Absorptive Muffler

Introduction

This model describes the pressure-wave propagation in a muffler for an internal combustion engine. The approach is generally applicable to analyzing the damping of propagation of harmonic pressure waves.

The model's purpose is to show how to analyze both inductive and resistive damping in pressure acoustics. The main output is the transmission loss for the frequency range 50 Hz–1500 Hz.

See also Eigenmodes in a Muffler, which computes the propagating modes in the chamber.

Model Definition

The muffler—schematically depicted in Figure 1—consists of a 24-liter resonator chamber with a section of the centered exhaust pipe included at each end. In the first version of the model the chamber is empty. In the second version it is lined with 15 mm of absorbing glass wool.

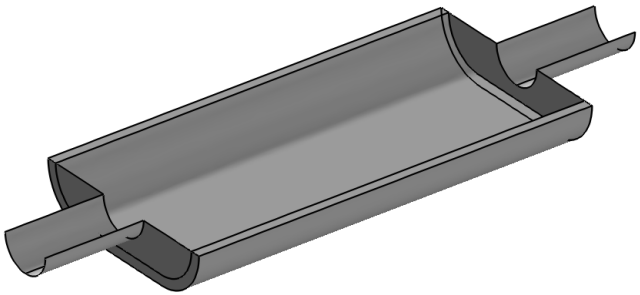


Figure 1: Geometry of the lined muffler with the upper half removed. The exhaust fumes enter through the left pipe and exit through the right pipe.

DOMAIN EQUATIONS

This model solves the problem in the frequency domain using the time-harmonic Pressure Acoustics interface. The model equation is a slightly modified version of the Helmholtz equation for the acoustic pressure, p:

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

where ρ is the density, c_s equals the speed of sound, and ω gives the angular frequency.

In the absorbing glass wool, the damping enters the equation as a complex speed of sound, $c_c = \omega/k_c$, and a complex density, $\rho_c = k_c Z_c/\omega$, where k_c is the complex wave number and Z_c equals the complex impedance.

For a highly porous material with a rigid skeleton, the well-known model of Delany and Bazley estimates these parameters as functions of frequency and flow resistivity. Using the original coefficients of Delany and Bazley (Ref. 1), the expressions are

$$k_c = k_a \cdot \left(1 + 0.098 \cdot \left(\frac{\rho_a f}{R_f}\right)^{-0.7} - i \cdot 0.189 \cdot \left(\frac{\rho_a f}{R_f}\right)^{-0.595}\right)$$

$$Z_c = Z_a \cdot \left(1 + 0.057 \cdot \left(\frac{\rho_a f}{R_f}\right)^{0.734} - i \cdot 0.087 \cdot \left(\frac{\rho_a f}{R_f}\right)^{-0.732}\right)$$

where R_f is the flow resistivity, and where $k_a = \omega/c_a$ and $Z_a = \rho_a c_a$ are the free-space wave number and impedance of air, respectively. You can find flow resistivities in tables. For glass-wool-like materials, Bies and Hansen (Ref. 2) give an empirical correlation

$$R_f = \frac{3.18 \cdot 10^{-9} \cdot \rho_{ap}^{1.53}}{d_{av}^2}$$

where ρ_{ap} is the material's apparent density and d_{av} is the mean fiber diameter. This model uses a rather lightweight glass wool with $\rho_{ap} = 12$ kg/m^3 and $d_{av} = 10$ µm.

BOUNDARY CONDITIONS

The boundary conditions are of three types.

- At the solid boundaries, which are the outer walls of the resonator chamber and the pipes, the model uses sound hard (wall) boundary conditions. The condition imposes that the normal velocity at the boundary is zero, and is specified mathematically by:

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

- The boundary condition at the inlet involves a combination of incoming and outgoing plane waves:

©2013 COMSOL

$$\mathbf{n} \cdot \frac{1}{\rho_0}\nabla p + ik\frac{p}{\rho_0} + \frac{i}{2k}\Delta_T p = \left(\frac{i}{2k}\Delta_T p_0 + (1 - (\mathbf{k} \cdot \mathbf{n}))ik\frac{p_0}{\rho_0} \right) e^{-ik(\mathbf{k} \cdot \mathbf{r})}$$

In this equation, p_0 represents the applied outer pressure, Δ_T is the boundary tangential Laplace operator, and i equals the imaginary unit (see Ref. 3). This boundary condition is valid as long as the frequency is kept below the cutoff frequency for the second propagating mode in the tube.

- At the outlet boundary, the model specifies an outgoing plane wave:

$$\mathbf{n} \cdot \frac{1}{\rho_0}\nabla p + i\frac{k}{\rho_0}p + \frac{i}{2k}\Delta_T p = 0$$

Results and Discussion

The pressure distribution in the absorptive muffler without the lining material is depicted in Figure 2 for the frequency f=1500 Hz. From the figure it is seen that at this frequency not only longitudinal standing waves exist but also transverse modes are present.

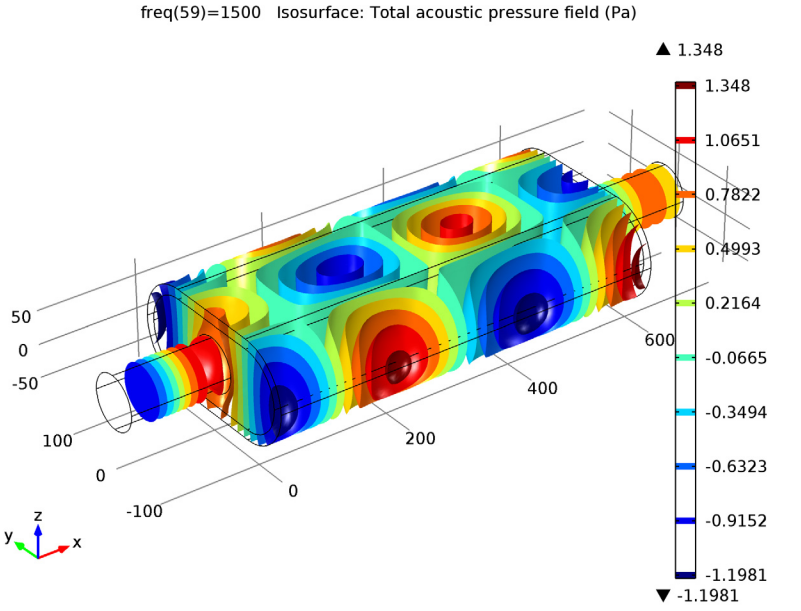


Figure 2: Pressure represented as an iso-barometric surface plot in the muffler.

An important parameter for a muffler is the transmission loss or attenuation. It is defined as the ratio between the incoming and outgoing acoustic energy. The attenuation d_w (in dB) of the acoustic energy is defined by the following equation:

$$d_w = 10\log\left(\frac{w_{\text{in}}}{w_{\text{out}}}\right)$$

Here w_{in} and w_{out} denote the incoming power at the inlet and the outgoing power at the outlet, respectively. You can calculate each of these quantities as an integral over the corresponding surface:

$$w_{\text{out}} = \int_{\partial\Omega} \frac{|p|^2}{2\rho c_s} dA$$

$$w_{\text{in}} = \int_{\partial\Omega} \frac{p_0^2}{2\rho c_s} dA$$

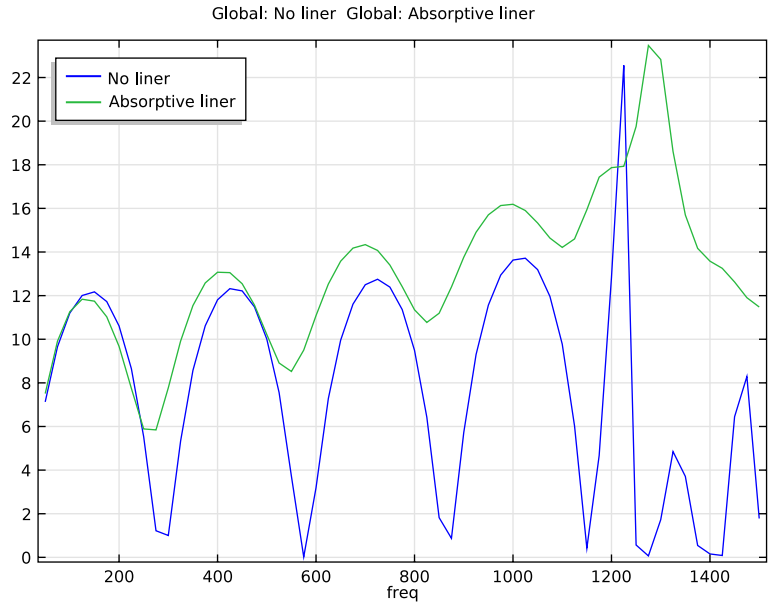


Figure 3: Comparison of the attenuation (dB) curves as function of frequency for the empty muffler and the muffler with absorptive lining. The first four dips are due to longitudinal resonances. In the muffler with absorbing lining the dips are still present, but the general trend is that the higher the frequency, the better the damping.

©2013 COMSOL

Figure 3 shows the result of a parametric frequency study. The two graphs represent the case of an empty muffler without any absorbing lining material (blue curve) and the case with a layer of glass wool lining on the chamber's walls (green line).

The graph for the undamped muffler shows that damping works rather well for most low frequencies. At frequencies higher than approximately 1250 Hz, the plot's behavior is more complicated and there is generally less damping. This is because, for such frequencies, the tube supports not only longitudinal resonances but also cross-sectional propagation modes. Not very far above this frequency a whole range of modes that are combinations of this propagation mode and the longitudinal modes participate, making the damping properties increasingly unpredictable. For an analysis of these modes, see the related model Eigenmodes in a Muffler. The glass-wool lining improves attenuation at the resonance frequencies as well as at higher frequencies.

Notes About the COMSOL Implementation

This model uses the Pressure Acoustics interface of the Acoustics Module. This interface has the Delany-Bazley coefficients built in. Therefore, the only damping parameter you need to supply is the flow resistivity.

The parametric solver provides results for a range of frequencies. The software computes integrals in the power expressions using boundary integration coupling variables, and it plots the resulting attenuation versus frequency.

The instructions take you through two versions of the model, first one with a completely hollow chamber, then one where the chamber is lined with glass wool.

References

1. M.A. Delany and E.N. Bazley, "Acoustic Properties of Fibrous Absorbent Materials," *Appl. Acoust.*, vol. 3, pp. 105–116, 1970.

2. D.A. Bies and C.H. Hansen, "Flow Resistance Information for Acoustical Design," *Appl. Acoust.*, vol. 13, Issue 5, September-October 1980, pp. 357–391.

3. D. Givoli and B. Neta, "High-order Non-reflecting Boundary Scheme for Time-dependent Waves," *J. Comp. Phys.*, vol. 186, pp. 24–46, 2003.

Model Library path: `Acoustics_Module/Industrial_Models/absorptive_muffler`

Modeling Instructions—Rigid Walls

The instructions take you through two versions of the model, first one with a completely hollow chamber with rigid walls, then one where the chamber is lined with glass wool.

MODEL WIZARD

1. Go to the **Model Wizard** window.
2. Click **Next**.
3. In the **Add physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
4. Click **Next**.
5. Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
6. 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 `absorptive_muffler_parameters.txt`.

   The parameters define the physical values and the geometrical dimensions of the system. The geometry is now parameterized and simply changing the value of a dimension in the parameters list will update the geometry automatically.

GEOMETRY 1

1. In the **Model Builder** window, under **Model 1** click **Geometry 1**.
2. In the **Geometry** settings window, locate the **Units** section.
3. From the **Length unit** list, choose **mm**.

 ©2013 COMSOL

Work Plane 1

1 Right-click **Model 1>Geometry 1** and choose **Work Plane**.

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

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

Rectangle 1

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

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

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

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

5 Locate the **Position** section. In the **xw** edit field, type `-W/2`.

6 In the **yw** edit field, type `-H/2`.

Fillet 1

1 Right-click **Plane Geometry** and choose **Fillet**.

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

3 On the object **r1**, select Points 1–4 only.

4 In the **Fillet** settings window, locate the **Radius** section.

5 In the **Radius** edit field, type `H/2`.

Rectangle 2

1 Right-click **Plane Geometry** and choose **Rectangle**.

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

3 In the **Width** edit field, type `W-2*D`.

4 In the **Height** edit field, type `H-2*D`.

5 Locate the **Position** section. In the **xw** edit field, type `-(W-2*D)/2`.

6 In the **yw** edit field, type `-(H-2*D)/2`.

Fillet 2

1 Right-click **Plane Geometry** and choose **Fillet**.

2 On the object **r2**, select Points 1–4 only.

3 In the **Fillet** settings window, locate the **Radius** section.

4 In the **Radius** edit field, type `(H-2*D)/2`.

5 Click the **Build All** button.

©2013 COMSOL

Extrude 1

1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Work Plane 1** and choose **Extrude**.

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (mm)
L

Cylinder 1

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

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

3 In the **Radius** edit field, type `R_io`.

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

5 Locate the **Position** section. In the **x** edit field, type `-L_io`.

6 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

7 In the **x** edit field, type `1`.

8 In the **z** edit field, type `0`.

Cylinder 2

1 Right-click **Geometry 1** and choose **Cylinder**.

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

3 In the **Radius** edit field, type `R_io`.

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

5 Locate the **Position** section. In the **x** edit field, type `L`.

6 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

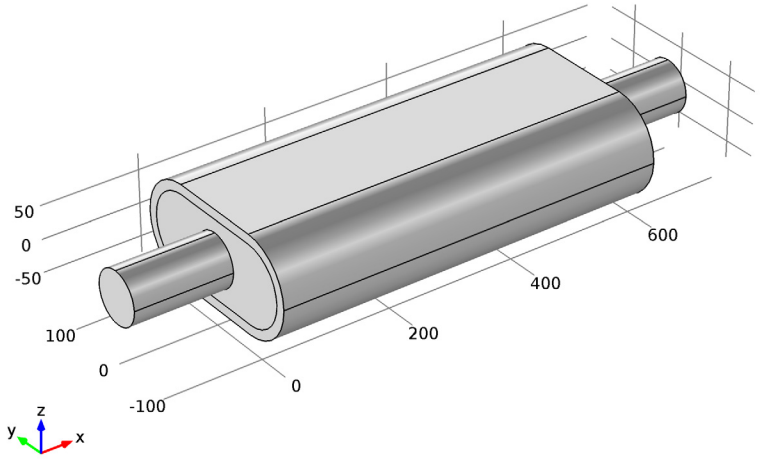
7 In the **x** edit field, type `1`.

8 In the **z** edit field, type `0`.

9 Click the **Build All** button.

©2013 COMSOL

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



DEFINITIONS

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 1** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Inlet` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

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

7 Select Boundary 1 only.

Explicit 2

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

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

3 Go to the **Rename Explicit** dialog box and type `Outlet` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

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

7 Select Boundary 28 only.

Integration 1

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

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

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

4 From the **Selection** list, choose **Inlet**.

Integration 2

1 Right-click **Definitions** and choose **Model Couplings>Integration**.

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

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

4 From the **Selection** list, choose **Outlet**.

You have now defined the integration coupling operators `intop1` and `intop2`, for integration over the inlet and outlet respectively. Use these operators in defining the in- and outgoing power, as defined in the Results and Discussion section.

Variables 1

1 Right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Description
w_in	intop1(p0^2/(2*acpr.rho*acpr.c))	Power of the incoming wave
w_out	intop2(abs(p)^2/(2*acpr.rho*acpr.c))	Power of the outgoing wave

MATERIALS

Material Browser

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

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click **Add Material to Model**.

 ©2013 COMSOL

Air

By default the first material you add applies on all domains so you need not alter the geometric scope settings.

In the second version of this model, you will insert a lining material in Domain 2. For now, the muffler is completely hollow.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN

Plane Wave Radiation 1

1. In the **Model Builder** window, under **Model 1** right-click **Pressure Acoustics, Frequency Domain** and choose **Plane Wave Radiation**.
2. Select Boundaries 1 and 28 only.

Incident Pressure Field 1

1. Right-click **Model 1>Pressure Acoustics, Frequency Domain>Plane Wave Radiation 1** and choose **Incident Pressure Field**.
2. In the **Incident Pressure Field** settings window, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet**.
4. Locate the **Incident Pressure Field** section. In the p_0 edit field, type `p0`.

   You have now specified the plane wave radiation condition to be active on both the inlet and outlet boundaries, but with an incident wave on the inlet. The remaining boundaries by default use the sound hard condition.

Now, add a second pressure acoustics model for the absorptive liner domain. You will deactivate this domain when configuring the first study step.

Pressure Acoustics Model 2

1. In the **Model Builder** window, right-click **Pressure Acoustics, Frequency Domain** and choose **Pressure Acoustics Model**.
2. Select Domain 2 only.
3. In the **Pressure Acoustics Model** settings window, locate the **Pressure Acoustics Model** section.
4. From the **Fluid model** list, choose **Macroscopic empirical porous models**.
5. In the R_f edit field, type `R_f`.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

To more easily locate and select a boundary use the wireframe rendering option for the graphics.

2 Click the **Wireframe Rendering** button on the Graphics toolbar.

3 Select Boundaries 6, 9, and 16 only.

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 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `343[m/s]/1500[Hz]/5`.

The global maximum element size is set equal to the minimal wavelength divided by 5, that is, $\lambda/5 = c/f_{\max}/5$, where c is the speed of sound.

5 Click the **Build All** button.

Swept 1

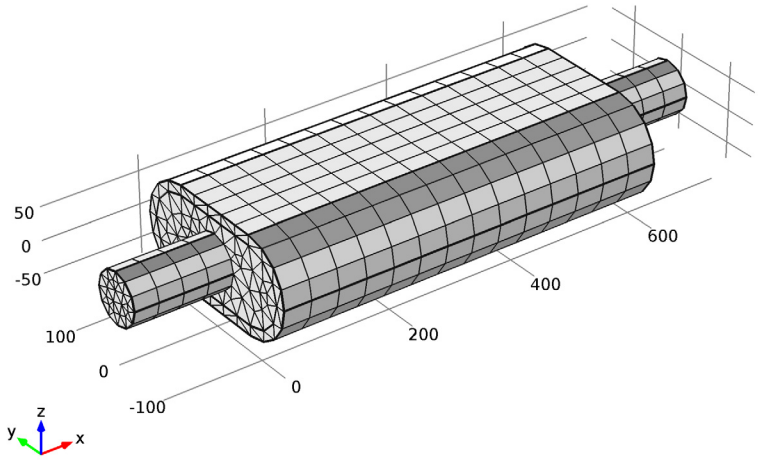
1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Now, simply push the **Build All** button and the mesher will automatically detect source and destination boundaries for the swept mesh.

2 In the **Settings** window, click **Build All**.

©2013 COMSOL

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



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(50,25,1500)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Model 1>Pressure Acoustics, Frequency Domain>Pressure Acoustics Model 2**.

6 Click **Disable**.

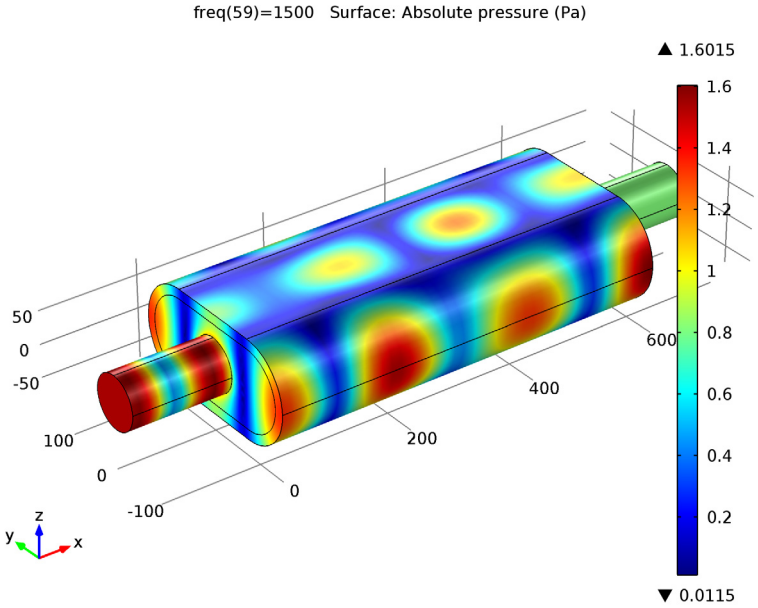
7 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Acoustic Pressure (acpr)

The first one of the default plots shows the pressure distribution on the walls of the muffler at the highest frequency, 1500 Hz. To get a better view of the standing wave pattern, you can plot the norm of the pressure instead of the real part of the pressure.

1. In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.
2. In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>Absolute pressure (acpr.absp)**.
3. Click the **Plot** button.

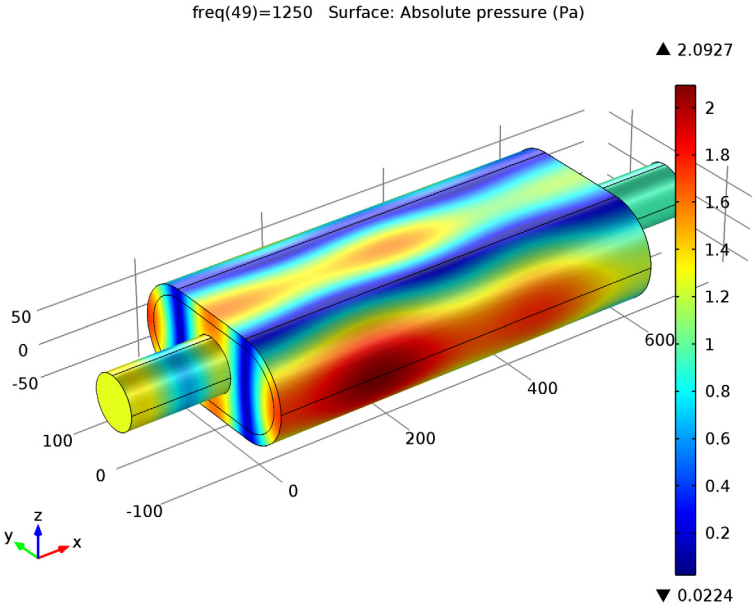


The pattern is very different at different frequencies. See for example what happens at 1250 Hz.

4. In the **Model Builder** window, click **Acoustic Pressure (acpr)**.
5. In the **3D Plot Group** settings window, locate the **Data** section.
6. From the **Parameter value (freq)** list, choose **1250**.

©2013 COMSOL

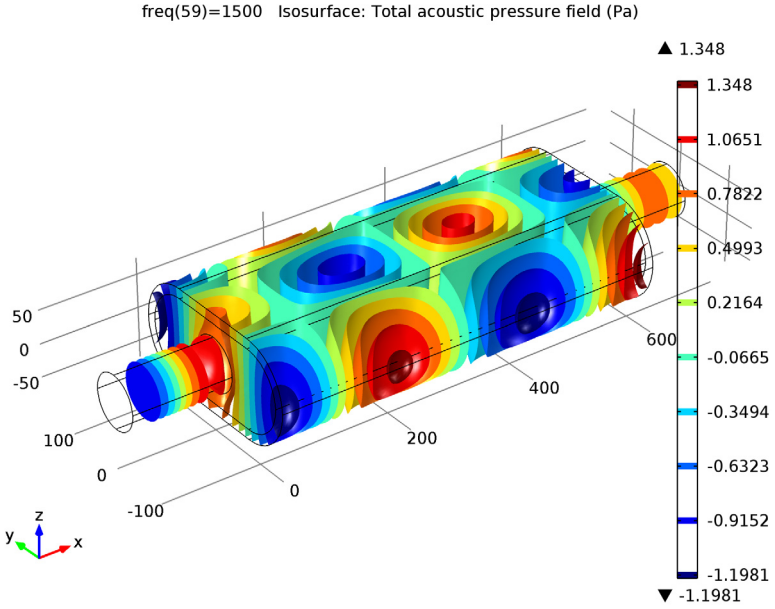
7 Click the **Plot** button.



At 1250 Hz, the absolute value of the pressure does not vary much with the x-coordinate. The reason is that this is just higher than the cutoff frequency for the first symmetric propagating mode, which is excited by the incoming wave. For a separate analysis of the propagating modes in the chamber, see the Eigenmodes in a Muffler model.

The two other default plot groups show the sound pressure level on the wall surface and the pressure inside the muffler as isosurfaces.

Acoustic Pressure, Isosurfaces (acpr)



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	Unit	Description
`10*log10(w_in/w_out)`		`No liner`

5 Click the **Plot** button.

The plot should be a reproduction of the blue curve in Figure 3.

Modeling Instructions—Lined Absorbing Walls

In this, the second version of the model, you will solve the model including a layer of absorptive glass wool on the muffler line. Continue working from where you left off with the model developed thus far and add a second study to keep your existing results intact.

 ©2013 COMSOL

ROOT

1 In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

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

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

3 Click **Finish**.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

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

3 Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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(50,25,1500)`.

4 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Acoustic Pressure, Isosurfaces (acpr)

You chose not to have new default plots generated. Once the solution process is finished you can use the existing plot groups and just switch the data set to see how the damping material affects the solution.

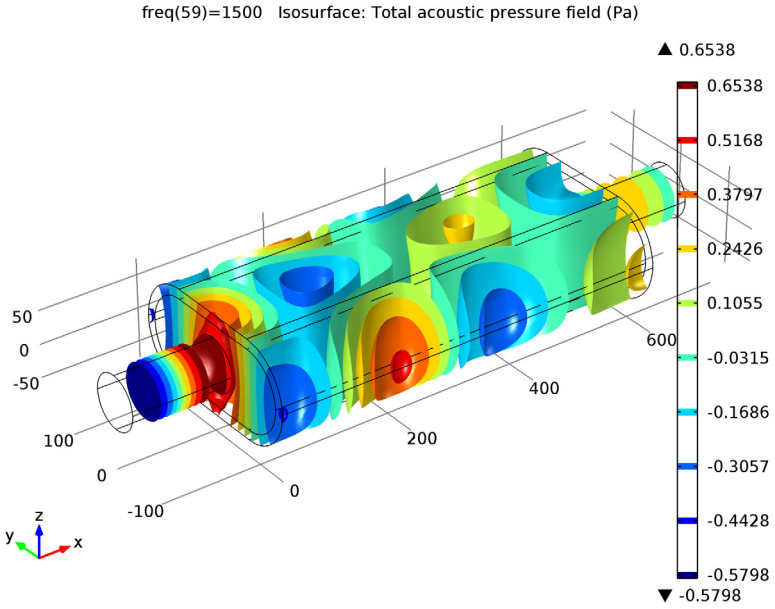
1 In the **Model Builder** window, click **Acoustic Pressure, Isosurfaces (acpr)**.

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

3 From the **Data set** list, choose **Solution 2**.

©2013 COMSOL

4 Click the **Plot** button.



At 1500 Hz, the pressure in the chamber is much lower than before. Proceed to study how the transmission has changed.

1D Plot Group 4

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Global**.

2 In the **Global** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`10*log10(w_in/w_out)`		`Absorptive liner`

5 Click the **Plot** button.

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

7 In the **1D Plot Group** settings window, click to expand the **Legend** section.

8 From the **Position** list, choose **Upper left**.

9 Right-click **1D Plot Group 4** and choose **Rename**.

©2013 COMSOL

10 Go to the **Rename 1D Plot Group** dialog box and type `Transmission Loss` in the **New name** edit field.

11 Click **OK**.

The plot should now look like that in Figure 3.

Now, create a plot that represents the intensity flux through the muffler system. Use streamlines that follow the intensity vector (flux of energy through the muffler). You can change between solutions and frequencies to study and visualize the muffler's sound-absorbing properties.

3D Plot Group 5

1 Right-click **Results** and choose **3D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **3D Plot Group 5** and choose **Streamline**.

3 In the **Streamline** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Pressure Acoustics, Frequency Domain>Intensity>Intensity (RMS) (acpr.Ix,acpr.Iy,acpr.Iz)**.

4 Select Boundary 1 only.

5 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

6 In the **Tube radius expression** edit field, type `2`.

7 Right-click **Results>3D Plot Group 5>Streamline 1** and choose **Color Expression**.

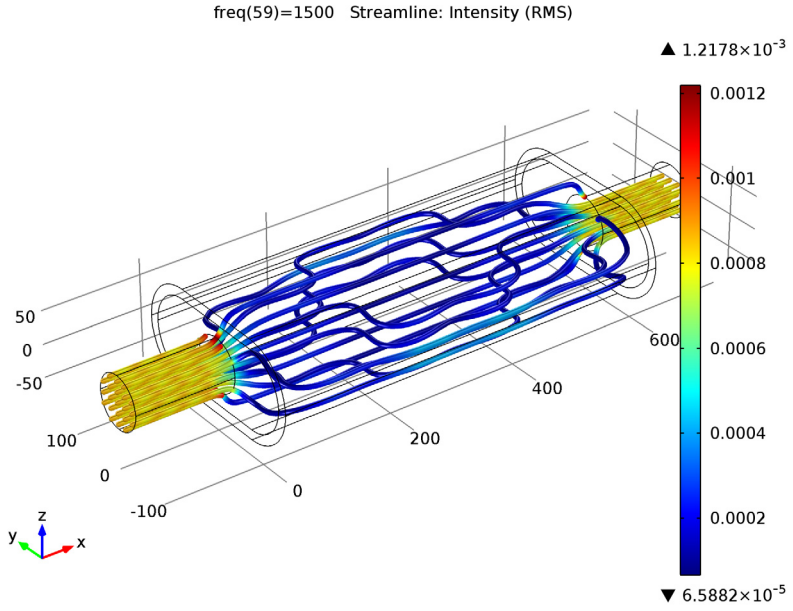
8 In the **Color Expression** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Pressure Acoustics, Frequency Domain>Intensity>Intensity magnitude (RMS) (acpr.I_rms)**.

9 Click the **Plot** button.

10 In the **Model Builder** window, right-click **3D Plot Group 5** and choose **Rename**.

11 Go to the **Rename 3D Plot Group** dialog box and type `Intensity` in the **New name** edit field.

I2 Click **OK**.



©2013 COMSOL