Model created in COMSOL Multiphysics 6.4

Transfer Impedance of a Perforate

Introduction

Perforates are plates with a distribution of small perforations or holes. They are used in muffler systems, sound absorbing panels, and in many other places as liners, where it is important to control attenuation precisely. As the perforations become smaller and smaller, the viscous and thermal losses become more important. The attenuation behavior, which is also frequency dependent, can be controlled by selecting the perforate size and distribution in a plate. Nonlinear loss mechanisms occur at higher sound levels or in the presence of a flow (through/bias or over/grazing the perforate). Only the linear effects due to viscosity and thermal conduction are studied in this tutorial model. These effects are modeled in detail using the *Thermoviscous Acoustics, Frequency Domain* interface.

Perforates have been theoretically studied for many years. Typically, analytical or semi-analytical models only apply for simple geometries. A numerical approach is necessary for systems where the holes have various cross sections, if the perforations are tapered, or if the distribution of holes is uneven or of mixed sizes.

In this model, a simple perforate with circular holes is studied. The transfer impedance, surface normal impedance, and attenuation coefficient of the system is determined. The transfer impedance determined in this detailed model can, for example, be used in a larger system simulation using the interior impedance condition that exists in the *Pressure Acoustics, Frequency Domain* interface.

Model Definition

A sketch of the perforate system simulated is depicted in Figure 1. The model uses the symmetries that exist in the geometry along the x and y directions.

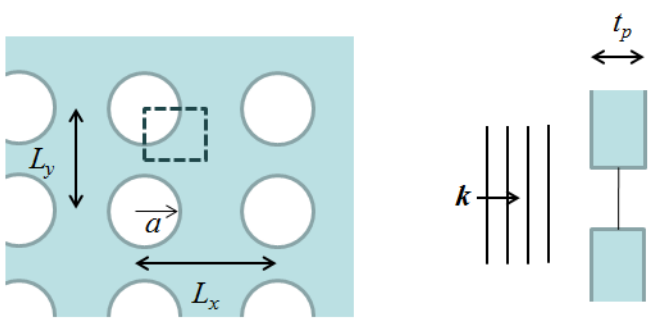


Figure 1: Schematic of the system, front and side views. The modeled region is marked by a box.

In Figure 1 the plate thickness is t_p, the hole radius is named a, the hole-hole x and y distances are named L_x and L_y, respectively. In this model, the default values are t_p = 1.0 mm, a = 0.3 mm, L_x = 1.4 mm, and L_y = 2.0 mm. A plane wave is incident from one side, modeled using the **Plane wave** port type option in the **Port** boundary condition of the Thermoviscous Acoustics, Frequency Domain physics interface.

The transfer impedance Z_{trans}, surface normal impedance Z_n, and absorption coefficient α are defined as

$$Z_{\mathrm{trans}} = \frac{\Delta p}{v_{\mathrm{hole}}} \qquad Z_{\mathrm{n}} = \frac{p}{\mathbf{v}\cdot\mathbf{n}} \qquad \alpha = 1-|R|^2 = 1-P_{\mathrm{r}} \tag{1}$$

where Δp is the pressure drop across the plate, v is the mean velocity in the perforation hole, $\mathbf{n}$ is the surface normal of the perforate, R is the reflection coefficient, and P_r is the power reflection coefficient. These expressions are defined as variables in the model (imported from a file), under the **Definitions** node.

The transfer impedance of a perforate (perforated plate) is a well-studied problem in acoustics. Several models exist: some are pure analytical and some are semi-analytical. In this tutorial model, the results obtained in the simulation are compared to a semi-analytical expression for the transfer impedance Z_{trans} presented in Ref. 3 and Ref. 4. The expression combines an expression for the linear losses including viscosity (given in Ref. 1) and an expression for nonlinear effects at high levels (given in Ref. 2). The model also includes hole-hole interaction effects (semi-empirical). In this tutorial, the COMSOL model only consider linear acoustics. We include thermal and viscous losses as well as hole-hole interaction fully. In Equation 2, retaining the nonlinear term is out of interest and shows how it can be added to an analytical expression in COMSOL Multiphysics. The magnitude of that term is very small with the chosen levels. The semi-analytical model is given by the expression

$$\begin{aligned}\frac{Z_{\mathrm{trans}}}{\rho_0 c_0} &= \left(t_{\mathrm{p}} + \frac{16a\Psi(\sqrt{\sigma})}{3\pi}\right)\frac{i\omega}{c_0\sigma\left[1-\frac{2}{k_{\mathrm{s}}a}\frac{J_1(k_{\mathrm{s}}a)}{J_0(k_{\mathrm{s}}a)}\right]} + \frac{1.2(1-\sigma^2)}{2c_0(\sigma C_{\mathrm{d}})^2}U_{\mathrm{rms}} \\ k_{\mathrm{s}}^2 &= -i\rho_0\omega/\mu\end{aligned} \tag{2}$$

where ρ_0 is the fluid density, c_0 the speed of sound, μ is the dynamic viscosity, J_n is the Bessel function of the first kind, k_s is the shear (viscous) wave number, U_{rms} the rms acoustic velocity in the hole, σ the porosity, and C_d is the discharge coefficient (a typical value is 0.76). Ψ is the so-called Fok function is defined by

$$\Psi(\sqrt{\sigma}) = 1 - 1.41(\sqrt{\sigma}) + 0.34(\sqrt{\sigma})^3 + 0.07(\sqrt{\sigma})^5 - 0.02(\sqrt{\sigma})^6 + 0.03(\sqrt{\sigma})^7 - 0.016(\sqrt{\sigma})^8 \tag{3}$$

The transfer impedance expression in Equation 2 does not include thermal effects. These can be added by modifying the linear part of the transfer impedance expression. The thermal effects are small for thin perforated plate (in the small $k \cdot t_p$ limit) where shear and viscous losses dominate, see Ref. 5.

Results and Discussion

The total acoustic pressure in the system is depicted in Figure 2 evaluated at 20 kHz. The acoustic particle velocity is depicted in Figure 3 at two different frequencies (400 Hz and 20 kHz). From the figure, the viscous boundary layer is easily seen as well as its frequency dependence (decreases with increasing frequency). Moreover, the end correction extent is also visualized here, as the part of the fluid moving axially, that extends out of the orifice. In Figure 4, the acoustic temperature variations can be seen. Here, as for the viscous boundary layer, the thermal boundary layer can be visualized.

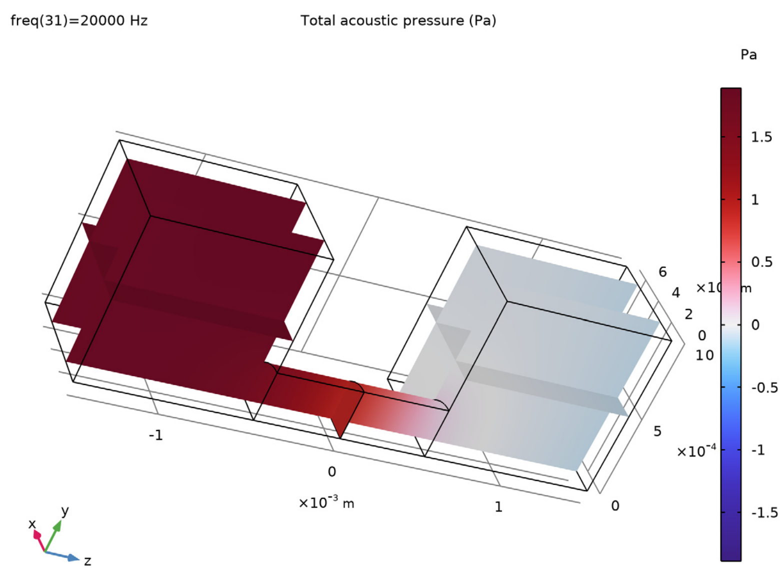


Figure 2: Pressure distribution in the system.

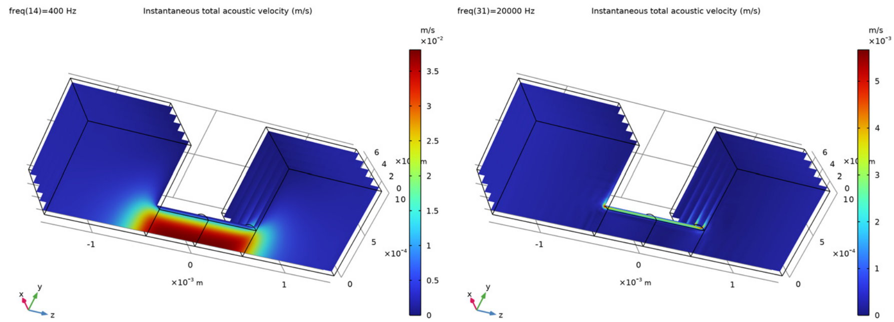


Figure 3: Velocity distribution at 400 Hz and 20 kHz.

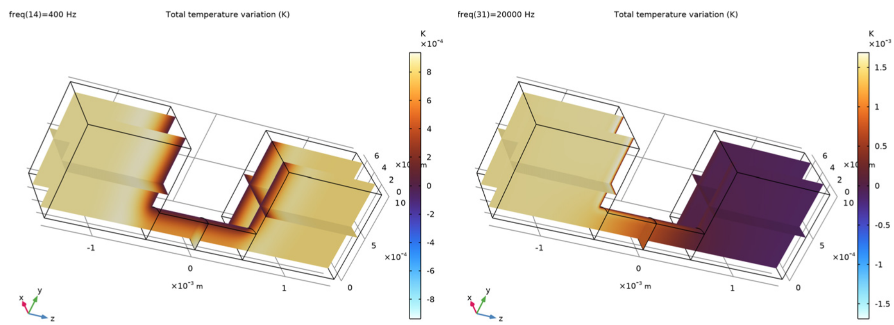


Figure 4: Temperature distribution at 400 Hz and 20 kHz.

The simulated value of the transfer impedance is depicted in Figure 5 and compared to the semi-analytical model given by Equation 2. The results are seen to coincide well. On the logarithmic y-axis scale, there is a constant difference between the curves. This corresponds to a factor multiplied on the linear scale. This could be due to the end correction and the hole-hole interaction modeled with the Fok function. In the present case, the holes are not equidistant in all directions. This can also make the simulation results deviate from the analytical assumptions.

The surface normal impedance experienced by a plane wave is depicted in Equation 6 and the normal absorption coefficient is depicted in Equation 7.

Finally, the instantaneous acoustic particle velocity is depicted using mirror datasets in the last Figure 8.

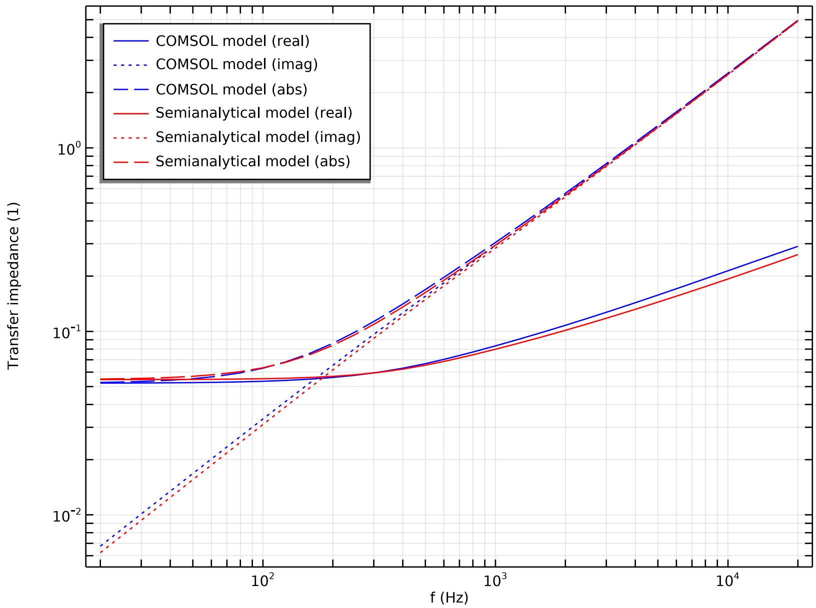


Figure 5: Transfer impedance as function of frequency. Comparing the COMSOL Multiphysics model results and the semianalytical expression.

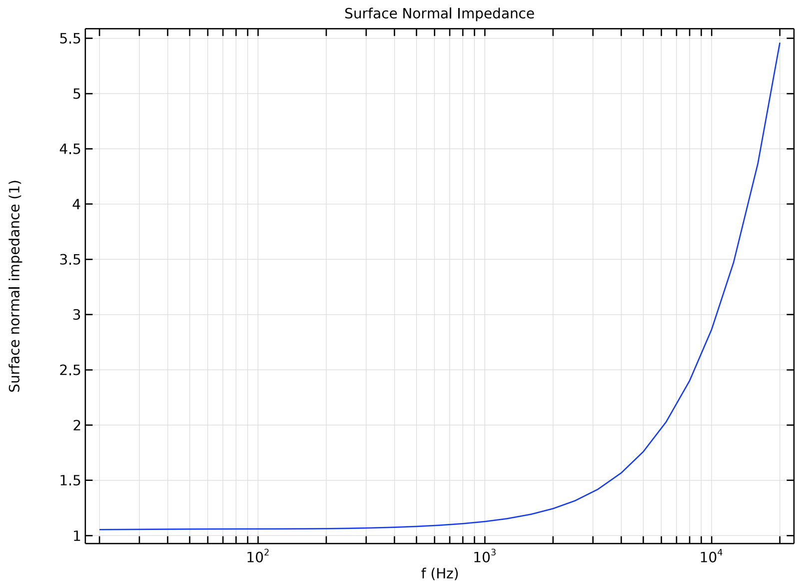


Figure 6: Surface normal impedance of the perforated plate.

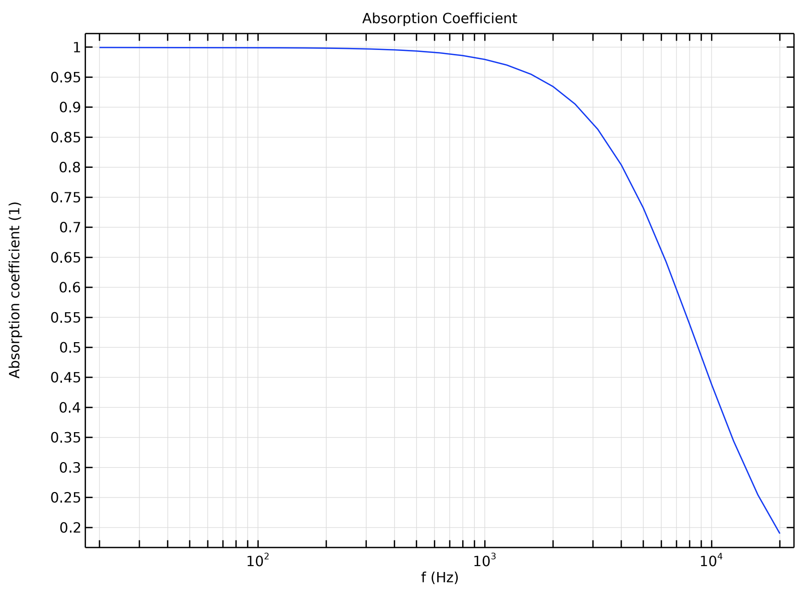


Figure 7: Absorption coefficient of the perforated plate.

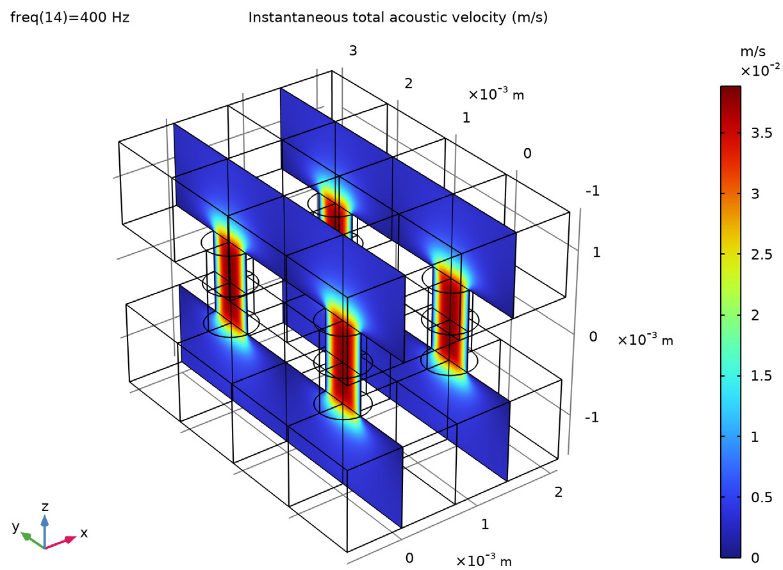


Figure 8: Instantaneous velocity plotted using the symmetry datasets.

Notes About the COMSOL Implementation

SOLVER

This model does not use the default direct solver, but an iterative approach with a so-called direct preconditioner. This is one of the predefined iterative solver suggestions generated by COMSOL Multiphysics (it is then simply to enable it). This solver strategy is described in the *Acoustics Module User's Guide* in the *Modeling with Thermoviscous Acoustics* section. The approach is ideal for medium sized pure thermoviscous acoustics models; the iterative solver is both faster and more memory efficient than the default direct solver.

The present model utilized the symmetries of the geometry to reduce the model size. Some perforated plate conflagrations may not have the same possibilities and thus require solving a larger problem. If the resulting problem becomes too large for the first iterative solver suggestion (used in this tutorial), the second iterative solver suggestion may be used. It is based on the domain decomposition (DD) method. This solver is very memory efficient but slower. An example can be found on the COMSOL homepage at:

- www.comsol.com/model/transfer-impedance-of-a-perforate-12585

MESH

When setting up the mesh, the parameter `dvisc` has been used as a measure of the viscous boundary layer thickness at the maximal study frequency. The viscous boundary layer thickness is given by

$$\delta_{\mathrm{visc}} = \sqrt{\frac{2\mu}{\omega\rho_0}}$$

which at 100 Hz is equal to 0.22 mm for air. Thus the expression `220[um]*sqrt(100[Hz]/fmax)` is used in the parameter list for `dvisc`.

The meshing sequence uses boundary layer operations to make sure that enough elements are present right at the edge of the hole, where the gradients in velocity are more steep.

ACOUSTIC PORTS

This model uses the **Port** feature available in various acoustics physics, including *Thermoviscous Acoustics, Frequency Domain*. The **Port** feature is used to prescribe a nonreflecting condition (an outlet) as well as to set up an incident wave at a boundary (a source). The **Plane wave** option makes it possible to represent an incoming or outgoing plane wave in any planar boundary with an arbitrary shape. The plane wave option does not take any wall effects into account and is not in general suited for waveguides. It can be used if all adjacent walls to the port are slip and adiabatic, or symmetry conditions.

By using the port feature, it is possible to easily compute the incoming, reflected, and transmitted power. For example, the expression for the power reflection coefficient P_r is given under **Definitions > Variables 1**, uses the power variables `ta.port1.P_out` and `ta.port1.P_in` available through the port definition, which represent the incoming and reflected energy going through the inlet.

References

1. I.B. Crandall, *Theory of Vibrating Systems and Sound*, D. Van Nostrand Company, New York, 1926.

2. T.H. Melling, “The acoustic impedance of perforates at medium and high sound pressure levels,” *J. Sound Vibration*, vol. 29, pp. 1–65, 1973.

3. T. Schultz, F. Liu, L. Cattafesta, M. Sheplak, and M. Jones, “A Comparison Study of Normal-Incidence Acoustic Impedance Measurements of a Perforated Liner,” NASA Technical Reports Server LF99-801, 2009.

4. T. Schultz, and others, “A Comparison Study of Normal-Incidence Acoustic Impedance Measurements of a Perforated Liner,” *15th AIAA/CEAS Aeroacoustics Conf.*, AIAA, pp. 2009–3301, 2009.

5. A.W. Nolle, “Small-Signal Impedance of Short Tubes,” *J. Acoust. Soc. Am.*, vol. 25, pp. 32–39, 1952.

Application Library path: `Acoustics_Module/Tutorials,` `_Thermoviscous_Acoustics/transfer_impedance_perforate`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Acoustics** > **Thermoviscous Acoustics** > **Thermoviscous Acoustics, Frequency Domain (ta)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Start by loading the global parameters that define the geometry and mesh parameters.

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 `transfer_impedance_perforate_parameters.txt`.

GEOMETRY 1

The next task is to set up the geometry of the perforate. The following instructions walk you through the steps of creating a fully parameterized geometry of one quarter of a hole as depicted in Figure 1.

If you want to skip these steps, the geometry sequence can be imported by right-clicking the **Geometry** node and selecting **Insert Sequence**. Browse to your COMSOL installation folder under `Multiphysics` > `applications` > `Acoustics_Module` > `Tutorials` and select `transfer_impedance_perforate.mph`. After this, just continue setting up the model from the **Definitions** section below.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `a`.

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

5 Locate the **Position** section. In the **z** text field, type `-tp/2`.

6 Click to expand the **Layers** section. Clear the **Layers on side** checkbox.

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

8 In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`tp/2`

Block 1 (blk1)

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

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

3 In the **Width** text field, type `Lx/2`.

4 In the **Depth** text field, type `Ly/2`.

5 In the **Height** text field, type `2*Lz+tp`.

6 Locate the **Position** section. In the **z** text field, type `-Lz-tp/2`.

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

Layer name	Thickness (m)
`Layer 1`	`Lz`

8 Find the **Layer position** subsection. Select the **Top** checkbox.

9 Click **Build Selected**.

Union 1 (uni1)

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

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

3 In the **Settings** window for **Union**, click **Build Selected**.

Delete Entities 1 (del1)

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

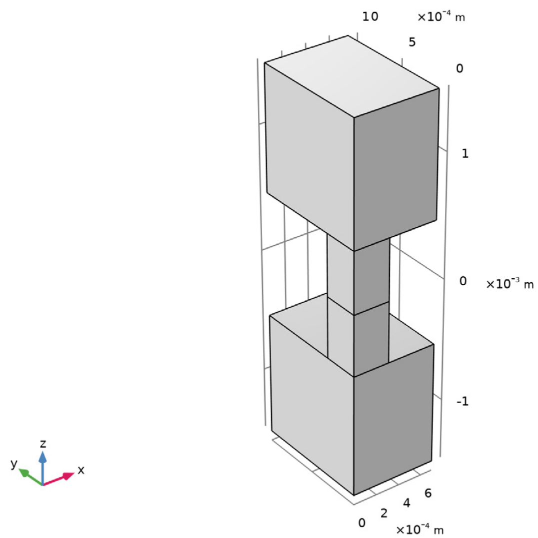
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **uni1**, select Domains 1, 2, and 7 only.

5 Click **Build All Objects**.

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



DEFINITIONS

Load the variables that define the transfer impedance, normal surface impedance, and absorption coefficient. The variables also define the semianalytical transfer impedance model defined in Equation 2.

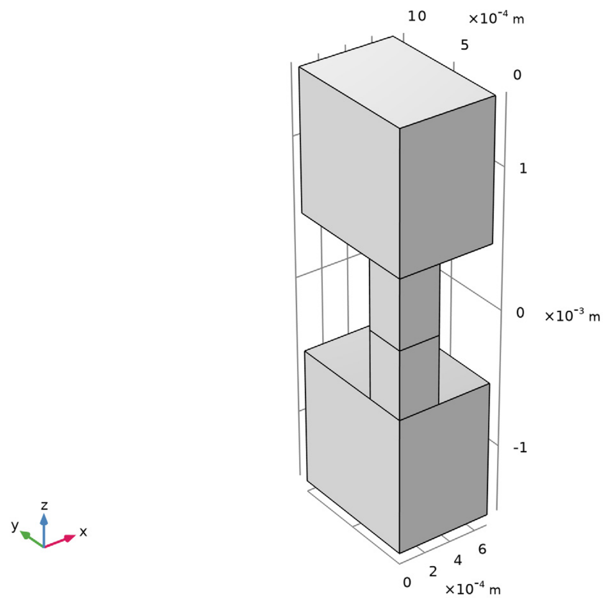
Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

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

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

4 Click **Load from File**.



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

Now, set up integration operators on the two sides of the plate (in and out) as well as in the center of the tube (mid). Create a selection for the symmetry planes.

Integration 1 (intop1)

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

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

3 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

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

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

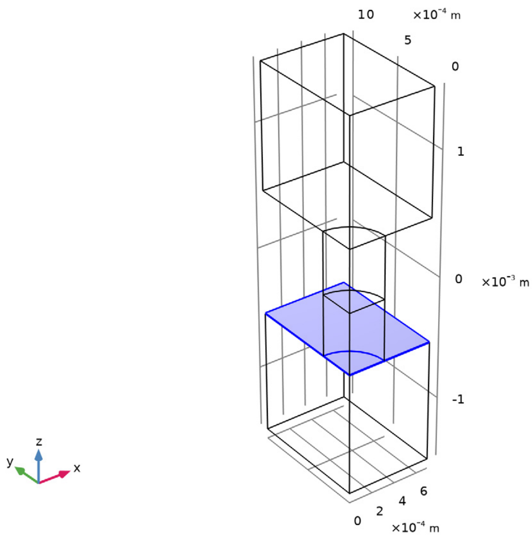
6 Select Boundaries 6 and 15 only.

It might be easier to select the boundaries by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

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

Integration 2 (intop2)

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



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

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

4 Select Boundaries 12 and 17 only.

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

Integration 3 (intop3)

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

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

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

4 Select Boundary 9 only.

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

Symmetry

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

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

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** checkbox, to simplify selection and click on one face on each symmetry planes. Alternatively select the following faces.

5 Select Boundaries 1, 2, 4, 5, 7, 8, 10, 11, and 18–21 only.

Wall

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

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

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 14–17 only.

MATERIALS

Add air as the material and set the bulk viscosity to 0.

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 Right-click and choose **Add to Component 1 (comp1)**.

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

Proceed to set up the physics.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Symmetry 1

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

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

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

Port 1

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

2 Select Boundary 3 only.

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

4 From the **Type of port** list, choose **Plane wave**.

5 Locate the **Incident Mode Settings** section. In the A_p^in text field, type `p0`.

Port 2

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

2 Select Boundary 13 only.

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

4 From the **Type of port** list, choose **Plane wave**.

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

MESH 1

Create a mesh that will resolve the acoustic boundary layers. The thickness of the viscous layer at `fmax` is defined as a parameter, `dvisc`. Use this parameter when setting up a boundary layer mesh.

Swept 1

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

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

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

4 Select Domains 2 and 3 only.

Size 1

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

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** checkbox. In the associated text field, type `a/6`.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `a`.

5 In the **Minimum element size** text field, type `dvisc/2`.

6 In the **Maximum element growth rate** text field, type `1.3`.

7 In the **Resolution of narrow regions** text field, type `4`.

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 **Geometric Entity Selection** section.

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

4 Select Domains 2 and 3 only.

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

Boundary Layer Properties

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

2 Select Boundaries 14 and 16 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 `0.4*dvisc`.

7 Click **Build Selected**.

Swept 2

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

Distribution 1

1 Right-click **Swept 2** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `8`.

5 In the **Element ratio** text field, type `2`.

Size 1

1 In the **Model Builder** window, right-click **Swept 2** and choose **Size**.

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

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

4 Select Edges 17 and 21 only.

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

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

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

8 Click **Build Selected**.

Boundary Layers 2

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

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

3 Clear the **Smooth transition to interior mesh** checkbox.

Boundary Layer Properties

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

2 Select Boundaries 6, 12, 15, and 17 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 `0.4*dvisc`.

7 Click **Build All**.

The generated mesh should look like the image above. By combining two **Boundary Layers**, the mesh has good resolution near the walls and also near the inlet and outlet of the perforate. Using the **Swept** structured mesh is also advantageous as it reduces the number of DOFs in the model, in particular for thermoviscous acoustics where serendipity elements are used as default. For more details see the Acoustics Module User's Guide.

STUDY 1

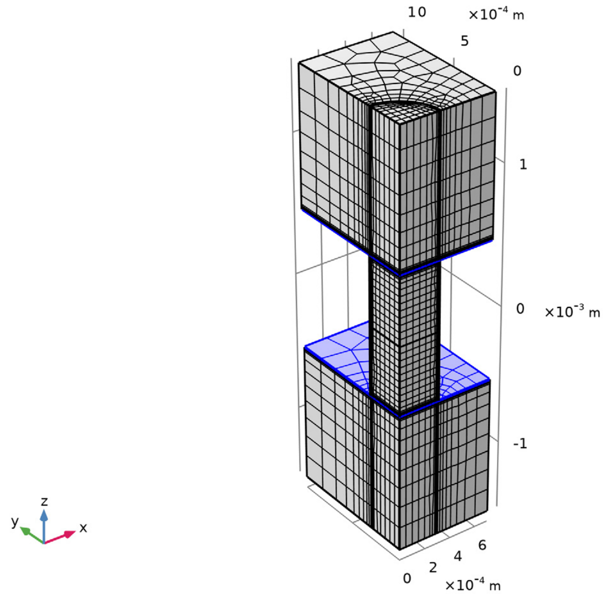
Step 1: Frequency Domain

Proceed to select the frequencies for which to solve the model. Use the built in **ISO preferred frequencies**.

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 Click **Range**.



4 In the **Range** dialog, choose **ISO preferred frequencies** from the **Entry method** list.

5 In the **Start frequency** text field, type `fmin`.

6 In the **Stop frequency** text field, type `fmax`.

7 From the **Interval** list, choose **1/3 octave**.

8 Click **Replace**.

This gives a list with the ISO preferred frequencies along the frequency range with 1/3 octave spacing.

In the next steps, set up an iterative solver to solve this thermoviscous acoustics problem. Because of the problem size (not too large) an iterative solver with a direct preconditioner is the best choice over the default direct solver. A discussion of different solver strategies is given in the modeling section under the Thermoviscous Acoustics chapter in the Acoustics Module User's Guide.

Start by generating the default solver, expand the nodes, and then enable the iterative solver suggestion that uses the direct preconditioner.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1** node.

4 Right-click **Study 1 > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1 > Suggested Iterative Solver (GMRES with Direct Precond.) (ta)** and choose **Enable**.

Step 1: Frequency Domain

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

RESULTS

Multislice

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

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

3 Find the **X-planes** subsection. In the **Planes** text field, type 2.

4 In the **Acoustic Pressure (ta)** toolbar, click **Plot**.

Acoustic Velocity (ta)

1 In the **Model Builder** window, under **Results** click **Acoustic Velocity (ta)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **400**.

4 In the **Acoustic Velocity (ta)** toolbar, click **Plot**.

Multislice

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

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

3 Find the **X-planes** subsection. In the **Planes** text field, type 2.

4 In the **Temperature Variation (ta)** toolbar, click **Plot**.

Temperature Variation (ta)

1 In the **Model Builder** window, click **Temperature Variation (ta)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **400**.

4 In the **Temperature Variation (ta)** toolbar, click **Plot**.

Now, create the transfer impedance plot depicted in Figure 5.

Transfer Impedance

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Transfer Impedance` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
4 Locate the **Plot Settings** section.
5 Select the **x-axis label** checkbox. In the associated text field, type `f (Hz)`.
6 Select the **y-axis label** checkbox. In the associated text field, type `Transfer impedance (1)`.
7 Locate the **Axis** section. Select the **x-axis log scale** checkbox.
8 Select the **y-axis log scale** checkbox.
9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Transfer Impedance** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
real(Ztrans)	1	COMSOL model (real)
imag(Ztrans)	1	COMSOL model (imag)
abs(Ztrans)	1	COMSOL model (abs)

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.
5 From the **Color** list, choose **Blue**.

Point Graph 1

1 In the **Model Builder** window, right-click **Transfer Impedance** and choose **Point Graph**.
2 Select Point 5 only.
3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `real(Ztrans_ana)`.
5 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Red**.
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
`Semianalytical model (real)`

Point Graph 2

1 Right-click **Transfer Impedance** and choose **Point Graph**.

2 Select Point 5 only.

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

4 In the **Expression** text field, type `imag(Ztrans_ana)`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** checkbox.

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

9 In the table, enter the following settings:

Legends
`Semianalytical model (imag)`

Point Graph 3

1 Right-click **Transfer Impedance** and choose **Point Graph**.

2 Select Point 5 only.

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

4 In the **Expression** text field, type `abs(Ztrans_ana)`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** checkbox.

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

9 In the table, enter the following settings:

Legends
`Semianalytical model (abs)`

10 In the **Transfer Impedance** toolbar, click **Plot**.

Create the surface impedance plot depicted in Figure 6.

Surface Normal Impedance

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

2 In the **Settings** window for **1D Plot Group**, type `Surface Normal Impedance` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section.

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

6 Select the **y-axis label** checkbox. In the associated text field, type `Surface normal impedance (1)`.

7 Locate the **Axis** section. Select the **x-axis log scale** checkbox.

8 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Surface Normal Impedance** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
abs(Zn)	1	

4 In the **Surface Normal Impedance** toolbar, click **Plot**.

Next, create the absorption coefficient plot depicted in Figure 7.

Absorption Coefficient

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

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

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section.

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

6 Select the **y-axis label** checkbox. In the associated text field, type `Absorption coefficient (1)`.

7 Locate the **Axis** section. Select the **x-axis log scale** checkbox.

8 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Absorption Coefficient** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`alpha`	1	

4 In the **Absorption Coefficient** toolbar, click **Plot**.

Finally, create a series of mirror datasets to plot the solution in a larger domain (using the symmetries of the model). This will reproduce the plot in Figure 8.

Mirror 3D 1

In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

Mirror 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

Mirror 3D 3

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 Locate the **Plane Data** section. From the **Plane type** list, choose **General**.

5 From the **Plane entry method** list, choose **Point and normal vector**.

6 Find the **Point** subsection. In the **x** text field, type `Lx/2`.

7 Find the **Normal vector** subsection. In the **z** text field, type `0`.

8 In the **x** text field, type `1`.

Mirror 3D 4

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

3 From the **Dataset** list, choose **Mirror 3D 3**.

4 Locate the **Plane Data** section. From the **Plane type** list, choose **General**.

5 From the **Plane entry method** list, choose **Point and normal vector**.

6 Find the **Point** subsection. In the **y** text field, type `Ly/2`.

7 Find the **Normal vector** subsection. In the **z** text field, type `0`.

8 In the **y** text field, type `1`.

Mirror Plot: Velocity

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Mirror Plot: Velocity` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 4**.

4 From the **Parameter value (freq (Hz))** list, choose **400**.

5 Locate the **Color Legend** section. Select the **Show units** checkbox.

Slice 1

1 Right-click **Mirror Plot: Velocity** and choose **Slice**.

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

3 In the **Expression** text field, type `ta.v_inst`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `2`.

5 In the **Mirror Plot: Velocity** toolbar, click **Plot**.

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