

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 semianalytical 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 α , the hole-hole x and y distances are named L_x and L_y , respectively. In this model, the default values are $t_p = 1.0 \text{ mm}$, a = 0.3 mm, $L_x = 1.4 \text{ mm}$, and $L_y = 2.0 \text{ mm}$. 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

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

where Δp is the pressure drop across the plate, v is the mean velocity in the perforation hole, **n** is the surface normal of the perforate, and R is the 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

$$\frac{Z_{\text{trans}}}{\rho_0 c_0} = \left(t_p + \frac{16a\Psi(\sqrt{\sigma})}{3\pi}\right) \frac{i\omega}{c_0 \sigma \left[1 - \frac{2}{k_s a} \frac{J_1(k_s a)}{J_0(k_s a)}\right]} + \frac{1.2(1 - \sigma^2)}{2c_0(\sigma C_d)^2} U_{\text{rms}}$$

$$k_s^2 = -i\rho_0 \omega/\mu$$
(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, $U_{\rm rms}$ 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

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

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

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 reflection coefficient *R* given under **Definitions>Variables I**, 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, F. Liu, L. Cattafesta, M. Sheplak, and M. Jones, "A Comparison Study of Normal-Incidence Acoustic Impedance Measurements of a Perforated Liner," 15th AIAA/CEAS Aeroacoustics conference, 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

- I In the Model Wizard window, click 间 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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

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 I (cyl1)

- I 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 check box.
- 7 Select the Layers on bottom check box.
- 8 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	tp/2

Block I (blkI)

- I 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 check box.

9 Click 틤 Build Selected.

Union I (uni I)

- I 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 틤 Build Selected.

Delete Entities 1 (del1)

- I In the Model Builder window, right-click Geometry I 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 unil, select Domains 1, 2, and 7 only.
- 5 Click 📑 Build All Objects.
- **6** Click the $4 \rightarrow$ **Zoom Extents** button in the **Graphics** toolbar.



DEFINITIONS

y**_ †** .

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 I

I In the Model Builder window, expand the Component I (compl)>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)

y 1 _ ×

- I In the Definitions toolbar, click 📂 Nonlocal Couplings and choose Integration.
- **2** Click the *Z***oom 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)

I 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)

y 1 - ×

- I 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

- I In the **Definitions** toolbar, click *A* **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** check box, 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

- I In the **Definitions** toolbar, click **here 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

- I In 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 Right-click and choose Add to Component I (compl).
- 5 In the Home toolbar, click Add Material to close the Add Material window.Proceed to set up the physics.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Symmetry I

- I In the Model Builder window, under Component I (compl) right-click Thermoviscous Acoustics, Frequency Domain (ta) and choose Symmetry.
- 2 In the Settings window for Symmetry, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Symmetry**.

Port I

- I 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^{in} text field, type p0.

Port 2

- I 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 I

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 I

- I 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 I

- I Right-click Swept I 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 check box. In the associated text field, type a/6.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I 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 All. Build All.

Boundary Layers 1

- I 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** check box.

Boundary Layer Properties

- I 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**.

Boundary Layers 2

- I 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 check box.

Boundary Layer Properties

- I In the Model Builder window, click Boundary Layer Properties.
- 2 Select Boundaries 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. 🏢

STUDY I

Step 1: Frequency Domain

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

- I In the Model Builder window, under Study I click Step I: Frequency Domain.
- 2 In the Settings window for Frequency Domain, locate the Study Settings section.
- 3 Click Range.



- 4 In the Range dialog box, 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.

y x

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 (soll)

- I In the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Stationary Solver I node.
- 4 Right-click Study I>Solver Configurations>Solution I (solI)>Stationary Solver I> Suggested Iterative Solver (GMRES with Direct Precon.) (ta) and choose Enable.

Step 1: Frequency Domain

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

RESULTS

Multislice

- I In the Model Builder window, expand the Acoustic Pressure (ta) node, then click Multislice.
- 2 In the Settings window for Multislice, locate the Coloring and Style section.
- 3 From the Scale list, choose Linear.
- 4 Locate the Multiplane Data section. Find the X-planes subsection. In the Planes text field, type 2.
- 5 In the Acoustic Pressure (ta) toolbar, click = Plot.

Acoustic Velocity (ta)

- I 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 **I** Plot.

Multislice

- I 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 **O** Plot.

Temperature Variation (ta)

- I 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 **O Plot**.

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

Transfer Impedance

- I In the Home toolbar, click 💿 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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** check box. In the associated text field, type f (Hz).
- 6 Select the y-axis label check box. In the associated text field, type Transfer Impedance (1).
- 7 Locate the Axis section. Select the x-axis log scale check box.
- 8 Select the **y-axis log scale** check box.
- 9 Locate the Legend section. From the Position list, choose Upper left.

Global I

- I 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)
<pre>imag(Ztrans)</pre>	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

- I 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 check box.
- 7 From the Legends list, choose Manual.
- 8 In the table, enter the following settings:

Legends

Semianalytical model (real)

Point Graph 2

- I 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 check box.
- 8 From the Legends list, choose Manual.
- **9** In the table, enter the following settings:

Legends

Semianalytical model (imag)

Point Graph 3

- I 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 check box.
- 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 **I** Plot.

Create the surface impedance plot depicted in Figure 6.

Surface Normal Impedance

- I In the Home toolbar, click 🗿 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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** check box. In the associated text field, type **f** (Hz).
- 6 Select the y-axis label check box. In the associated text field, type Surface normal impedance (1).
- 7 Locate the Axis section. Select the x-axis log scale check box.
- 8 Locate the Legend section. Clear the Show legends check box.

Global I

- I 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

I In the Home toolbar, click 🗿 Add Plot Group and choose ID Plot Group.

- 2 In the Settings window for ID 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** check box. In the associated text field, type **f** (Hz).
- 6 Select the y-axis label check box. In the associated text field, type Absorption coefficient (1).
- 7 Locate the Axis section. Select the x-axis log scale check box.
- 8 Locate the Legend section. Clear the Show legends check box.

Global I

- I 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 **I** 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 I

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

Mirror 3D 2

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

Mirror 3D 3

- I 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

- I 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

- I 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 check box.

Slice 1

- I 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 间 Plot.
- 6 Click the **O** Zoom Extents button in the Graphics toolbar.