Model created in COMSOL Multiphysics 6.4

Flow Duct — Modes with Impedance Condition

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 including the wall lining (impedance condition). Results are presented for the case with a background flow and for the lined duct walls.

Model Definition

Depicted in Figure 1, the setup and geometry of this model closely follow those of the Flow Duct model. Specifically, this model is only solved for the condition with a liner (impedance condition) on the outer boundary of the flow duct.

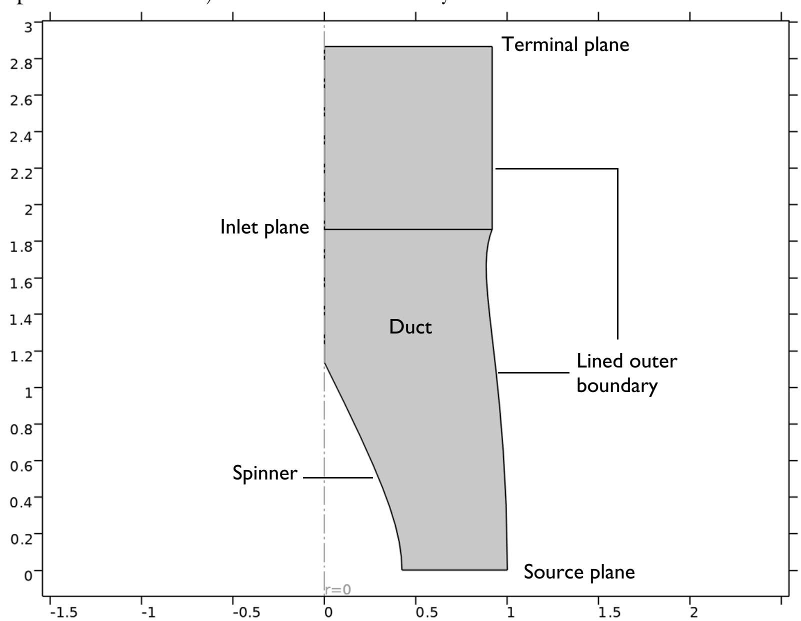


Figure 1: Schematic of the flow duct geometry including reference planes.

Moreover, the modes that are used for the **Port** boundary conditions are also computed including the impedance condition. In the Flow Duct model, the modes are computed for the hard-wall condition. To study the dynamics of the out-of-plane wave number in the complex plane, the background mean flow is solved for several different Mach numbers M. An example of such a plot for the source plane is depicted in Figure 2.

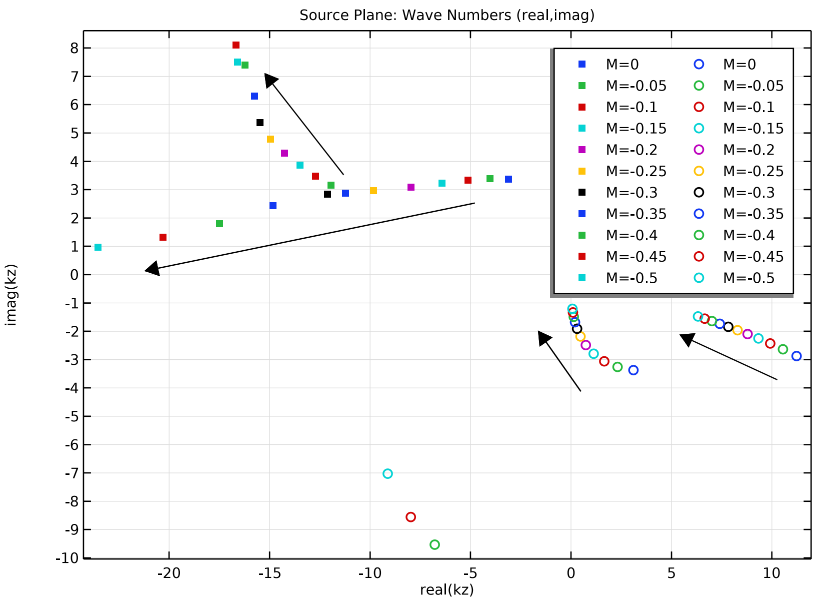


Figure 2: Dynamics of the out-of-plane wave number in the complex plane for increasing background mean flow Mach number.

In Figure 2, the location of the out-of-plane wave number in the complex plane is depicted for a range of Mach numbers M (increasing values in the direction of the arrows). Moreover, different symbols are used for the modes that are incoming and outgoing, or rather the symbols indicate if the mode is moving in the positive or negative z direction. The condition is based on the sign of the integral of the axial component of the intensity vector:

$$\int_{\Gamma} (\mathbf{I} \cdot \mathbf{e}_z) dA \tag{1}$$

where $\mathbf{I}$ is the intensity vector and $\mathbf{e}_z$ is the unit vector in the axial direction. In Figure 2, the dots represent modes propagating in the positive direction (incoming at the source plane), whereas circles are modes propagating in the negative direction (outgoing at the source plane).

Just like for the Flow Duct model, one of the configurations from Ref. 1 is studied here. This is the case in which 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 the parameter space by varying these quantities independently. Several additional cases are examined in the reference paper.

Results and Discussion

THE MEAN-FLOW FIELD

The mean flow velocity for $M = -0.5$ is depicted in Figure 3.

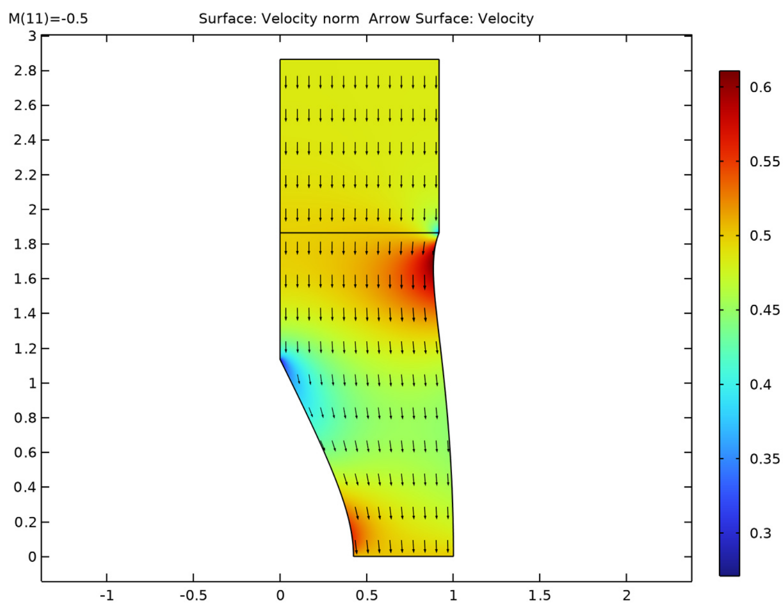


Figure 3: Mean flow velocity field.

SOURCE AND TERMINAL PLANE MODES

The source plane modes are represented in a 2D plot in Figure 4 for the azimuthal angle zero. The plot also includes a legend that shows the axial propagation direction. The first radial mode used to excite the system is represented in Figure 5 in the revolved geometry including the azimuthal dependency. Finally, the propagating modes at the terminal plane are depicted in Figure 6, again in a 2D plot for the azimuthal angle zero.

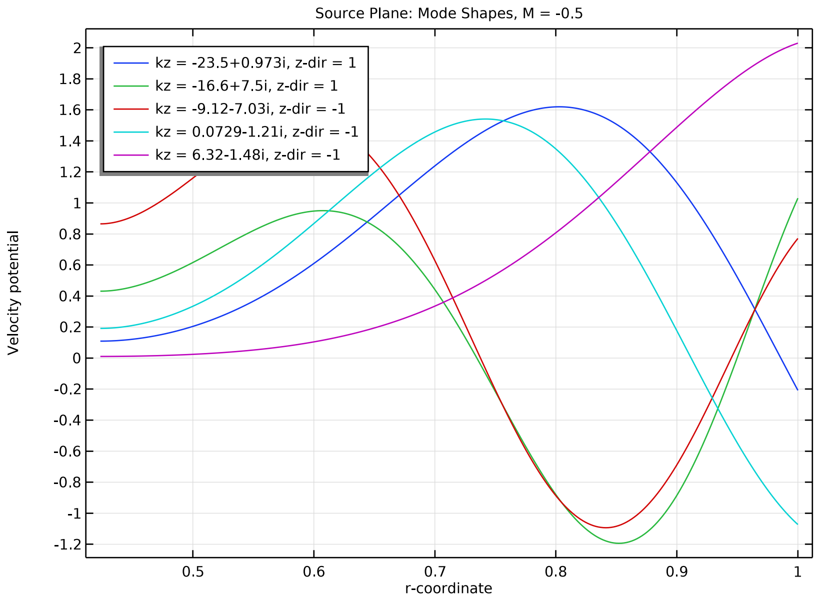


Figure 4: The source plane modes shapes (2D representation for azimuthal angle 0). The legend shows if the propagation direction is in the positive (incident) or negative (outgoing) axial direction.

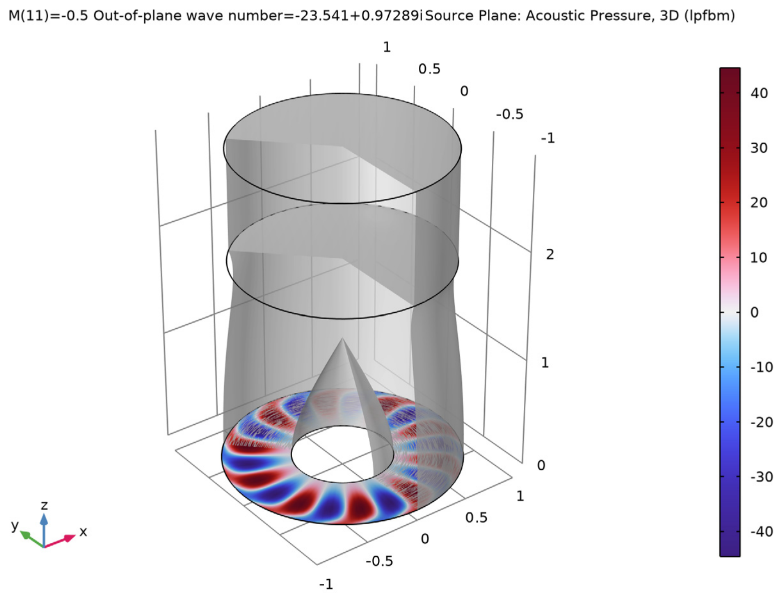


Figure 5: First radial (incident) mode used to excite the system at the source plane, here represented in the revolved geometry including the azimuthal dependency.

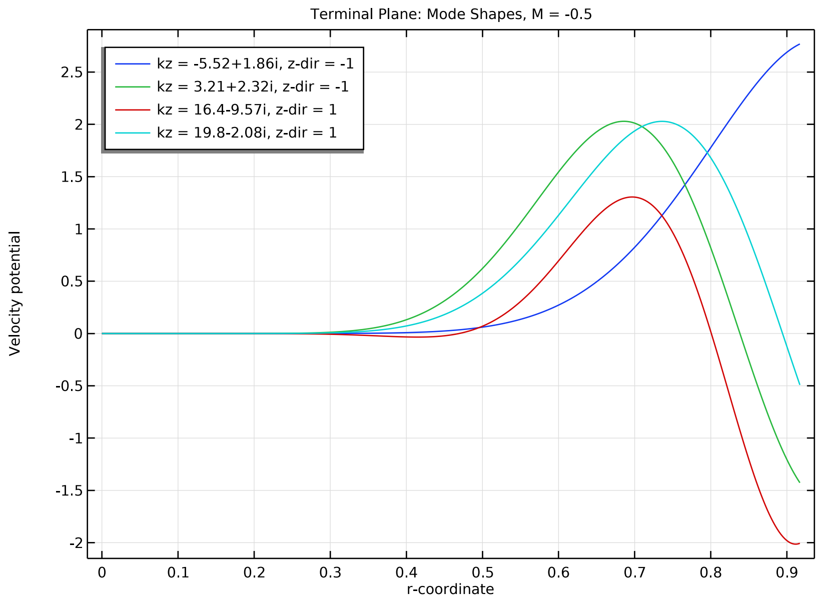


Figure 6: The terminal plane modes shapes (2D representation for azimuthal angle 0). The legend shows if the propagation direction is in the positive (outgoing) or negative (incident) axial direction.

THE AEROACOUSTIC FIELD

The normalized pressure fields, for the lined case with a background mean flow (M = –0.5), shown in Figure 7, 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: 26.60 dB for the COMSOL Multiphysics solution versus 27.20 dB for the FEM solution, as shown in Table 1 in Ref. 1. This agreement is slightly better than for the results presented in the Flow Duct tutorial (computed to be 28.0 dB), where the modes are computed based on the hard wall.

Finally, the acoustic intensity magnitude and normalized direction is depicted in Figure 8. The figure shows that there is no mismatch between the imposed acoustic field at the source plane and the lined boundary. In the case where the computed modes do not match the boundary conditions perfectly, a small accommodation length is necessary.

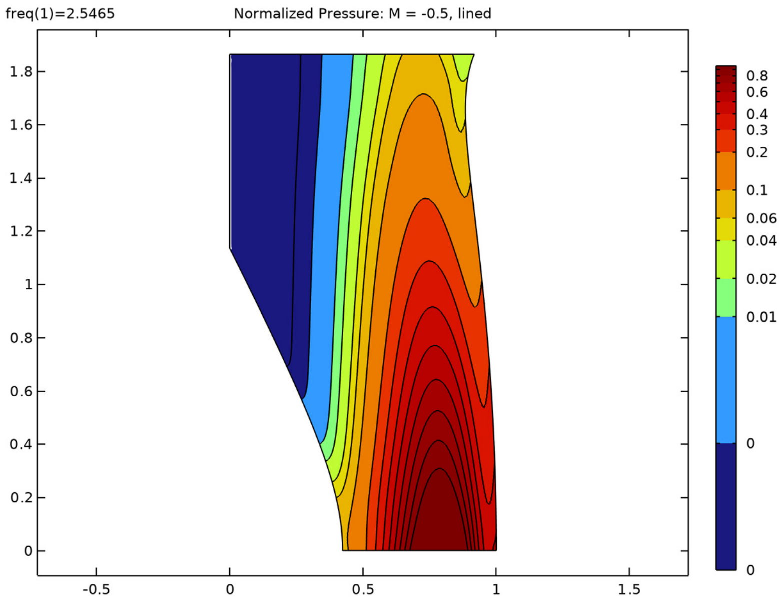


Figure 7: Normalized acoustic pressure field for the lined configuration studied here.

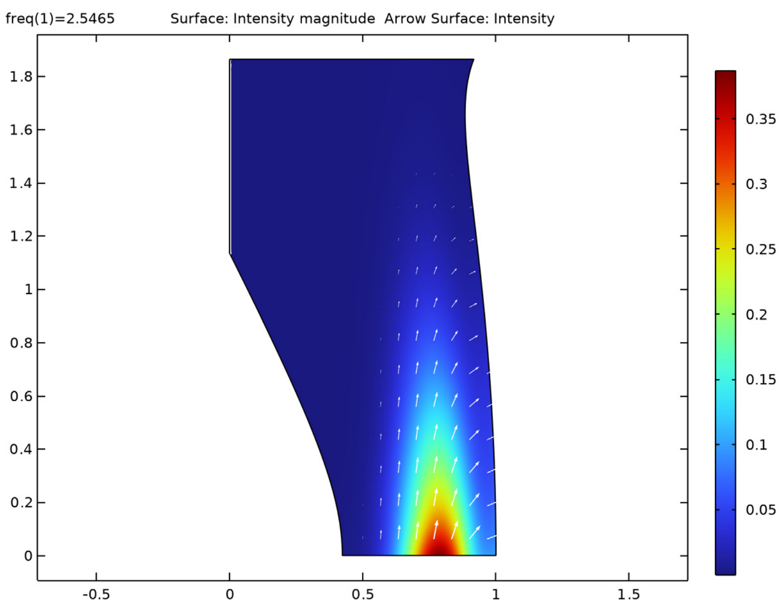


Figure 8: Acoustic intensity magnitude and direction.

Notes About the COMSOL Implementation

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 wavenumber. 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.

Reference

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.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/flow_duct_impedance`

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_impedance_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`.

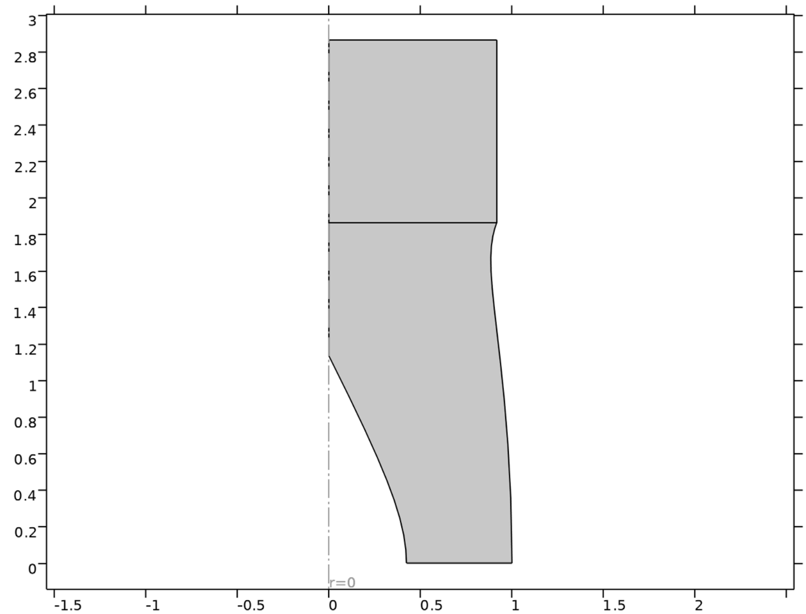
4 Locate the **Position** section. In the **z** text field, type `zi`.

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_sp` 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 **Source Plane**.

Integration 3 (intop3)

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)**.

Impedance 1

1 In the **Physics** toolbar, click **Points** and choose **Impedance**.
2 Select Point 7 only.
3 In the **Settings** window for **Impedance**, locate the **Impedance** section.
4 In the Z_n text field, type `Zw`.

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`.

Linearized Potential Flow Model 1

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)**.

Impedance 1

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

2 Select Point 6 only.

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

4 In the Z_n text field, type `Zw`.

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.

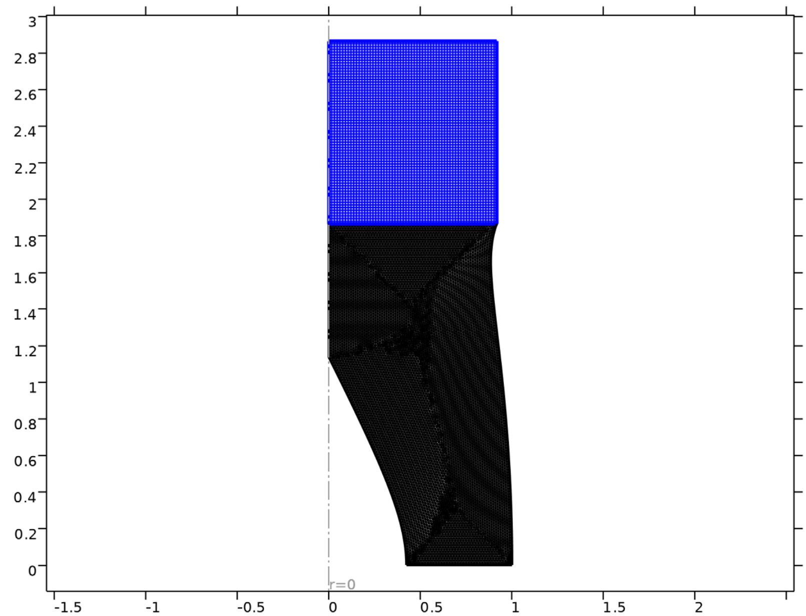
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.015`.

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 several 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)	`range(0,-0.05,-0.5)`	

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

RESULTS

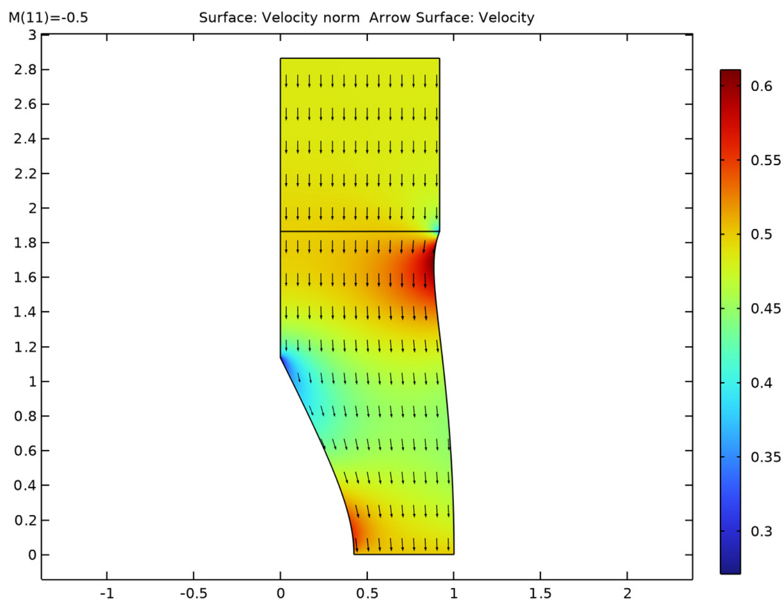
Arrow Surface 1

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**.

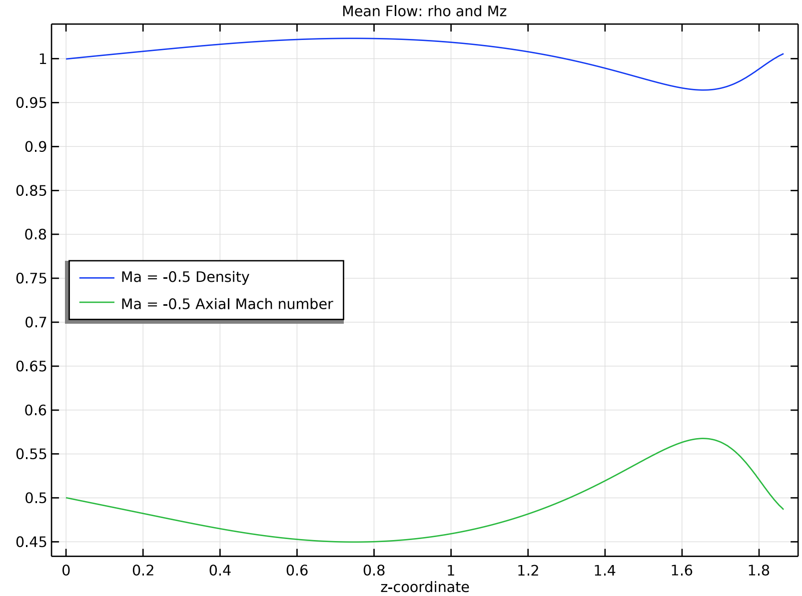
5 In the **Expression** text field, type `z`.

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 to limit the 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 all the 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)	`range(0,-0.05,-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 `10`.

6 In the **Maximum number of modes** text field, type `20`.

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 `-10`.

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

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 **Mode Analysis** study step, the section called **Filtering and Sorting**. In this section, it is possible to filter out certain modes based on global expressions. It is also in this section that the sorting of the modes is controlled. In this model, they are arranged by ascending real part of the wave number (the default behavior).

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

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 and impedance is present and when higher order azimuthal modes are analyzed. In this case, it is necessary to plot the axial intensity vector or compute its integral to identify the propagation directions. This is done by modifying the next default plot, here at the source plane.

RESULTS

Source Plane: Acoustic Pressure and Axial Intensity

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**.

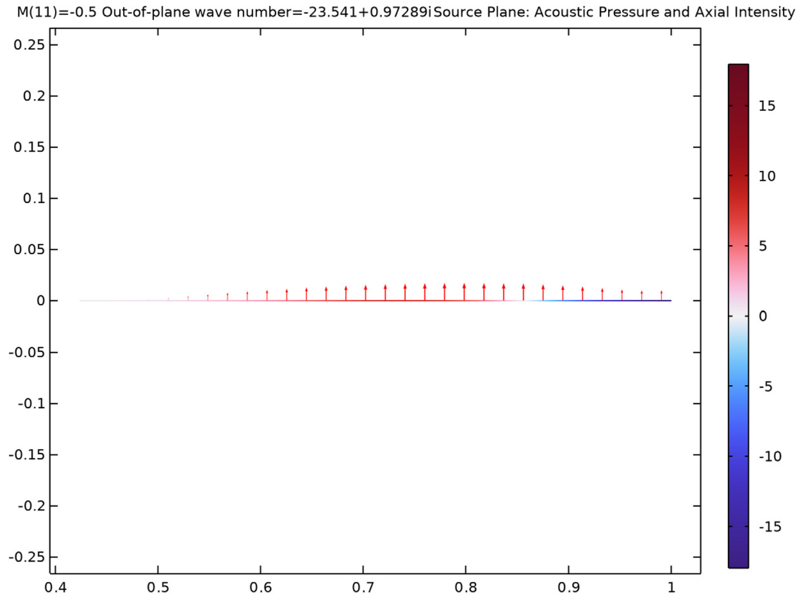
Arrow Line 1

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.



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)`.

Source Plane: Acoustic Pressure, 3D (lpfbm)

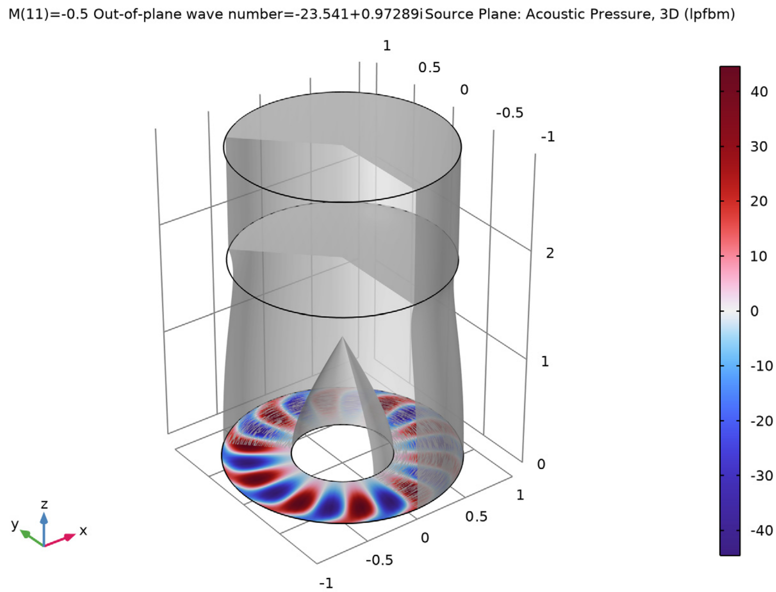
1 In the **Model Builder** window, click **Source Plane: Acoustic Pressure, 3D (lpfbm)**.

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

3 From the **Out-of-plane wave number** list, choose **-23.541+0.97289i**.

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

This plot shows the first radial mode used to excite the system.



Next, set up a plot that shows the mode shapes at the source plane. To identify the propagation direction, the integral of the axial intensity over the boundary is computed. If it is negative (-1), the propagation is in the -*z* direction, and vice versa if it is positive (1).

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 Click to expand 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**.

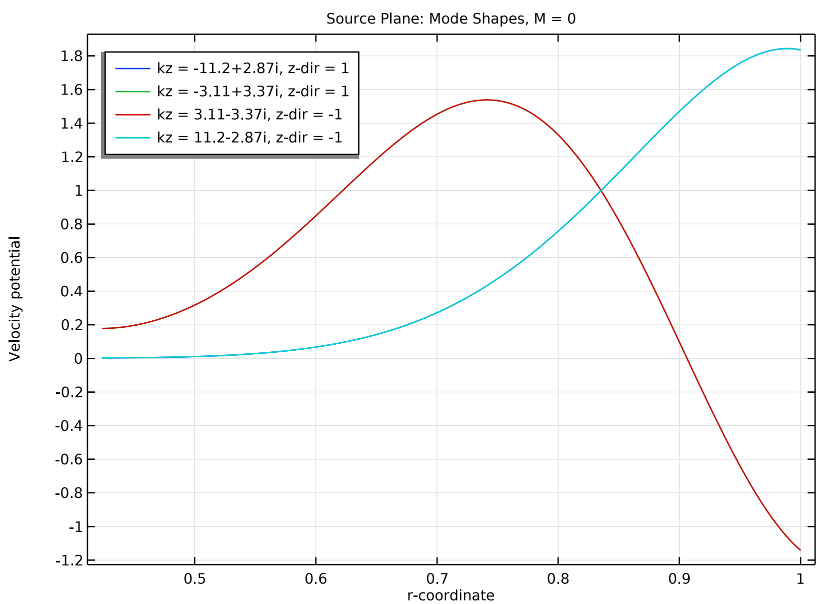
6 In the **Expression** text field, type `r`.

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

8 From the **Legends** list, choose **Evaluated**.

9 In the **Legend** text field, type `kz = eval(lpfbm.kz), z-dir = eval(if(intop_sp(lpfbm.Iz)>0,1,-1))`.

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



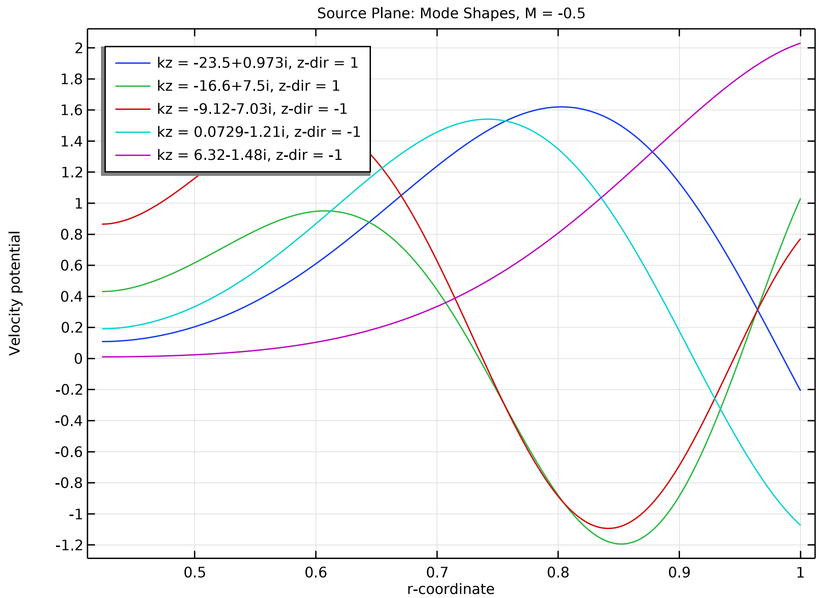
Source Plane: Mode Shapes, M = -0.5

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**.



Source Plane: Wave Numbers (real,imag)

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

2 In the **Settings** window for **1D Plot Group**, type `Source Plane: Wave Numbers (real, imag)` 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 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `real(kz)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `imag(kz)`.

8 Locate the **Legend** section. In the **Number of columns** text field, type `2`.

Global 1

1 Right-click **Source Plane: Wave Numbers (real,imag)** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
imag(lpfbm.kn)		

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

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

6 In the **Expression** text field, type `real(lpfbm.kn)`.

7 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

Filter 1

1 Right-click **Global 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Point Selection** section.

3 In the **Logical expression for inclusion** text field, type `intop_sp(lpfbm.Iz)>0`.

Global 2

1 In the **Model Builder** window, under **Results** > **Source Plane: Wave Numbers (real,imag)** right-click **Global 1** and choose **Duplicate**.

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

3 From the **Color** list, choose **Cycle (reset)**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

Filter 1

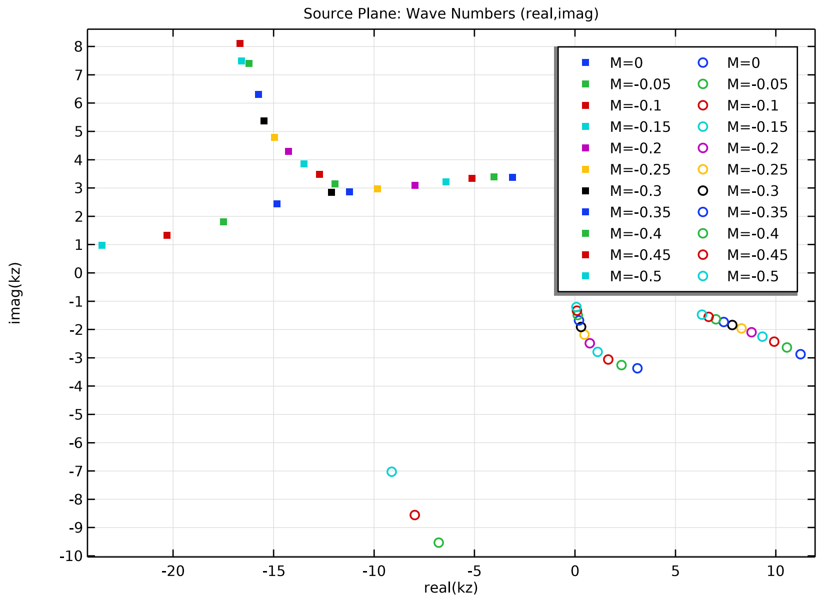
1 In the **Model Builder** window, expand the **Global 2** node, then click **Filter 1**.

2 In the **Settings** window for **Filter**, locate the **Point Selection** section.

3 In the **Logical expression for inclusion** text field, type `intop_sp(lpfbm.Iz)<0`.

4 In the **Source Plane: Wave Numbers (real,imag)** toolbar, click **Plot**.

This plot shows the dynamics of the wave number in the complex plane for an increasing flow. The incident and outgoing modes have different symbols.



Now, proceed and set up the same analysis for the terminal plane, including study and results, as discussed above.

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)	range(0,-0.05,-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 10.

6 In the **Maximum number of modes** text field, type 20.

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 -10.

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

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)**.

15 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 Click to expand 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 Click to expand 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`.

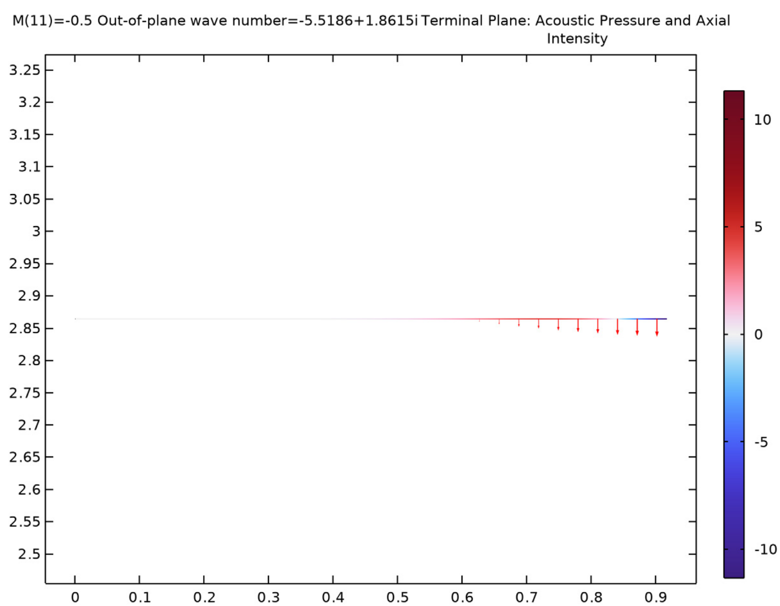
5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

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.



Revolution 2D 2

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

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

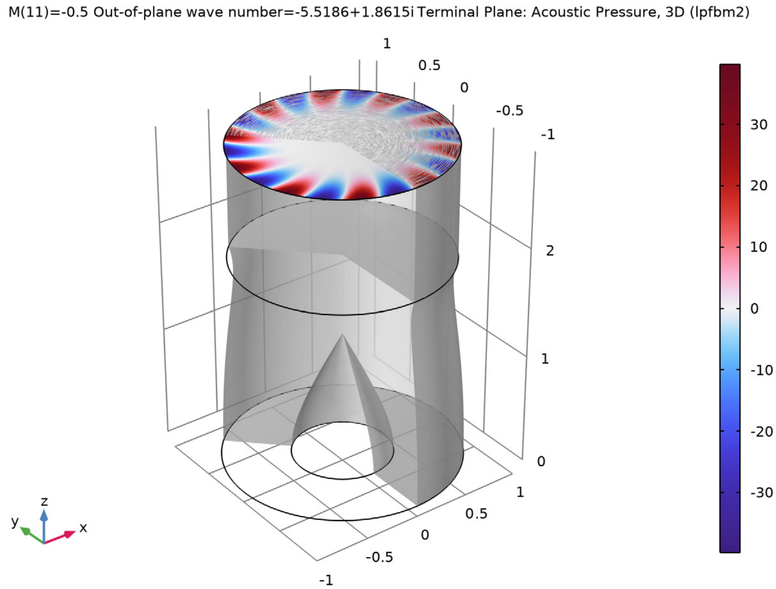
Surface

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 (sol16)**.

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

5 Click to expand 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**.

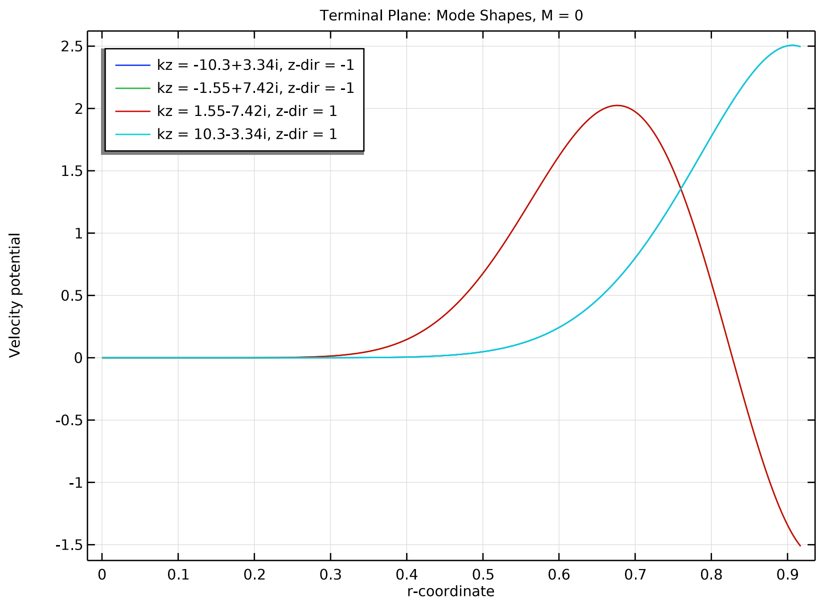
6 In the **Expression** text field, type `r`.

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

8 From the **Legends** list, choose **Evaluated**.

9 In the **Legend** text field, type `kz = eval(lpfbm2.kz), z-dir = eval(if(intop_tp(lpfbm2.Iz)>0,1,-1))`.

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



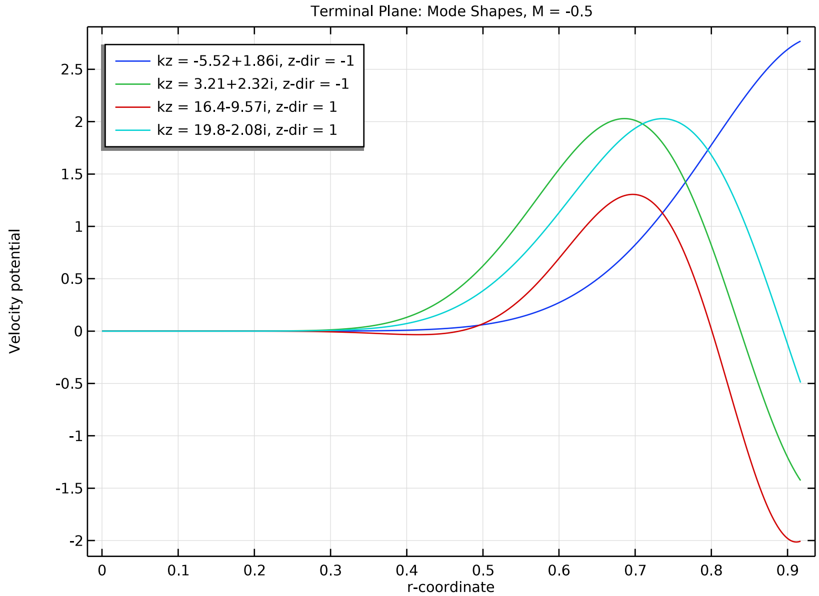
Terminal Plane: Mode Shapes, M = -0.5

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**.



Terminal Plane: Wave Numbers (real,imag)

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

2 In the **Settings** window for **1D Plot Group**, type `Terminal Plane: Wave Numbers (real,imag)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Terminal Plane Modes/ Parametric Solutions 2 (sol16)**.

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

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `real(kz)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `imag(kz)`.

8 Locate the **Legend** section. In the **Number of columns** text field, type `2`.

Global 1

1 Right-click **Terminal Plane: Wave Numbers (real,imag)** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
imag(lpfbm2.kn)		

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

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

6 In the **Expression** text field, type `real(lpfbm2.kn)`.

7 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

Filter 1

1 Right-click **Global 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Point Selection** section.

3 In the **Logical expression for inclusion** text field, type `intop_tp(lpfbm2.Iz)>0`.

Global 2

1 In the **Model Builder** window, under **Results** > **Terminal Plane: Wave Numbers (real,imag)** right-click **Global 1** and choose **Duplicate**.

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

3 From the **Color** list, choose **Cycle (reset)**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

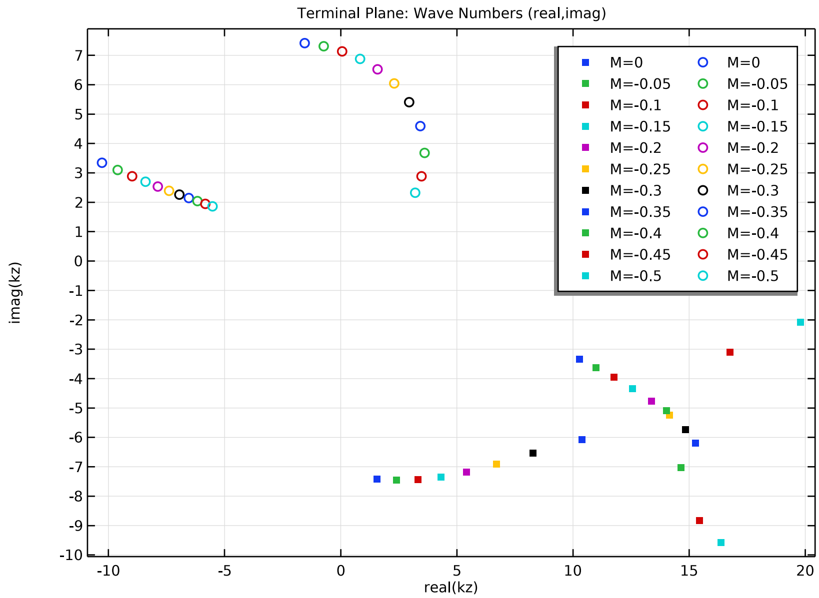
Filter 1

1 In the **Model Builder** window, expand the **Global 2** node, then click **Filter 1**.

2 In the **Settings** window for **Filter**, locate the **Point Selection** section.

3 In the **Logical expression for inclusion** text field, type `intop_tp(lpfbm2.Iz)<0`.

4 In the **Terminal Plane: Wave Numbers (real,imag)** 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 terminal planes. 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** 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**.

The two outgoing modes (propagating in the negative z direction) have index number 4 and 5, as can be seen in the `Source Plane: Mode Shapes, M = -0.5` plot. The other outgoing mode, that is solved for, has index number 3. This mode is a convected evanescent mode. This is more clearly seen in the `Source Plane: Wave Numbers (real,imag)` plot, where the mode appears.

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

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

4 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,5))`.

5 In the k_n^{out} text field, type `withsol('sol3',lpfbm.kn,setval(M,-0.5),setind(lambda,5))`.

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

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

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

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

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

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

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

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 Outgoing Mode Settings** section. In the $\phi_n{}^{out}$ text field, type `withsol('sol16',phi_tp,setval(M,-0.5),setind(lambda,3))`.

5 In the $k_n{}^{out}$ text field, type `withsol('sol16',lpfbm2.kn,setval(M,-0.5), setind(lambda,3))`.

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 Outgoing Mode Settings** section. In the $\phi_n{}^{out}$ text field, type `withsol('sol16',phi_tp,setval(M,-0.5),setind(lambda,4))`.

5 In the $k_n{}^{out}$ text field, type `withsol('sol16',lpfbm2.kn,setval(M,-0.5), setind(lambda,4))`.

Solve the frequency domain model for the flow (M = -0.5) cases with liner (finite impedance). 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 = -0.5, LINED)

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

Step 1: Frequency Domain

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 **Last**.

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**.

RESULTS

Normalized Pressure: M = -0.5, lined

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

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 28 (sol28)**.

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 = -0.5, 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 = -0.5, 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**.

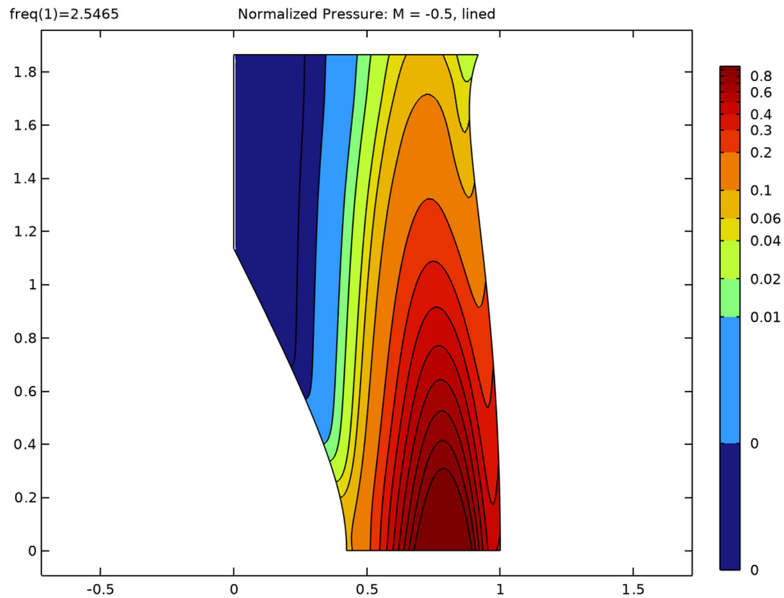
6 Clear the **Color legend** checkbox.

Normalized Pressure: M = -0.5, lined

1 In the **Model Builder** window, click **Normalized Pressure: M = -0.5, lined**.

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

3 Click the Zoom Extents button in the Graphics toolbar.



Intensity

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4 - Frequency Domain (M = -0.5, lined)/Solution 28 (sol28)**.

Surface 1

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

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

3 In the **Expression** text field, type `lpff.I_mag`.

Intensity

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

2 In the **Settings** window for **2D Plot Group**, click to expand the **Selection** section.

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

4 Select Domain 1 only.

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

Arrow Surface 1

1 Right-click **Intensity** and choose **Arrow Surface**.

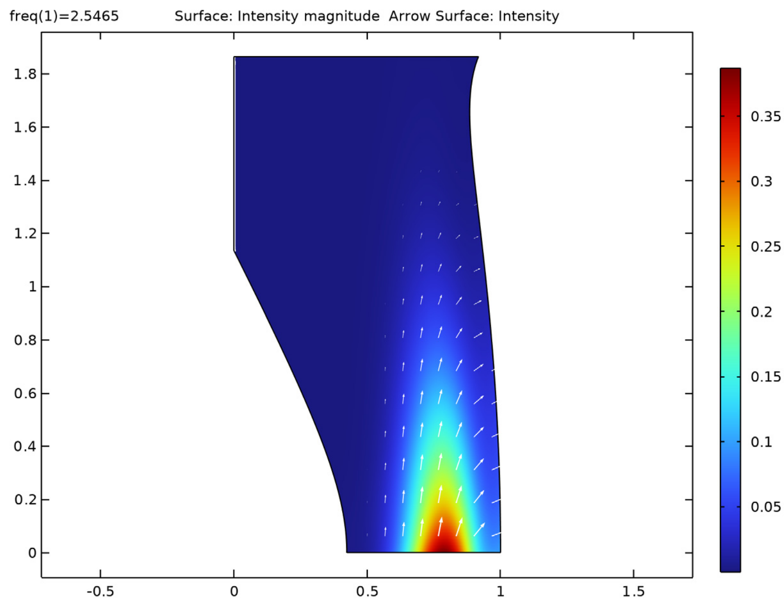
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Linearized Potential Flow, Frequency Domain** > **Intensity** > **lpff.Ir,lpff.Iz - Intensity**.

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** toolbar, click **Plot**.

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



Finally, create an evaluation group for computing the attenuation of the propagating mode.

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 = -0.5, lined)/ Solution 28 (sol28)**.

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**.