Model created in COMSOL Multiphysics 6.4

Acoustics of a Particulate-Filter-Like System

Introduction

This is a model of the acoustics in a particulate-filter-like system. Real systems, like diesel particulate filters (DPFs), are designed to remove/filter soot (diesel particles) from the exhaust of diesel engine vehicles. The porous medium in such systems are typically structured with long air-filled ducts. To simplify this model, the filter geometry is assumed to be axisymmetric and the ducts are represented by long cylindrical groves inside a porous material plug. Although the main function of a particulate filter is filtering of the exhaust flow, the filter also has acoustic damping properties that relate to the muffler system.

The model analyzes the acoustic properties of the simplified 2D axisymmetric particulate-filter like geometry using the Acoustic–Poroelastic Waves Interaction multiphysics interface. This interface describes the small-deformation elastic waves propagating in a porous material coupled to waves in a fluid by an Acoustic–Porous Boundary multiphysics coupling. The model accounts for the coupled displacement and is thus a fluid-structure interaction problem.

Model Definition

Three aligned cylinders make up the particulate filter system under study: an inlet, an outlet, and a main filter cylinder. The particulate filter is located inside the filter cylinder. Figure 1 shows a sketch of a cross section in the *rz*-plane of the 2D axisymmetric geometry. The filter in the central region is of length L_{filter} = 200 mm with a filter radius of R_{filter} = 150 mm. The inlet and outlet pipe radii are R_{tube} = 50 mm. The filter consists of a structured air-filled porous material (the brown region), which could be a silicon carbide matrix. The air-filled groves (light blue) have a width of d_{h} = 5 mm and the porous walls are of thickness h_{t} = 3.2 mm. At the end of each grove there is an impermeable steel plug (black). The rest of the system is filled with air.

The wide groves are used to simplify the model. In real DPF systems the groves are replaced by long slender ducts with a typical width of 1–2 mm and the porous walls have a typical width of 0.3–0.5 mm.

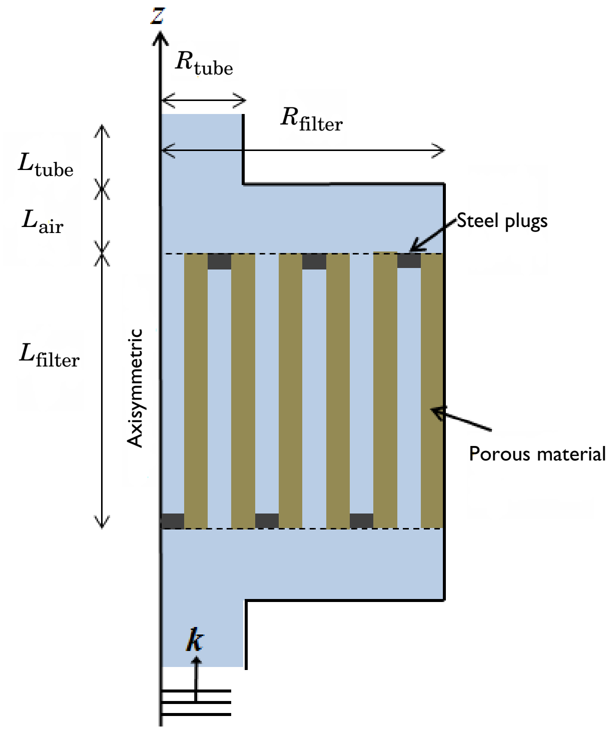


Figure 1: Geometry of the particulate filter with dimensions indicated.

The porous material is assumed to be isotropic with the material parameters as listed in Table 1.

TABLE 1: MATERIAL PARAMETERS OF THE POROUS MATRIX.

PARAMETER	VALUE	DESCRIPTION
E	20 GPa	Young's modulus
ν	0.4	Poisson's ratio
ρ_d	1000 kg/m^3	Drained matrix density
α_B	0.3	Biot–Willis coefficient
ε_p	0.3	Porosity
κ_p	10^{-11} m^2	Permeability of porous matrix
τ	1	Tortuosity factor

Note that the Biot–Willis coefficient is equal to the porosity for rigid porous materials and is equal to 1 for a soft porous material (or a suspension of solid in liquid). The fluid parameters are those of air including the compressibility, χ, which for an ideal gas is equal to $(p_0)^{-1}$, where p_0 is the absolute pressure (here 1 atm).

The filter is characterized acoustically by the transmission loss, TL (given in dB), as a function of the frequency, f. It is defined as

$$\mathrm{TL}(f) = 20\log\left(\left|\frac{p_{\mathrm{incident}}}{p_{\mathrm{out}}}\right|\right)$$

where p_{incident} is the incident inlet pressure and p_{out} is the outlet pressure. You solve the model for the frequency interval 20 Hz–2000 Hz.

When setting up the porous material model, you also need to specify whether to use the low-frequency (default) or high-frequency range approximation for the fluid viscosity. The transition between the two ranges is defined by the reference frequency f_c given by the expression

$$f_c = \frac{\varepsilon_p \mu}{2\pi\kappa\rho_{\mathrm{f}}}$$

where ρ_{f} is the fluid density (for air 1.2 kg/m^3) and μ is the dynamic viscosity of the fluid (for air $1.8\cdot 10^{-5}$ Pa·s). Using the above material parameters gives a reference frequency of the order 100 kHz. Thus, the low-frequency range applies to the current problem.

The acoustic transmission loss TL through the axisymmetric simplified particulate filter is determined for the frequency range 20 Hz to 2000 Hz and depicted in Figure 2.

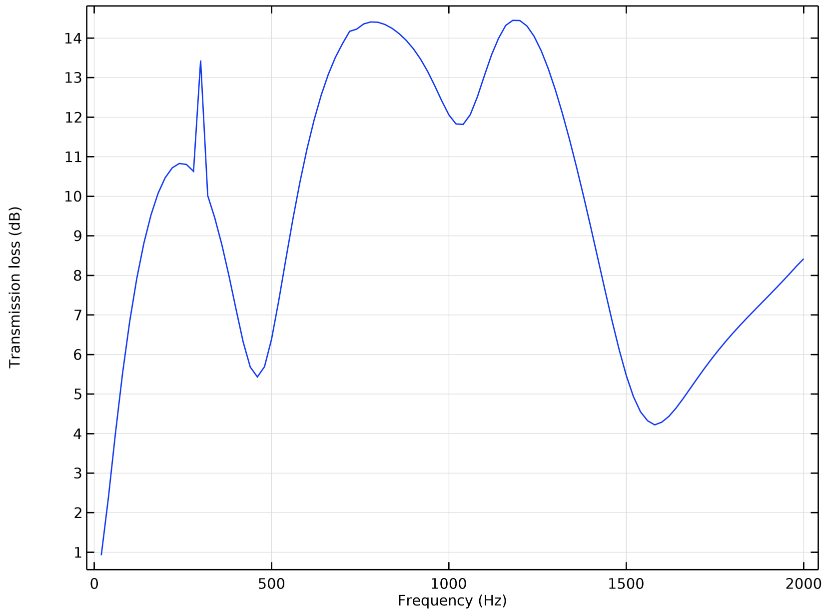


Figure 2: Transmission through the simple particulate filter as a function of frequency.

The loss is seen to be of the same order of magnitude as in real particulate filters (like diesel particulate filters, DPFs) the porous medium is often, as mentioned, structured with long ducts that decrease the acoustic damping while retaining good filtering properties. In this axisymmetric model the ducts take the form of cylindrical slits in 3D, which may introduce some nonstandard resonances in the filter. Moreover, in a real exhaust system there is an interaction between the exhaust flow and the acoustics (here the no-flow situation is studied), and the temperature is higher than 20°C (as used here). Other physical effects include acoustic–structure and poroelastic–structure interactions with the exhaust pipe system. The present simplified model enables isolating the acoustics problem from other physical phenomena.

Figure 3 depicts the pressure distribution inside the particulate filter model for 20 Hz and for 2 kHz.

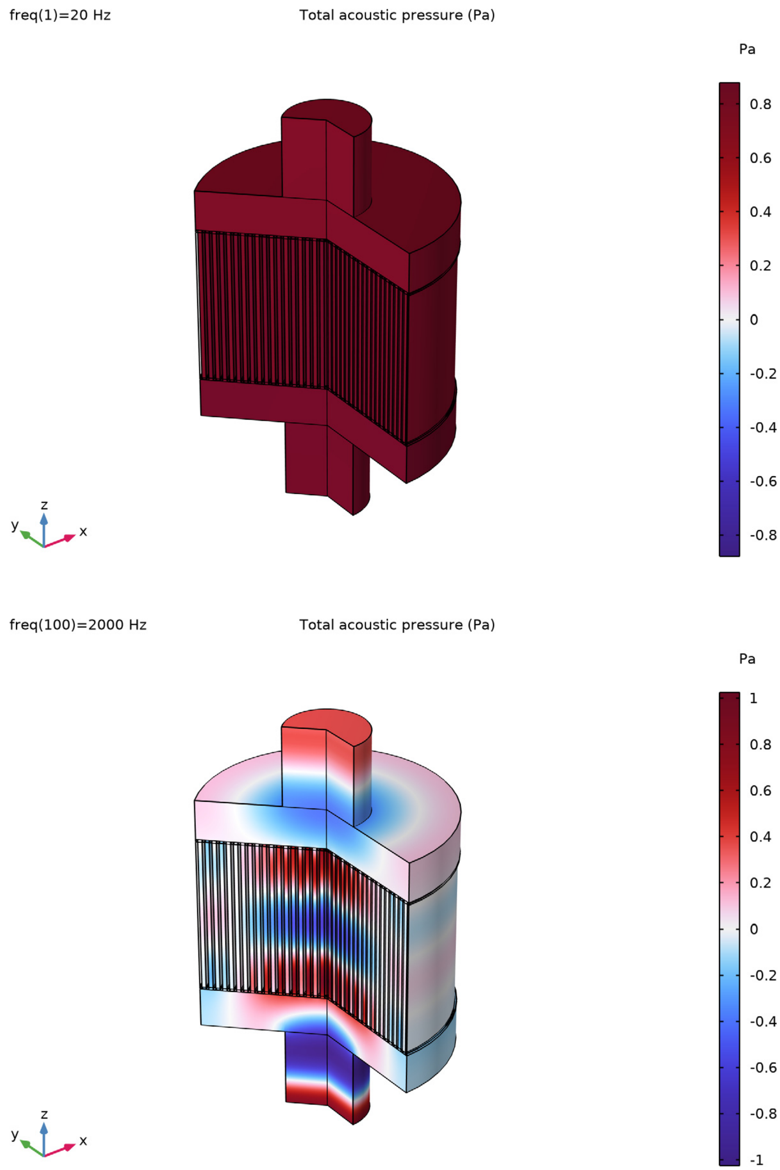


Figure 3: Pressure distribution inside the particulate filter for $f = 20$ Hz (top) and $f = 2$ kHz (bottom).

Application Library path: `Acoustics_Module/Automotive/acoustics_particulate_filter`

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** > **Acoustic–Structure Interaction** > **Acoustic–Solid–Poroelastic Waves Interaction**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

6 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 `acoustics_particulate_filter_parameters.txt`.

The parameters loaded here define the geometric dimensions and the tortuosity parameter used in the poroelastic model. Because the geometry is now parameterized, changing the dimensions in the parameters list will update the geometry automatically.

GEOMETRY 1

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 `Rtube`.

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

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 `Rfilter`.

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

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

Rectangle 3 (r3)

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 `Rfilter`.

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

5 Locate the **Position** section. In the **z** text field, type `Ltube+Lair`.

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

Layer name	Thickness (m)
Layer 1	ht

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

Rectangle 4 (r4)

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 `Rfilter`.

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

5 Locate the **Position** section. In the **z** text field, type `Ltube+Lair+Lfilter`.

Rectangle 5 (r5)

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 `Rtube`.

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

5 Locate the **Position** section. In the **z** text field, type `Ltube+2*Lair+Lfilter`.

Rectangle 6 (r6)

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 `dh`.

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

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

6 In the **z** text field, type `Ltube+Lair`.

Array 1 (arr1)

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

2 Select the object **r6** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **r size** text field, type `18`.

5 Locate the **Displacement** section. In the **r** text field, type `(dh+ht)`.

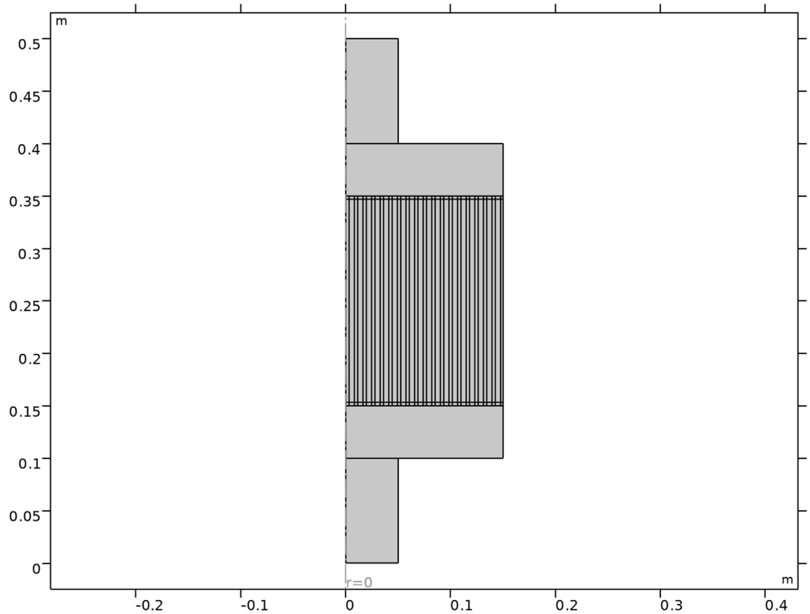
Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

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

The geometry of the diesel particulate filter should look like the figure below.



DEFINITIONS

All domains

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

2 In the **Settings** window for **Box**, type `All domains` in the **Label** text field.

Poroelastic Waves Domains

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

2 In the **Settings** window for **Explicit**, type `Poroelastic Waves Domains` in the **Label** text field.

3 Select Domains 3–5, 11–13, 17–19, 23–25, 29–31, 35–37, 41–43, 47–49, 53–55, 59–61, 65–67, 71–73, 77–79, 83–85, 89–91, 95–97, 101–103, 107–109, and 113–115 only.

Steel Domains

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

2 In the **Settings** window for **Explicit**, type `Steel Domains` in the **Label** text field.

3 Select Domains 10, 14, 22, 26, 34, 38, 46, 50, 58, 62, 70, 74, 82, 86, 94, 98, 106, and 110 only.

Air Domains

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

2 In the **Settings** window for **Difference**, type `Air Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog, select **All domains** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click **Add**.

8 In the **Add** dialog, in the **Selections to subtract** list, choose **Poroelastic Waves Domains** and **Steel Domains**.

9 Click **OK**.

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.

By default, the first material applies to all domains except where overridden by subsequent materials in the **Materials** node.

Next, create a poroelastic material with user-defined material parameters for the solid matrix.

MATERIALS

SiC matrix

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `SiC matrix` in the **Label** text field.

The material properties for the SiC matrix will be defined after the physics interface settings.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Steel AISI 4340**.

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

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

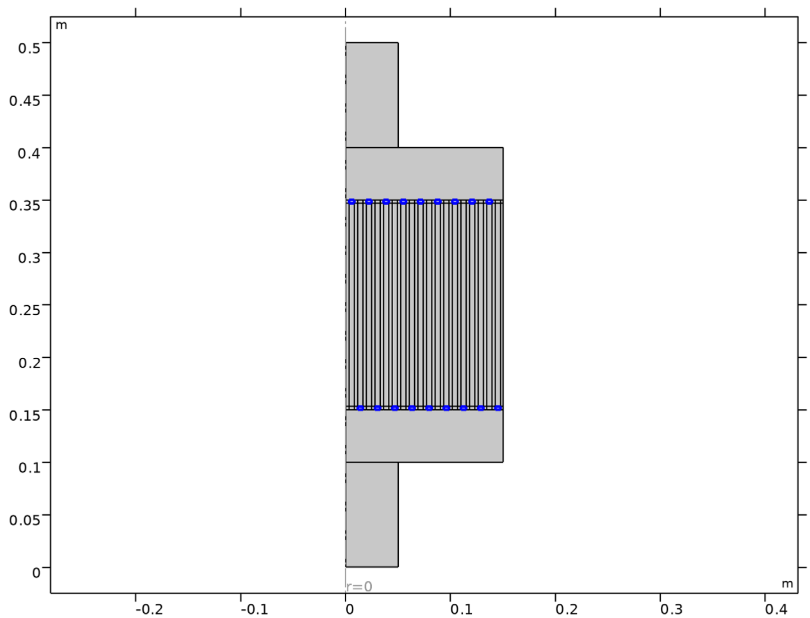
MATERIALS

Steel AISI 4340 (mat3)

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

2 From the **Selection** list, choose **Steel Domains**.

The steel plugs are the domains selected in the figure below.



POROELASTIC WAVES (PELW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Poroelastic Waves (pelw)**.

2 In the **Settings** window for **Poroelastic Waves**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Poroelastic Waves Domains**.

Having defined the materials, you can specify the domain settings.

Poroelastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Poroelastic Waves (pelw)** click **Poroelastic Material 1**.

2 In the **Settings** window for **Poroelastic Material**, locate the **Poroelastic Model** section.

3 From the **Model** list, choose **Biot (viscous losses)**.

4 Locate the **Porous Matrix Properties** section. From the **Porous elastic material** list, choose **SiC matrix (mat2)**.

5 From the **Specify** list, choose **Young's modulus and Poisson's ratio**.

Use a linear elastic material model for the steel-plug domains.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Steel Domains**.

POROELASTIC WAVES (PELW)

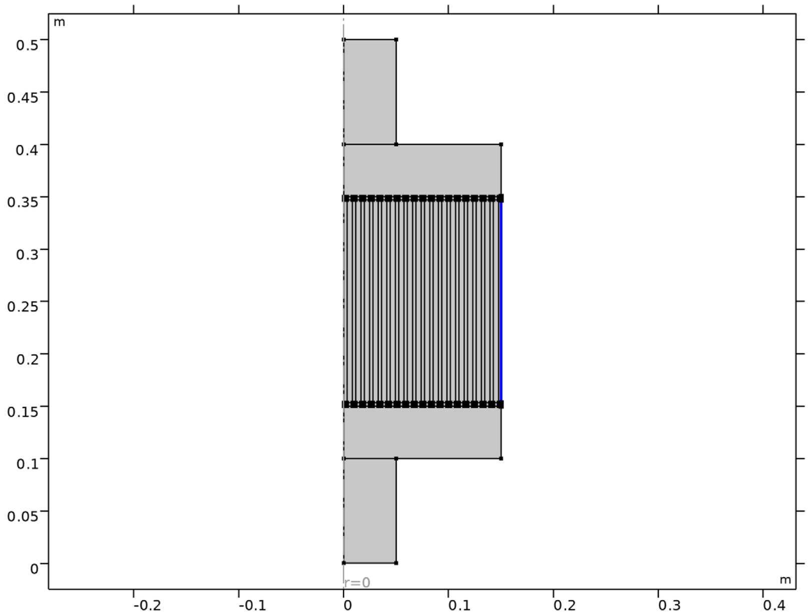
In the **Model Builder** window, under **Component 1 (comp1)** click **Poroelastic Waves (pelw)**.

Fixed Constraint 1

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

2 Select Boundaries 273–275 only.

The porous matrix is assumed to be glued to an outer casing at the boundary highlighted in the figure below; hence the fixed constraint boundary condition. This boundary is also sound hard.



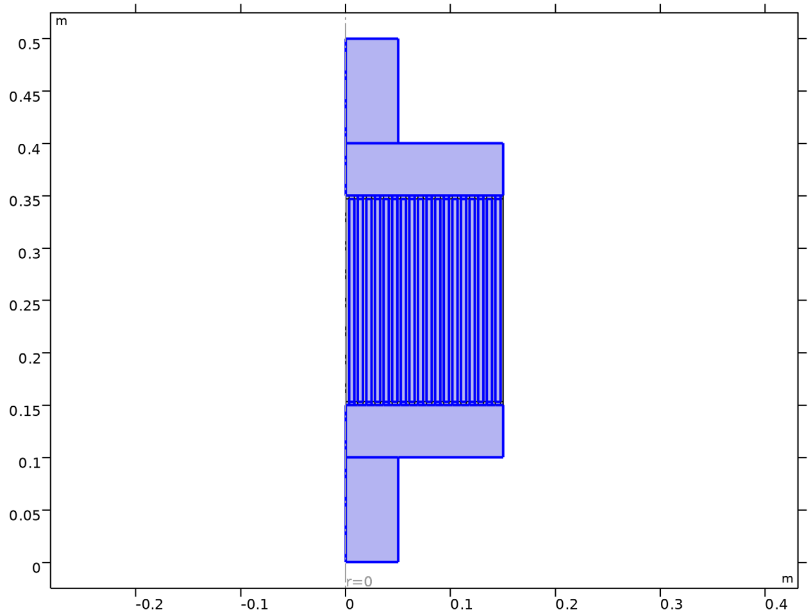
PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air Domains**.

The selected air domain where Pressure Acoustics apply is depicted in the figure below. It consists of the inlet and outlet as well as the thin air groves inside the particulate filter.



Port 1

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

2 Select Boundary 2 only.

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

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

5 Locate the **Incident Mode Settings** section. In the $A_{\mathrm{p}}{}^{\mathrm{in}}$ text field, type `p0`.

Port 2

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

2 Select Boundary 15 only.

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

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

You have now specified the domain settings. The red cross decoration for the Air and the SiC matrix nodes under **Materials** indicates that there are still undefined material parameters. To discover what is missing, return to the **Materials** node before proceeding with the boundary conditions.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Air (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Compressibility of fluid	chif	1/1[atm]	1/Pa	Basic

Recall that the compressibility of an ideal gas at the pressure P0 equals 1/P0.

SiC matrix (mat2)

1 In the **Model Builder** window, click **SiC matrix (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	1e-11	m²	Basic
Tortuosity factor	tau_iso ; tauii = tau_iso, tauij = 0	tauP	1	Poroacoustics model
Young's modulus	E	20[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.4	1	Young's modulus and Poisson's ratio
Density	rho	1000	kg/m³	Basic
Porosity	epsilon	0.3	1	Basic
Biot-Willis coefficient	alphaB	0.3	1	Poroelastic material

MULTIPHYSICS

The coupling between the acoustic and the porous domain, and between the acoustic and the elastic domain is automatically set up under the **Multiphysics** node. Click on the node and then inspect the **Acoustic–Porous Boundary 1** node and the **Acoustic–Structure Boundary 1** node. Both nodes have the selection automatically set to **All boundaries** - this default setting automatically couples all relevant boundaries.

MESH 1

In this model, the mesh is set up manually. Proceed by directly adding the desired mesh component.

Use a mapped mesh with boundary layers added to resolve the continuity condition between the poroelastic and pressure acoustic domains.

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** section.

3 From the **Predefined** list, choose **Extra fine**.

Mapped 1

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

2 In the **Settings** window for **Mapped**, click to expand the **Reduce Element Skewness** section.

3 Select the **Adjust edge mesh** checkbox.

Size 1

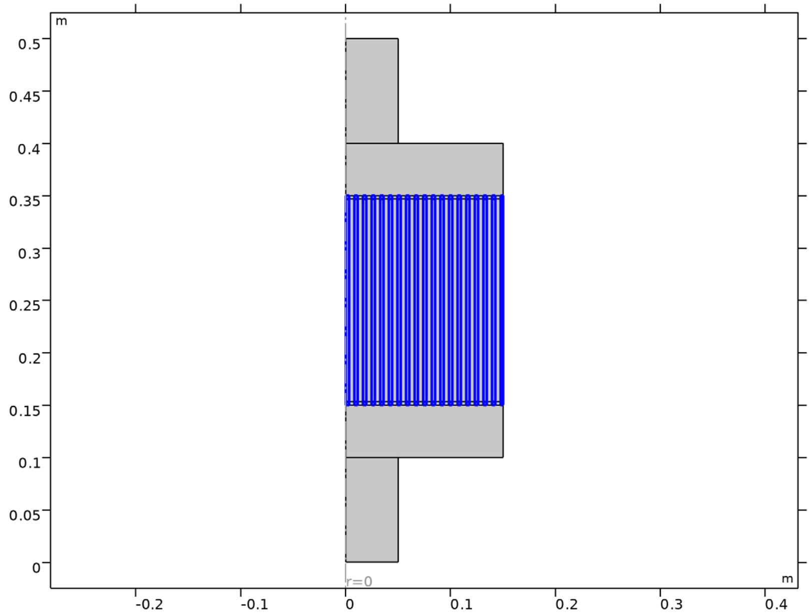
1 Right-click **Mapped 1** and choose **Size**.

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

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

Select all the domains that make up the particulate filter, for easy selection use the **Select Box** utility.

4 From the **Selection** list, choose **Poroelastic Waves Domains**.



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 `ht`.

8 Click **Build All**.

Boundary Layers 1

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

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

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

4 Select Domains 2 and 6 only.

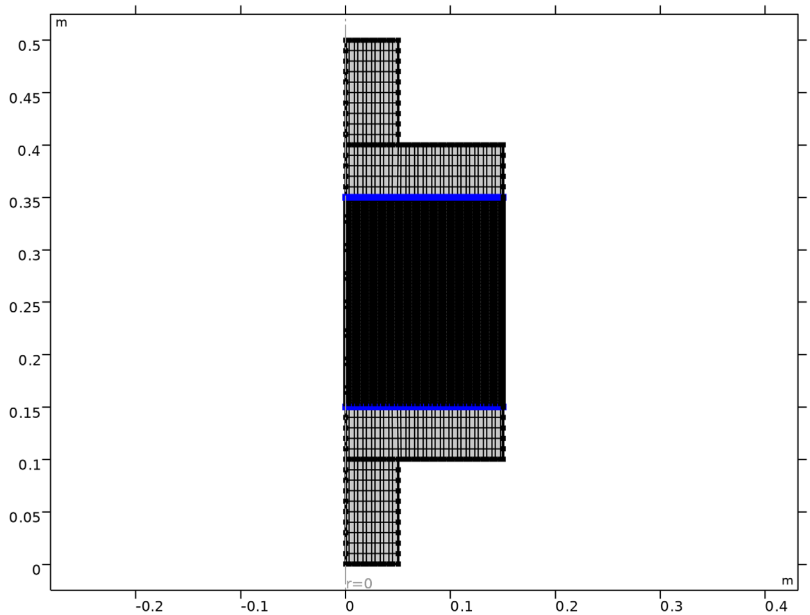
5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** checkbox.

Boundary Layer Properties

Now, select all the boundaries between the particulate filter and the air, for easy selection use the **Select Box** utility. You can also paste in the numbers given below.

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

2 Select Boundaries 6, 12, 17, 22, 24, 29, 31, 36, 38, 43, 45, 50, 52, 57, 59, 64, 66, 71, 73, 78, 80, 85, 87, 92, 94, 99, 105, 110, 112, 117, 119, 124, 126, 131, 133, 138, 140, 145, 147, 152, 154, 159, 161, 166, 168, 173, 175, 180, 182, 187, 189, 194, 196, 201, 203, 208, 210, 215, 217, 222, 224, 229, 231, 236, 238, 243, 245, 250, 252, 257, 259, 264, 266, and 271 only.



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

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

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

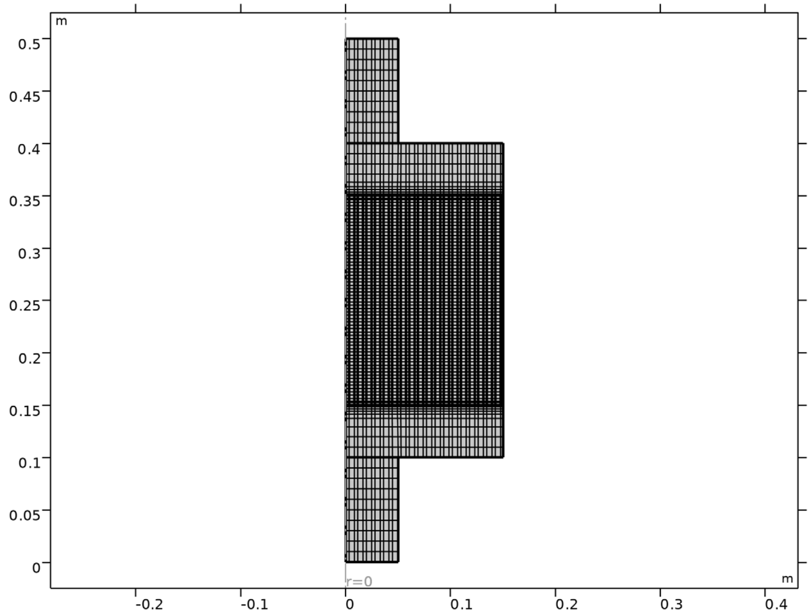
6 In the **Thickness** text field, type `ht/2`.

7 Click **Build All**.

When modeling poroelastic waves coupled to pressure acoustics, it is good practice to include a boundary layer mesh on the interface between the two physics. This is due to the fact that large gradients in the dependent variables appear here.

The mesh should look like the figure below.

8 In the **Model Builder** window, click **Mesh 1**.



STUDY 1

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 1** 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 `range(20,20,2000)`.
4. In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
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. From the **Selection** list, choose **Steel Domains**.
5. Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
6. In the **Model Builder** window, expand the **Stress (solid)** node.

Deformation

1. In the **Model Builder** window, expand the **Results** > **Stress (solid)** > **Surface 1** node, then click **Deformation**.
2. In the **Settings** window for **Deformation**, locate the **Scale** section.
3. Select the **Scale factor** checkbox. In the associated text field, type `1`.
4. In the **Stress (solid)** toolbar, click **Plot**.

Stress, 3D (solid)

1. In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.
2. In the **Settings** window for **3D Plot Group**, click to expand the **Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Steel Domains**.
5. Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
6. In the **Model Builder** window, expand the **Stress, 3D (solid)** node.

Deformation

1. In the **Model Builder** window, expand the **Results** > **Stress, 3D (solid)** > **Surface 1** node, then click **Deformation**.
2. In the **Settings** window for **Deformation**, locate the **Scale** section.
3. Select the **Scale factor** checkbox. In the associated text field, type `1`.
4. In the **Stress, 3D (solid)** toolbar, click **Plot**.

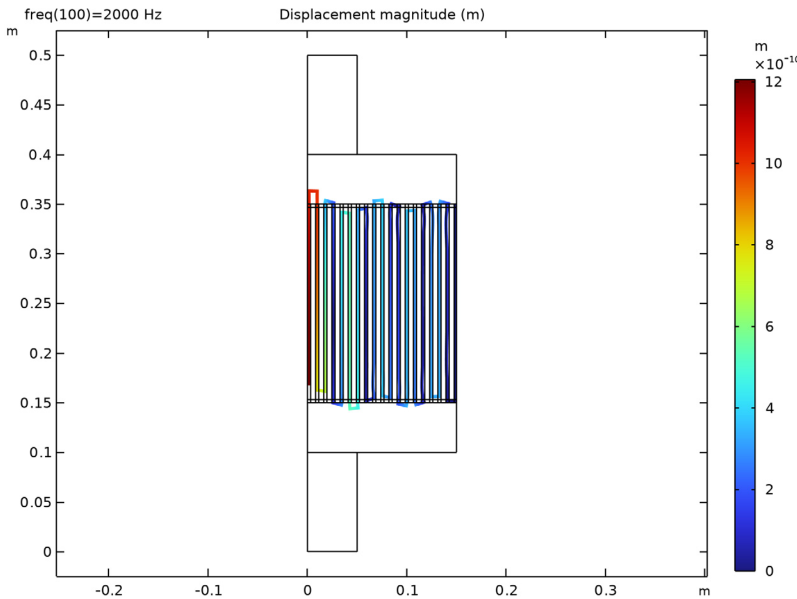
Surface 2

1. In the **Model Builder** window, right-click **Displacement (pelw)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.disp`.
4. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface**.
5. Click to expand the **Title** section. From the **Title type** list, choose **None**.

Deformation 1

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

The plot should now look like the figure below. It depicts the displacement magnitude.



Surface 2

1 Right-click **Displacement, 3D (pelw)** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.disp`.

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

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Deformation 1

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

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

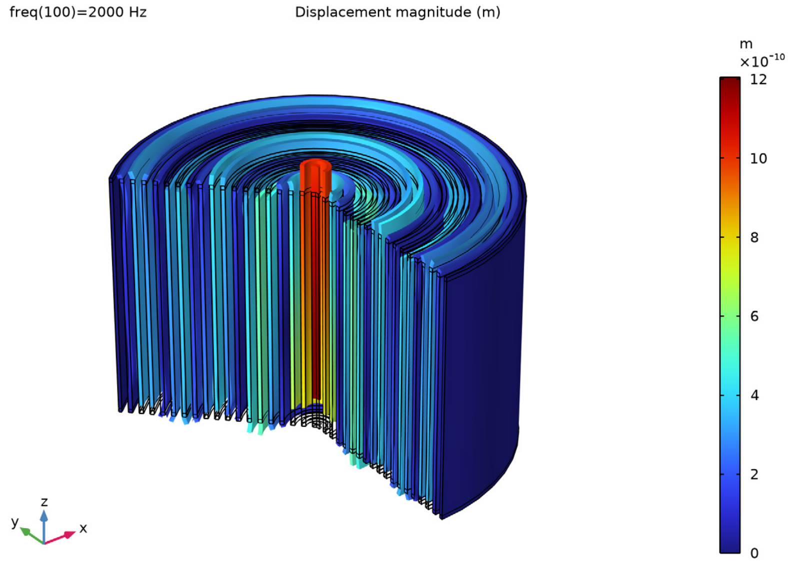
3 In the **R-component** text field, type `cos(rev1phi)*u2`.

4 In the **PHI-component** text field, type `sin(rev1phi)*v2`.

Displacement, 3D (pelw)

Displacement in the porous matrix at 2 kHz, is depicted in the figure below.

1 In the **Model Builder** window, under **Results** click **Displacement, 3D (pelw)**.

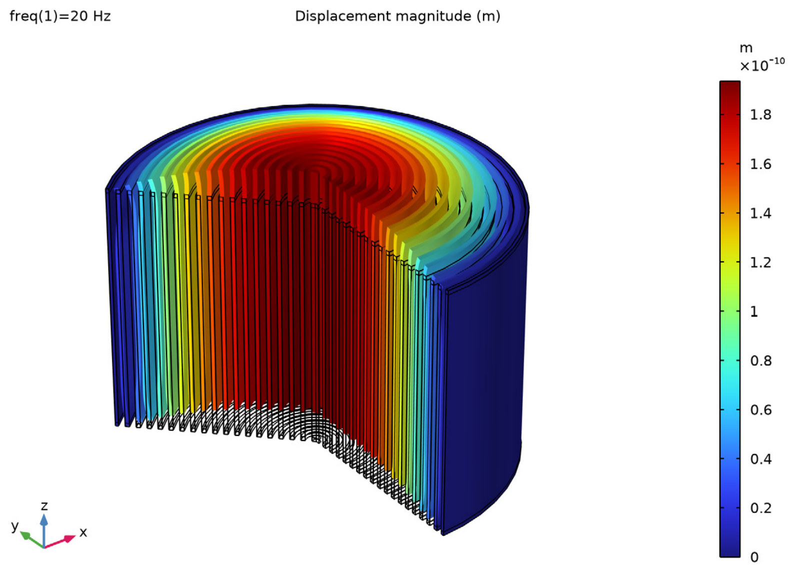


2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **20**.

4 In the **Displacement, 3D (pelw)** toolbar, click **Plot**.

Depicted below, the displacement magnitude in the porous matrix after changing the evaluation frequency to 20 Hz.

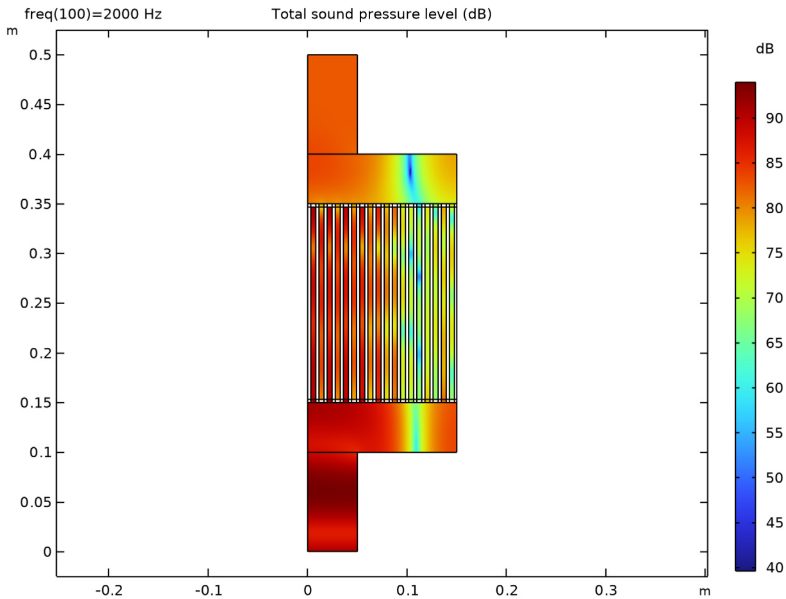


Sound Pressure Level (acpr)

1 In the **Model Builder** window, click **Sound Pressure Level (acpr)**.

2 In the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

The sound pressure level in dB (relative to 20 µPa) at 2 kHz, is depicted below.



Acoustic Pressure, 3D (acpr)

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

2 In the **Acoustic Pressure, 3D (acpr)** toolbar, click **Plot**.

The plot should look like the one depicted in Figure 3 (bottom).

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

4 From the **Parameter value (freq (Hz))** list, choose **20**.

5 In the **Acoustic Pressure, 3D (acpr)** toolbar, click **Plot**.

The plot, now evaluated at 20 Hz, should look like the one depicted in Figure 3 (top).

SPL at Inlet and Outlet

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

2 In the **Settings** window for **1D Plot Group**, type `SPL at Inlet and Outlet` in the **Label** text field.

Octave Band 1

1 In the **SPL at Inlet and Outlet** toolbar, click **More Plots** and choose **Octave Band**.

Select boundary as the geometric entity and the pressure is automatically averaged on that boundary.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

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

4 Select Boundary 15 only.

5 Locate the **Plot** section. From the **Quantity** list, choose **Continuous power spectral density**.

Change the style to octave or 1/3 octave if you like the SPL given in bands.

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

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

8 In the table, enter the following settings:

Legends
`SPL at outlet`

Octave Band 2

1 Right-click **Octave Band 1** and choose **Duplicate**.

Change the selection for the boundary.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

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

4 Click **Clear Selection**.

5 Select Boundary 2 only.

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

Legends
`SPL at inlet`

SPL at Inlet and Outlet

1 In the **Model Builder** window, click **SPL at Inlet and Outlet**.

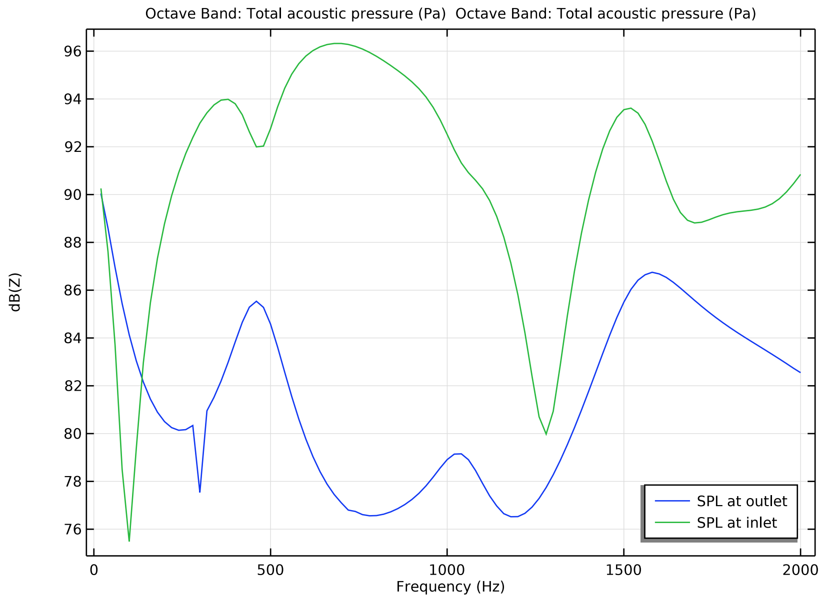
2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 Locate the **Axis** section. Clear the **x-axis log scale** checkbox.

5 In the **SPL at Inlet and Outlet** toolbar, click **Plot**.

The plot should look like the figure below. It depicts the sound pressure level in dB (relative to µPa) at the inlet (blue line) and the outlet (green line).



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.

Octave Band 1

1. In the **Transmission Loss** toolbar, click **More Plots** and choose **Octave Band**.
2. In the **Settings** window for **Octave Band**, locate the **Selection** section.
3. From the **Geometric entity level** list, choose **Global**.
4. Locate the **y-Axis Data** section. In the **Expression** text field, type `acpr.port1.P_in`.
5. From the **Expression type** list, choose **Power**.
6. In the **Power reference** text field, type `acpr.port2.P_out`.
7. Locate the **Plot** section. From the **Quantity** list, choose **Continuous power spectral density**.

Transmission Loss

1 In the **Model Builder** window, click **Transmission Loss**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

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

6 Locate the **Axis** section. Clear the **x-axis log scale** checkbox.

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

The plot should look like the one depicted in Figure 2.