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 slits. In this model the filter is approximated with a uniform porous material plug. Although the main function of a particulate filter is filtering of the exhaust flow, the filter also has acoustic damping properties which relate to the muffler system.

In this model the acoustic properties of a simplified 2D axisymmetric particulate-filter like geometry are analyzed using the The Poroelastic Waves Interface. The poroelastic wave model describes the small-deformation elastic waves propagating in a porous material coupled to waves in a fluid. The model accounts for the coupled displacement and is thus a fluid-structure interaction problem.

©2012 COMSOL

Model Definition

The present model is that of a simple particulate filter made up of three aligned cylinders: an inlet, an outlet, and a main filter cylinder. The *rz*-plane of the 2D axisymmetric geometry is depicted in Figure 1.

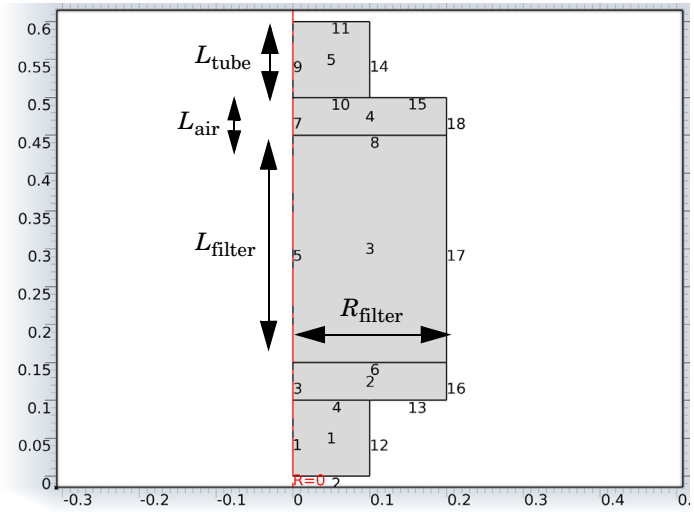


Figure 1: Geometry of the DPF with dimensions indicated: porous filter length, L_{filter}; inlet/outlet length, L_{tube}; air gap length, L_{air}; and filter radius, R_{filter}

The filter has an air-filled porous material, which could be a silicon carbide matrix, in the central region of length L_{filter} (Domain 3) and air in the rest of the system. The porous material is assumed to be isotropic with 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

 ©2012 COMSOL

parameters are that 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 T_{loss} (given in dB) as function of frequency f:

$$T_{\text{loss}}(f) = 20\log\left(\left|\frac{p_{\text{in}}}{p_{\text{out}}}\right|\right)$$

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

$$f_c = \frac{\varepsilon_p \mu}{2\pi\kappa\rho_f}$$

where ρ_f is the fluid (air) density 1.2 kg/m^3 and μ is the dynamic viscosity of the fluid (for air $1.8\cdot10^{-5}$ 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.

Results and Discussion

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

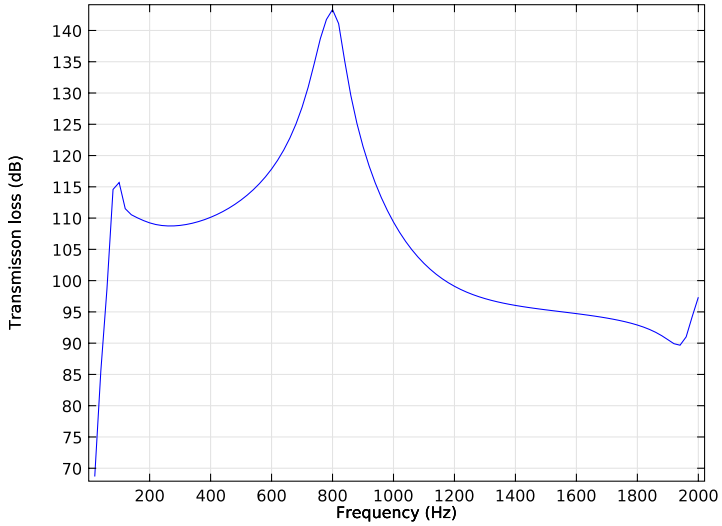


Figure 2: Transmission through the simple particulate filter as a function of frequency.

The loss is seen to be substantial and with resonances. In real particulate filters (like diesel particulate filters, DPFs) the porous medium is often, as mentioned, structured with long slits which decreases the acoustic damping while retaining good filtering properties. Moreover, in a real exhaust system there is an interaction between the flow and the acoustics, 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.

Model Library path: `Acoustics_Module/Tutorial_Models/acoustics_particulate_filter`

©2012 COMSOL

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D axisymmetric** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Acoustics>Acoustic-Structure Interaction>Poroelastic Waves (elw)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `acoustics_particulate_filter_parameters.txt`.

The parameters loaded here define the incident acoustic pressure, the geometrical dimensions, and the tortuosity parameter used in the poroelastic model. Because the geometry is now parametrized, changing the dimensions in the parameters list will update the geometry automatically.

GEOMETRY 1

Rectangle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `Rtube`.

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

Rectangle 2

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

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `Rfilter`.

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

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

Rectangle 3

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

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `Rfilter`.

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

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

Rectangle 4

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

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `Rfilter`.

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

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

Rectangle 5

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

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `Rtube`.

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

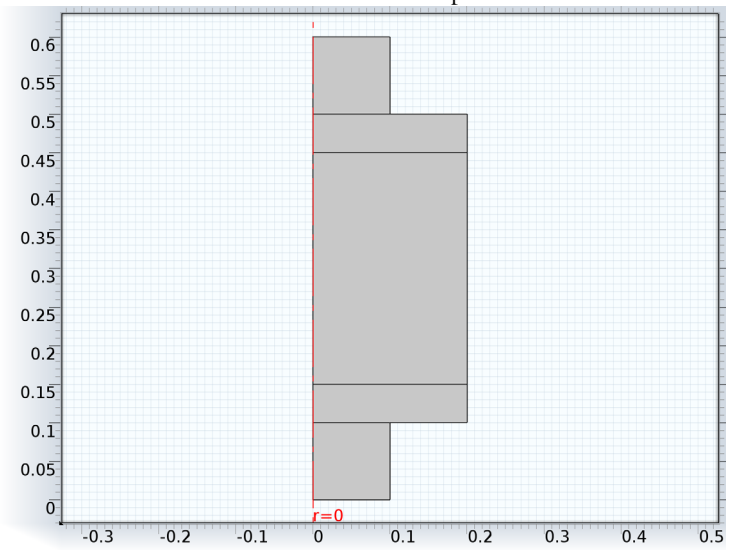
5 Locate the **Position** section. In the **z** edit field, type `Ltube+2*Lair+Lfilter`.

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Form Union** and choose **Build Selected**.

 ©2012 COMSOL

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



3 *Geometry of the diesel particulate filter, including boundary and domain labels.*

DEFINITIONS

Create integration coupling operators for calculating the inlet and outlet pressures, which you need in order to compute the transmission loss.

Integration 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `intop_in`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Select Point 2 only.

Integration 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `intop_out`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Select Point 5 only.

MATERIALS

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

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

4 Right-click and choose **Add Material to Model** from the menu.

Air

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

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

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `SiC matrix` in the **New name** edit field.

4 Click **OK**.

SiC matrix

1 In the **Material** settings window, locate the **Material Properties** section.

2 In the **Material properties** tree, select **Basic Properties>Density**.

3 Click **Add to Material**.

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

PROPERTY	NAME	VALUE
Density	rho	1000

5 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Poisson's Ratio**.

6 Click **Add to Material**.

 ©2012 COMSOL

7 Locate the **Material Contents** section. In the table, enter the following settings:

PROPERTY	NAME	VALUE
Poisson's ratio	nu	0.4

8 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Young's Modulus**.

9 Click **Add to Material**.

10 Locate the **Material Contents** section. In the table, enter the following settings:

PROPERTY	NAME	VALUE
Young's modulus	E	20 [GPa]

11 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Poroelastic Material>Biot-Willis coefficient (alphaB)**.

12 Click **Add to Material**.

13 Locate the **Material Contents** section. In the table, enter the following settings:

PROPERTY	NAME	VALUE
Biot-Willis coefficient	alphaB	0.3
Porosity	epsilon	0.3
Permeability	kappa	1e-11

POROELASTIC WAVES

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

Poroelastic Material 1

1 In the **Model Builder** window, expand the **Poroelastic Waves** node, then click **Poroelastic Material 1**.

2 In the **Poroelastic Material** settings window, locate the **Poroelastic Material** section.

3 From the **Porous elastic material** list, choose **SiC matrix**.

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

5 In the τ edit field, type `tauP`.

Pressure Acoustics Model 1

1 In the **Model Builder** window, right-click **Poroelastic Waves** and choose the domain setting **Pressure Acoustics, Frequency Domain>Pressure Acoustics Model**.

2 Select Domains 1, 2, 4, and 5 only.

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

MATERIALS

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Compressibility of fluid	chif	`1/1[atm]`

Recall that the compressibility of an ideal gas at pressure P_0 equals $1/P_0$.

POROELASTIC WAVES

Porous, Fixed Constraint 1

1 In the **Model Builder** window, right-click **Model 1>Poroelastic Waves** and choose the boundary condition **Porous Material>Porous, Fixed Constraint**.

 ©2012 COMSOL

2 Select Boundary 17 only.

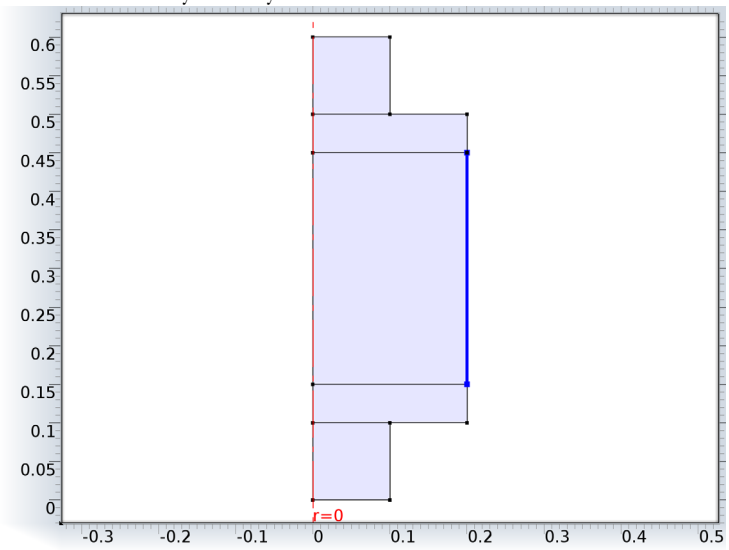


Figure 3: The porous matrix is assumed to be glued to an outer casing; hence the fixed constraint boundary condition.

Continuity 1

Continuity is automatically selected on the internal boundaries.

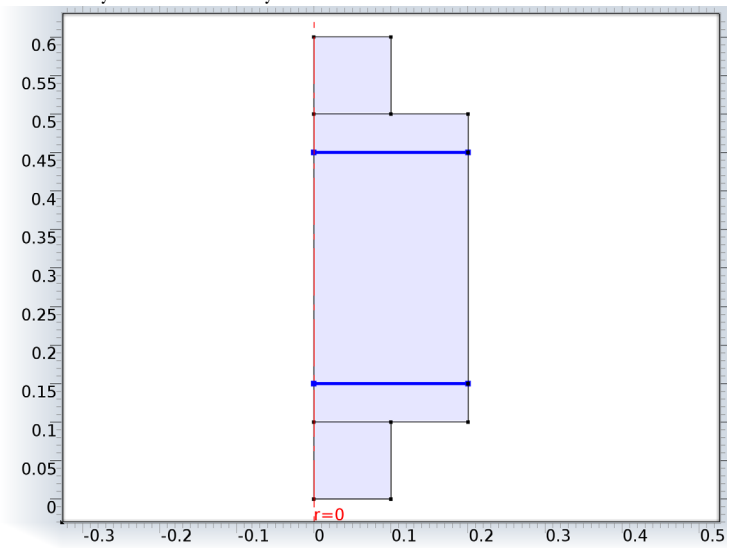


Figure 4: Continuity conditions.

Plane Wave Radiation 1

1 In the **Model Builder** window, right-click **Poroelastic Waves** and choose the boundary condition **Pressure Acoustics, Frequency Domain>Plane Wave Radiation**.

2 Select Boundaries 2 and 11 only.

3 Right-click **Model 1>Poroelastic Waves>Plane Wave Radiation 1** and choose the boundary condition **Incident Pressure Field**.

Incident Pressure Field 1

4 To restrict the incoming plane wave to the inlet, click on Boundary 11 (the outlet), then right-click to remove this boundary from the selection.

5 In the **Incident Pressure Field** settings window, locate the **Incident Pressure Field** section.

6 In the p_0 edit field, type `p0`.

MESH 1

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

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

 ©2012 COMSOL

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

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

Boundary Layer Properties

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 Select Boundaries 6 and 8 only.

3 Click the **Build All** button.

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.

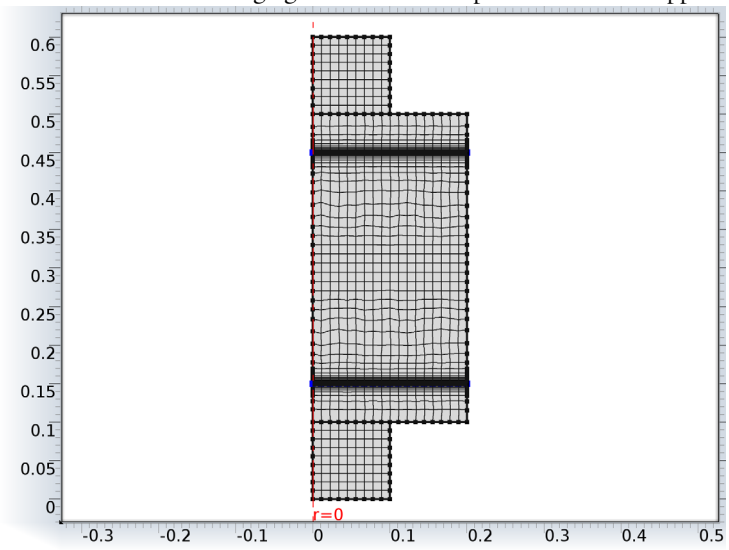


Figure 5: Mapped mesh including boundary layer meshes on the interfaces between the poroelastic and the pressure acoustic domains.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** edit field, type `range(20,20,2000)`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

2D Plot Group 1

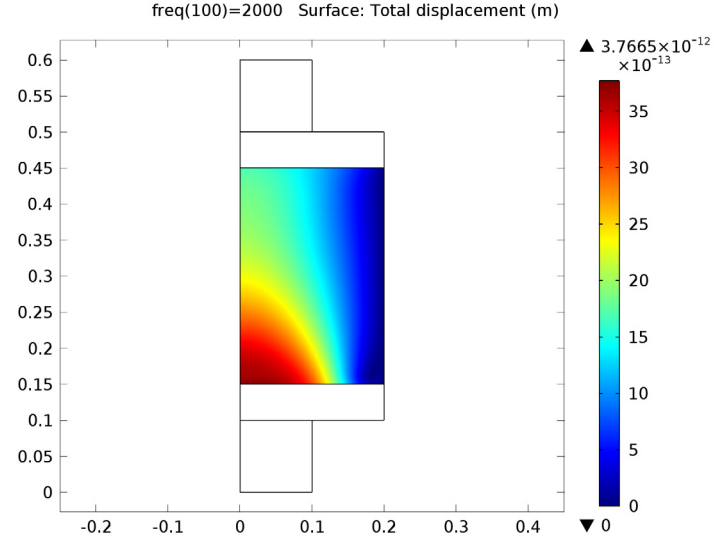


Figure 6: The first plot you see after solving depicts the total displacement.

©2012 COMSOL

3D Plot Group 2

1 In the **Model Builder** window, click **Results>3D Plot Group 2**.

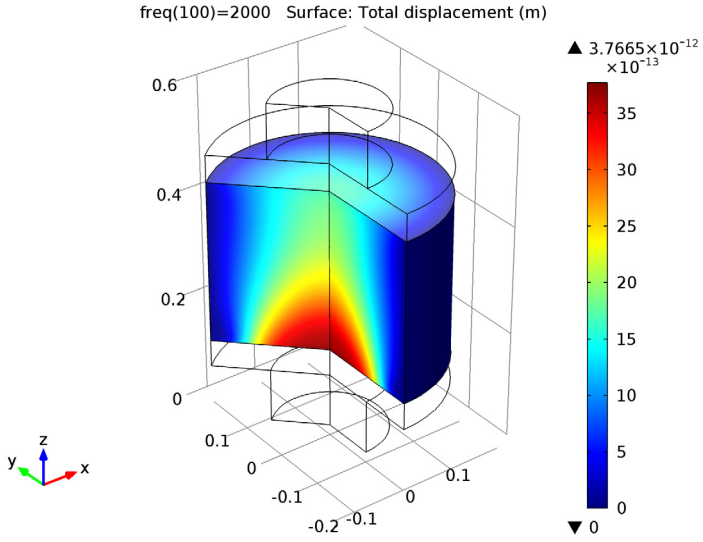


Figure 7: Total displacement in the porous matrix at 2 kHz.

2 In the **Model Builder** window, under **Results** click **3D Plot Group 2**.

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

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

5 Click the **Plot** button.

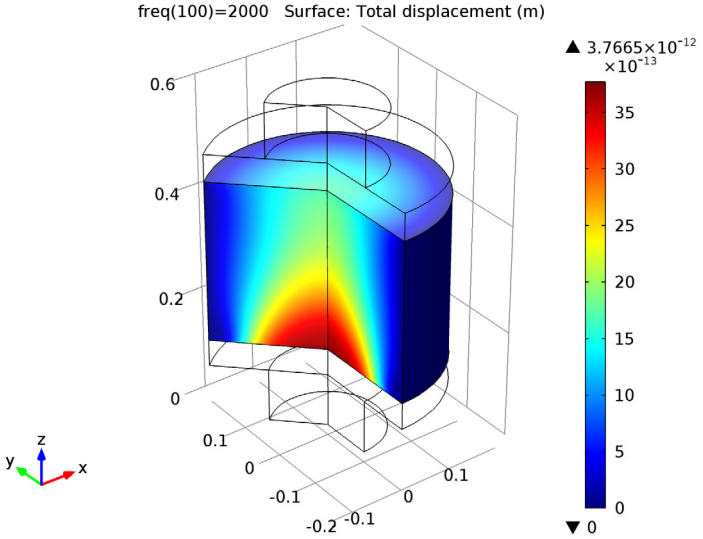


Figure 8: Total displacement in the porous matrix at 20 Hz.

2D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 3** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Poroelastic Waves (Pressure Acoustics, Frequency Domain)>Sound pressure level (elw.Lp)**.

 ©2012 COMSOL

4 Click the **Plot** button.

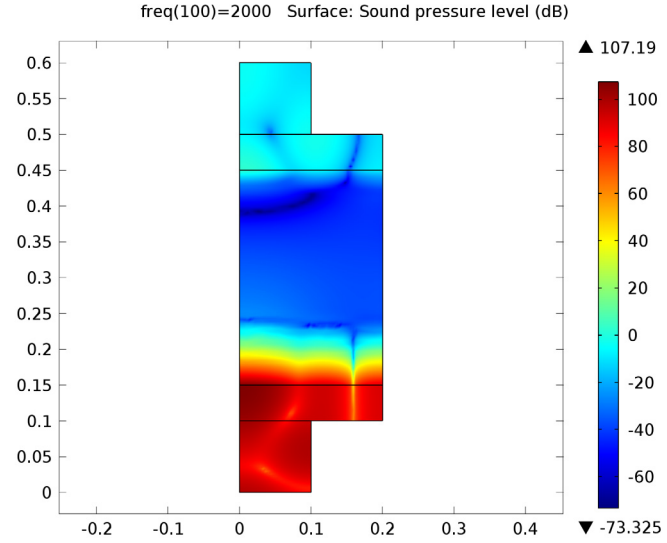


Figure 9: The sound pressure level in dB (relative to 20 mPa) at 2 kHz.

1D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 4** and choose **Global**.

3 In the **Global** settings window, locate the **y-Axis Data** section.

4 In the table, enter the following settings:

EXPRESSION
`20*log10(abs(intop_in(p)))`
`20*log10(abs(intop_out(p)))`

5 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

LEGENDS
`pressure at inlet`
`pressure at outlet`

7 Click the **Plot** button.

8 In the **Model Builder** window, click **1D Plot Group 4**.

9 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

10 Select the **x-axis label** check box.

11 In the associated edit field, type `Frequency (Hz)`.

12 Select the **y-axis label** check box.

13 In the associated edit field, type `Sound pressure level (dB)`.

14 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

15 Click the **Plot** button.

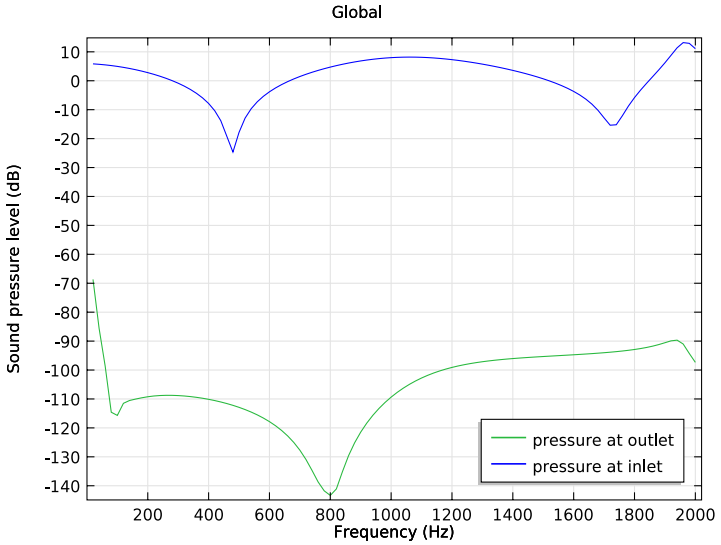


Figure 10: Sound pressure levels in dB (relative to 1 Pa) at the inlet (blue line) and the outlet (green line).

1D Plot Group 5

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 5** and choose **Global**.

3 In the **Global** settings window, locate the **y-Axis Data** section.

4 In the table, enter the following settings:

EXPRESSION
`20*log10(abs(intop_in(p0)/intop_out(p)))`

5 Locate the **Legends** section. Clear the **Show legends** check box.

6 In the **Model Builder** window, click **1D Plot Group 5**.

 ©2012 COMSOL

7 In the **1D Plot Group** settings window, locate the **Title** section.

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

9 Locate the **Plot Settings** section. Select the **x-axis label** check box.

10 In the associated edit field, type `Frequency (Hz)`.

11 Select the **y-axis label** check box.

12 In the associated edit field, type `Transmisson loss (dB)`.

13 Click the **Plot** button.

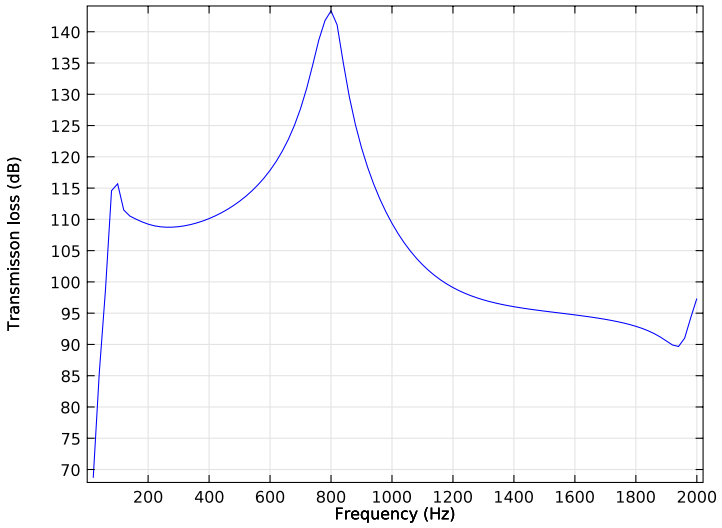


Figure 11: Transmission loss versus frequency.

 ©2012 COMSOL