Model created in COMSOL Multiphysics 6.4

Helmholtz Resonator with Flow: Imported Fluid Flow from CGNS Data

Introduction

Helmholtz resonators are used in exhaust systems, as they can attenuate a specific narrow frequency band. The presence of a flow in the system alters the acoustic properties of the resonator and the transmission loss of the subsystem. In this tutorial model, a Helmholtz resonator is located as a side branch to a main duct. The transmission loss through the main duct is investigated when a flow is introduced.

This model is an extension of the Helmholtz Resonator with Flow: Interaction of Flow and Acoustics tutorial. In this version, the mean flow is imported and mapped using the **Imported Fluid Flow** interface and the **CFD Data (CGNS)** function. The model is set up for the case where the Mach number is equal 0.1. The acoustics problem is then solved using the **Linearized Navier–Stokes, Frequency Domain** interface.

Note: Evaluation of CGNS data in COMSOL is only supported on Windows.

The CGNS data imported in this model is courtesy of Resolvent Denmark PS (resolvent.com).

Model Definition

The model setup and flow conditions follow the definitions in the Helmholtz Resonator with Flow: Interaction of Flow and Acoustics tutorial. In the present model, only the case where the Mach number is equal to 0.1 is studied.

In this model, the flow is not solved for; instead, it is imported using the **Imported Fluid Flow** interface and the **CFD Data (CGNS)** function. The workflow is integrated with the **Background Fluid Flow Coupling** multiphysics coupling and the dedicated **Mapping** study.

Results and Discussion

The mapped background pressure and the magnitude of the background velocity is depicted in Figure 1. A comparison between the mapped z-velocity and the values of the CGNS function data, in a cross section through the main pipe, is depicted in Figure 2. Notice that the mapped data is not completely identical to the function values. There are different ways to control the mapping in the **Imported Fluid Flow** interface, both in terms of the amount of smoothing but also with regards to constraints that can be set on the mapping.

The acoustic pressure resulting from solving the linearized Navier–Stokes equations is depicted in Figure 3. Finally, the transmission loss of the system is depicted in Figure 4. The values can be compared to the results in the Helmholtz Resonator with Flow: Interaction of Flow and Acoustics tutorial.

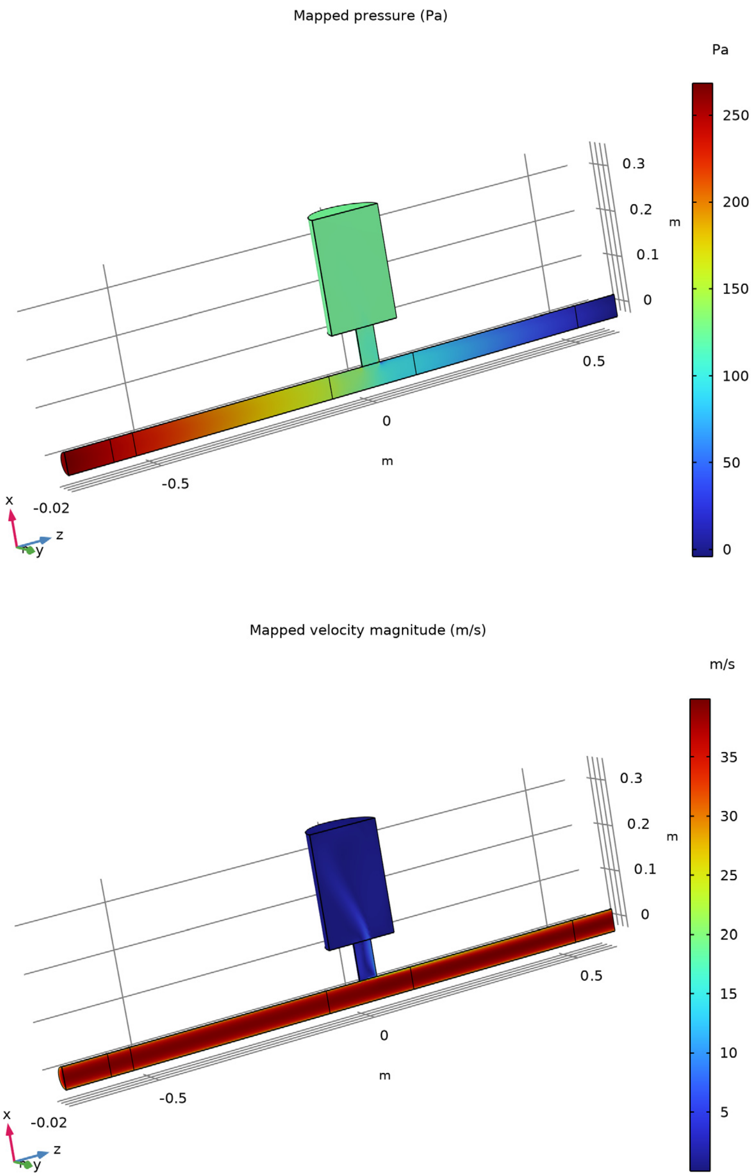


Figure 1: The mapped flow pressure (top) and velocity (bottom).

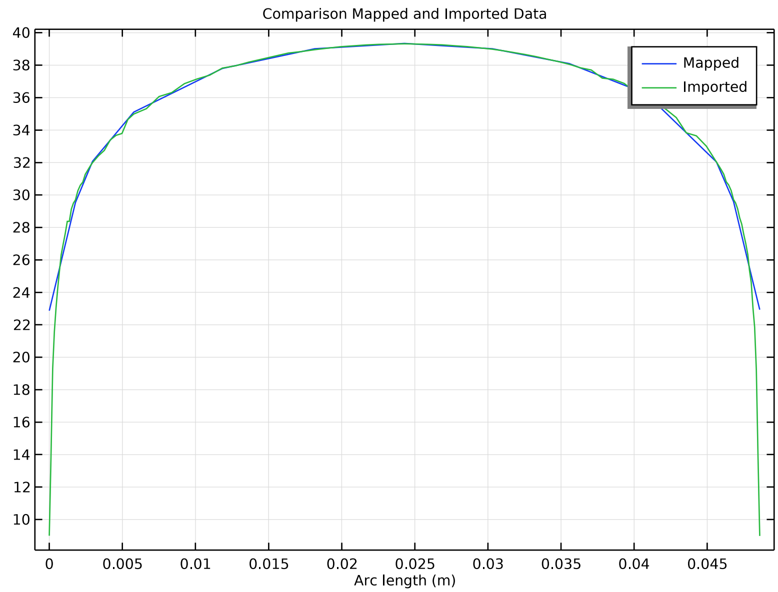


Figure 2: Comparison of the mapped z-velocity and the values of the CGNS function evaluated at the same points.

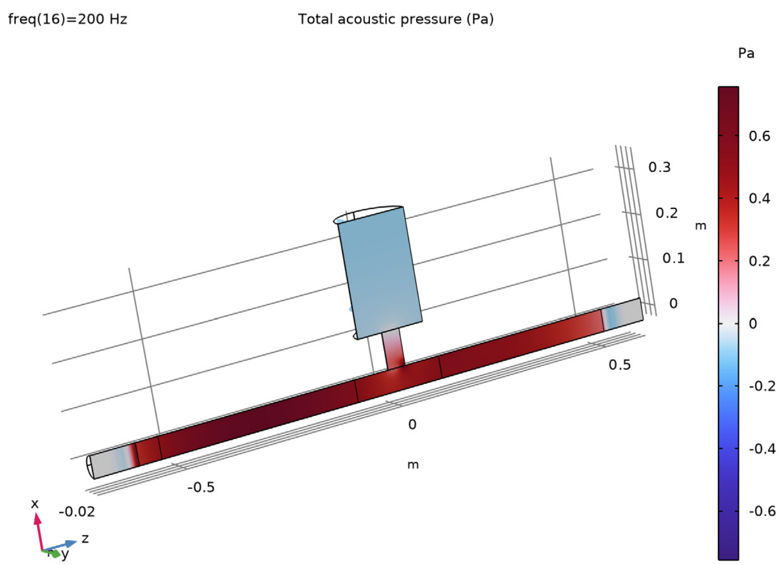


Figure 3: Acoustic pressure distribution at 200 Hz.

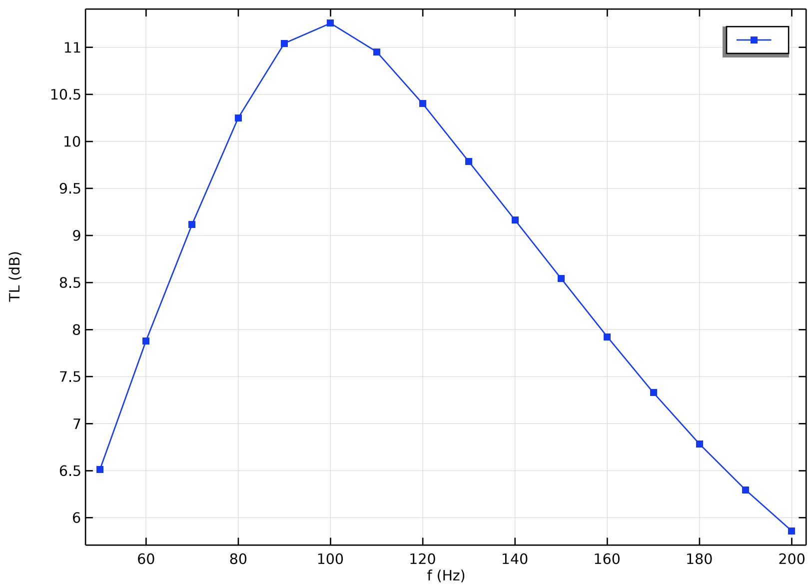


Figure 4: Transmission loss TL of the resonator system.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resonator_with_flow_cgns`

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 **3D**.
2 In the **Select Physics** tree, select **Acoustics** > **Aeroacoustics** > **Imported Fluid Flow (iff)**.
3 Click **Add**.

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

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

The model setup initially follows the setup in the `helmholtz_resonator_with_flow` tutorial.

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 `helmholtz_resonator_with_flow_cgns_parameters.txt`.

GEOMETRY 1

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `Dmain/2`.

4 In the **Height** text field, type `Lin+Lout+2*Lpml`.

5 Locate the **Position** section. In the **z** text field, type `-Lin-Lpml`.

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

Layer name	Thickness (m)
Layer 1	Lpml

7 Clear the **Layers on side** checkbox.

8 Select the **Layers on bottom** checkbox.

9 Select the **Layers on top** checkbox.

Cylinder 2 (cyl2)

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

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

3 In the **Radius** text field, type `Dneck/2`.

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

5 Locate the **Position** section. In the **x** text field, type `Dmain/2-0.2*Lneck`.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 3 (cyl3)

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

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

3 In the **Radius** text field, type `Dreson/2`.

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

5 Locate the **Position** section. In the **x** text field, type `Dmain/2+Lneck`.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 4 (cyl4)

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

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

3 In the **Radius** text field, type `Dmain/2`.

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

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

Cylinder 5 (cyl5)

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

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

3 In the **Radius** text field, type `Dmain/2`.

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

5 Locate the **Position** section. In the **z** text field, type `-0.1`.

6 Click **Build Selected**.

Partition Domains 1 (pard1)

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

2 On the object **cyl2**, select Domain 1 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 From the **Partition with** list, choose **Extended faces**.

5 On the object **cyl1**, select Boundaries 12 and 15 only.

6 From the **Repair tolerance** list, choose **Relative**.

7 Click **Build All Objects**.

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 **pard1**, select Domain 1 only.

5 Click **Build Selected**.

Union 1 (uni1)

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

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

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Partition Objects 1 (par1)

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

2 Select the object **uni1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 Click **Build Selected**.

Delete Entities 2 (del2)

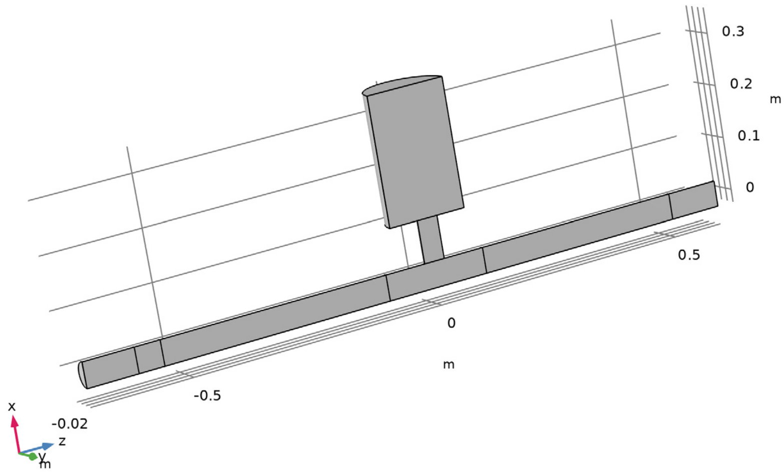
1 Right-click **Geometry 1** and choose **Delete Entities**.

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 **par1**, select Domains 2, 4, 6, 8, 10, 12, 14, and 16 only.

5 Click **Build All Objects**.



Rotate the geometry in the **Graphics** window. The geometry should look like the figure above.

DEFINITIONS

Symmetry

1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 2, 5, 8, 11, 14, 17, 29, and 34 only.

5 In the **Label** text field, type `Symmetry`.

Walls

1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 1, 4, 7, 10, 13, 16, 20–25, 27, 28, 30–32, and 35 only.

5 In the **Label** text field, type `Walls`.

Integration 1 (intop1)

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

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

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

4 Select Boundary 6 only.

5 In the **Operator name** text field, type `intop_in`.

Integration 2 (intop2)

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

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

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

4 Select Boundary 18 only.

5 In the **Operator name** text field, type `intop_out`.

Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 1 and 6 only.

Variables - Incident Plane Wave

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

2 In the **Settings** window for **Variables**, type `Variables - Incident Plane Wave` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

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.

Now, proceed and set up the **Imported Fluid Flow** physics. The physics refer to a **CFD Data (CGNS)** function, found in **Global Definitions**, that links to the external `.cgns` data. The dedicated interface and **Mapping** study ensures that the external data is mapped consistently to the COMSOL mesh and variables. This is essential to get accurate solutions as well as an efficient solving procedure.

GLOBAL DEFINITIONS

CFD Data (CGNS) 1

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **CFD Data (CGNS)**.

2 In the **Settings** window for **CFD Data (CGNS)**, locate the **Functions** section.

3 Click **Browse**.

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

Add units for the imported flow quantities.

5 In the table, enter the following settings:

Function name	Arguments	Function unit
Total_velocity	x,y,z,t	m/s
x_velocity	x,y,z,t	m/s
y_velocity	x,y,z,t	m/s
z_velocity	x,y,z,t	m/s
Turbulent_viscosity	x,y,z,t	Pa*s
Pressure	x,y,z,t	Pa
Density	x,y,z,t	kg/m^3

IMPORTED FLUID FLOW (IFF)

Check that the discretization of the imported flow is the same as the aeroacoustics model. For linearized Navier-Stokes linear discretization is the default. You can also inspect the variable names given to the imported quantities.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Imported Fluid Flow (iff)**.

2 In the **Settings** window for **Imported Fluid Flow**, click to expand the **Discretization** section.

3 Click to expand the **Dependent Variables** section.

Flow Import 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Imported Fluid Flow (iff)** click **Flow Import 1**.

2 In the **Settings** window for **Flow Import**, locate the **Variables to Map** section.

3 From the **Imported flow data** list, choose **CFD Data (CGNS) 1**.
4 From the p list, choose **Pressure**.
5 From the ρ list, choose **Density**.
6 From the **u** list, choose **Map components**.
7 From the u list, choose **x_velocity**.
8 From the v list, choose **y_velocity**.
9 From the w list, choose **z_velocity**.
10 From the μ list, choose **Turbulent_viscosity**.

Symmetry 1

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

The symmetry condition ensures that the mapped solution is symmetric, in particular that no numerical pollution create flow through the symmetry plane.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Symmetry**.

Proceed and add the **Background Fluid Flow** multiphysics coupling, it couples the imported flow to the acoustics.

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**.

LINEARIZED NAVIER–STOKES, FREQUENCY DOMAIN (LNSF)

Background Acoustic Fields 1

1 In the **Physics** toolbar, click **Domains** and choose **Background Acoustic Fields**.
2 Select Domain 2 only.
3 In the **Settings** window for **Background Acoustic Fields**, locate the **Background Acoustic Fields** section.
4 In the p_b text field, type `pb`.

5 Specify the $\mathbf{u}_\mathrm{b}$ vector as

ub	x
vb	y
wb	z

6 In the T_b text field, type `Tb`.

Symmetry 1

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

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

3 From the **Selection** list, choose **Symmetry**.

Now, create the acoustic mesh, following the same procedure as in the `helmholtz_resonator_with_flow` tutorial. It is not necessary to have a CFD mesh in this case as the imported data is mapped directly to the acoustic mesh.

MESH 1

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

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

4 Select Domains 2–5, 7, and 8 only.

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 `Dmain/8`.

5 In the **Minimum element size** text field, type `Dmain/15`.

Size 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

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

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

4 Select Boundary 26 only.

5 Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `Dmain/15`.

Swept 1

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

Distribution 1

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

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

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

Boundary Layers 1

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

Boundary Layer Properties

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

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

3 From the **Selection** list, choose **Walls**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `3`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `Dmain/60`.

7 Click **Build All**.

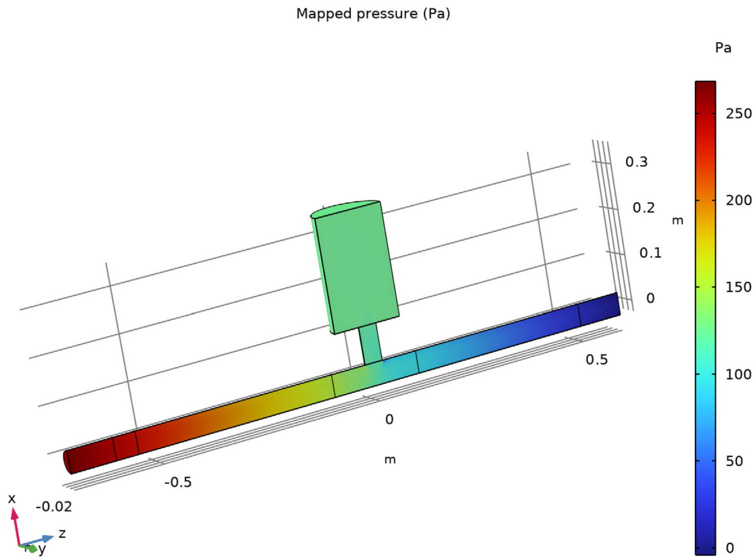
STUDY 1

Step 1: Mapping

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

RESULTS

Mapped Pressure (iff)



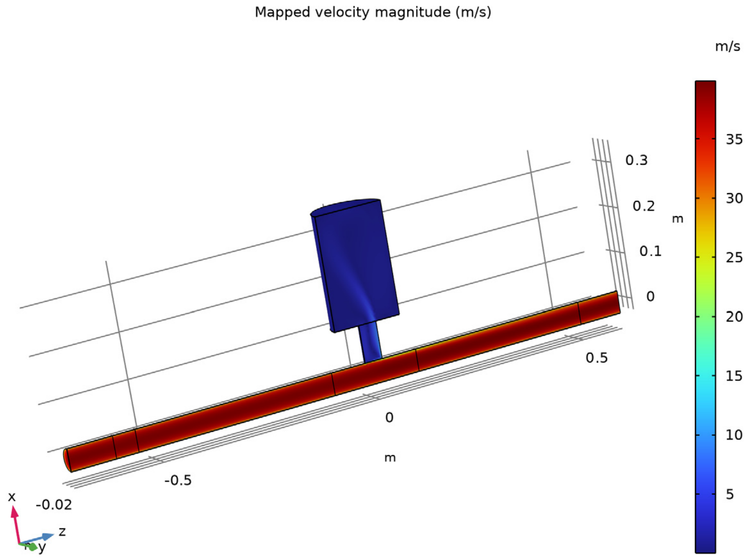
Mapped Velocity Magnitude (iff)

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Mapped Velocity Magnitude (iff)` in the **Label** text field.
3 Locate the **Color Legend** section. Select the **Show units** checkbox.

Surface 1

1 Right-click **Mapped Velocity Magnitude (iff)** 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)** > **Imported Fluid Flow** > **iff.U_map - Mapped velocity magnitude - m/s**.

3 In the **Mapped Velocity Magnitude (iff)** toolbar, click **Plot**.



ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Imported Fluid Flow (iff)**.

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

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

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

STUDY 2

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `range(50,10,200)`.

Make sure to point to the solution of the Mapping study.

3 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**.

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

5 From the **Study** list, choose **Study 1, Mapping**.

Generate the default solvers and enable the first iterative solver suggestion.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** node.

4 Right-click **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** > **Suggested Iterative Solver (GMRES with Direct Precond.) (lnsf)** and choose **Enable**.

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

RESULTS

Multislice

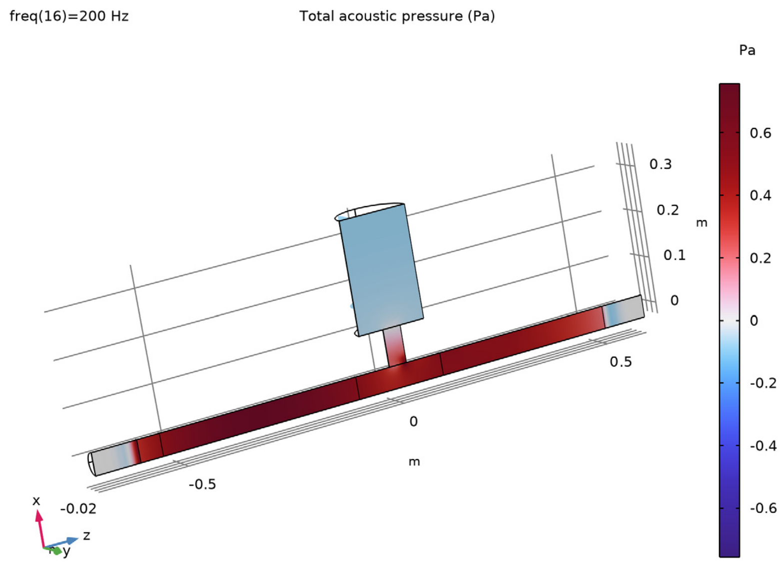
1 In the **Model Builder** window, expand the **Acoustic Pressure (lnsf)** node, then click **Multislice**.

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

3 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type `0`.

5 In the **Acoustic Pressure (lnsf)** toolbar, click **Plot**.



Multislice

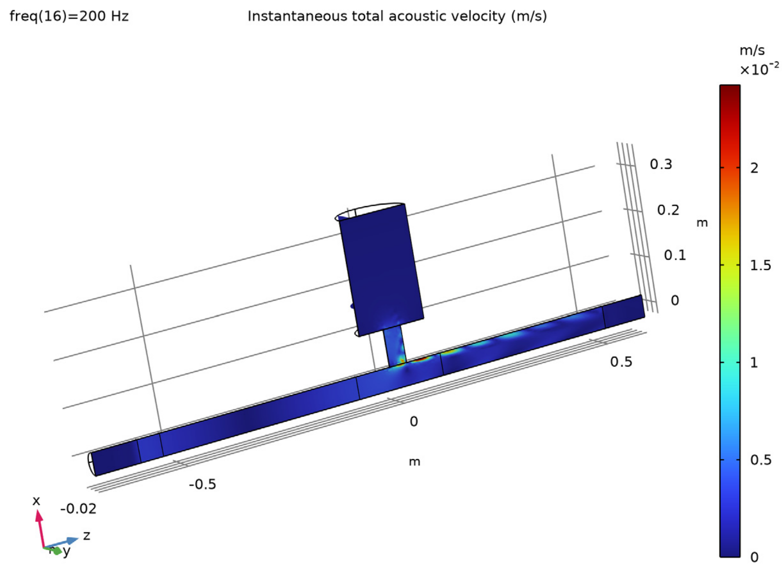
1 In the **Model Builder** window, expand the **Acoustic Velocity (lnsf)** node, then click **Multislice**.

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

3 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type `0`.

5 In the **Acoustic Velocity (lnsf)** toolbar, click **Plot**.



Multislice

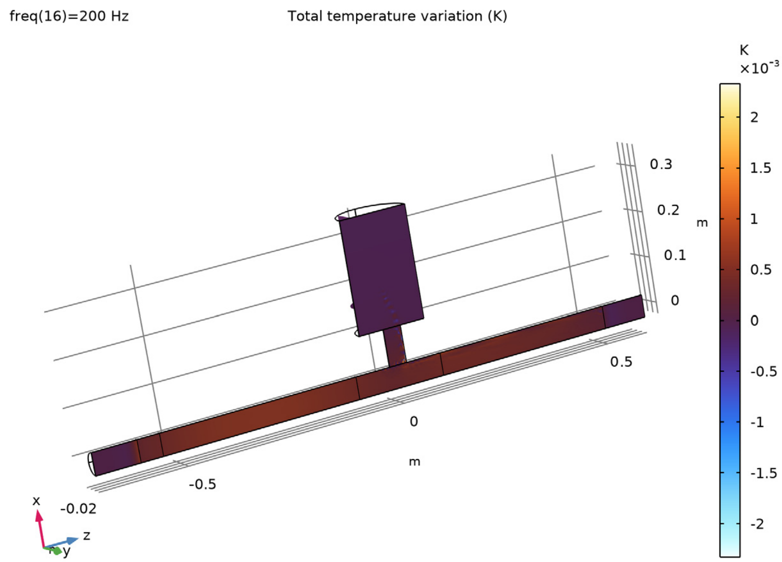
1 In the **Model Builder** window, expand the **Temperature Variation (lnsf)** node, then click **Multislice**.

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

3 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

4 In the **Coordinates** text field, type 0.

5 In the **Temperature Variation (lnsf)** toolbar, click **Plot**.



Transmission Loss

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section.

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

6 Select the **y-axis label** checkbox. In the associated text field, type `TL (dB)`.

Global 1

1 Right-click **Transmission Loss** and choose **Global**.

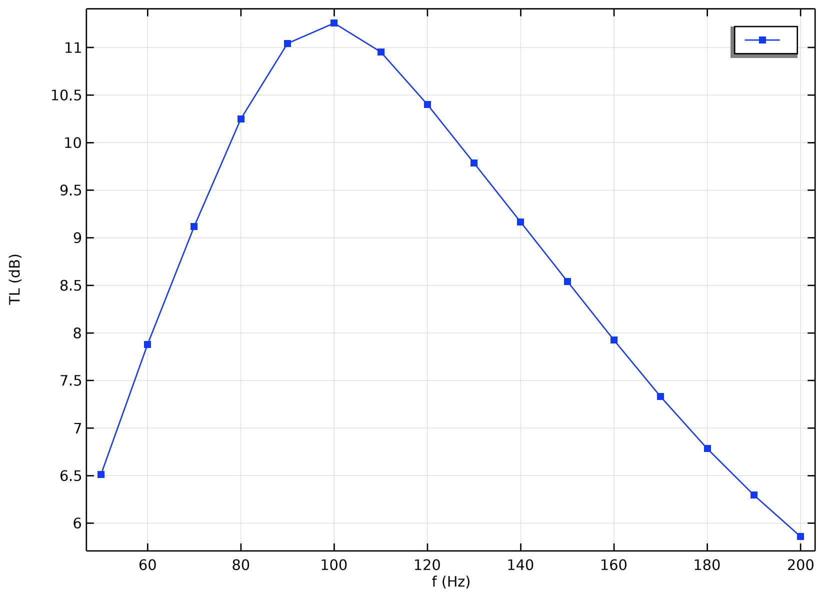
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(intop_in(pb)/intop_out(lnsf.p_t)))`		

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

5 In the **Transmission Loss** toolbar, click **Plot**.



Comparison Mapped and Imported Data

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

2 In the **Settings** window for **1D Plot Group**, type `Comparison Mapped and Imported Data` in the **Label** text field.

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

Line Graph 1

1 Right-click **Comparison Mapped and Imported Data** and choose **Line Graph**.

2 Select Edge 12 only.

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

4 In the **Expression** text field, type `w_map`.

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

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Mapped

Line Graph 2

1 In the **Model Builder** window, right-click **Comparison Mapped and Imported Data** and choose **Line Graph**.

2 Select Edge 12 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Global definitions** > **Functions** > **z_velocity(x, y, z, t) - CFD Data (CGNS) 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `z_velocity(x, y, z, 0)`.

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

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Imported

8 In the **Comparison Mapped and Imported Data** toolbar, click **Plot**.