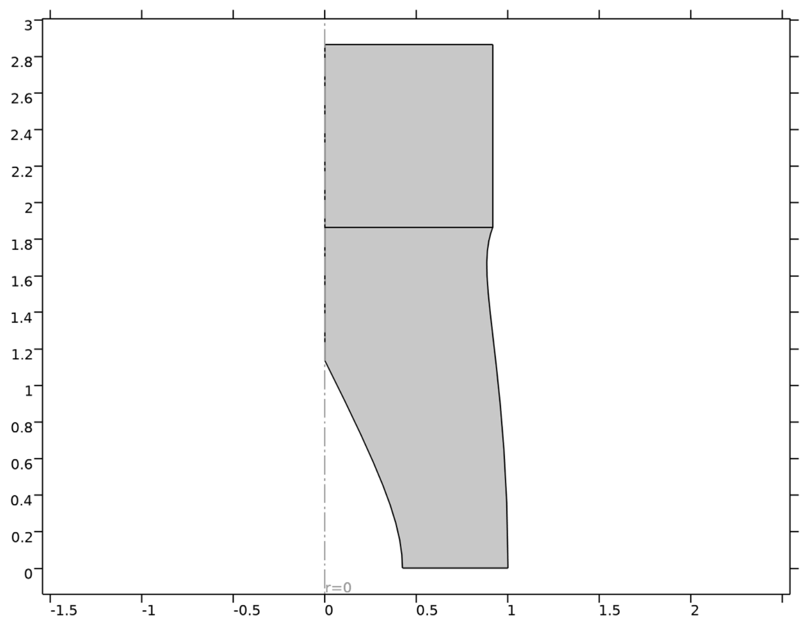
4 Locate the **Position** section. In the **z** text field, type `zi`.

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**.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2 In the **Settings** window for **Integration**, type `intop_tp` 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 **Terminal Plane**.

Now proceed and set up the physics for the Compressible Potential Flow as well as the two Boundary Mode interfaces. The latter two are used to compute the propagating modes at the source and the terminal planes. The modes are used in the Port boundary conditions used for the frequency domain analysis.

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**.

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 **Source Plane**.

4 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**.

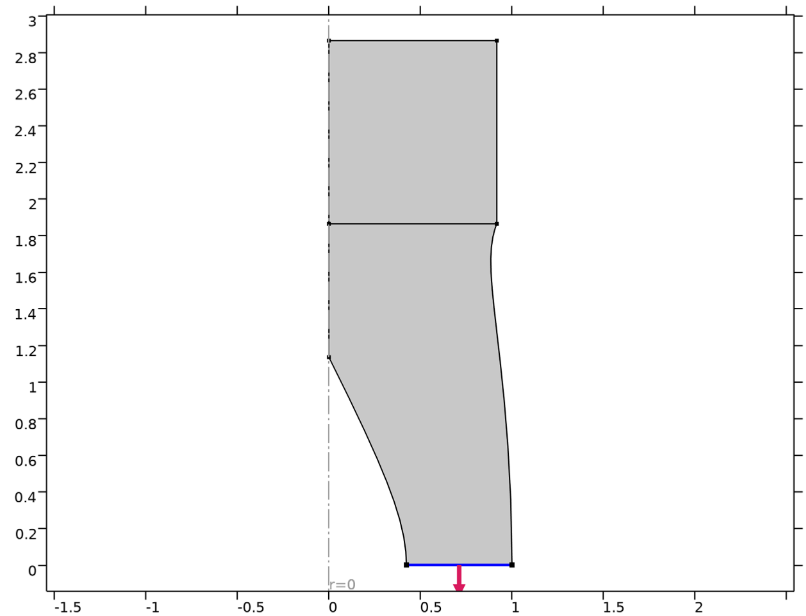
2 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Model Input** section.

3 From the $\mathbf{u}_0$ list, choose **Velocity (cpf/cpf1)**.

4 Locate the **Fluid Properties** section. From the ρ_0 list, choose **Density (cpf)**.

5 From the c_0 list, choose **Speed of sound (cpf/cpf1)**.

Notice the orientation of the outward propagating normal, the red arrow seen in the **Graphics** window. For consistency it should point out of the domains. It is possible to reverse the orientation using the **Reverse normal direction** checkbox. This is not necessary here.



LINEARIZED POTENTIAL FLOW, BOUNDARY MODE 2 (LPFBM2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Boundary Mode 2 (lpfbm2)**.
2 In the **Settings** window for **Linearized Potential Flow, Boundary Mode**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Terminal Plane**.
4 Locate the **Linearized Potential Flow Equation Settings** section. In the m text field, type `m`.

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Linearized Potential Flow, Boundary Mode 2 (lpfbm2)** click **Linearized Potential Flow Model 1**.
2 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Model Input** section.
3 From the $\mathbf{u}_0$ list, choose **Velocity (cpf/cpf1)**.
4 Locate the **Fluid Properties** section. From the ρ_0 list, choose **Density (cpf)**.
5 From the c_0 list, choose **Speed of sound (cpf/cpf1)**.

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`.

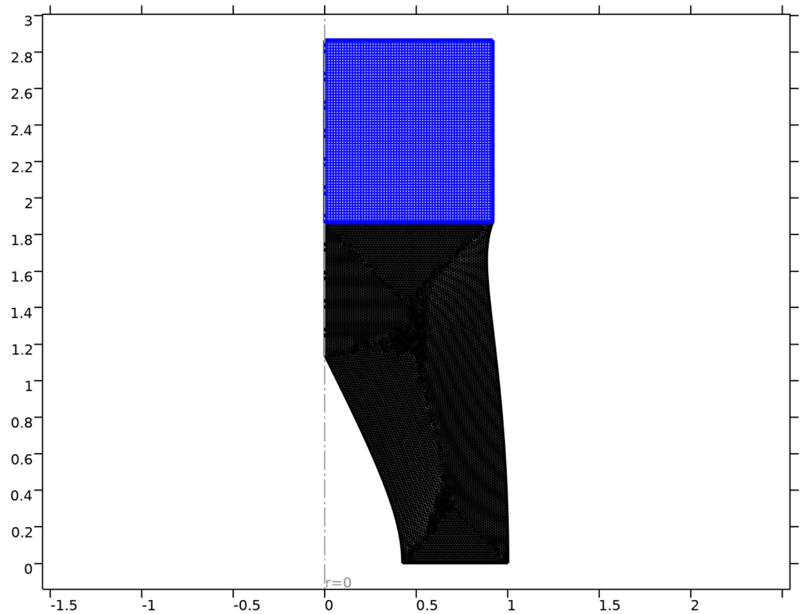
Size

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

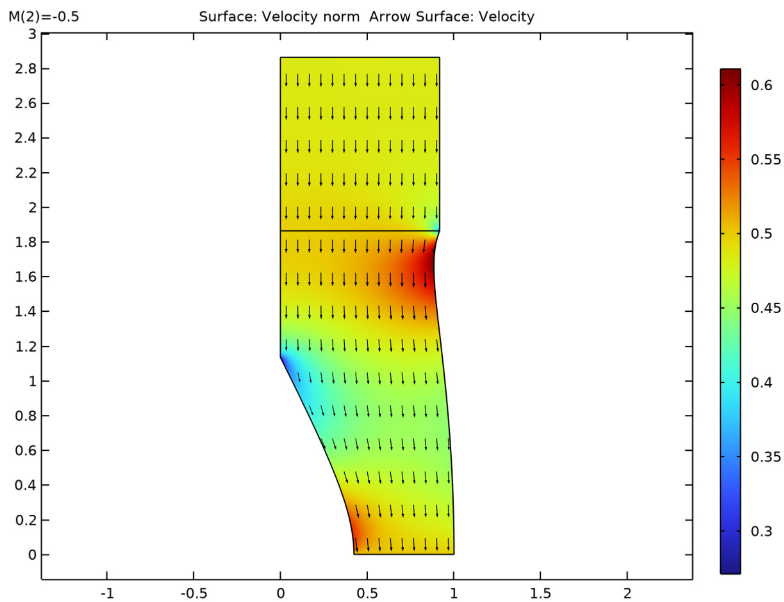
Arrow Surface 1

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**.

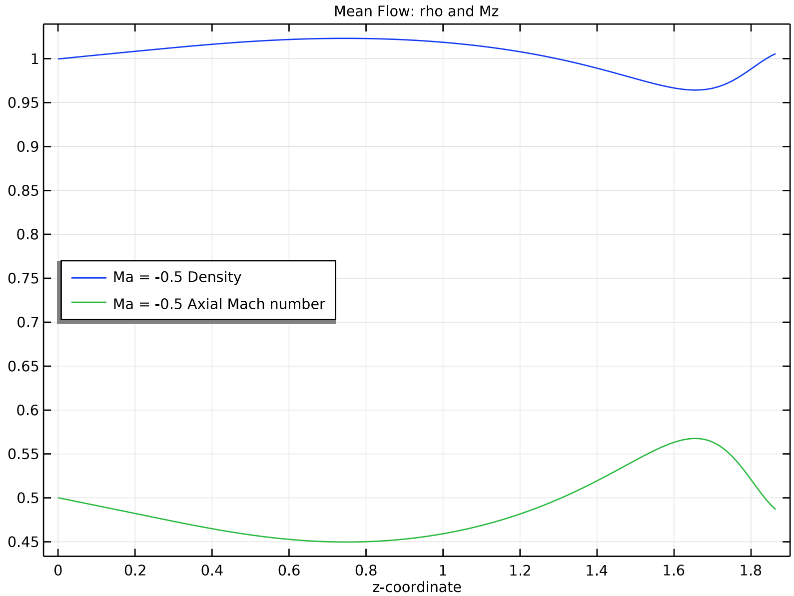
5 In the **Expression** text field, type `z`.

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**.



Proceed to compute and analyze the mode shapes for the source plane. The **Rectangle** mode search method will be used as it can filter out all the evanescent (nonpropagating) modes by setting a small range for the imaginary part of the out-of-plane wave number. The real part lies between `-k0max_abs` and `+k0max_abs` (the maximal absolute wave number) computed in the parameters list. Note that an additional small margin is used for the interval. A parametric sweep is also performed solving for the two Mach numbers.

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** checkboxes for **Compressible Potential Flow (cpf)**, **Linearized Potential Flow, Boundary Mode 2 (lpfbm2)**, and **Linearized Potential Flow, Frequency Domain (lpff)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Mode Analysis**.

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

STUDY 2 - SOURCE PLANE MODES

In the **Settings** window for **Study**, type `Study 2 - Source Plane Modes` 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	

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** list, choose **Rectangle**.

5 In the **Approximate number of modes** text field, type `5`.

6 In the **Maximum number of modes** text field, type `10`.

7 Find the **Rectangle search region** subsection. In the **Smallest real part (Out-of-plane wave number)** text field, type `-1.1*k0max_abs`.

8 In the **Largest real part (Out-of-plane wave number)** text field, type `1.1*k0max_abs`.

9 In the **Smallest imaginary part (Out-of-plane wave number)** text field, type `-0.1`.

10 In the **Largest imaginary part (Out-of-plane wave number)** text field, type `0.1`.

11 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**.

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

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

14 From the **Solution** list, choose **Solution 1 (sol1)**.

Notice in the **Settings** window the section called **Filtering and Sorting**. This section contains settings for filtering the eigenmodes, as well as settings for sorting (or ordering) of the eigenmodes in the output. At the source plane both outgoing and incident modes are needed, so no filtering is performed. At the terminal plane however, only outgoing modes are needed (no source is present). The setup of the next study shows how such a filtering can be easily set up.

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

RESULTS

Source Plane: Acoustic Pressure and Axial Intensity

When setting up the Port boundary conditions it is necessary to know which modes are outgoing and incident. Simply knowing the sign of the computed out-of-plane wave number is not enough when a background flow is present and when higher order azimuthal modes are analyzed. In this case it is advantageous to plot the axial intensity vector to identify the propagation directions. This is done by modifying the next default plot, here at the source plane.

1 In the **Settings** window for **2D Plot Group**, type `Source Plane: Acoustic Pressure and Axial Intensity` in the **Label** text field.

2 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.

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

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

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

Arrow Line 1

1 Right-click **Source Plane: Acoustic Pressure and Axial Intensity** and choose **Arrow Line**.

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

3 Locate the **Expression** section. In the **R-component** text field, type `0`.

4 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `30`.

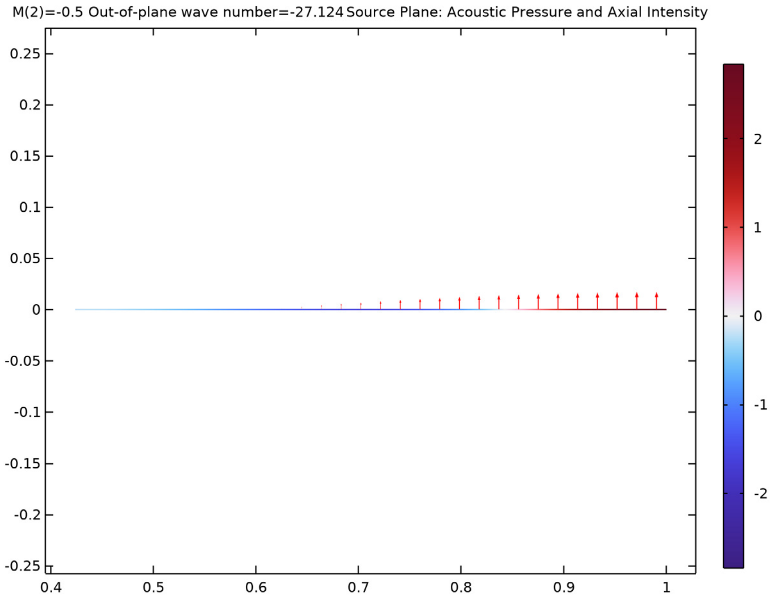
5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

Source Plane: Acoustic Pressure and Axial Intensity

1 In the **Model Builder** window, click **Source Plane: Acoustic Pressure and Axial Intensity**.

2 In the **Source Plane: Acoustic Pressure and Axial Intensity** toolbar, click **Plot**.

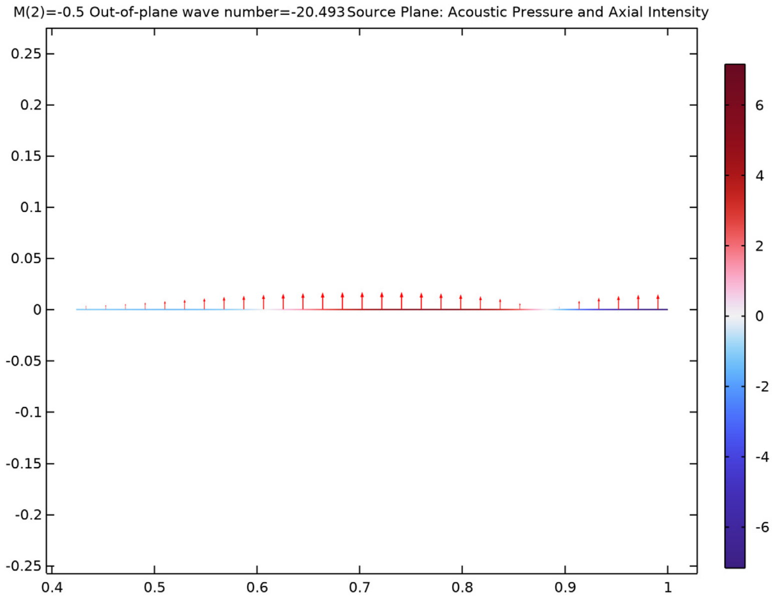
3 Click the  **Zoom Extents** button in the **Graphics** toolbar.



4 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

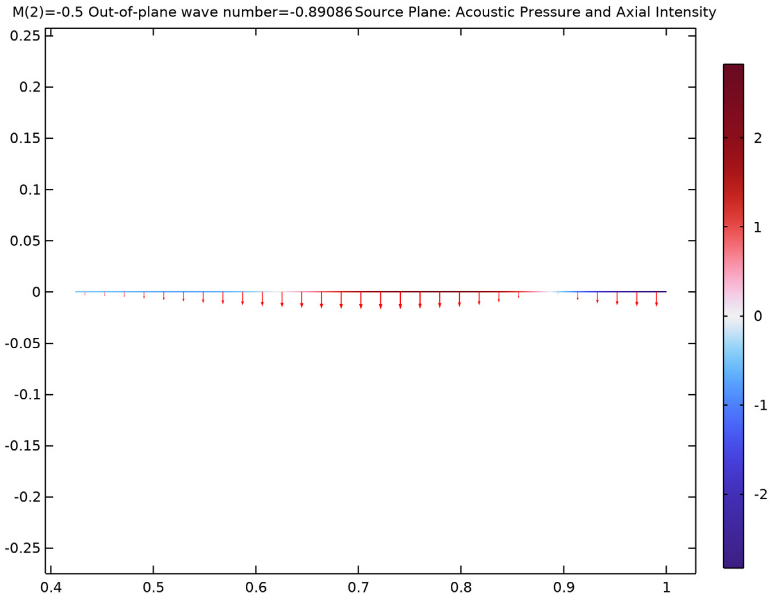
5 From the **Out-of-plane wave number** list, choose **-20.493**.

6 In the **Source Plane: Acoustic Pressure and Axial Intensity** toolbar, click **Plot**.



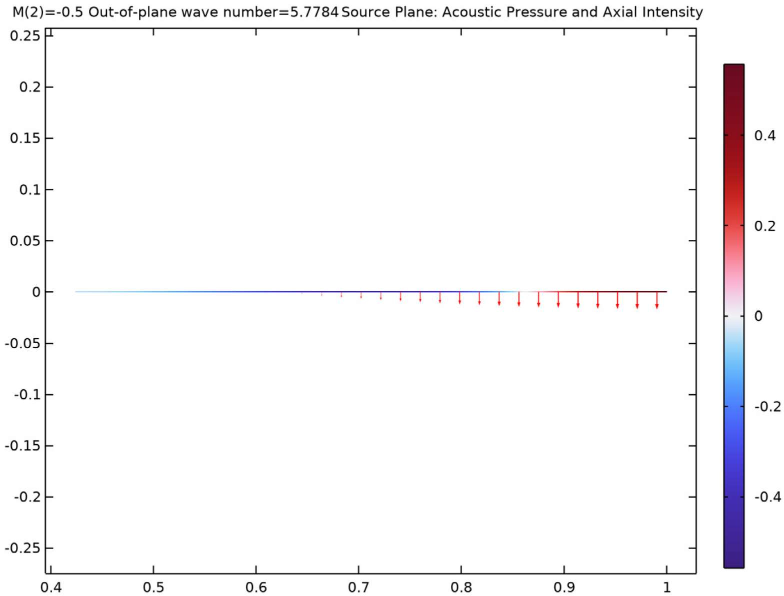
7 From the **Out-of-plane wave number** list, choose **-0.89086**.

8 In the **Source Plane: Acoustic Pressure and Axial Intensity** toolbar, click **Plot**.



9 From the **Out-of-plane wave number** list, choose **5.7784**.

10 In the **Source Plane: Acoustic Pressure and Axial Intensity** toolbar, click **Plot**.



Looking at the four out-of-plane wave numbers (for the M = -0.5 case) it can be seen that the first two are inward propagating and the last two are outward propagating.

Revolution 2D 1

1 In the **Model Builder** window, under **Results > Datasets** click **Revolution 2D 1**.

2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.

3 From the **Number of layers** list, choose **Fine**.

The **Azimuthal mode number** option can be used when evaluating the dependent variable (the velocity potential). However, here we will evaluate the pressure. The azimuthal component will be added manually in the plot, using the defined phi variable.

Source Plane: Acoustic Pressure, 3D (lpfbm)

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

2 In the **Settings** window for **3D Plot Group**, type `Source Plane: Acoustic Pressure, 3D (lpfbm)` in the **Label** text field.

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

Surface 2

1 Right-click **Source Plane: Acoustic Pressure, 3D (lpfbm)** and choose **Surface**.

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

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

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

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 In the **Settings** window for **Selection**, locate the **Revolution Selection** section.

4 Clear the **Evaluate the start cap** checkbox.

5 Clear the **Evaluate the end cap** checkbox.

Transparency 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Transparency**.

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

3 Find the **Transparency** subsection. Set the **Transparency** value to **0.2**.

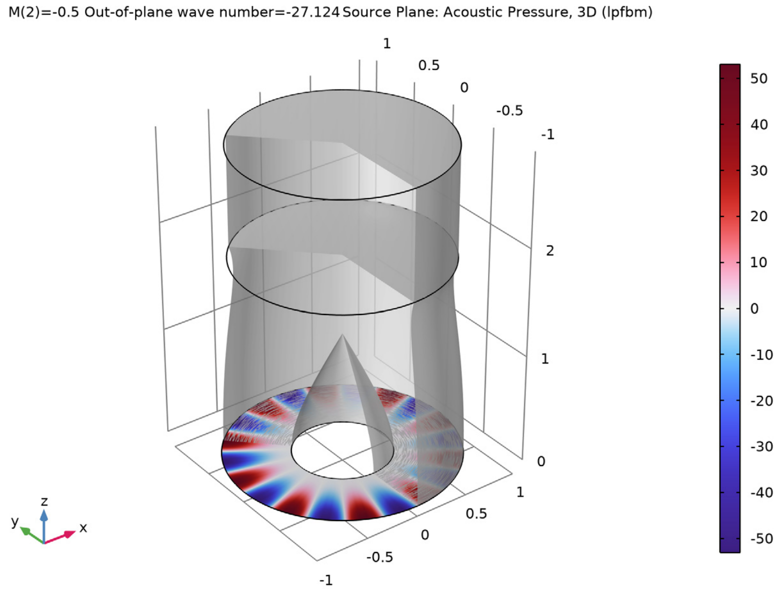
Surface

1 In the **Model Builder** window, under **Results** > **Source Plane: Acoustic Pressure, 3D (lpfbm)** click **Surface**.

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

3 In the **Expression** text field, type `lpfbm.p*exp(-i*m*rev2phi)`.

4 In the **Source Plane: Acoustic Pressure, 3D (lpfbm)** toolbar, click **Plot**.



Next, set up a plot that shows the mode shapes at the source plane. Notice the small difference in the shape of the outgoing and incident modes. This is due to the fact that flow is not uniform at this boundary. In the frequency domain analysis the model will be excited by the first incident radial mode. For the M = -0.5 case, it can be seen to have the wave number -27.1 and it is number 1 in the list. See also the previous plot to verify the propagation direction.

Source Plane: Mode Shapes, M = 0

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

2 In the **Settings** window for **1D Plot Group**, type `Source Plane: Mode Shapes, M = 0` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Source Plane Modes/ Parametric Solutions 1 (sol3)**.

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

5 Locate the **Title** section. From the **Title type** list, choose **Label**.

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

Line Graph 1

1 Right-click **Source Plane: Mode Shapes, M = 0** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

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

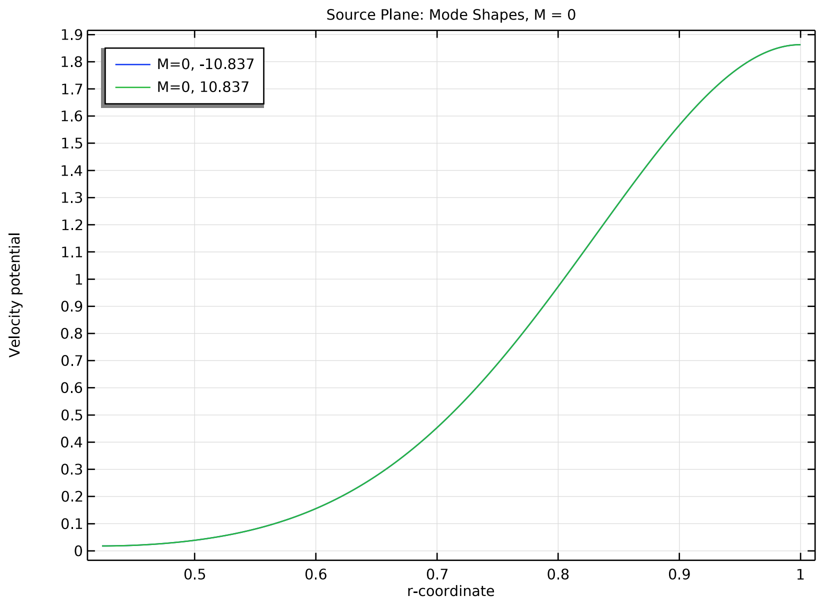
4 Locate the **y-Axis Data** section. In the **Expression** text field, type `phi_sp`.

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

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

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

8 In the **Source Plane: Mode Shapes, M = 0** toolbar, click **Plot**.



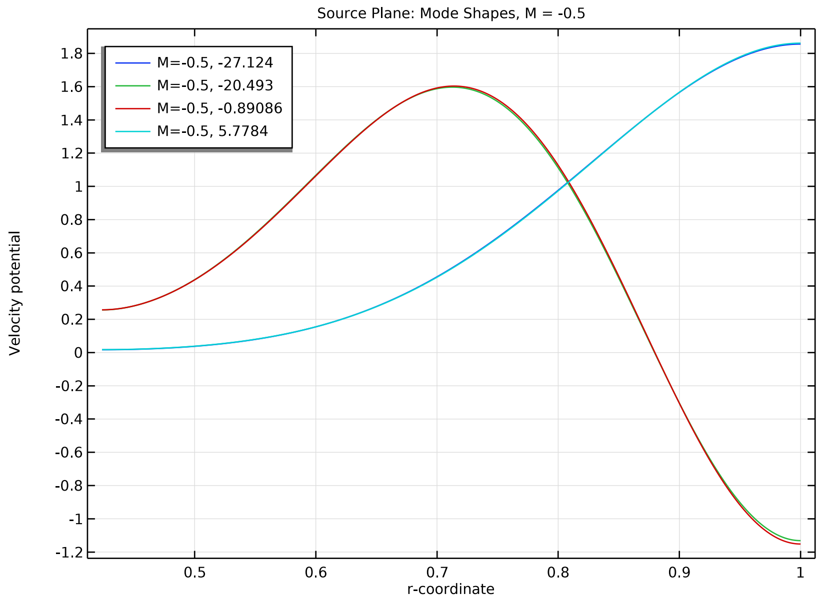
Source Plane: Mode Shapes, M = -0.5

1 In the **Model Builder** window, right-click **Source Plane: Mode Shapes, M = 0** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Source Plane: Mode Shapes, M = -0.5` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (M)** list, choose **Last**.

4 In the **Source Plane: Mode Shapes, M = -0.5** toolbar, click **Plot**.



Now, proceed and set up the same analysis for the terminal plane, including study and results, as discussed above. Add a filtering expression to only store outgoing modes at the terminal plane.

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** checkboxes for **Compressible Potential Flow (cpf)**, **Linearized Potential Flow, Boundary Mode (lpfbm)**, and **Linearized Potential Flow, Frequency Domain (lpff)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Mode Analysis**.

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

STUDY 3 - TERMINAL PLANE MODES

In the **Settings** window for **Study**, type `Study 3 - Terminal Plane Modes` 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`	

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** list, choose **Rectangle**.
5 In the **Approximate number of modes** text field, type `5`.
6 In the **Maximum number of modes** text field, type `10`.
7 Find the **Rectangle search region** subsection. In the **Smallest real part (Out-of-plane wave number)** text field, type `-1.1*k0max_abs`.
8 In the **Largest real part (Out-of-plane wave number)** text field, type `1.1*k0max_abs`.
9 In the **Smallest imaginary part (Out-of-plane wave number)** text field, type `-0.1`.
10 In the **Largest imaginary part (Out-of-plane wave number)** text field, type `0.1`.
11 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
12 From the **Method** list, choose **Solution**.
13 From the **Study** list, choose **Study 1 - Background Flow, Stationary**.
14 From the **Solution** list, choose **Solution 1 (sol1)**.

Now, set up an expression for filtering the eigenmodes such that only outgoing modes are stored for the terminal plane. The filter expression is the integral of the axial intensity over the port. For the modes to be outgoing, this quantity should be positive.

15 Click to expand the **Filtering and Sorting** section. Find the **Filtering** subsection. In the table, enter the following settings:

Filter expression (store if true or >0)	Description
`comp1.intop_tp(comp1.lpfbm2.Iz)`	`Outgoing modes only`

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

RESULTS

Terminal Plane: Acoustic Pressure and Axial Intensity

1 In the **Settings** window for **2D Plot Group**, type `Terminal Plane: Acoustic Pressure and Axial Intensity` in the **Label** text field.

2 Locate the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.

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

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

5 Locate the **Title** section. From the **Title type** list, choose **Label**.

Arrow Line 1

1 Right-click **Terminal Plane: Acoustic Pressure and Axial Intensity** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Linearized Potential Flow, Boundary Mode 2** > **Intensity** > **lpfbm2.Ir,lpfbm2.Iz - Intensity**.

3 Locate the **Expression** section. In the **R-component** text field, type `0`.

4 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `30`.

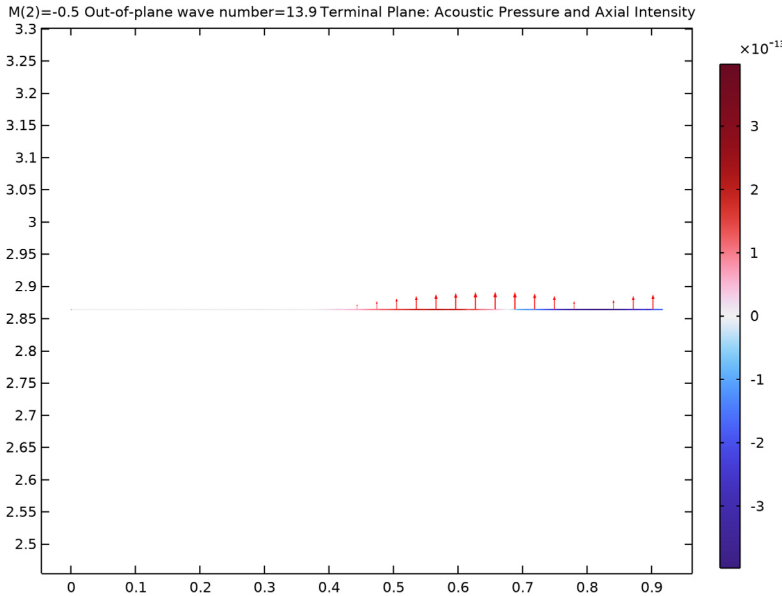
5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

Terminal Plane: Acoustic Pressure and Axial Intensity

1 In the **Model Builder** window, click **Terminal Plane: Acoustic Pressure and Axial Intensity**.

2 In the **Terminal Plane: Acoustic Pressure and Axial Intensity** toolbar, click **Plot**.

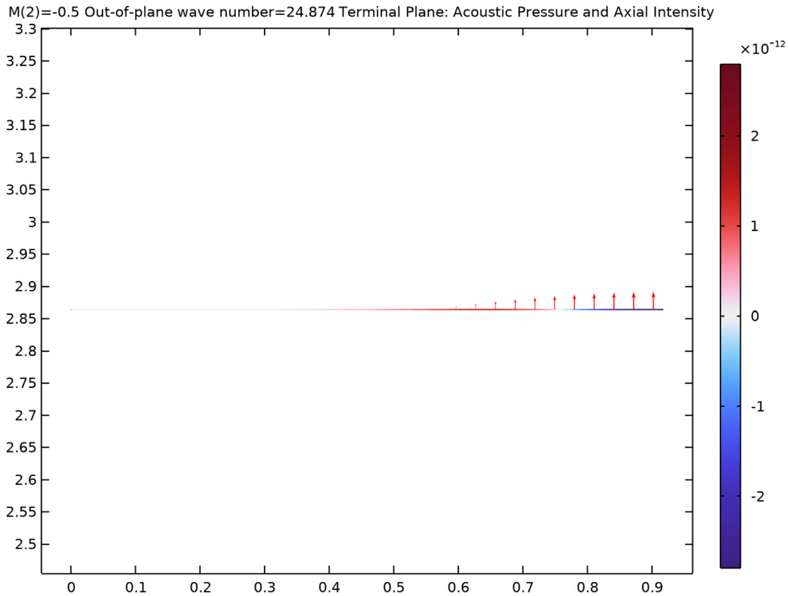
3 Click the **Zoom Extents** button in the **Graphics** toolbar.



4 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

5 From the **Out-of-plane wave number** list, choose **24.874**.

6 In the **Terminal Plane: Acoustic Pressure and Axial Intensity** toolbar, click **Plot**.



Revolution 2D 2

1 In the **Model Builder** window, under **Results** > **Datasets** click **Revolution 2D 2**.

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

3 From the **Number of layers** list, choose **Fine**.

Terminal Plane: Acoustic Pressure, 3D (lpfbm2)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, 3D (lpfbm2)**.

2 In the **Settings** window for **3D Plot Group**, type `Terminal Plane: Acoustic Pressure, 3D (lpfbm2)` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Surface 2

1 Right-click **Terminal Plane: Acoustic Pressure, 3D (lpfbm2)** and choose **Surface**.

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

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

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

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

Selection 1

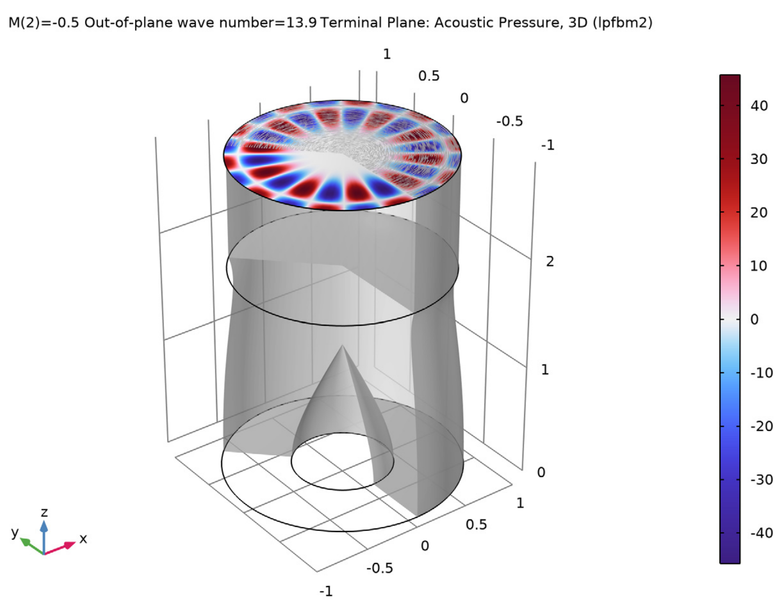
1. Right-click **Surface 2** and choose **Selection**.
2. Click in the **Graphics** window and then press Ctrl+A to select both domains.
3. In the **Settings** window for **Selection**, locate the **Revolution Selection** section.
4. Clear the **Evaluate the start cap** checkbox.
5. Clear the **Evaluate the end cap** checkbox.

Transparency 1

1. In the **Model Builder** window, right-click **Surface 2** and choose **Transparency**.
2. In the **Settings** window for **Transparency**, locate the **Transparency** section.
3. Find the **Transparency** subsection. Set the **Transparency** value to **0.2**.

Surface

1. In the **Model Builder** window, under **Results** > **Terminal Plane: Acoustic Pressure, 3D (lpfbm2)** click **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `lpfbm2.p*exp(-i*m*rev3phi)`.
4. In the **Terminal Plane: Acoustic Pressure, 3D (lpfbm2)** toolbar, click **Plot**.



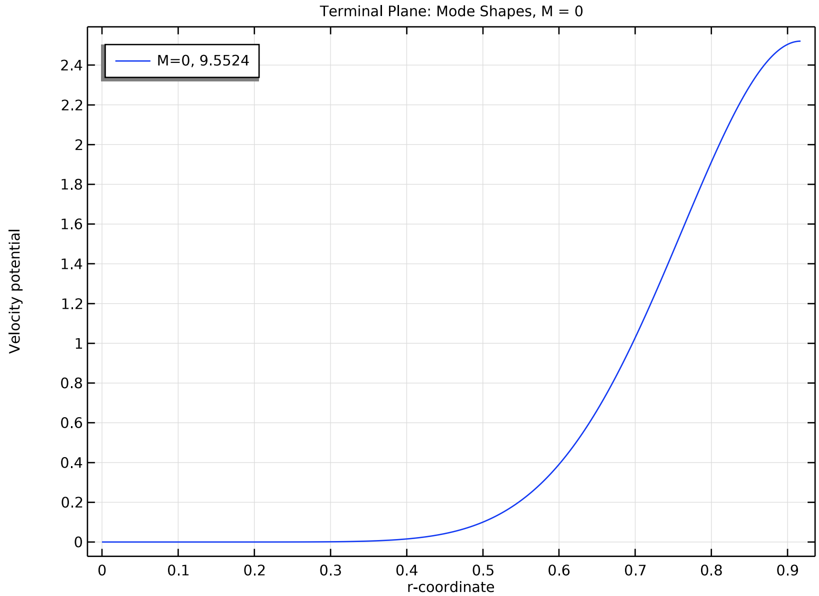
Terminal Plane: Mode Shapes, M = 0

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Terminal Plane: Mode Shapes, M = 0` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Terminal Plane Modes/Parametric Solutions 2 (sol7)**.
4 From the **Parameter selection (M)** list, choose **First**.
5 Locate the **Title** section. From the **Title type** list, choose **Label**.
6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Terminal Plane: Mode Shapes, M = 0** and choose **Line Graph**.
2 In the **Settings** window for **Line Graph**, locate the **Selection** section.
3 From the **Selection** list, choose **Terminal Plane**.
4 Locate the **y-Axis Data** section. In the **Expression** text field, type `phi_tp`.
5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6 In the **Expression** text field, type `r`.
7 Locate the **Legends** section. Select the **Show legends** checkbox.

8 In the **Terminal Plane: Mode Shapes, M = 0** toolbar, click **Plot**.



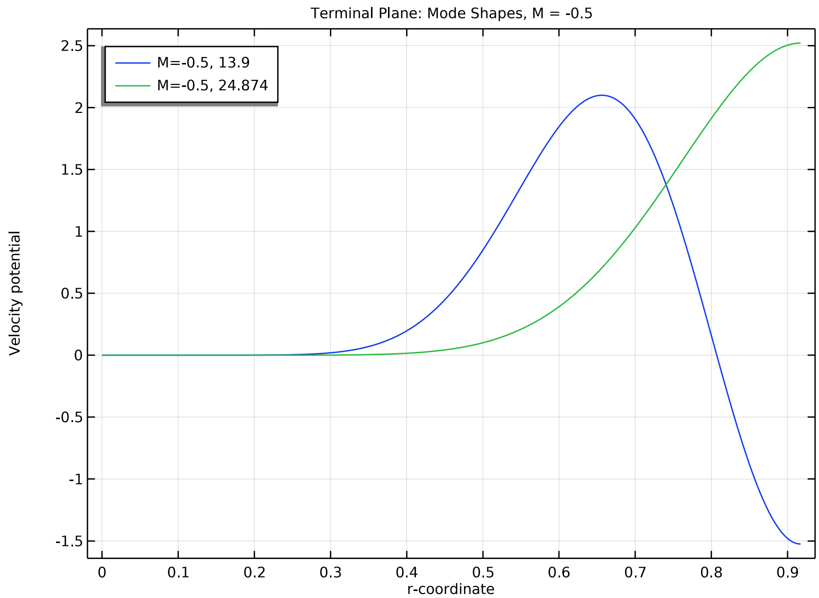
Terminal Plane: Mode Shapes, M = -0.5

1 In the **Model Builder** window, right-click **Terminal Plane: Mode Shapes, M = 0** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Terminal Plane: Mode Shapes, M = -0.5` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (M)** list, choose **Last**.

4 In the **Terminal Plane: Mode Shapes, M = -0.5** 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 set up of the Port conditions. The ports are divided into those applied at the source and inlet planes as well as their use in the flow and no flow cases. To reference the mode shapes and wave numbers the `withsol()` operator is used. Using the `setval()` and `setind()` statements it is possible to pick the desired mode.

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 (Ingard–Myers)** 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 (for M = 0)

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 In the **Label** text field, type `Port 1 (for M = 0)`.

5 Locate the **Port Outgoing Mode Settings** section. In the ϕ_n^{out} text field, type `withsol('sol3',phi_sp,setval(M,0),setind(lambda,2))`.

6 In the k_n^{out} text field, type `withsol('sol3',lpfbm.kn,setval(M,0), setind(lambda,2))`.

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

8 In the ϕ_n^{in} text field, type `withsol('sol3',phi_sp,setval(M,0),setind(lambda, 1))`.

9 In the k_n^{in} text field, type `withsol('sol3',lpfbm.kn,setval(M,0), setind(lambda,1))`.

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

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

Port 2 (for M = -0.5)

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 In the **Label** text field, type `Port 2 (for M = -0.5)`.

5 Locate the **Port Outgoing Mode Settings** section. In the ϕ_n^{out} text field, type `withsol('sol3',phi_sp,setval(M,-0.5),setind(lambda,4))`.

6 In the k_n^{out} text field, type `withsol('sol3',lpfbm.kn,setval(M,-0.5),setind(lambda,4))`.

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

8 In the ϕ_n^{in} text field, type `withsol('sol3',phi_sp,setval(M,-0.5),setind(lambda,1))`.

9 In the k_n^{in} text field, type `withsol('sol3',lpfbm.kn,setval(M,-0.5),setind(lambda,1))`.

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

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

Port 3 (for M = -0.5)

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 In the **Label** text field, type `Port 3 (for M = -0.5)`.

5 Locate the **Port Outgoing Mode Settings** section. In the ϕ_n^{out} text field, type `withsol('sol3',phi_sp,setval(M,-0.5),setind(lambda,3))`.

6 In the k_n^{out} text field, type `withsol('sol3',lpfbm.kn,setval(M,-0.5),setind(lambda,3))`.

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 4 (for M = 0)

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

2 In the **Settings** window for **Port**, type `Port 4 (for M = 0)` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Terminal Plane**.

4 Locate the **Port Outgoing Mode Settings** section. In the ϕ_n^{out} text field, type `withsol('sol7',phi_tp,setval(M,0),setind(lambda,1))`.

5 In the k_n^{out} text field, type `withsol('sol7',lpfbm2.kn,setval(M,0),setind(lambda,1))`.

Port 5 (for M = -0.5)

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 In the **Label** text field, type `Port 5 (for M = -0.5)`.

5 Locate the **Port Outgoing Mode Settings** section. In the ϕ_n^{out} text field, type `withsol('sol7',phi_tp,setval(M,-0.5),setind(lambda,1))`.

6 In the k_n^{out} text field, type `withsol('sol7',lpfbm2.kn,setval(M,-0.5),setind(lambda,1))`.

Port 6 (for M = -0.5)

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 In the **Label** text field, type `Port 6 (for M = -0.5)`.

5 Locate the **Port Outgoing Mode Settings** section. In the ϕ_n^{out} text field, type `withsol('sol7',phi_tp,setval(M,-0.5),setind(lambda,2))`.

6 In the k_n^{out} text field, type `withsol('sol7',lpfbm2.kn,setval(M,-0.5),setind(lambda,2))`.

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** checkboxes for **Compressible Potential Flow (cpf)**, **Linearized Potential Flow, Boundary Mode (lpfbm)**, and **Linearized Potential Flow, Boundary Mode 2 (lpfbm2)**.

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, LINED)

In the **Settings** window for **Study**, type `Study 4 - Frequency Domain (M = 0, lined)` in the **Label** text field.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 4 - 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 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)** > **Source Plane** > **Port 2 (for M = -0.5)**, **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Source Plane** > **Port 3 (for M = -0.5)**, **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Terminal Plane** > **Port 5 (for M = -0.5)**, and **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Terminal Plane** > **Port 6 (for M = -0.5)**.

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 4 - Frequency Domain (M = 0, lined)**.

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** checkboxes for **Compressible Potential Flow (cpf)**, **Linearized Potential Flow, Boundary Mode (lpfbm)**, and **Linearized Potential Flow, Boundary Mode 2 (lpfbm2)**.

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, HARD)

In the **Settings** window for **Study**, type `Study 5 - Frequency Domain (M = 0, hard)` in the **Label** text field.

1 In the **Model Builder** window, under **Study 5 - 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**, **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Source Plane** > **Port 2 (for M = -0.5)**, **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Source Plane** > **Port 3 (for M = -0.5)**, **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Terminal Plane** > **Port 5 (for M = -0.5)**, and **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Terminal Plane** > **Port 6 (for M = -0.5)**.

6 Click **Disable**.

7 Locate 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 5 - 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** checkboxes for **Compressible Potential Flow (cpf)**, **Linearized Potential Flow, Boundary Mode (lpfbm)**, and **Linearized Potential Flow, Boundary Mode 2 (lpfbm2)**.

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 6 - FREQUENCY DOMAIN (M = -0.5, LINED)

In the **Settings** window for **Study**, type `Study 6 - Frequency Domain (M = -0.5, lined)` in the **Label** text field.

1 In the **Model Builder** window, under **Study 6 - 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 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)** > **Source Plane** > **Port 1 (for M = 0)** and **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Terminal Plane** > **Port 4 (for M = 0)**.

6 Click **Disable**.

7 Locate 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 6 - Frequency Domain (M = -0.5, lined)**.

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** checkboxes for **Compressible Potential Flow (cpf)**, **Linearized Potential Flow, Boundary Mode (lpfbm)**, and **Linearized Potential Flow, Boundary Mode 2 (lpfbm2)**.

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 7 - FREQUENCY DOMAIN (M = -0.5, HARD)

In the **Settings** window for **Study**, type `Study 7 - Frequency Domain (M = -0.5, hard)` in the **Label** text field.

1 In the **Model Builder** window, under **Study 7 - 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**, **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Source Plane** > **Port 1 (for M = 0)**, and **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain (lpff)** > **Terminal Plane** > **Port 4 (for M = 0)**.

6 Click **Disable**.

7 Locate 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 7 - 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 4 - Frequency Domain (M = 0, lined)/Solution 10 (sol10)**.

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 Locate 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**.

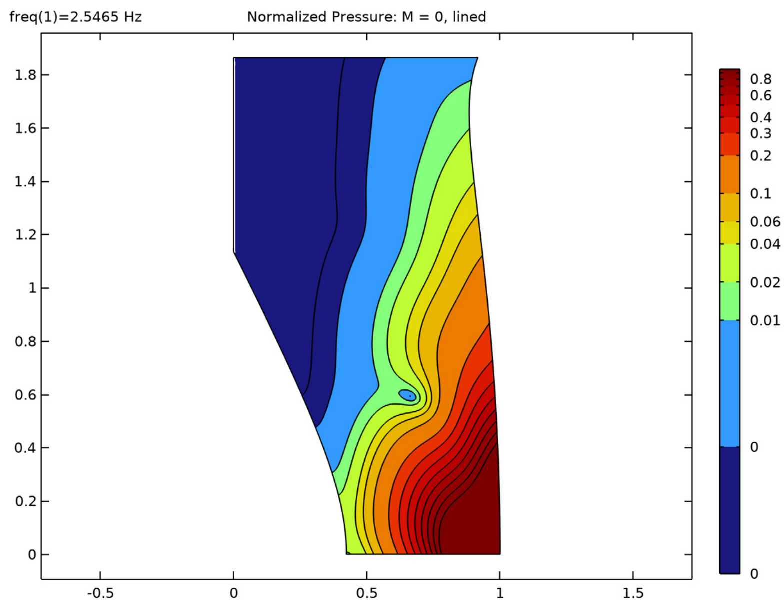
4 From the **Coloring** list, choose **Uniform**.

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

6 Clear the **Color legend** checkbox.

Normalized Pressure: M = 0, lined

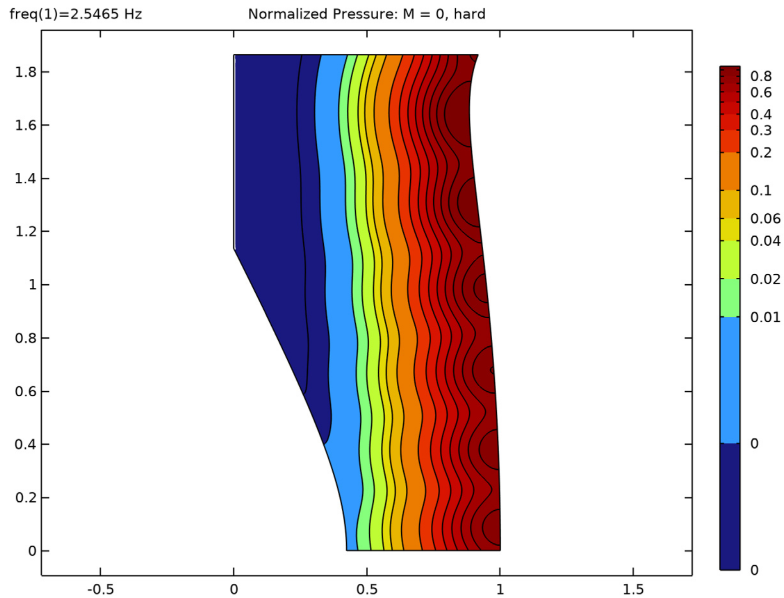
1 In the **Model Builder** window, click **Normalized Pressure: M = 0, lined**.
2 In the **Normalized Pressure: M = 0, lined** toolbar, click **Plot**.
3 Click the **Zoom Extents** button in the **Graphics** toolbar.



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 5 - Frequency Domain (M = 0, hard)/Solution 11 (sol11)**.

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



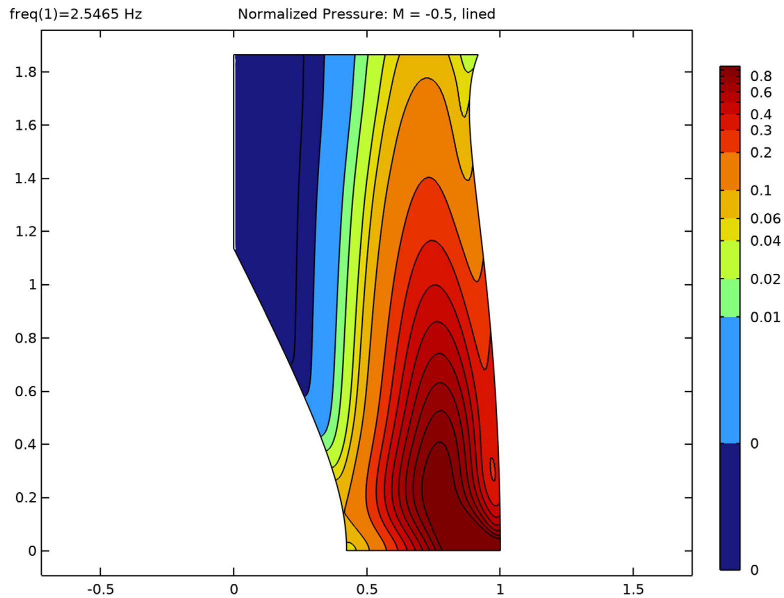
Normalized Pressure: M = -0.5, lined

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 6 - Frequency Domain (M = -0.5, lined)/Solution 12 (sol12)**.

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



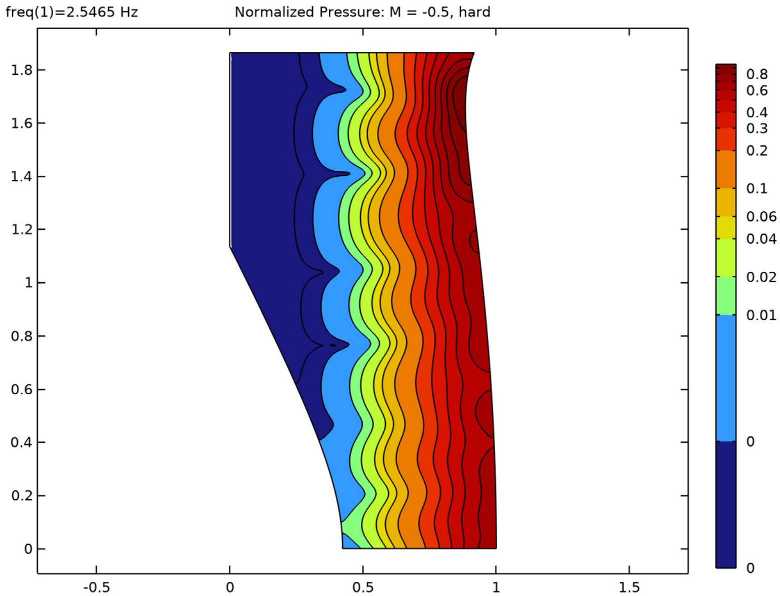
Normalized Pressure: M = -0.5, hard

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 7 - Frequency Domain (M = -0.5, hard)/Solution 13 (sol13)**.

4 In the **Normalized Pressure: 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 4 - Frequency Domain (M = 0, lined)/ Solution 10 (sol10)**.

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 6 - Frequency Domain (M = -0.5, lined)/ Solution 12 (sol12)**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`10*log10(lpff.port2.P_in/ intop_ip(lpff.Iz))`		`M = -0.5, lined`

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

Finally, create evaluation groups that help to identify the mode solution index related to a specific mode wave number and flow condition.

Source Plane: Mode Solution Index

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Source Plane: Mode Solution Index` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Source Plane Modes/ Parametric Solutions 1 (sol3)**.
4 Locate the **Transformation** section. Select the **Transpose** checkbox.

Global Evaluation 1

1 Right-click **Source Plane: Mode Solution Index** and choose **Global Evaluation**.
2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3 From the **Table columns** list, choose **Inner solutions**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`lpfbm.kn`		`kn`

5 In the **Source Plane: Mode Solution Index** toolbar, click **Evaluate**.

Terminal Plane: Mode Solution Index

1 In the **Model Builder** window, right-click **Source Plane: Mode Solution Index** and choose **Duplicate**.

2 In the **Settings** window for **Evaluation Group**, type `Terminal Plane: Mode Solution Index` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Terminal Plane Modes/ Parametric Solutions 2 (sol7)**.

Global Evaluation 1

1 In the **Model Builder** window, expand the **Terminal Plane: Mode Solution Index** node, then click **Global Evaluation 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`lpfbm2.kn`		`kn`

4 In the **Terminal Plane: Mode Solution Index** toolbar, click **Evaluate**.