Automotive Muffler

Introduction

This example describes the pressure wave propagation in a muffler for an explosion engine. The approach is general for analysis of damping of harmonic pressure waves.

The purpose of the application is to show how to treat 3D acoustics in a fairly complex geometry consisting of several separate sections and pipes divided by thin perfectly rigid walls. The analysis gives the transmission loss in the frequency range 100 Hz–1000 Hz.

Model Definition

The model geometry consists of three separate resonator chambers divided by thin walls. The inlet and the outlet correspond to the connection in the direction of the engine and of free air, respectively.

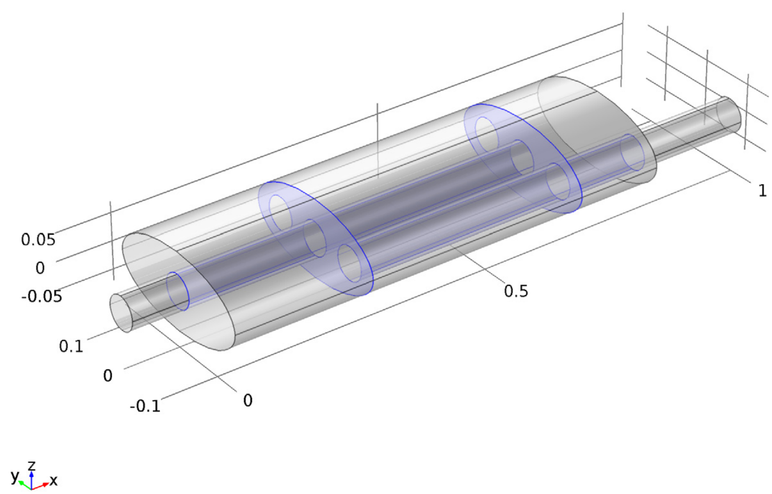


Figure 1: The geometry of a muffler. The exhaust fumes enter through the left pipe, pass the three resonator chambers, and exit through the right pipe.

DOMAIN EQUATIONS

You solve the problem in the frequency domain using the Pressure Acoustics, Frequency Domain interface. The model equation is a slightly modified Helmholtz equation for the acoustic pressure p:

$$\nabla \cdot \left(-\frac{\nabla p}{\rho}\right) - \frac{\omega^2 p}{c^2 \rho} = 0$$

where ρ is the density, c is the speed of sound, and ω is the angular frequency. The density needs to be included in the equation in cases where variations in density in different materials exist. The model assumes that in the low-frequency range, reactive damping prevails. Resistive damping is therefore not included.

BOUNDARY CONDITIONS

The boundary conditions are of three different types. At all the solid boundaries, which include the outer walls of the muffler, the dividing walls between the resonator chambers, and the walls of the pipes, sound hard (wall) boundary conditions are used:

$$\left(-\frac{\nabla p}{\rho}\right) \cdot \mathbf{n} = 0$$

At the inlet boundary, a combination of an incoming and an outgoing plane waves is assumed:

$$\left(-\frac{\nabla p}{\rho}\right) \cdot \mathbf{n} = \frac{i\omega}{\rho c} p - \frac{2 i\omega}{\rho c} p_0$$

In this equation p_0 denotes the applied outer pressure and i the imaginary unit. At the outlet boundary, an outgoing plane wave is set:

$$\left(-\frac{\nabla p}{\rho}\right) \cdot \mathbf{n} = \frac{i\omega}{\rho c} p$$

Results and Discussion

Figure 2 visualizes the pressure field in the muffler at a frequency of 490 Hz using a boundary plot of the absolute value of the pressure and an isosurface plot of the pressure.

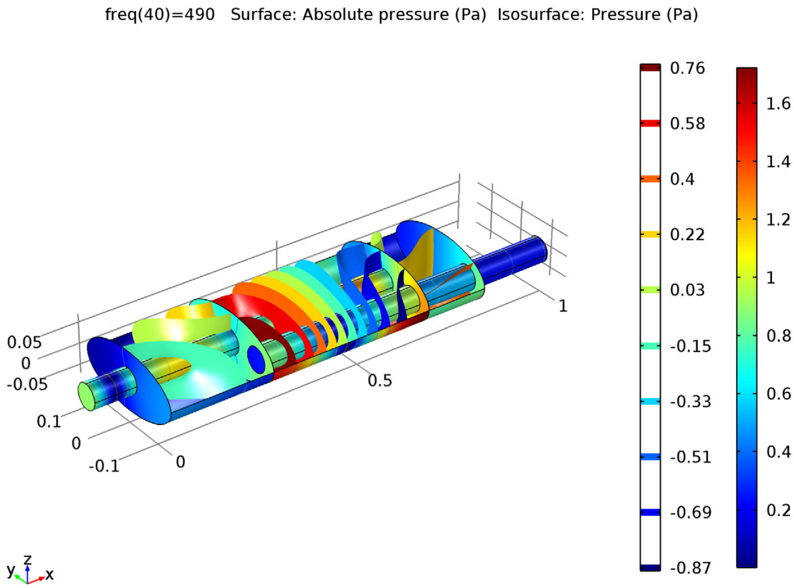


Figure 2: The solution at 490 Hz. The real value of the pressure is plotted as isosurfaces, and the absolute value of the pressure is displayed as a boundary plot on the inner walls of the muffler.

The following equation defines the transmission loss in the muffler:

$$\mathrm{TL} = 10\log\left(\frac{P_{\mathrm{in}}}{P_{\mathrm{out}}}\right)$$

Here, P_{in} and P_{out} denote the acoustic effect at the inlet and outlet, respectively. The acoustic effect is calculated using the following equations:

$$P_{\mathrm{in}} = \int_{\partial\Omega} \frac{p_0^{\ 2}}{2\rho c} dA$$

$$P_{\mathrm{out}} = \int_{\partial\Omega} \frac{|p_c|^2}{2\rho c} dA$$

Figure 3 shows the result of a parametric frequency study. This plot reveals that the damping is better at higher frequencies, with the exception of several deep dips throughout the frequency range. The dips correspond to the resonance frequencies for different parts of the muffler system.

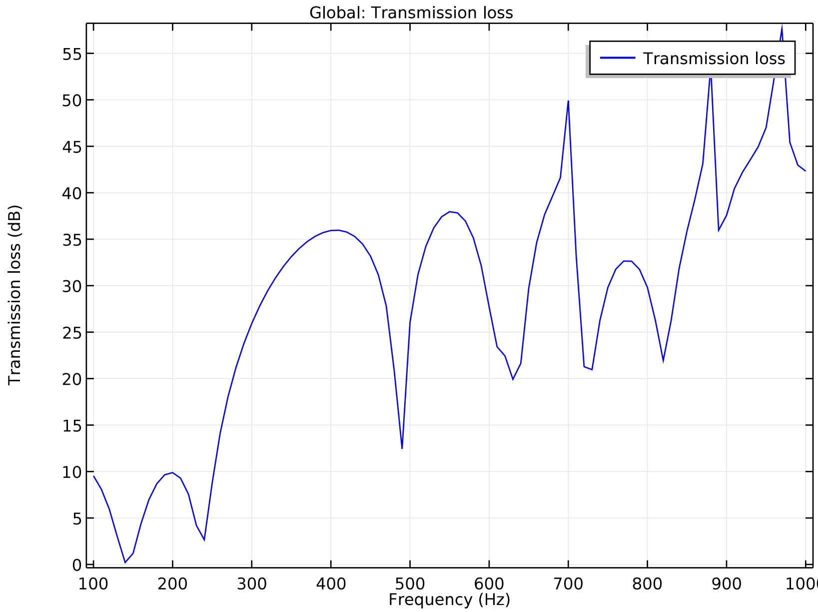


Figure 3: The damping (dB) in the muffler as a function of the frequency (Hz).

Application Library path: `COMSOL_Multiphysics/Acoustics/automotive_muffler`

Modeling Instructions

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

NEW

1 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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

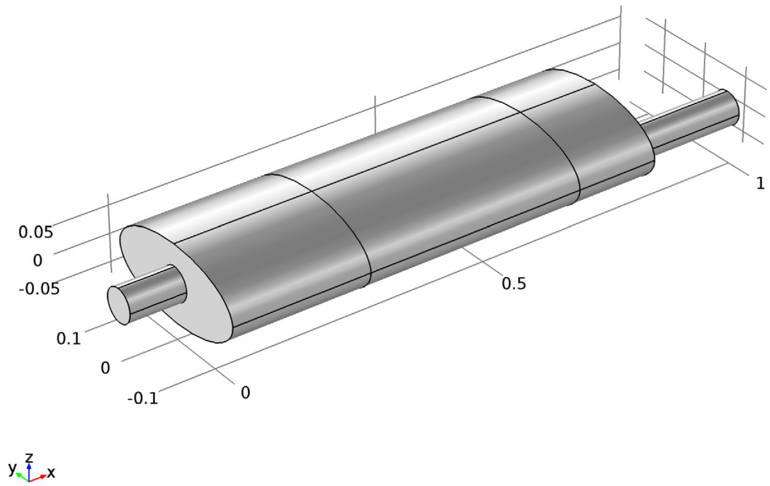
Name	Expression	Value	Description
p0	1[Pa]	1 Pa	Inlet pressure amplitude

Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 On the **Geometry** toolbar, click **Import/Export** and choose **Insert Sequence**.

2 Browse to the application's Application Library folder and double-click the file automotive_mufler.mph.

3 Click **Build all** on the **Geometry** toolbar.



DEFINITIONS

Define integration operators for the inlet and outlet, then use these to calculate the attenuation.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component 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 1 only.

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

6 Right-click **Component 1 (comp1)>Definitions>Integration 1 (intop1)** and choose **Rename**.

7 In the **Rename Integration** dialog box, type `inlet` in the **New label** text field.

8 Click **OK**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component 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 50 only.

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

6 Right-click **Component 1 (comp1)>Definitions>Integration 2 (intop2)** and choose **Rename**.

7 In the **Rename Integration** dialog box, type `outlet` in the **New label** text field.

8 Click **OK**.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
P_in	intop_inlet(p0^2/(2*acpr.rho*acpr.c))	W	Incoming power
P_out	intop_outlet(p*conj(p)/(2*acpr.rho*acpr.c))	W	Outgoing power
TL	10*log10(P_in/P_out)		Transmission loss

Note that you need to define the incoming power using p0 rather than the pressure variable, p, because p is the sum of incident and reflected pressure waves.

Create a selection to simplify setting up the interior boundary conditions.

Explicit 1

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

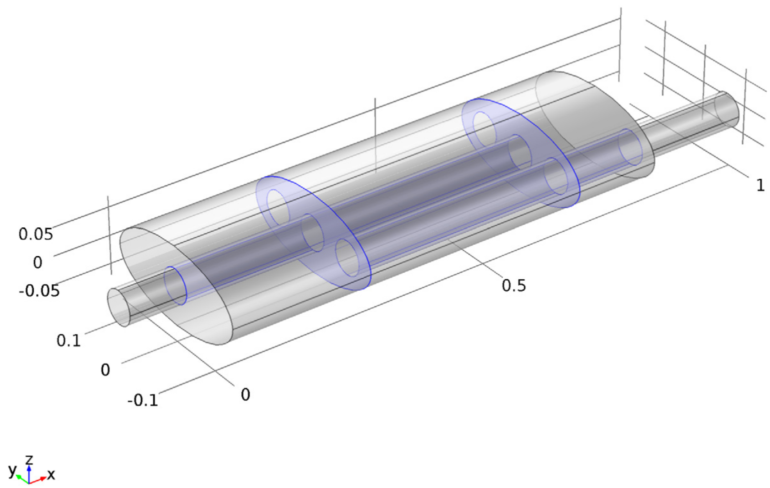
2 In the **Settings** window for Explicit, locate the **Input Entities** section.

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

4 Click the **Transparency** button on the **Graphics** toolbar.

5 Select Boundaries 10, 11, 13, 14, 16, 20–23, 25, 26, 28, 29, 32, and 36–39 only.

To do this, click the **Paste Selection** button next to the **Selection** box, paste the text: 10, 11, 13, 14, 16, 20–23, 25, 26, 28, 29, 32, 36–39 in the text field of the dialog box that opens, and finally click **OK**.



6 Right-click **Component 1 (comp1)>Definitions>Explicit 1** and choose **Rename**.

7 In the **Rename Explicit** dialog box, type `interior boundaries` in the **New label** text field.

8 Click **OK**.

ADD MATERIAL

1 On the **Home** 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 **Add to Component** in the window toolbar.

MATERIALS

By default, the first material you add applies for all domains. In the Material Contents table you can see which material properties the physics interfaces use, in this case the density and the speed of sound. Notice that these quantities are functions of the temperature and (in the case of the density) the ambient pressure.

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics 1

In the **Model Inputs** section you can read off and, if desired, modify the temperature and absolute pressure at which the expressions for the air density and speed of sound are calculated. For this model, use the default settings.

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundary 1 only.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Settings** window for Incident Pressure Field, locate the **Incident Pressure Field** section.

3 In the p_0 text field, type `p0`.

Plane Wave Radiation 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundary 50 only.

3 Click the **Transparency** button on the **Graphics** toolbar to return to the default transparency state.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 In the **Settings** window for Interior Sound Hard Boundary (Wall), locate the **Boundary Selection** section.

3 From the **Selection** list, choose **interior boundaries**.

MESH 1

Use the default physics-controlled mesh as a starting point and then modify the maximum element size so that you get 10 elements per wavelength for the highest frequency in the sweep, that is 1 kHz.

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

Size

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

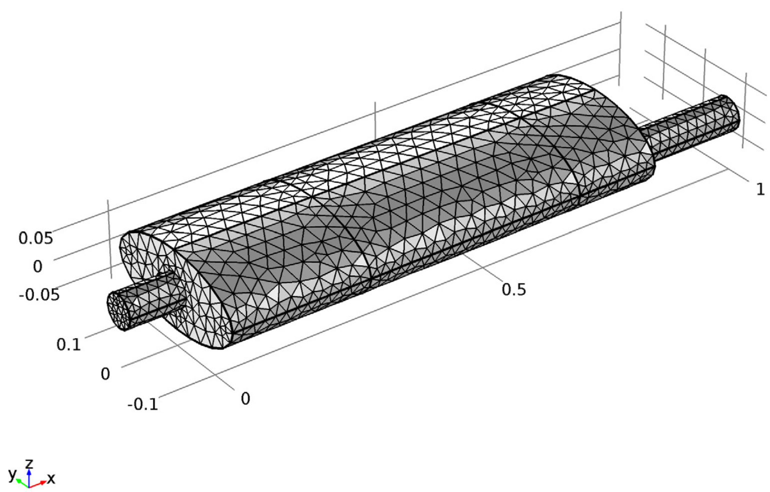
3 In the **Settings** window for Size, locate the **Element Size** section.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `343[m/s]/1[kHz]/10`.

This corresponds to one 10th of the shortest wavelength.

6 Click the **Build All** button.



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

3 In the **Frequencies** text field, type `range(100,10,1000)`.

   This computes the solution for 91 equally spaced frequencies from 100 Hz to 1000 Hz. If you want to run a faster analysis, try the same frequency range but with a step of 100 Hz instead (to do so, type `range(100,100,1000)`).

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

Before visualizing the acoustic pressure field, add a selection to the default solution data set that filters out the upper muffler boundaries for a better view.

Data Sets

1 On the **Results** toolbar, click **Selection**.

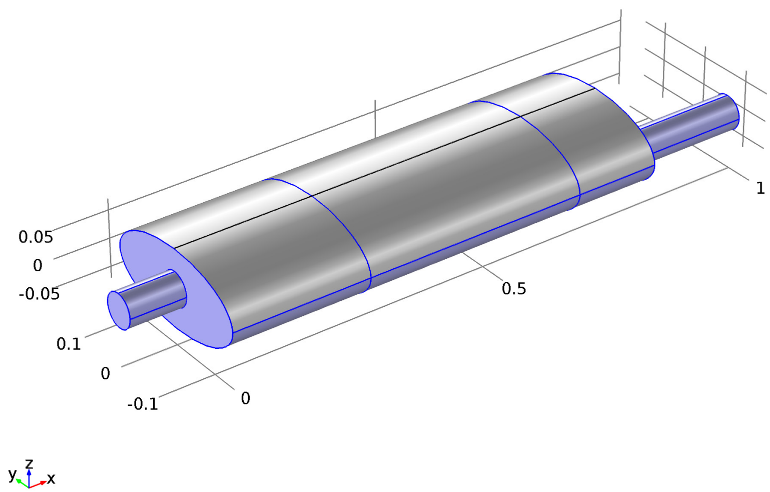
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **All boundaries**.

5 Ctrl+click to highlight the six top faces of the muffler, then right-click to remove them from the selection.

Alternatively, you can click the **Paste Selection** button next to the **Selection** box and then paste the text "1–7, 9–14, 16, 17, 19–29, 31–33, 35–41, 43–50" in the text field of the dialog box that opens before clicking **OK**.



Acoustic Pressure (acpr)

Reproduce the plot in Figure 2 by following these steps.

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **490**.

4 In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.

5 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.

6 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

7 In the **Model Builder** window, right-click **Acoustic Pressure (acpr)** and choose **Isosurface**.

8 In the **Settings** window for Isosurface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>p - Pressure**.

9 Locate the **Levels** section. In the **Total levels** text field, type `10`.

What is a suitable number of isosurface levels for the isosurface plot varies with the frequency. At frequencies with low damping many of the isosurfaces tend to congregate inside the pipe.

10 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

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

Finally, reproduce the plot of attenuation versus frequency shown in Figure 3.

1D Plot Group 4

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

2 On the **1D Plot Group 4** toolbar, click **Global**.

3 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>TL - Transmission loss**.

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

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

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

7 In the associated text field, type `Frequency (Hz)`.

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

9 In the associated text field, type `Transmission loss (dB)`.

10 On the **1D Plot Group 4** toolbar, click **Plot**.

Notice the deep dip in the damping around 490 Hz caused by the resonance in the second chamber. If you plot the pressure in the muffler at other dips, resonances in the other chambers appear.