Model created in COMSOL Multiphysics 6.4

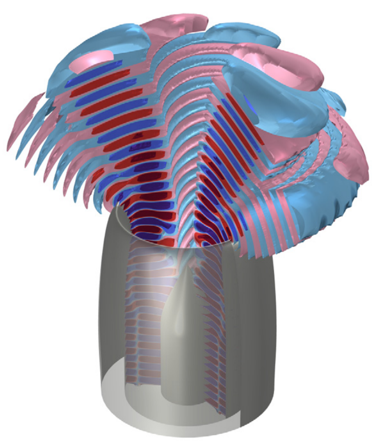
Generic Nacelle with an Acoustic Liner

Introduction

In this model, the sound propagation from a generic nacelle of an aircraft engine is modeled. This is an aeroacoustic problem with a background flow field through the nacelle. Two different modeling approaches are demonstrated in the model:

- One using Compressible Potential Flow for the background flow field and Linearized Potential Flow, Frequency Domain for the acoustics
- A second, more comprehensive, method using Turbulent Flow for the background flow field and Linearized Navier–Stokes, Frequency Domain for the acoustic field

An acoustic liner is used to dampen the sound from the engine. It is modeled by an impedance on the boundary. In the model with the Linearized Potential Flow, there is no turbulent boundary layer in the background flow field. To include the effect of the turbulent boundary layer, the Brambley impedance boundary condition is used (see Ref. 1 and Ref. 2).



Model Definition

The model uses a 2D axisymmetric geometry of the air domain at the inlet of a generic nacelle from an aircraft engine; see Figure 1. In the acoustic simulations, a PML region is used to dampen the acoustic waves traveling away from the nacelle. The acoustic liner is placed on the inside of the outer nacelle wall.

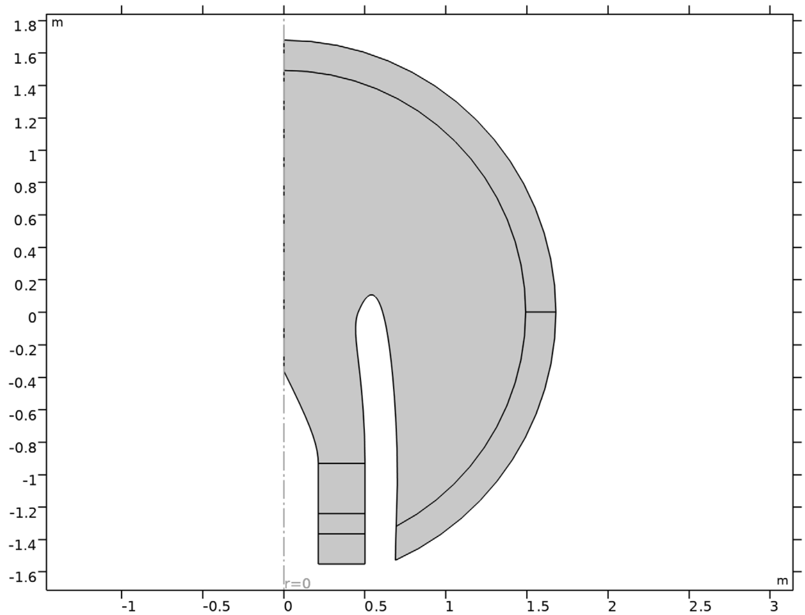


Figure 1: Geometry of the air domain at the inlet of a generic nacelle.

The mode investigated has an angular wavenumber $m = 4$ and a radial wavenumber of $n = 1$. In the Linearized Potential Flow interface, a Port feature is used to excite the mode, whereas in the Linearized Navier–Stokes interface, the propagating modes are calculated with a boundary-mode study and subsequently excited with a Background Flow Field feature.

In the Linearized Potential Flow, the impedance model used is the Brambley impedance model, which includes the effect of the turbulent boundary layer in the background flow field. The implemented impedance model assumes that the turbulent boundary layer is linear. The Brambley boundary condition can be seen as an extension to the classical Ingard–Myers impedance condition. The model compares the two methods and investigates the impact of an acoustic liner.

Results and Discussion

The background flow field modeled with a turbulent flow and a potential flow for the two models are slightly different. The most important difference is that the potential flow by definition does not have a turbulent boundary layer at the nacelle wall. There are slight differences in the maximum flow velocity but the total mass flow through the nacelle is the same. Figure 2 shows the background flow field modeled with the potential flow.

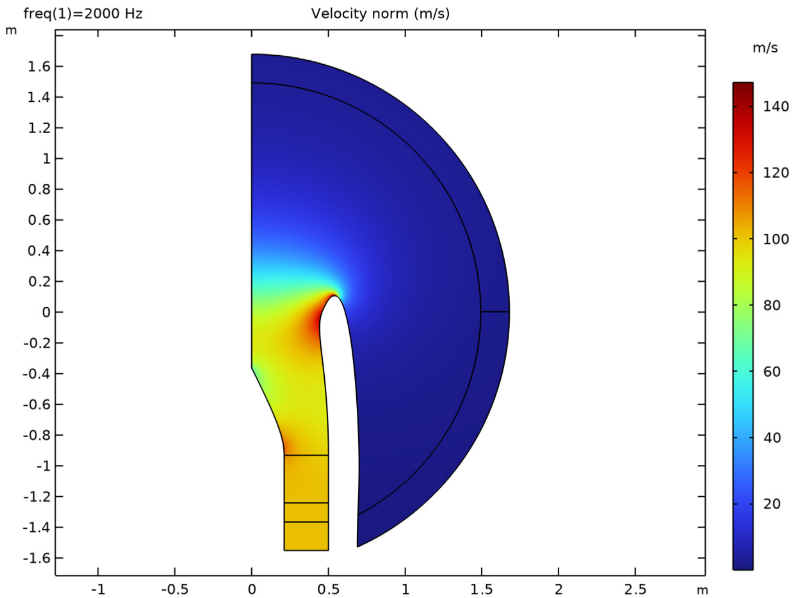


Figure 2: Background flow through the nacelle from Compressible Potential Flow.

Figure 3 shows the sound pressure level for the excited mode at 2000 Hz for the Linearized Potential Flow. The excited mode has the azimuthal wavenumber $m = 4$ and radial wavenumber $n = 1$.

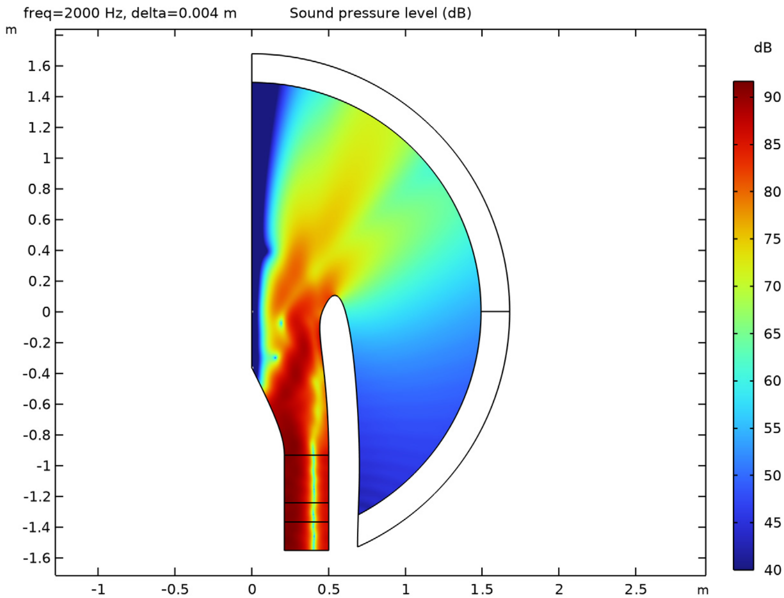


Figure 3: Sound pressure level of the mode m = 4 and n = 1 at 2000 Hz, modeled with Linearized Potential Flow.

The transmitted sound pressure level is modeled for with and without an acoustic liner. Figure 4 shows the angle dependency of the transmitted sound for both setups. For the Brambley boundary condition the thickness of the turbulent boundary layer has to be estimated in this model. The best fit is for a boundary-layer thickness of 2.2 mm. In reality, the boundary-layer thickness is not constant across the acoustic liner.

Figure 5 shows the angle-dependent insertion loss of the acoustic liner in the generic nacelle for both studies. Again a boundary layer thickness of 2.2 mm is used.

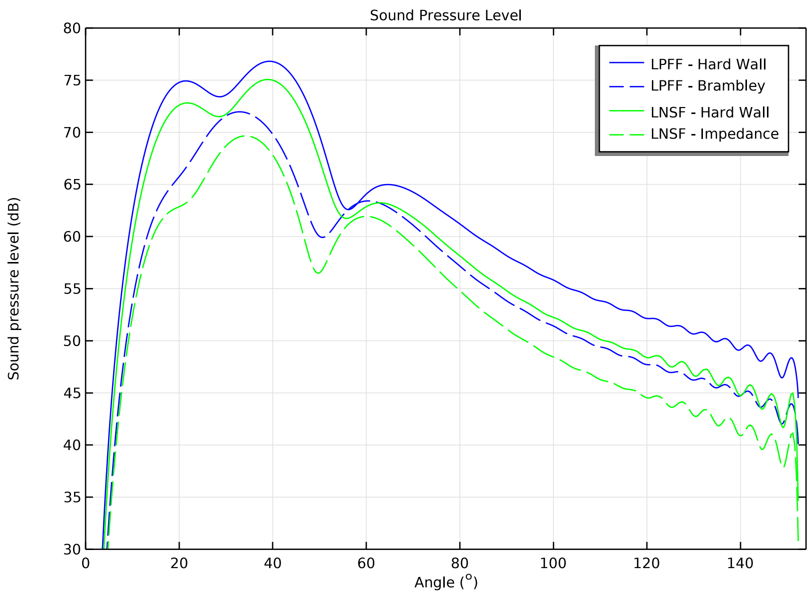


Figure 4: Angle-dependent sound pressure level of the acoustic fields. Results are shown for both modeling methods and with and without the inserted acoustic liner.

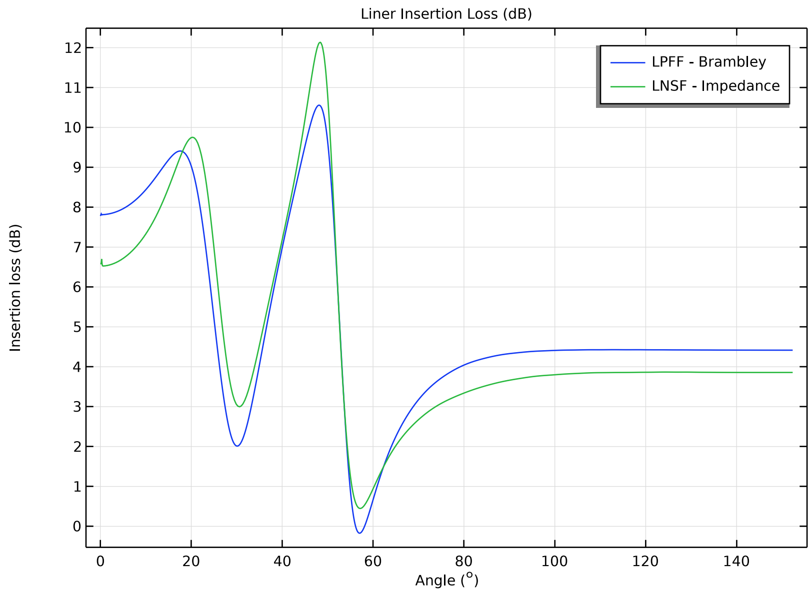


Figure 5: Angle-dependent insertion loss of the acoustic liner in dB.

Notes About the COMSOL Implementation

To excite the correct modes in Linearized Navier–Stokes, the modes are calculated with a boundary-mode study, whose solutions are subsequently used as a background-pressure field.

References

1. E. Brambley, “Well-Posed Boundary Condition for Acoustic Liners in Straight Ducts with Flow,” *AIAA J.*, vol. 49, no. 6, pp. 1272–1282, 2011.

2. L.A. Seki, A.M. Spillere, and J.A. Cordioli, “Numerical implementation of an impedance boundary condition considering a finite boundary layer,” *Proc. AIAA Aviation 2021 Forum*, AIAA 2021-2168, 2021.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/generic_nacelle_liner`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

2 In the **Select Physics** tree, select **Acoustics** > **Aeroacoustics** > **Compressible Potential Flow (cpf)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics** > **Aeroacoustics** > **Linearized Potential Flow, Frequency Domain (lpff)**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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 `generic_nacelle_liner_parameters.txt`.

Build the geometry using parametric curves.

GEOMETRY 1

Parametric Curve 1 (pc1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** node.

2 Right-click **Geometry 1** and choose **More Primitives** > **Parametric Curve**.

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

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

5 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 **Expressions** section.

3 In the **r** text field, type `1/2*(0.64212-sqrt(0.04777+0.98234*s^2))`.

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

5 Locate the **Parameter** section. In the **Maximum** text field, type `0.7`.

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

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

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

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 **pc2**, select Point 1 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.

Offset 1 (off1)

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

2 Select the object **ls2** only.

3 In the **Settings** window for **Offset**, locate the **Options** section.

4 In the **Distance** text field, type `-zi/3`.

5 Click **Build Selected**.

Offset 2 (off2)

1 Right-click **Offset 1 (off1)** and choose **Duplicate**.

2 Select the object **off1** only.

Line Segment 3 (ls3)

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

2 On the object **ls2**, select Point 1 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 **off2**, select Point 1 only.

Line Segment 4 (ls4)

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

2 On the object **ls2**, select Point 2 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 **off2**, select Point 2 only.

Line Segment 5 (ls5)

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 In the **z** text field, type `-2/3*zi`.

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

6 In the **z** text field, type `-2/3*zi`.

7 In the **r** text field, type `1.5*zi`.

Interpolation Curve 1 (ic1)

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

2 In the **Settings** window for **Interpolation Curve**, locate the **Interpolation Points** section.

3 In the table, enter the following settings:

r (m)	z (m)
ri	zi
ri*1.1	zi*1.09
1.5*ri	1/3*zi
1.5*ri	-2/3*zi

4 Locate the **End Conditions** section. From the **Condition at endpoint** list, choose **Tangent direction**.

5 In the **r** text field, type 0.

6 In the **z** text field, type -1.

Convert to Solid 1 (csol1)

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

2 In the **Settings** window for **Convert to Solid**, locate the **Input** section.

3 From the **Input objects** list, choose **All objects**.

4 Click **Build All Objects**.

Circle 1 (c1)

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

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

3 In the **Radius** text field, type 1.8*zi.

4 In the **Sector angle** text field, type 180.

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

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.2*zi

Union 1 (uni1)

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

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

3 From the **Input objects** list, choose **All objects**.

4 Click **Build Selected**.

Delete Entities 1 (del1)

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

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **uni1**, select Domains 1–3 and 9–11 only.

5 Click **Build Selected**.

Move 1 (mov1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the object **del1** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

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

5 Click **Build All Objects**.

Offset 3 (off3)

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

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

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

4 On the object **mov1**, select Boundary 5 only.

5 Locate the **Options** section. In the **Distance** text field, type `-0.2*zi`.

6 Click **Build Selected**.

Ignore Edges 1 (ige1)

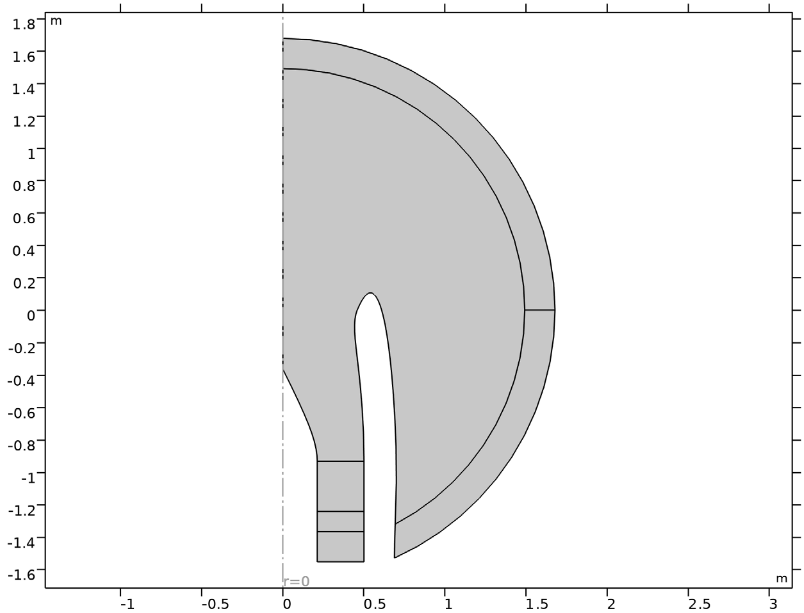
1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Boundary 19 only.

Form Union (fin)

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

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



ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Air**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Perfectly Matched Layer**.

3 Select Domains 2 and 6 only.

4 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

5 From the **Coordinate stretching type** list, choose **Rational**.

6 Locate the **Geometry** section. From the **Type** list, choose **User defined**.

7 In the table, enter the following settings:

	Distance function (m)	Thickness (m)
Direction 1	`sqrt(r^2+z^2)-1.6*zi`	`0.2*zi`

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

9 In the **Typical wavelength** text field, type `lam0`.

10 In the **PML scaling curvature parameter** text field, type `3`.

To set up the perfectly matched layer, use the user-defined option. Here, the distance function is written as a user defined expression.

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 v_{ref} text field, type `Ma*c0`.

Mass Flow 1

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

2 Select Boundary 5 only.

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

4 In the v_{n} text field, type `-cpf.vref`.

Normal Flow 1

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

2 Select Boundaries 16 and 17 only.

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)

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`.

Port 1

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

Add a port feature to excite the desired mode. The mode has the radial wave number `n` and angular wave number `m`.

2 Select Boundary 5 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Annular**.

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

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

7 Locate the **Port Incident Mode Settings** section. From the **Define incident wave** list, choose **Amplitude (pressure)**.

8 In the A_p^{in} text field, type `p0`.

Impedance - Brambley

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

2 Select Boundary 20 only.

3 In the **Settings** window for **Impedance**, type `Impedance - Brambley` in the **Label** text field.

4 Locate the **Impedance** section. From the **Impedance model** list, choose **Brambley (finite boundary layer)**.

5 In the Z_n text field, type `lpff.rho0*lpff.c0*Zrel`.

6 In the δ text field, type `delta`.

Use the Brambley (finite boundary layer) option. It takes the specific impedance and the thickness of the turbulent boundary layer of the background flow as inputs.

MESH 1

Distribution 1

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

2 Select Boundaries 3, 14, and 22 only.

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

4 In the **Number of elements** text field, type `12`.

Size

1 In the **Model Builder** window, 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 `0.01`.

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 Domains 1 and 3–5 only.

5 Click **Build Selected**.

Mapped 1

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

Boundary Layers 1

In the **Mesh** toolbar, click **Boundary Layers**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundaries 11–13 and 20–22 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

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

5 Click **Build All**.

STUDY 1 - LPFF - HARD WALL

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

2 In the **Settings** window for **Study**, type `Study 1 - LPFF - Hard Wall` in the **Label** text field.

Step 2: Frequency Domain

1 In the **Study** toolbar, click **Frequency Domain**.

Disable the impedance boundary condition in the study to model a sound hard wall.

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

3 In the **Frequencies** text field, type `f0`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

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

6 Right-click and choose **Disable**.

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

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

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

STUDY 2 - LPFF - BRAMBLEY

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

2 In the **Study** toolbar, click **Frequency Domain**.

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

2 In the **Frequencies** text field, type `f0`.

3 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
delta (Turbulent boundary layer thickness)		m

6 In the table, click to select the cell at row number 1 and column number 2.

7 Click **Range**.

8 In the **Range** dialog, type `0` in the **Start** text field.

9 In the **Step** text field, type `0.2[mm]`.

10 In the **Stop** text field, type `4[mm]`.

11 Click **Replace**.

ADD COMPONENT

In the **Model Builder** window, right-click **Study 2 - LPFF - Brambley** and choose **2D Axisymmetric**.

GEOMETRY 2

Import 1 (imp1)

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

In Component 2, set up the same model with the physics interfaces Turbulent Flow and Linearized Navier–Stokes.

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

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Geometry 1**.

5 Click **Import**.

6 Click **Build All Objects**.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Air**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

ADD PHYSICS

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

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

3 In the tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, SST (spf)**.

4 Click the **Add to Component 2** button in the window toolbar.

5 In the tree, select **Acoustics** > **Aeroacoustics** > **Linearized Navier–Stokes, Frequency Domain (lnsf)**.

6 Click the **Add to Component 2** button in the window toolbar.

7 In the tree, select **Acoustics** > **Aeroacoustics** > **Linearized Navier–Stokes, Boundary Mode (lnsbm)**.

8 Click the **Add to Component 2** button in the window toolbar.

9 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

LINEARIZED NAVIER–STOKES, BOUNDARY MODE (LNSBM)

Add a Linearized Navier–Stokes, Boundary Mode interface, to model the correct modes to excite.

1 In the **Settings** window for **Linearized Navier–Stokes, Boundary Mode**, locate the **Boundary Selection** section.

2 Click **Clear Selection**.

3 Select Boundary 9 only.

TURBULENT FLOW, SST (SPF)

Inlet 1

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

2 Select Boundaries 16 and 17 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

Outlet 1

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

2 Select Boundary 5 only.

3 In the **Settings** window for **Outlet**, locate the **Boundary Condition** section.

4 From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. Click the **Average velocity** button.

6 In the U_{av} text field, type `c0*Ma`.

MULTIPHYSICS

Background Fluid Flow Coupling 1 (bffc1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain > Background Fluid Flow Coupling**.

2 In the **Settings** window for **Background Fluid Flow Coupling**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Variables to Map** section. Select the **Map the turbulent viscosity** checkbox.

DEFINITIONS (COMP2)

Perfectly Matched Layer 2 (pml2)

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

2 Select Domains 2 and 6 only.

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

4 From the **Type** list, choose **User defined**.

5 In the table, enter the following settings:

	Distance function (m)	Thickness (m)
Direction 1	`sqrt(r^2+z^2)-1.6*zi`	`0.2*zi`

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

7 In the **Typical wavelength** text field, type `lam0`.

8 In the **PML scaling curvature parameter** text field, type `3`.

ARTIFICIAL DOMAINS

Perfectly Matched Layer 3 (pml3)

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 **User defined**.

5 In the table, enter the following settings:

	Distance function (m)	Thickness (m)
Direction 1	`-z-(5/3-1/5)*zi`	`0.2*zi`

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

7 In the **Typical wavelength** text field, type `lam0`.

8 In the **PML scaling curvature parameter** text field, type `3`.

LINEARIZED NAVIER–STOKES, FREQUENCY DOMAIN (LNSF)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Linearized Navier–Stokes, Frequency Domain (lnsf)**.

2 In the **Settings** window for **Linearized Navier–Stokes, Frequency Domain**, locate the **Linearized Navier–Stokes Equation Settings** section.

3 Select the **Out-of-plane mode extension** checkbox.

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

Linearized Navier–Stokes Model 1

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Linearized Navier–Stokes, Frequency Domain (lnsf)** click **Linearized Navier–Stokes Model 1**.

2 In the **Settings** window for **Linearized Navier–Stokes Model**, locate the **Model Input** section.

3 In the T_0 text field, type `T0`.

Wall 1

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

2 In the **Settings** window for **Wall**, locate the **Mechanical** section.

3 From the **Mechanical condition** list, choose **Slip (perfect)**.

4 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

Impedance 1

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

2 Select Boundary 20 only.

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

4 In the Z_n text field, type `lnsf.rho0*lnsf.c0*Zrel`.

Background Acoustic Fields 1

1 In the **Physics** toolbar, click **Domains** and choose **Background Acoustic Fields**.

Use one of the propagating modes found with the boundary mode analysis.

2 Select Domain 4 only.

3 In the **Settings** window for **Background Acoustic Fields**, locate the **Background Acoustic Fields** section.

4 In the p_b text field, type `withsol('sol8',genext1(p3)/maxop1(p3)*exp(-i*lnsbm.kn*z),setind(lambda,n+1))*p0`.

5 Specify the $\mathbf{u}_b$ vector as

`withsol('sol8',genext1(u3)/maxop1(p3)*exp(-i*lnsbm.kn*z),setind(lambda,n+1))*p0`	r
`withsol('sol8',genext1(v3)/maxop1(p3)*exp(-i*lnsbm.kn*z),setind(lambda,n+1))*p0`	phi
`withsol('sol8',genext1(w3)/maxop1(p3)*exp(-i*lnsbm.kn*z),setind(lambda,n+1))*p0`	z

6 In the T_b text field, type `withsol('sol8',genext1(T2)/maxop1(p3)*exp(-i*lnsbm.kn*z),setind(lambda,n+1))*p0`.

DEFINITIONS (COMP2)

General Extrusion 1 (genext1)

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

2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.

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

4 Select Boundary 9 only.

5 Locate the **Destination Map** section. Clear the **z-expression** text field.

6 Locate the **Source** section. Select the **Use source map** checkbox.

7 Clear the **z^i-expression** text field.

Maximum 1 (maxop1)

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

2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.

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

4 Select Boundary 9 only.

LINEARIZED NAVIER–STOKES, BOUNDARY MODE (LNSBM)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Linearized Navier–Stokes, Boundary Mode (lnsbm)**.

2 In the **Settings** window for **Linearized Navier–Stokes, Boundary Mode**, locate the **Linearized Navier–Stokes Equation Settings** section.

3 Select the **Out-of-plane mode extension** checkbox.

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

STUDY 1 - LPFF - HARD WALL

In the **Model Builder** window, collapse the **Study 1 - LPFF - Hard Wall** node.

STUDY 2 - LPFF - BRAMBLEY

In the **Model Builder** window, collapse the **Study 2 - LPFF - Brambley** node.

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

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

STUDY 3 - CFD

In the **Settings** window for **Study**, type `Study 3 - CFD` in the **Label** text field.

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 In the table, clear the **Use** checkboxes for **Linearized Navier–Stokes, Frequency Domain (lnsf)**, **Linearized Navier–Stokes, Boundary Mode (lnsbm)**, and **Background Fluid Flow Coupling 1 (bffc1)**.

4 Click **Build All**.

5 Click to collapse the **Sequence Type** section. Click to collapse the **Physics-Controlled Mesh** section. Click to expand the **Sequence Type** section. Click to expand the **Physics-Controlled Mesh** section. Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Mesh 2** click **Free Triangular 1**.

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

Boundary Layers 1

1 In the **Model Builder** window, click **Boundary Layers 1**.

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

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 Select Boundaries 4, 6, 8, 11–13, and 19–22 only.

3 In the **Settings** window for **Boundary Layer Properties**, click **Build All**.

4 In the **Model Builder** window, click **Mesh 2**.

5 In the **Settings** window for **Mesh**, type `CFD Mesh` in the **Label** text field.

ACO MESH

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

2 In the **Settings** window for **Mesh**, type `Aco Mesh` in the **Label** text field.

Distribution 1

1 Right-click **Aco Mesh** and choose **Distribution**.

2 Select Boundaries 3, 4, 11, 14, and 22 only.

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

4 In the **Number of elements** text field, type `16`.

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 Domains 1, 4, and 5 only.

Mapped 1

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

Size

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

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

3 In the **Maximum element size** text field, type `0.005`.

4 Locate the **Element Size** section. Click the **Predefined** button.

5 From the **Predefined** list, choose **Extremely fine**.

6 Click **Build All**.

7 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.01`.

8 Click **Build All**.

STUDY 1 - LPFF - HARD WALL

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1 - LPFF - Hard Wall** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, clear the checkbox for **Component 2 (comp2)**.

Step 2: Frequency Domain

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

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 2 (comp2)**.

4 Right-click and choose **Disable in Solvers**.

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

STUDY 2 - LPFF - BRAMBLEY

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 2 - LPFF - Brambley** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, clear the checkbox for **Component 2 (comp2)**.

Step 2: Frequency Domain

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

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, clear the checkbox for **Component 2 (comp2)**.

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

STUDY 3 - CFD

Step 1: Wall Distance Initialization

1 In the **Model Builder** window, under **Study 3 - CFD** click **Step 1: Wall Distance Initialization**.

2 In the **Settings** window for **Wall Distance Initialization**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 2 (comp2)** > **Definitions** > **Artificial Domains** > **Perfectly Matched Layer 2 (pml2)**.

5 Right-click and choose **Disable**.

6 In the tree, select **Component 2 (comp2)** > **Definitions** > **Artificial Domains** > **Perfectly Matched Layer 3 (pml3)**.

7 Right-click and choose **Disable**.

Step 2: Stationary

1 In the **Model Builder** window, click **Step 2: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

4 Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 2 (comp2)** > **Definitions** > **Artificial Domains** > **Perfectly Matched Layer 2 (pml2)**.

6 Right-click and choose **Disable**.

7 In the tree, select **Component 2 (comp2)** > **Definitions** > **Artificial Domains** > **Perfectly Matched Layer 3 (pml3)**.

8 Right-click and choose **Disable**.

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

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Mapping**.

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

STUDY 4

Step 1: Mapping

1 In the **Settings** window for **Mapping**, locate the **Data to Map** section.

2 From the **Source** list, choose **Study 3 - CFD, Stationary**.

3 Click to expand the **Destination Mesh Selection** section. In the **Model Builder** window, click **Study 4**.

4 In the **Settings** window for **Study**, type `Study 4 - Mapping` in the **Label** text field.

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

LINEARIZED NAVIER–STOKES, BOUNDARY MODE (LNSBM)

Linearized Navier–Stokes Model 1

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Linearized Navier–Stokes, Boundary Mode (lnsbm)** click **Linearized Navier–Stokes Model 1**.

2 In the **Settings** window for **Linearized Navier–Stokes Model**, locate the **Model Input** section.

3 From the $\mathbf{u}_0$ list, choose **Mapped velocity (bffc1)**.

4 From the p_0 list, choose **Mapped pressure (bffc1)**.

5 In the T_0 text field, type `T0`.

Wall 1

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

2 In the **Settings** window for **Wall**, locate the **Mechanical** section.

3 From the **Mechanical condition** list, choose **Slip (perfect)**.

4 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Air**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air 1 (mat3)

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

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

3 Select Boundary 9 only.

ADD STUDY

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

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

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

STUDY 5

Step 1: Mode Analysis

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

2 In the **Mode analysis frequency** text field, type `f0`.

3 From the **Search method around shift** list, choose **Larger real part**.

4 Select the **Search for modes around shift** checkbox.

5 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**.

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

7 From the **Study** list, choose **Study 4 - Mapping, Mapping**.

8 Click to expand the **Filtering and Sorting** section. Find the **Sorting** subsection. From the **Ordering** list, choose **Descending**.

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

10 In the **Settings** window for **Study**, type `Study 5 - Boundary Mode` in the **Label** text field.

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

ADD STUDY

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

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

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

STUDY 6

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `f0`.

3 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

4 Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 2 (comp2) > Linearized Navier–Stokes, Frequency Domain (lnsf) > Impedance 1**.

6 Right-click and choose **Disable**.

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

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

9 From the **Study** list, choose **Study 4 - Mapping, Mapping**.

10 In the **Model Builder** window, click **Study 6**.

11 In the **Settings** window for **Study**, type `Study 6 - LNS - Hard Wall` in the **Label** text field.

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

ADD STUDY

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

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

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

STUDY 7 - LNS - IMPEDANCE

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

2 In the **Frequencies** text field, type `f0`.

3 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

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 4 - Mapping, Mapping**.

7 In the **Model Builder** window, click **Study 7**.

8 In the **Settings** window for **Study**, type `Study 7 - LNS - Impedance` in the **Label** text field.

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

RESULTS

Acoustic Pressure (lpff), Acoustic Pressure (lpff) 1, Acoustic Pressure, 3D (lpff), Acoustic Pressure, 3D (lpff) 1, Mean Flow Velocity (cpf), Mean Flow Velocity (cpf) 1, Mean Flow Velocity, 3D (cpf), Mean Flow Velocity, 3D (cpf) 1, Sound Pressure Level (lpff), Sound Pressure Level (lpff) 1, Sound Pressure Level, 3D (lpff), Sound Pressure Level, 3D (lpff) 1
Right-click and choose **Group**.

Linearized Potential Flow
In the **Settings** window for **Group**, type `Linearized Potential Flow` in the **Label** text field.

Pressure (spf), Velocity (spf), Velocity, 3D (spf), Wall Resolution (spf)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf)**, **Pressure (spf)**, **Velocity, 3D (spf)**, and **Wall Resolution (spf)**.
2 Right-click and choose **Group**.

CFD

In the **Settings** window for **Group**, type `CFD` in the **Label** text field.

Acoustic Pressure (lnsbm), Acoustic Velocity (lnsbm), Background Mean Flow (lnsbm), Temperature Variation (lnsbm)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (lnsbm)**, **Acoustic Velocity (lnsbm)**, **Temperature Variation (lnsbm)**, and **Background Mean Flow (lnsbm)**.
2 Right-click and choose **Group**.

Boundary Mode Analysis

In the **Settings** window for **Group**, type `Boundary Mode Analysis` in the **Label** text field.

Acoustic Pressure (lnsf), Acoustic Pressure (lnsf) 1, Acoustic Velocity (lnsf), Acoustic Velocity (lnsf) 1, Background Mean Flow (lnsf), Background Mean Flow (lnsf) 1, Temperature Variation (lnsf), Temperature Variation (lnsf) 1

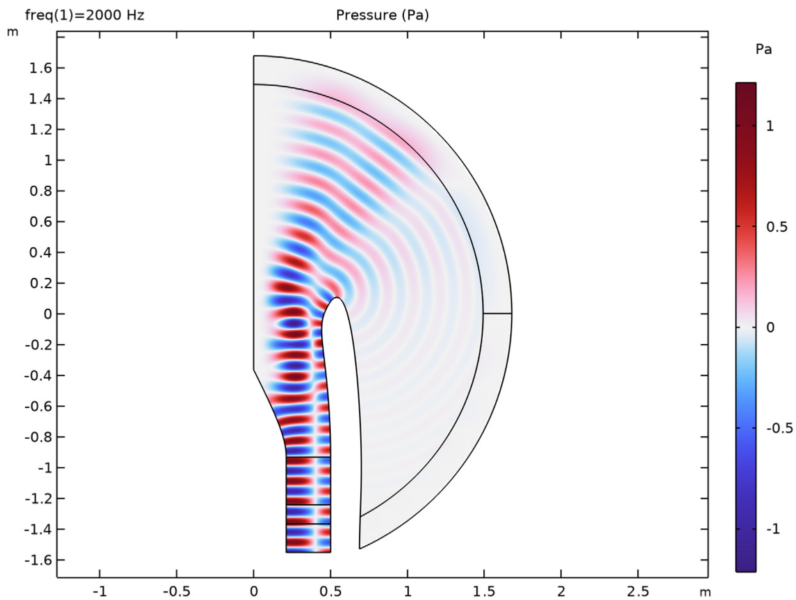
1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (lnsf)**, **Acoustic Velocity (lnsf)**, **Temperature Variation (lnsf)**, **Background Mean Flow (lnsf)**, **Acoustic Pressure (lnsf) 1**, **Acoustic Velocity (lnsf) 1**, **Temperature Variation (lnsf) 1**, and **Background Mean Flow (lnsf) 1**.
2 Right-click and choose **Group**.

Linearized Navier Stokes

In the **Settings** window for **Group**, type `Linearized Navier Stokes` in the **Label** text field.

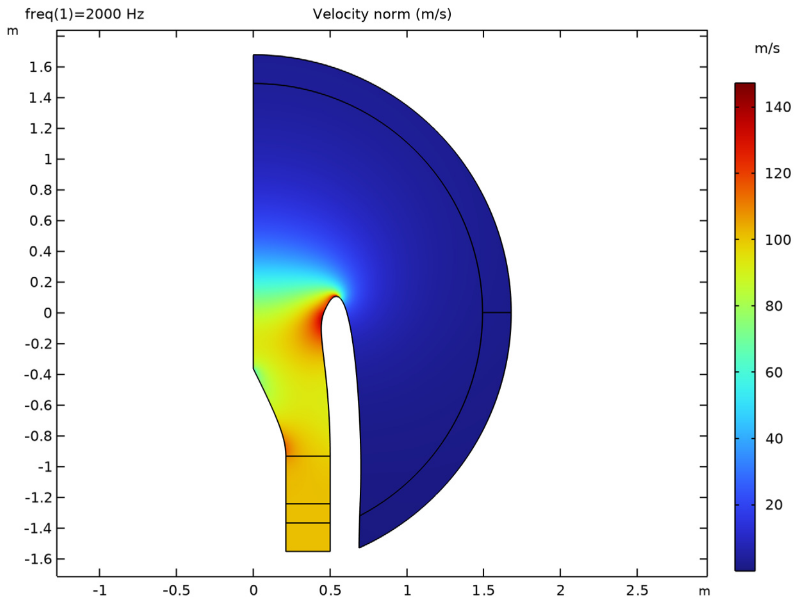
Acoustic Pressure (lpff)

In the **Model Builder** window, under **Results** > **Linearized Potential Flow** click **Acoustic Pressure (lpff)**.



Mean Flow Velocity (cpf)

In the **Model Builder** window, click **Mean Flow Velocity (cpf)**.



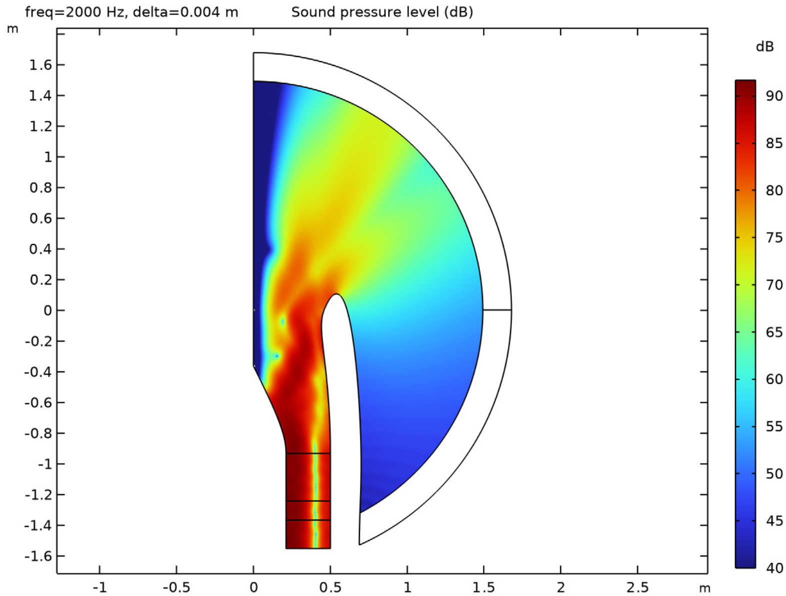
Surface

1. In the **Model Builder** window, expand the **Sound Pressure Level (lpff) 1** node, then click **Surface**.
2. In the **Settings** window for **Surface**, click to expand the **Range** section.
3. Select the **Manual color range** checkbox.
4. In the **Minimum** text field, type 40.

Sound Pressure Level (lpff) 1

1. In the **Model Builder** window, click **Sound Pressure Level (lpff) 1**.
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 Domains 1 and 3–5 only.

5 In the **Sound Pressure Level (lpff) 1** toolbar, click **Plot**.



Distance function - PML

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

2 In the **Settings** window for **2D Plot Group**, type `Distance function - PML` in the **Label** text field.

3 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

Surface 1

1 Right-click **Distance function - PML** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Definitions > Perfectly Matched Layer 1 > pml1.dDist - Distance function - m**.

Distance function - PML

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

Surface 2

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

2 From the **Dataset** list, choose **Study 6 - LNS - Hard Wall/Solution 9 (18) (sol9)**.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)** > **Definitions** > **Perfectly Matched Layer 2** > **pml2.dDist - Distance function - m**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Distance function - PML

Right-click **Results** > **Distance function - PML** > **Surface 2** and choose **Surface**.

Surface 3

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

2 From the **Dataset** list, choose **Study 6 - LNS - Hard Wall/Solution 9 (18) (sol9)**.

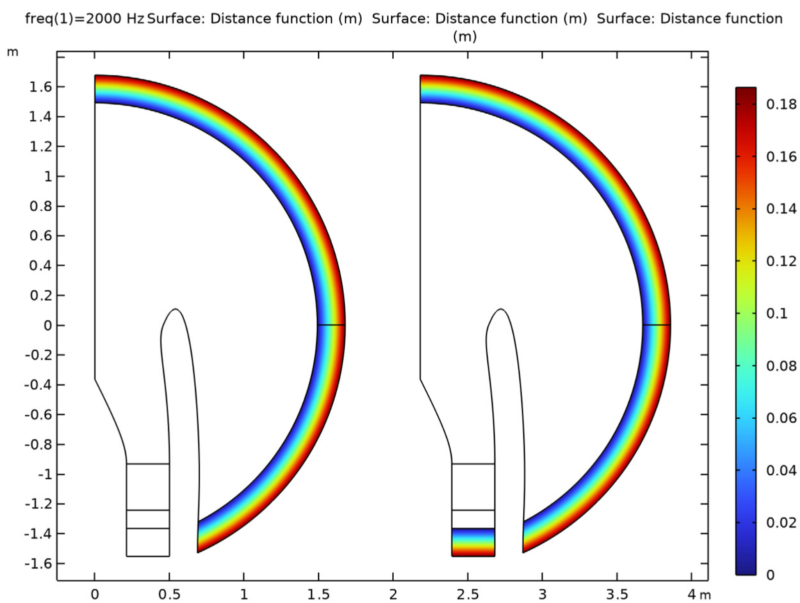
3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)** > **Definitions** > **Perfectly Matched Layer 3** > **pml3.dDist - Distance function - m**.

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

5 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

6 In the **Index** text field, type 1.

7 In the **Distance function - PML** toolbar, click **Plot**.



Sound Pressure Level

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Sound Pressure Level` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.
4 Locate the **Plot Settings** section.
5 Select the **x-axis label** checkbox. In the associated text field, type `Angle (\deg)`.
6 Select the **y-axis label** checkbox. In the associated text field, type `Sound pressure level (dB)`.
7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.
8 In the **y minimum** text field, type `30`.
9 In the **y maximum** text field, type `80`.
10 In the **x minimum** text field, type `0`.
11 In the **x maximum** text field, type `154`.

LPFF - Hard Wall

1 Right-click **Sound Pressure Level** and choose **Line Graph**.
2 In the **Settings** window for **Line Graph**, type `LPFF - Hard Wall` in the **Label** text field.
3 Select Boundaries 15 and 18 only.
4 Locate the **y-Axis Data** section. In the **Expression** text field, type `up(lpff.Lp)`.
5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6 In the **Expression** text field, type `atan2(r,z)`.
7 From the **Unit** list, choose **°**.
8 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Blue**.
9 Click to expand the **Legends** section. Select the **Show legends** checkbox.
10 Find the **Include** subsection. Select the **Label** checkbox.
11 Clear the **Solution** checkbox.

LPFF - Brambley

1 Right-click **LPFF - Hard Wall** and choose **Duplicate**.
2 In the **Settings** window for **Line Graph**, type `LPFF - Brambley` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - LPFF - Brambley/ Solution 3 (5) (sol3)**.
4 From the **Parameter selection (delta)** list, choose **From list**.

5 In the **Parameter values (delta (m))** list box, select **0.0022**.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

LNSF - Hard Wall

1 Right-click **LPFF - Brambley** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, type `LNSF - Hard Wall` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 6 - LNS - Hard Wall/Solution 9 (18) (sol9)**.

4 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

5 Select Boundaries 15 and 18 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `up(comp2.lnsf.Lp_t)`.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

8 Find the **Line style** subsection. From the **Line** list, choose **Solid**.

LNSF - Impedance

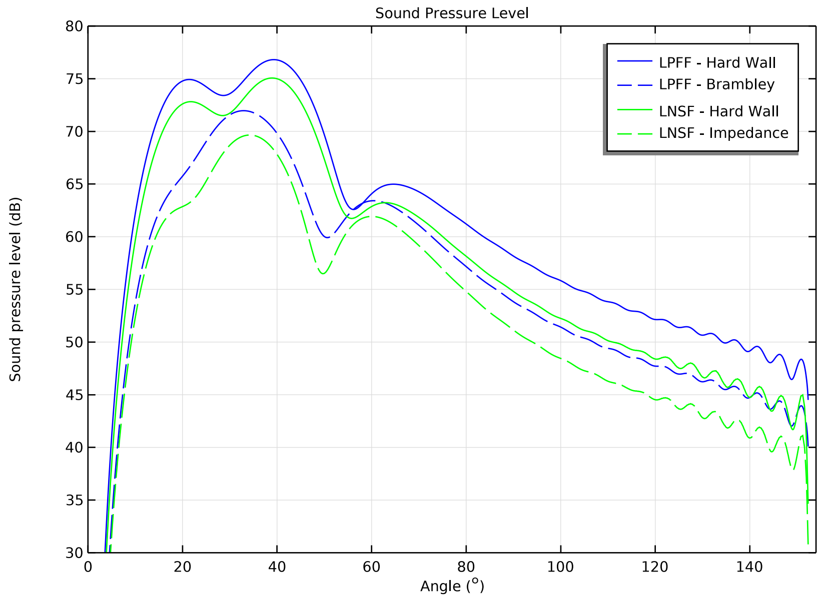
1 Right-click **LNSF - Hard Wall** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, type `LNSF - Impedance` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 7 - LNS - Impedance/Solution 10 (20) (sol10)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 In the **Sound Pressure Level** toolbar, click **Plot**.



Liner Insertion Loss (dB)

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

2 In the **Settings** window for **1D Plot Group**, type `Liner Insertion Loss (dB)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - LPFF - Brambley/Solution 3 (5) (sol3)**.

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

5 In the **Parameter values (delta (m))** list box, select **0.0022**.

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

7 Locate the **Plot Settings** section.

8 Select the **x-axis label** checkbox. In the associated text field, type `Angle (\deg)`.

9 Select the **y-axis label** checkbox. In the associated text field, type `Insertion loss (dB)`.

LPFF - Brambley

1 Right-click **Liner Insertion Loss (dB)** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `LPFF - Brambley` in the **Label** text field.

3 Select Boundaries 15 and 18 only.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `withsol('sol1', up(lpff.Lp))-up(lpff.Lp)`.

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

6 In the **Expression** text field, type `atan2(r,z)`.

7 From the **Unit** list, choose **°**.

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

9 Find the **Include** subsection. Select the **Label** checkbox.

10 Clear the **Solution** checkbox.

LNSF - Impedance

1 Right-click **LPFF - Brambley** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, type `LNSF - Impedance` in the **Label** text field.

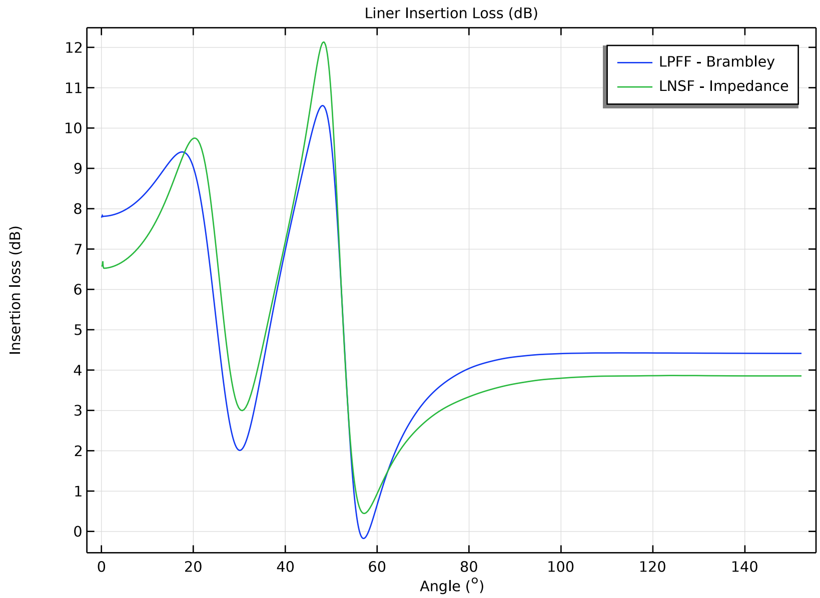
3 Locate the **Data** section. From the **Dataset** list, choose **Study 7 - LNS - Impedance/ Solution 10 (20) (sol10)**.

4 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

5 Select Boundaries 15 and 18 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `withsol('sol9', up(comp2.lnsf.Lp_t))-up(comp2.lnsf.Lp_t)`.

7 In the **Liner Insertion Loss (dB)** toolbar, click **Plot**.



Background Flow

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

2 In the **Settings** window for **1D Plot Group**, type `Background Flow` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Radius (r)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Background flow (m/s)`.

Compressible Flow

1 Right-click **Background Flow** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Compressible Flow` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - LPFF - Hard Wall/Solution 1 (1) (sol1)**.

4 Select Boundary 10 only.

5. Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Compressible Potential Flow** > **cpf.normV - Velocity norm - m/s**.
6. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
7. In the **Expression** text field, type `r`.
8. Locate the **Legends** section. Select the **Show legends** checkbox.
9. Find the **Include** subsection. Select the **Label** checkbox.
10. Clear the **Solution** checkbox.

Turbulent Flow - CFD Mesh

1. Right-click **Compressible Flow** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 3 - CFD/Solution 5 (10) (sol5)**.
4. Locate the **Selection** section. Click to select the **Activate Selection** toggle button.
5. Select Boundary 10 only.
6. Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)** > **Turbulent Flow, SST** > **Velocity and pressure** > **spf.U - Velocity magnitude - m/s**.
7. In the **Label** text field, type `Turbulent Flow - CFD Mesh`.

Turbulent Flow - Aco Mesh

1. Right-click **Turbulent Flow - CFD Mesh** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 4 - Mapping/Solution 7 (14) (sol7)**.
4. Locate the **Selection** section. Click to select the **Activate Selection** toggle button.
5. Select Boundary 10 only.
6. In the **Label** text field, type `Turbulent Flow - Aco Mesh`.

7 In the **Background Flow** toolbar, click **Plot**.