Created in COMSOL Multiphysics 6.0

Flow Duct

Introduction

The modeling of aircraft-engine noise attenuation is a central problem in the field of computational aeroacoustics (CAA). In this example you simulate the harmonically time-varying acoustic field from a turbofan engine under various background flow conditions (a convected acoustic simulation) and calculate the attenuation of the acoustic noise made possible by introducing a layer of lining inside the engine duct. The noise is introduced as a source computed from a boundary mode analysis.

Model Definition

Assume that the flow in the axisymmetric duct is compressible, inviscid, perfectly isentropic, and irrotational. In this case the acoustic field is well described by the linearized potential flow equations. The Linearized Potential Flow, Frequency Domain interface is used to set up the model.

The flow is in this model described by Euler's equations for an ideal gas (assuming adiabatic processes):

$$\frac{\partial\tilde{\rho}}{\partial t} + \nabla\cdot(\tilde{\rho}\tilde{\mathbf{v}}) = 0$$

$$\tilde{\rho}\left(\frac{\partial\tilde{\mathbf{v}}}{\partial t} + \tilde{\mathbf{v}}\cdot\nabla\tilde{\mathbf{v}}\right) + \nabla\tilde{p} = 0$$

$$\gamma\tilde{p} = \tilde{\rho}^{\gamma} \qquad \tilde{c}^2 = \frac{d\tilde{p}}{d\tilde{\rho}} = \tilde{\rho}^{\gamma-1}$$

Here $\tilde{\rho}$ is the density, $\tilde{\mathbf{v}}$ equals the velocity, $\tilde{p}$ denotes the pressure, $\tilde{c}$ equals the speed of sound, and γ is the constant ratio of the specific heats at constant pressure and volume. The variables are made dimensionless by division by suitable combinations of a reference duct radius R_∞, a reference speed of sound c_∞, and a reference density ρ_∞.

LINEARIZED POTENTIAL FLOW EQUATIONS

Because the flow is assumed irrotational, you can describe the velocity field, $\tilde{\mathbf{v}} = \tilde{\mathbf{v}}(r, z, t)$, in terms of a potential $\tilde{\phi}$, defined by the equation $\tilde{\mathbf{v}} = \nabla\tilde{\phi}$. The basic time- and space-dependent variables describing the flow are then the velocity potential and the density, $\tilde{\rho}$. These variables (and the velocity field itself) are split into a stationary mean-flow part and a harmonically time-varying acoustic part:

$$\tilde{\phi} = \Phi + \phi e^{i\omega t} \qquad \tilde{\mathbf{v}} = \mathbf{V} + \mathbf{v}e^{i\omega t} \qquad \tilde{\rho} = \rho_0 + \rho e^{i\omega t}$$

where ϕ, $\mathbf{v}$, and ρ are the acoustic variations to the potential, velocity, and density, respectively. Also assume that the amplitudes of the acoustic variables are small compared to the corresponding mean-flow quantities. This allows for a linearization of the equations of motion and the equation of state. The linearized potential flow equations for the acoustic variables are

$$i\omega\rho + \nabla \cdot (-\rho_0 \nabla\phi + \rho\mathbf{V}) = 0$$
$$\rho_0(i\omega\phi + \mathbf{V} \cdot \nabla\phi) = p$$
$$p = c_0^2 \rho$$

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

GEOMETRY AND BOUNDARY CONDITIONS

The duct geometry 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.

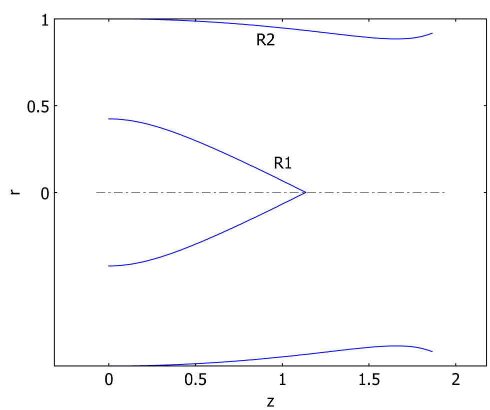


Figure 1: The duct geometry.

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 $0 \leq z' = z/L \leq 1$, 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*.

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.

To facilitate the COMSOL Multiphysics modeling, add a set of auxiliary domains to the geometry:

- A cylindrical domain — adjoined at the inlet plane and extending to the *terminal plane*, $z = 2.86393$ — 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 at the terminal plane.
- PML domains, adjoined at the source and terminal planes, allow you to conveniently implement nonreflecting boundary conditions for the aeroacoustic field. At the source plane, the PML domain is split into three annular sections with the innermost and outermost sections damping both in the axial and radial directions, while the central one is damping only in the axial direction. For more information about PMLs, see the *COMSOL Multiphysics Reference Manual*.

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 $r = 0$.

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

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

  where Z is the impedance. 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.

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 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.

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 appear in the cross-section plots in Figure 3.

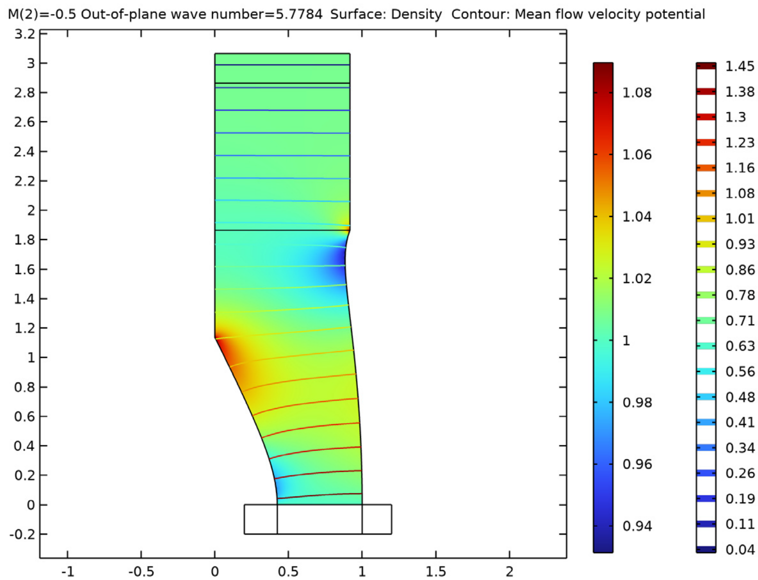


Figure 2: Mean-flow velocity potential and density for source-plane Mach number $M = -0.5$.

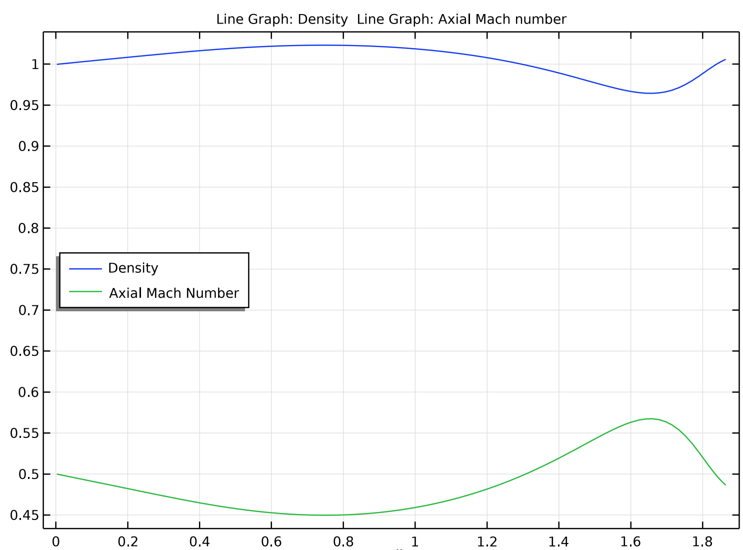


Figure 3: Mean-flow cross section plot at a sample radius of 0.8.

THE NOISE SOURCE

With the solution for the mean-flow field at hand, it is possible to calculate the corresponding eigenmodes for the acoustic field at the source plane. Figure 4 shows the resulting velocity-potential profile for the lowest mode. This is the boundary mode used as the source of the acoustic noise field in the duct for the case $M = -0.5$.

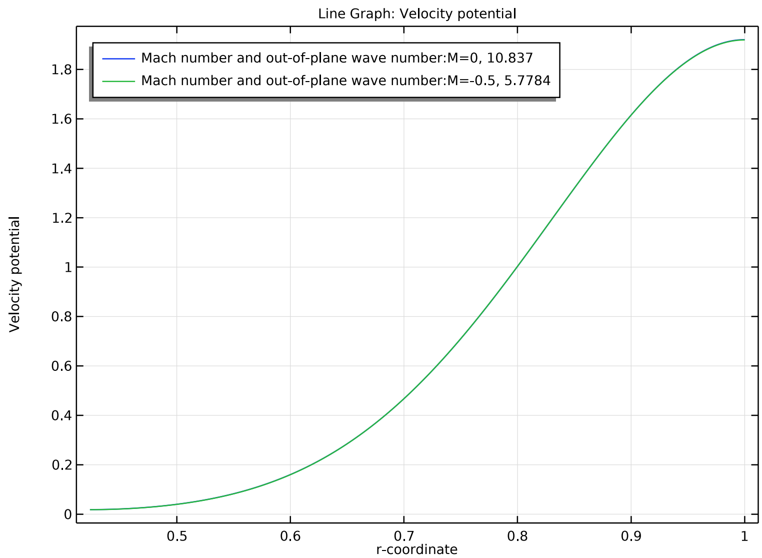


Figure 4: The first axial boundary mode at the source plane (z = 0) for the case of a background flow with Mach number $M = -0.5$.

THE AEROACOUSTIC FIELD

The pressure fields for the case without a background mean flow, shown in Figure 5, very closely match those for the corresponding finite element model (FEM) solutions presented in Figure 6 of Ref. 1. Similarly, the results for the attenuation between the source and inlet planes in the lined-wall case are in good agreement: 50.6 dB for the COMSOL Multiphysics solution versus 51.6 dB for the FEM solution in Ref. 1.

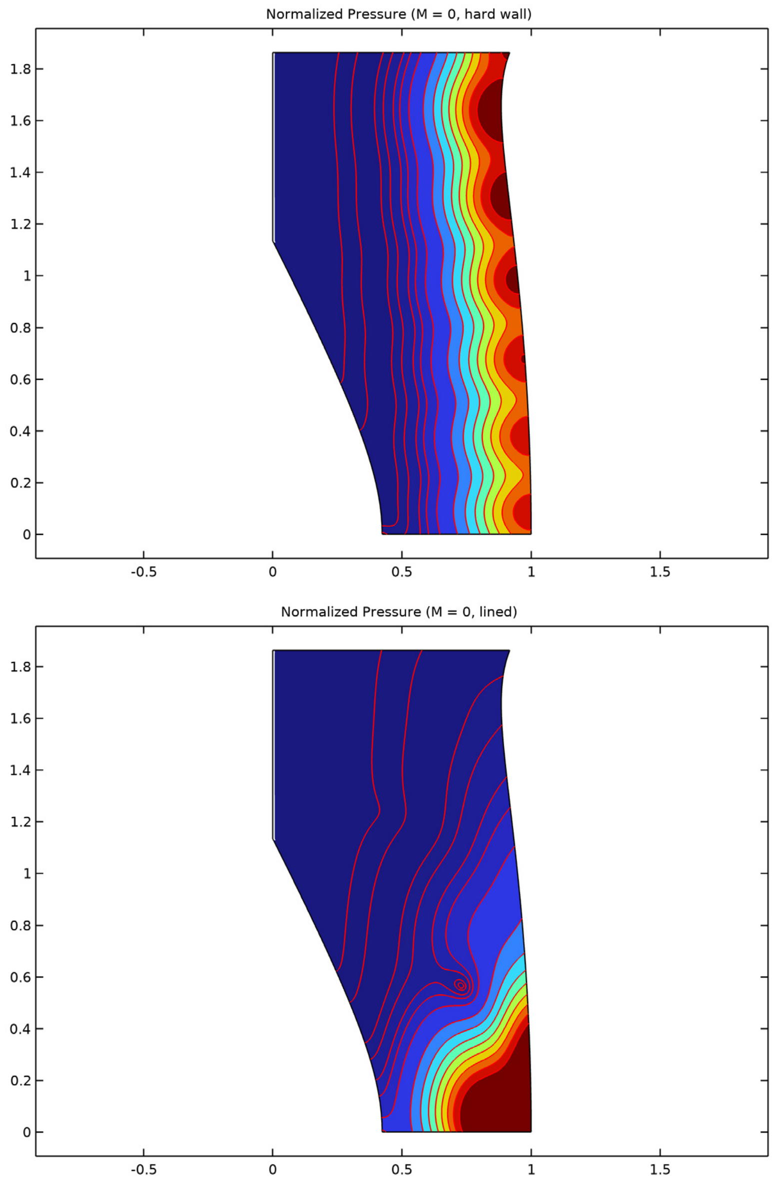


Figure 5: Acoustic pressure field for the cases of hard (top) and lined (bottom) duct wall with no mean flow and at circumferential mode number m = 10 and angular frequency ω *= 16.*

Turning to the case with a mean flow, the pressure field for the hard-wall case in the upper image of Figure 6 closely resembles the FEM solution obtained by Rienstra and Eversman in Ref. 1. For the lined-wall case in the lower image, although the agreement is still quite good, you can note some differences, especially near the source plane. This observation extends to the attenuation, for which the calculated value of 25.2 dB differs slightly more from the value of 27.2 dB obtained in Ref. 1.

However, these discrepancies have a natural explanation: 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. The lowest mode for the lined-wall case is a linear combination of the two forward-propagating hard-wall modes. Thus, the noise source term used to obtain the FEM solution visualized in the lower plot of Figure 6 is not optimally adapted to the duct, and it is consequently not maximally attenuated.

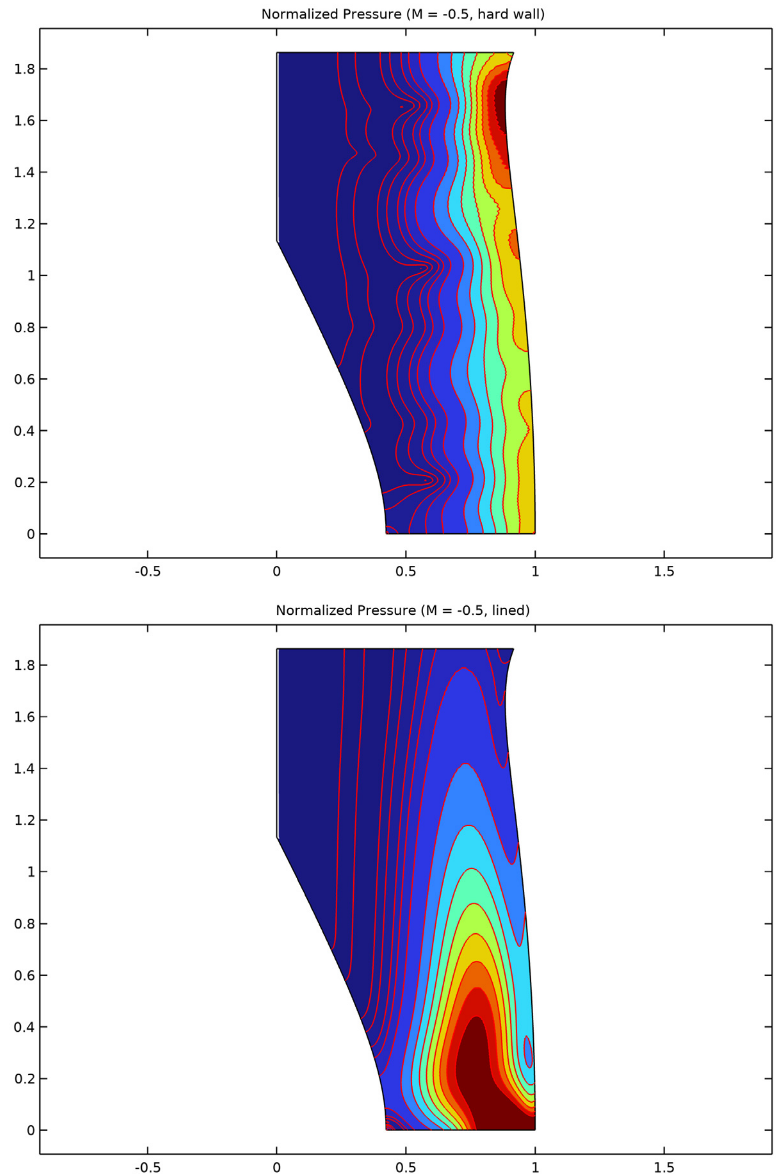


Figure 6: Acoustic pressure distribution for the cases of hard (top) and lined (bottom) duct wall with mean flow ($M = -0.5$) and at circumferential mode number $m = 10$ and angular frequency $\omega = 16$.

Notes About the COMSOL Implementation

The model involves three physics interfaces, the last of which is used twice:

- *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* (aebm) — for calculating the boundary eigenmode to be used as the source of the acoustic noise in the background mean-flow.
- *Linearized Potential Flow, Frequency Domain* (ae, ae2) — for modeling the time-harmonic acoustic field above and below the source plane.

After an initial modeling stage — consisting of creating the geometry and the mesh, then defining parameters, variables, and component couplings — you proceed with three consecutive stages corresponding to the items in the list above.

As explained in the Model Definition section, this model uses nondimensional variables obtained by dividing each variable by a suitable reference quantity of the same dimension. The reference length is the duct radius at the source plane (which is why it has the value 1). The mean-flow density and speed of sound at the source plane ($z = 0$) complete the set of reference variables.

References

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

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

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

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/flow_duct`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Acoustics>Aeroacoustics> Compressible Potential Flow (cpf)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Aeroacoustics>Linearized Potential Flow, Boundary Mode (aebm)**.
5 Click **Add**.
6 In the **Velocity potential** text field, type `phi_b`.
7 In the **Select Physics** tree, select **Acoustics>Aeroacoustics>Linearized Potential Flow, Frequency Domain (ae)**.
8 Click **Add** twice.
9 Click **Study**.
10 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.
11 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 and physical properties including the liner impedance.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `flow_duct_parameters.txt`.

GEOMETRY 1

First import the duct geometry, which is supplied in the form of an MPHBIN-file.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `flow_duct.mphbin`.

5 Click **Import**.

Next, add the auxiliary cylindrical domain between the inlet plane at z = 1.86393 and the terminal plane at z = 2.86393, including a PML layer.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.91705`.

4 In the **Height** text field, type `1.2`.

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

6 Click to expand the **Layers** section. Clear the **Layers on bottom** check box.

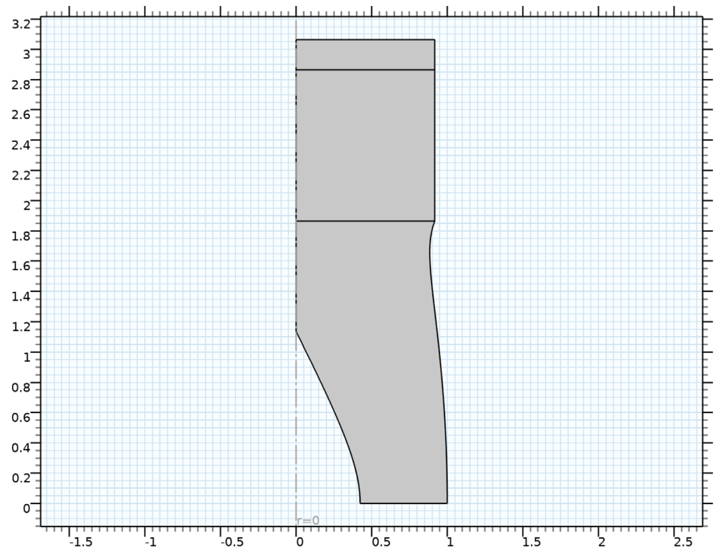
7 Select the **Layers on top** check box.

8 In the table, enter the following settings:

Layer name	Thickness
`Layer 1`	`0.2`

9 Click **Build Selected**.

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



Finally, attach the cylindrical PML domain at the outlet. This domain is divided into three as the PML here will be damping both in the axial and radial directions.

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `1`.

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

5 Locate the **Position** section. In the **r** text field, type `0.2`.

6 In the **z** text field, type `-0.2`.

7 Click **Build Selected**.

Thicken 1 (thi1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Thicken**.

2 In the **Settings** window for **Thicken**, locate the **Input** section.

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

4 On the object **imp1**, select Boundary 34 only.

5 Select the **Keep input objects** check box.

6 Locate the **Options** section. From the **Offset** list, choose **Asymmetric**.

7 In the **Upside thickness** text field, type `0.2`.

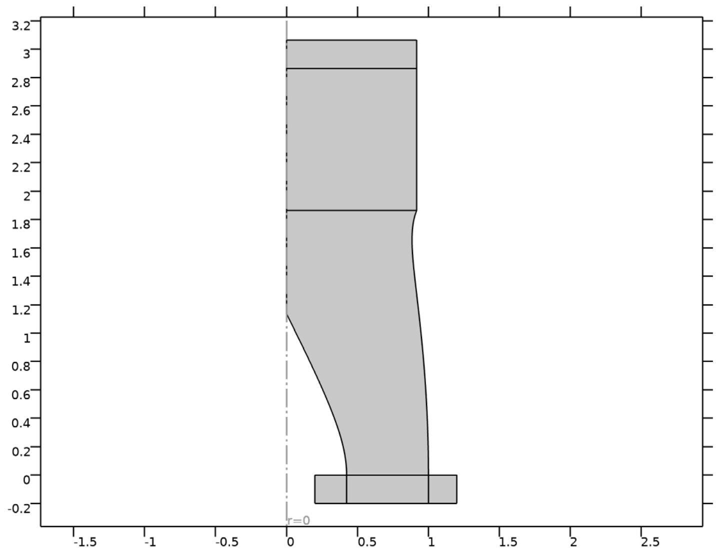
8 Click **Build All Objects**.

Form Union (fin)

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

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

The model geometry is now complete. The axisymmetric model geometry including duct domain and auxiliary domains.



MESH 1

Create a user-defined mapped mesh that is sufficiently fine to resolve the small-scale acoustic perturbations by following the instructions below.

Mapped 1

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

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 1 only.

This is the duct domain's boundary along the symmetry axis.

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

4 In the **Number of elements** text field, type `39`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 3 only.

   This is the symmetry-axis boundary segment for the auxiliary domain above the duct.

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

4 In the **Number of elements** text field, type `60`.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 6 and 43 only.

   These are the source-plane and terminal-plane boundaries. Note that you can make the selection by clicking the Paste Selection button and typing the indices in the dialog box that opens.

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

4 In the **Number of elements** text field, type `40`.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 5, 19, 20, 96, and 97 only.

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

4 In the **Number of elements** text field, type `18`.

Distribution 5

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type `1`.

Distribution 6

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 8–18, 22–39, and 66–94 only.

   You can do this most easily by copying the text `8-18, 22-39, 66-94` and then clicking in the **Selection** box and pressing Ctrl+V or by using the **Paste Selection** dialog box.

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

4 In the **Number of elements** text field, type `3`.

Distribution 7

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 40 and 95 only.

Distribution 8

1 Right-click **Mapped 1** and choose **Distribution**.

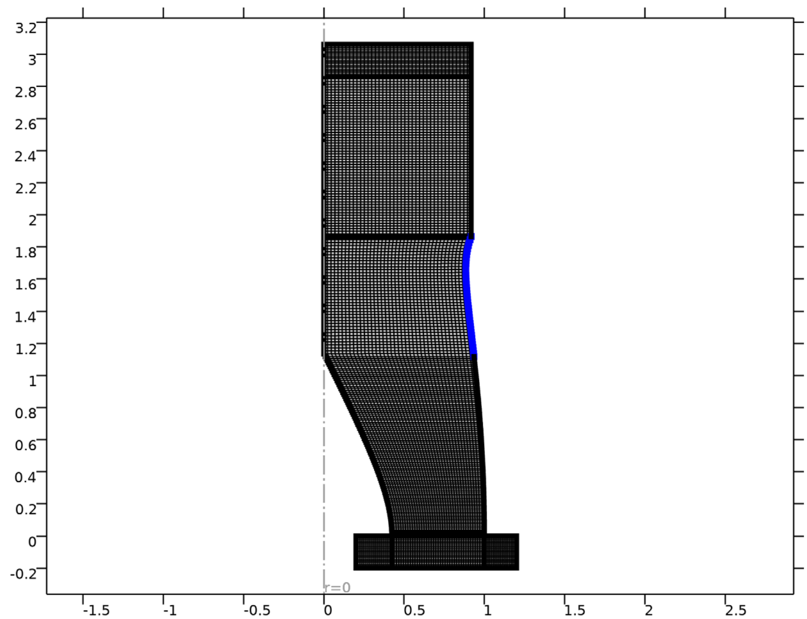
2 Select Boundaries 44–59 and 62–65 only.

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

4 In the **Number of elements** text field, type 2.

5 Click **Build All**.

The finished mesh should look like that in the figure below.



DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local 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 Domains 1 and 2 only.

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

Name	Expression	Description
Mz	-cpf.Vz	Axial Mach number

Next, define an expression for the source mode's intensity component normal to the source boundary.

Variables 2

1 In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.

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

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

4 Select Boundary 43 only.

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

Name	Expression	Description
Iz_src	A^2*0.5*real((aebm.p/cpf.rhoref+ aebm.Vr*aebm.vr-aebm.Vz*aebm.ikz* phi_b)*conj(rho*aebm.Vz-cpf.rhoref* aebm.ikz*phi_b))	Source mode intensity, normal component

Proceed by defining nonlocal integration couplings for the source and inlet planes for use in computing the attenuation.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_src` in the **Operator name** text field.

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

4 Select Boundary 43 only.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_inl` in the **Operator name** text field.

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

4 Select Boundary 4 only.

Using these couplings, define variables for the power through the source and inlet planes.

Variables 3

1. In the **Definitions** toolbar, click **Local Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. In the table, enter the following settings:

Name	Expression	Description
W_src	intop_src(Iz_src)	Power through source plane
W_inl	intop_inl(ae.Iz)	Power through inlet plane

The integral is automatically performed in the full azimuthal direction, because of the option selected in the integration coupling.

Because the variables you just defined cannot be evaluated for the same solution datasets, it is not possible to define a variable for the attenuation. Instead, create an analytic function that returns the attenuation when supplied with two power values.

Attenuation

1. In the **Definitions** toolbar, click **Analytic**.
2. In the **Settings** window for **Analytic**, type `Attenuation` in the **Label** text field.
3. In the **Function name** text field, type `dw`.
4. Locate the **Definition** section. In the **Expression** text field, type `10*log10(w_src/w_in)`.
5. In the **Arguments** text field, type `w_src, w_in`.

The Background Flow and the Source

In the following modeling steps you derive the stationary background as well as the acoustic source at the duct entrance. Both are used when modeling the time-harmonic acoustic perturbations that you will set up following this.

You calculate the stationary flow field using the Compressible Potential Flow interface defined on the duct geometry (Domain 1) and on the auxiliary region (Domain 2) appended at the inlet plane (z = 1.86393). Impose a mass-flow boundary condition at the source plane and a normal-flow condition at the terminal plane (z = 2.86393). The duct wall and the spinner are both impervious to the flow.

As the source generating the acoustic field in the duct, use a single boundary mode imposed at z = 0. More specifically, take this mode to be the lowest propagating axial mode in the duct computed in the background flow field from the previous stage of the modeling process. The subsequent instructions demonstrate how to derive this boundary mode.

Proceed with setting up the background flow physics and couple it to the mode analysis model. Then solve everything coupled for two values of the Mach number (M = 0 and M = -0.5). The solution datasets will include both background flow and source data.

COMPRESSIBLE POTENTIAL FLOW (CPF)

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

2 Select Domains 1–3 only.

3 In the **Settings** window for **Compressible Potential Flow**, locate the **Reference Values** section.

4 In the p_{ref} text field, type `cpf.rhoref^gamma/gamma`.

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

6 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 Select Boundary 7 only.

Mass Flow 1

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

2 Select Boundary 43 only.

Now set up Linearized Potential Flow, Boundary Mode physics for the boundary mode source calculation.

LINEARIZED POTENTIAL FLOW, BOUNDARY MODE (AEBM)

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

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

3 Click **Clear Selection**.

4 Select Boundary 43 only.

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

Linearized Potential Flow Model 1

The following settings couple the aeroacoustics boundary mode to the background flow:

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

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

3 From the ρ_0 list, choose **Density (cpf)**.

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

5 Specify the **V** vector as

cpf.Vr	r
cpf.Vz	z

6 In the **Model Builder** window, collapse the **Linearized Potential Flow, Boundary Mode (aebm)** node.

STUDY 1 - BACKGROUND AND SOURCE

Set up the solver. Step 1 solves the stationary background field and Step 2 is the boundary mode analysis. The solution of the stationary step (the background flow) is automatically used in the mode analysis step.

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

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

Mode Analysis

1 In the **Study** toolbar, click **Study Steps** and choose **Other>Mode Analysis**.

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

3 From the **Transform** list, choose **Out-of-plane wave number**.

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

5 Select the **Desired number of modes** check box.

6 In the associated text field, type `1`.

7 From the **Mode search method around shift** list, choose **Larger real part**.

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:

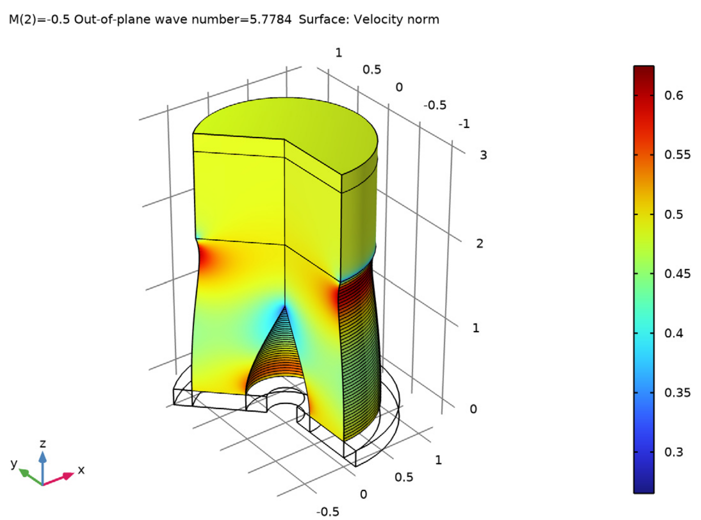
Parameter name	Parameter value list	Parameter unit
M (Mean flow Mach number)	0 -0.5	

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

RESULTS

Mean Flow Velocity, 3D (cpf)

The second default plot group is a 225° revolution plot of the velocity potential (of the first plot group).



In the plot you can also inspect the value of the out-of-plane wave number. Setting up the solver you only looked for the first propagating mode for each value of the Mach number. For M = 0 you have 10.84 and for M = -0.5 you have 5.778. The strong background flow has shifted the wave number.

NOTE: If you want to investigate more modes you can change the settings in the Mode Analysis study step. Set the desired number of modes, for example 12, and then set the search to Closest in absolute value. Solve again. Now, inspecting the Out-of-plane wave number list (for M = -0.5) you will find four solutions with a purely real wave number, three of them positive and one negative. In other words, there are four propagating waves, three of which propagate in the positive z direction and one in the opposite direction. The strong background flow has shifted the wave numbers, which in the absence of a mean flow would be symmetrically distributed around zero (if you select M = 0). If you had set these options you would have need to select the correct out-of-plane wave number in subsequent actions. Now you only have one wave number, the forward propagating one, for each M value.

To reproduce the plot shown in Figure 2 create a new 2D plot group.

Stationary: rho and Phi

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Stationary: rho and Phi` in the **Label** text field.

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

Surface 1

1 Right-click **Stationary: rho and Phi** and choose **Surface**.

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

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

Contour 1

1 In the **Model Builder** window, right-click **Stationary: rho and Phi** and choose **Contour**.

2 In the **Stationary: rho and Phi** toolbar, click **Plot**.

Compare the result with Figure 2.

Proceed with creating the figures in Figure 3 and Figure 4. First create a new dataset to get a detailed view of the density and velocity profiles along the length of the duct.

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.

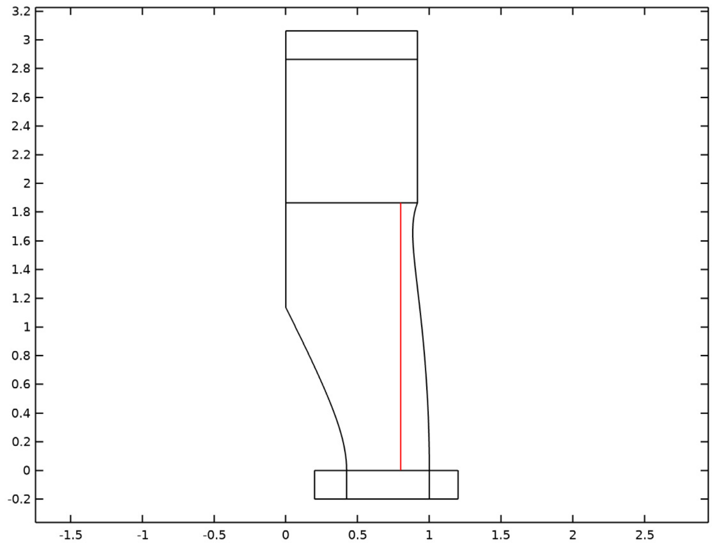
3 In row **Point 1**, set **R** to `0.8`.

4 In row **Point 2**, set **R** to `0.8` and **z** to `1.86393`.

5 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Background and Source/ Parametric Solutions 1 (sol3)**.

6 Click **Plot**.

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



Stationary: rho and Mz

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

2 In the **Settings** window for **1D Plot Group**, type `Stationary: 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 **From list**.

5 In the **Parameter values (M)** list, select **-0.5**.

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

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

Line Graph 1

1 Right-click **Stationary: 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. From the **Legends** list, choose **Manual**.

7 Select the **Show legends** check box.

8 In the table, enter the following settings:

Legends
Density

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

Legends
Axial Mach Number

5 In the **Stationary: rho and Mz** toolbar, click **Plot**.

The resulting plot should closely resemble that in Figure 3.

Boundary Mode Potential: phi_b

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

2 In the **Settings** window for **1D Plot Group**, type Boundary Mode Potential: phi_b in the **Label** text field.

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

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

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

Line Graph 1

1 Right-click **Boundary Mode Potential: phi_b** and choose **Line Graph**.

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

3 Click to select the **Activate Selection** toggle button.

4 Select Boundary 43 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type phi_b.

6 Locate the **Legends** section. Select the **Show legends** check box.

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

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

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

10 In the **Boundary Mode Potential: phi_b** toolbar, click **Plot**.

The resulting plot of the desired boundary modes should closely resemble that in Figure 4. Note that the shapes of the two seem identical. Here the real part is plotted. Change the plot parameter to `imag(phi_p)` and you will see a difference.

The Acoustic Field

Equipped with the solution derived in the previous stage, you can now go on to simulate the acoustic field. Model the noise source through judicious choices of boundary conditions at the source plane ($z = 0$) for the two Linearized Potential Flow, Frequency Domain interfaces. Furthermore, implement nonreflecting boundary conditions at both ends of the duct geometry by using the auxiliary PML domains that you added to the model earlier in the geometry creation steps.

Here are the detailed instructions for the procedure.

DEFINITIONS

Also, create a nonlocal maximum coupling for the duct domain and use it to define a normalized absolute pressure variable.

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

2 Select Domain 1 only.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Description
pabsn	abs(ae.p)/comp1.maxop1(ae.p)	Normalized pressure

Perfectly Matched Layer 1 (pml1)

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

2 Select Domain 3 only.

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

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

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

6 In the **Typical wavelength** text field, type `ae.c0/f`.

Perfectly Matched Layer 2 (pml2)

1 Right-click **Perfectly Matched Layer 1 (pml1)** and choose **Duplicate**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 4–6 only.

5 Locate the **Scaling** section. In the **Typical wavelength** text field, type `ae2.c0/f`.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (AE)

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

2 Select Domains 1–3 only.

3 In the **Settings** window for **Linearized Potential Flow, Frequency Domain**, click to expand the **Equation** section.

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

Couple the mean-flow field to the linearized potential flow (LPF) model by using the dedicated Multiphysics coupling. Verify that the correct physics are coupled. The multiphysic coupling is not used for the second LPF physics. The latter represents a truncation domain with constant flow conditions.

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 (AE)

Normal Mass Flow 1

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

2 Select Boundary 43 only.

3 In the **Settings** window for **Normal Mass Flow**, locate the **Normal Mass Flow** section.

4 In the m_n text field, type `rho*(-aebm.ikz*A*phi_b)`.

Impedance 1

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

2 Select Boundaries 44–59 and 62–95 only.

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

4 In the Z_i text field, type `Zw/flc2hs(z/zi,b)`.

The reason behind using the smoothed Heaviside function `flc2hs` is to make the impedance a continuous (albeit abruptly changing) function across the interfaces between regions with and without an acoustic lining. This is a condition required for the equivalence of Myers's original impedance boundary condition and its weak reformulation due to Eversman used here to hold (see Ref. 3).

Impedance 2

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

2 Select Boundaries 60 and 61 only.

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

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

The second linearized potential flow interface is used to set up an open domain at the outlet. Since this interface has a different dependent variable (`phi2`), it allows to set up a port-like condition at the outlet with a discontinuous dependent variable (only the scattered field is absorbed).

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN 2 (AE2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Frequency Domain 2 (ae2)**.

2 Select Domains 4–6 only.

3 In the **Settings** window for **Linearized Potential Flow, Frequency Domain**, locate the **Linearized Potential Flow Equation Settings** section.

4 In the m text field, type `m`.

Linearized Potential Flow Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Linearized Potential Flow, Frequency Domain 2 (ae2)** click **Linearized Potential Flow Model 1**.

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

3 From the ρ_0 list, choose **User defined**. In the associated text field, type `rho0`.

4 From the c_0 list, choose **User defined**. In the associated text field, type `c0`.

5 Specify the **V** vector as

0	r
M	z

Linearized Potential Flow Model 2

1 Right-click **Component 1 (comp1)>Linearized Potential Flow, Frequency Domain 2 (ae2)> Linearized Potential Flow Model 1** and choose **Duplicate**.

2 Select Domains 4 and 6 only.

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

4 Specify the **V** vector as

0	r
0	z

Velocity Potential 1

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

The potential is set equal to only the scattered potential from the main domain, that is, the total potential `phi` minus the incident potential `A*phi_b`.

2 Select Boundary 43 only.

3 In the **Settings** window for **Velocity Potential**, locate the **Velocity Potential** section.

4 In the ϕ_0 text field, type `phi-A*phi_b`.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN 2 (AE2)

In the **Model Builder** window, collapse the **Component 1 (comp1)>Linearized Potential Flow, Frequency Domain 2 (ae2)** node.

Next compute the acoustic fields for the case with (M = -0.5) and without (M = 0) background flow as well as with and without the wall lining. Do this by adding four frequency domain studies, select the desired background flow solution, and enable or disable the impedance boundary condition using the **Modify physics tree and variables for study step** option in the solver step.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

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

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

6 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces> Frequency Domain**.

7 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

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

9 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Frequency Domain**.

10 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

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

12 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Frequency Domain**.

13 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

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

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

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

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

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

3 Clear the **Generate default plots** check box.

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

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Frequency Domain (M = 0, lined)** click **Step 1: Frequency Domain**.

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

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

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

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

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

7 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

8 From the **Use** list, choose **M=0 (sol4)**.

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

STUDY 3 - FREQUENCY DOMAIN (M = -0.5, LINED)

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

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 3 - Frequency Domain (M = -0.5, lined)`.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3 - 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 **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 and Source, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

8 From the **Use** list, choose **M=-0.5 (sol5)**.

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

STUDY 4 - FREQUENCY DOMAIN (M = 0, HARD WALL)

1 In the **Model Builder** window, click **Study 4**.

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 4 - Frequency Domain (M = 0, hard wall)`.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 4 - Frequency Domain (M = 0, hard wall)** 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** check box.

5 In the tree, select **Component 1 (Comp1)>Linearized Potential Flow, Frequency Domain (Ae)>Impedance 1**.

6 Click **Disable**.

7 In the tree, select **Component 1 (Comp1)>Linearized Potential Flow, Frequency Domain (Ae)>Impedance 2**.

8 Click **Disable**.

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

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

11 From the **Study** list, choose **Study 1 - Background and Source, Mode Analysis**.

12 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

13 From the **Use** list, choose **M=0 (sol4)**.

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

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

1 In the **Model Builder** window, click **Study 5**.

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 5 - Frequency Domain (M = -0.5, hard wall)`.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 5 - Frequency Domain (M = -0.5, hard wall)** 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** check box.
5 In the tree, select **Component 1 (Comp1)>Linearized Potential Flow, Frequency Domain (Ae)>Impedance 1**.
6 Click **Disable**.
7 In the tree, select **Component 1 (Comp1)>Linearized Potential Flow, Frequency Domain (Ae)>Impedance 2**.
8 Click **Disable**.
9 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
10 From the **Method** list, choose **Solution**.
11 From the **Study** list, choose **Study 1 - Background and Source, Mode Analysis**.
12 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.
13 From the **Use** list, choose **M=-0.5 (sol5)**.
14 In the **Home** toolbar, click **Compute**.

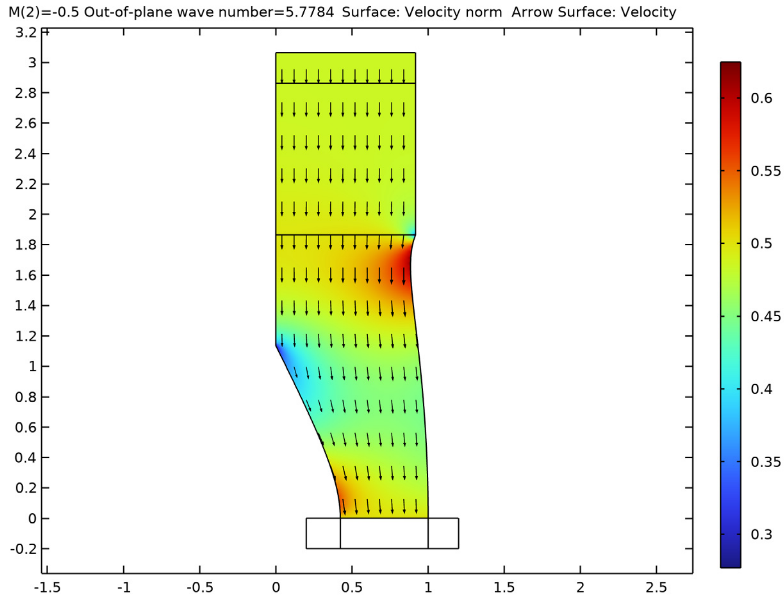
RESULTS

First, have a look at the mean flow velocity and add arrow to understand the flow direction. This is also the direction that defines the outlet and inlet of the model. You can also inspect the remaining default plots generated by the first study.

Arrow Surface 1

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



Pressure: M = 0, lined

Now, proceed to create the results depicted in Figure 5 and in Figure 6. Start by creating one plot and set it up, then simply duplicate the plot and select the correct dataset. After generating the plots proceed to calculating the attenuation of the system under **Derived Values** in the **Results** node.

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

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

4 In the **Title** text area, type `Normalized Pressure (M = 0, lined)`.

5 Clear the **Parameter indicator** text field.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

Contour 1

1 Right-click **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 **Legend type** list, choose **Line**.
8 Clear the **Color legend** check box.

Contour 2

1 In the **Model Builder** window, right-click **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 **Coloring** list, choose **Uniform**.
7 Clear the **Color legend** check box.
8 In the **Pressure: M = 0, lined** toolbar, click **Plot**.

The plot does not look exactly like the one at the top of Figure 5, this is because only the solution inside the flow duct is of interest. Add selections to the datasets to restrict plotting to this domain.

Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)

In the **Model Builder** window, under **Results>Datasets** click **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 1 only.

Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)

In the **Model Builder** window, under **Results>Datasets** click **Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 1 only.

Study 4 - Frequency Domain (M = 0, hard wall)/Solution 8 (sol8)

In the **Model Builder** window, under **Results>Datasets** click **Study 4 - Frequency Domain (M = 0, hard wall)/Solution 8 (sol8)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 1 only.

Study 5 - Frequency Domain (M = -0.5, hard wall)/Solution 9 (sol9)

In the **Model Builder** window, under **Results>Datasets** click **Study 5 - Frequency Domain (M = -0.5, hard wall)/Solution 9 (sol9)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 1 only.

Pressure: M = 0, lined

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

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

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

This should look like the lower plot in Figure 5.

Pressure: M = -0.5, lined

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

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

3 Locate the **Title** section. In the **Title** text area, type `Normalized Pressure (M = -0.5, lined)`.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)**.

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

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

This should look like the lower plot in Figure 6.

Pressure: M = 0, hard wall

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

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

3 Locate the **Title** section. In the **Title** text area, type `Normalized Pressure (M = 0, hard wall)`.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 4 - Frequency Domain (M = 0, hard wall)/Solution 8 (sol8)**.

5 In the **Pressure: M = 0, hard wall** toolbar, click **Plot**.

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

This should look like the upper plot in Figure 5.

Pressure: M = -0.5, hard wall

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

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

3 Locate the **Title** section. In the **Title** text area, type `Normalized Pressure (M = -0.5, hard wall)`.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 5 - Frequency Domain (M = -0.5, hard wall)/Solution 9 (sol9)**.

5 In the **Pressure: M = -0.5, hard wall** toolbar, click **Plot**.

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

This should look like the upper plot in Figure 6.

Finally, calculate the attenuation. But first, determine the power at the source and the duct inlet. These numerical values are used to evaluate the attenuation using the attenuation function `dw` you have created.

Global Evaluation: W_src (M = 0)

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

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_src (M = 0)` in the **Label** text field.

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

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

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

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

Expression	Unit	Description
W_src		Power through source plane

7 Click **Evaluate**.

Global Evaluation: W_src (M = -0.5)

1 Right-click **Global Evaluation: W_src (M = 0)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_src (M = -0.5)` in the **Label** text field.

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

4 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

Global Evaluation: W_inl (M = 0, lined)

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_inl (M = 0, lined)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

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

Expression	Unit	Description
W_inl		Power through inlet plane

5 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

Global Evaluation: W_inl (M = -0.5, lined)

1 Right-click **Global Evaluation: W_inl (M = 0, lined)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_inl (M = -0.5, lined)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)**.

4 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

Global Evaluation: Attenuation (M = 0, lined)

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

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: Attenuation (M = 0, lined)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

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

Expression	Unit	Description
comp1.dw(0.02796, 2.43e-7)		Attenuation

5 Click **Evaluate**.

The result should be approximately 50.6 dB, which is quite close to that in Ref. 1 (51.6 dB). The reason is that both use the same source mode as a result of the fact that there is a single forward-propagating mode in the flow-free case, thus making the two calculations directly comparable.

Global Evaluation: Attenuation (M = -0.5, lined)

1 Right-click **Global Evaluation: Attenuation (M = 0, lined)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: Attenuation (M = -0.5, lined)` in the **Label** text field.

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

Expression	Unit	Description
comp1.dw(9.5194e-3, 2.8650e-5)		Attenuation

4 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

The result should now be roughly to 25.2 dB. Compare this result to the value of approximately 27 dB obtained for the corresponding quantity in Ref. 1. In contrast to that paper, the source mode used in these calculations was derived for the case of a hard duct wall, whereas the lowest mode for the lined-wall case would be a linear combination of the two forward-propagating hard-wall modes. For this reason, the noise source is not an eigenmode and is, consequently, not maximally attenuated.