Generic 711 Coupler: An Occluded Ear-Canal Simulator

Introduction

This is a model of an 711 coupler[1], an occluded ear-canal simulator that follows the specifications given in the IEC 60318-4 international standard (Ref. 1). A coupler is a device for measuring the acoustic output of sound sources with a calibrated microphone. The microphone is coupled to the source by a cavity of known shape and volume (see Figure 1 bottom). The 711 coupler approximates the acoustic transfer impedance of the inner part of the ear canal from the tip of an ear plug (ear insert or ear mould) located at the reference plane to the eardrum (see Figure 1 top right). It is thus a device that is intended to have the same acoustic properties as the average occluded human ear-canal and eardrum system, approximately from the second bend in the ear canal to the eardrum. The 711 coupler is intended for measurements of hearing aids and earphones that are coupled to the ear by means of an insert earphone.
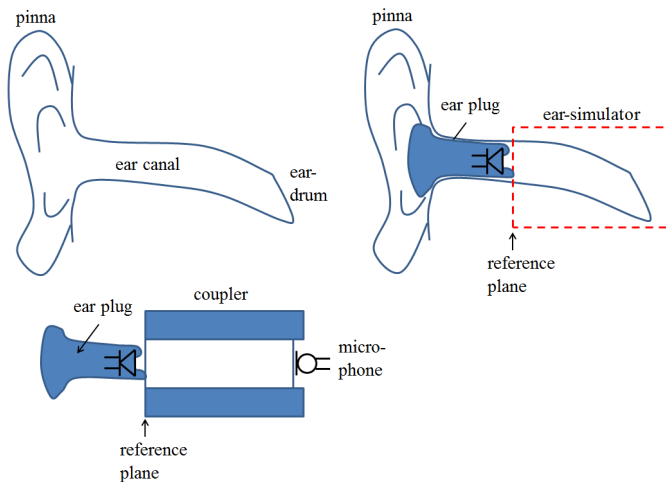

Figure 1: (top left) sketch of the ear with pinna, ear canal, and eardrum, (top right) ear canal occluded by an earplug with a loudspeaker, the occluded-ear canal is the part that the coupler is intended to model, (bottom) ear plug placed at the reference plane of the acoustic coupler including the location of the recording microphone.

1. Here 711 are the last digits in the IEC 60711 standard (1981), which is the standard that the current updated IEC 60318-4 replaces. This has given the occluded ear-canal simulator its commonly used name.

Using a coupler for measurement of the acoustic response enables standardized measurements and comparisons. Coupler measurements on ear-canal simulators are also often used for prototype development and testing. Such measurements do, however, not include leakage between ear mold and ear canal. As the coupler represents a normal average human ear it does not mimic the large acoustic performance variations that exist between individual ears (see for example, Ref. 2).
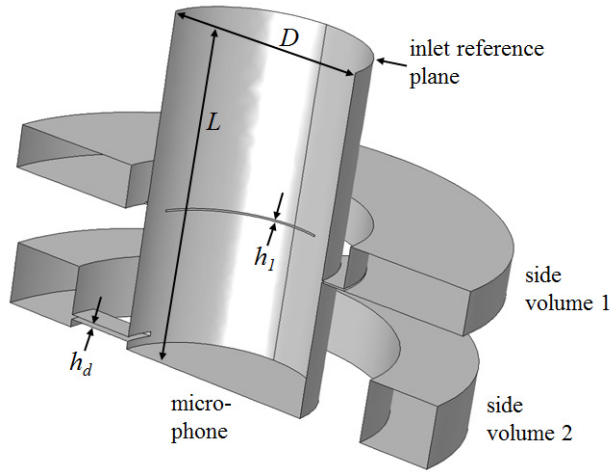

Figure 2: Sketch of the geometry used in the model (cut in half due to symmetry). The main volume is a cylinder of length L and diameter D. The two side volumes are attached to the main volume via slender slits of height h_1 and h_2.

Model Definition

GEOMETRY

The coupler is made of hard nonporous material and it is terminated at one end by a measurement microphone. This end corresponds to the eardrum (the tympanic membrane) and the measured microphone pressure should correspond the to that perceived by the eardrum. The human eardrum has a nontrivial acoustic behavior (see Ref. 3); the coupler has to account for

1 the acoustic energy losses at the eardrum, and

2 the acoustics of the cylinder like volume of the ear canal.

In order for the coupler to do this it is constructed as a main cylinder of length L and diameter D with two attached side volumes, see Figure 2. The side volumes are connected to the main volume via shallow slits of height h_1 and h_2. The coupler

geometry meets the requirements of the IEC 60318-4 international standard (Ref. 1) and besides certain details corresponds to the Brüel & Kjær Ear Simulator Type 4157.

The inclusion of the side volumes and the slits is necessary to mimic the complex eardrum mechanical losses using an acoustic system. In the coupler the losses are mainly due to the high thermal and viscous damping in the slits. The diameter D of the main cylinder is given by the standard and is 7.5 mm. The length of the cylinder is prescribed by the IEC standard to be such as to produce a half-wavelength resonance at around 13.5 kHz. In this model L = 12.5 mm, which gives a resonance at 13.8 kHz. The slit heights are h_1 = 69 μm and h_2 = 170 μm.

THERMOACOUSTICS

As the thermal and viscous losses are important the finite element model is set up in COMSOL using the The Thermoacoustics, Frequency Domain Interface. This enables the direct inclusion and modeling of the thermal and viscous losses in the slits. The losses arise in the viscous and thermal boundary layers that are characterized by the length scales

$$\delta_v = \sqrt{\frac{2\mu}{\omega\rho}} \qquad \delta_{th} = \sqrt{\frac{2k}{\omega\rho C_p}} \tag{1}$$

where ω is the angular frequency, μ is the dynamic viscosity, k is the coefficient of thermal conduction, ρ is density, and C_p is the specific heat at constant pressure. In the modeled frequency range, from 100 Hz to 20 kHz, this yields length scales from 220 μm to 15 μm (for air at 23° C). Thermal and viscous losses are hence important in most of the frequency range when comparing these length scales to the slit heights. However, they need not be included in the main cylinder and the side volumes, where pressure acoustics may be used.

COUPLER CHARACTERIZATION

The 711 coupler is characterized in the IEC 60318-4 standard in terms of its transfer impedance Z_{trans} and the microphone response L_{mic} (for a constant volume displacement source):

$$Z_{trans} = \frac{p_{mic}}{Q_{in}}$$

$$L_{mic} = 10\log\left(\frac{\langle p_{mic}\rangle^2}{p_{ref}^2}\right) = 10\log(\langle p_{mic}\rangle^2) - L_{ref} \tag{2}$$

where Q_{in} is the volume flow rate at the inlet reference plane, $\langle p_{mic} \rangle$ is the root mean square (rms) pressure at the measurement microphone, p_{ref} is a reference pressure (here the rms pressure at 500 Hz), and L_{ref} is the corresponding reference level. The transfer impedance of a coupler is easily measured using, for example, a microphone as a sound source (it has a high output impedance and thus a nearly constant Q_{in}). Moreover, the pressure is directly determined by the measurement microphone. In a real ear, Z_{trans} is somewhat more complicated to measure because it requires the insertion of a probe tube into the ear to measure p_{mic} (now the pressure at the eardrum). The transfer impedance and microphone response are specified in the frequency range 100 Hz to 10 kHz. Above 10 kHz the 711 coupler does not simulate a human ear.

Generally, to mimic an ear with a coupler one should require both systems to have the same two-port parameters. These are four parameters that relate pressure and volume velocity at the inlet and outlet. If only the acoustic input load and the pressure at the eardrum are of interest, the transfer impedance characterizes the coupler together with the input impedance Z_{in}. The latter is defined as

$$Z_{in} = \frac{p_{in}}{Q_{in}} \tag{3}$$

where p_{in} is the pressure at the inlet reference plane.

Modeling an acoustic coupler is of interest for design and for optimizing the acoustic response with respect to different input systems (see for example Ref. 5). The coupler cannot always be modeled as a simple acoustic load (impedance). Interactions may exist between the sound source outlets (hearing aid ear-mold or earplug) and the coupler acoustics. Nonplane waves may, for example, propagate into the coupler.

BOUNDARY CONDITIONS

At the inlet reference plane a constant volume source is applied by specifying the inward normal acceleration a_n of the boundary. Because the acceleration is twice the time derivative of the position, this corresponds to multiplication with $(i\omega)^2$ in the frequency domain, so that

$$a_n = (i\omega)^2 d_0 \tag{4}$$

where d_0 is the sound source displacement.

At the location of the measurement microphone, an impedance corresponding to the mechanical properties of a Brüel & Kjær 4192 microphone is specified (see Ref. 4 pp 6–18). The impedance is given by

$$Z_{mic} = \frac{1}{i\omega C_{mic}} + R_{mic} + i\omega L_{mic} \qquad (5)$$

where $C_{mic} = 0.62 \cdot 10^{-13}$ m^5/N is the acoustic compliance, $R_{mic} = 119 \cdot 10^6$ N·s/m^5 is the acoustic mass, and $L_{mic} = 710$ kg/m^4 is the acoustic resistance.

To reduce the model size, model only half of the geometry and use symmetry conditions on both the acoustic and thermoacoustic domains.

Results and Discussion

Figure 3 depicts the transfer impedance for the system as a function of frequency. The modeled system is seen to comply well with the IEC standard curve.
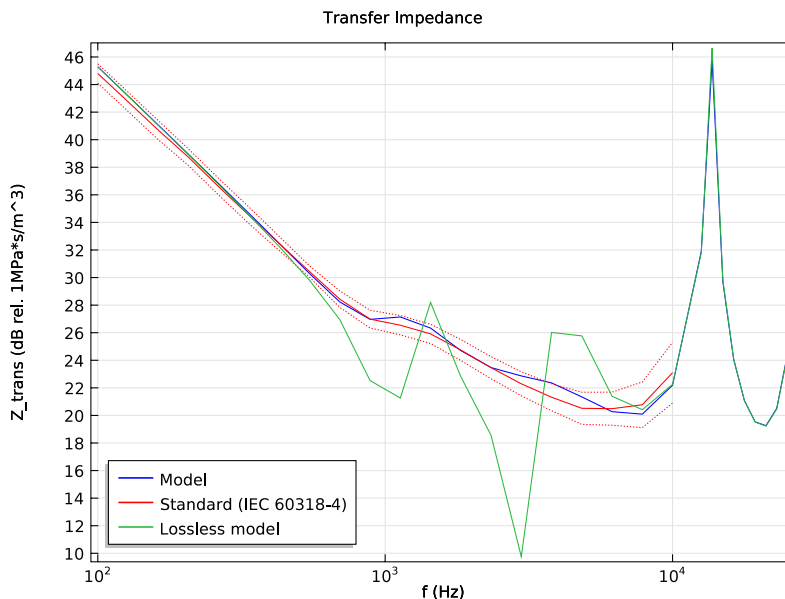

Figure 3: Transfer impedance as function of frequency for the model (blue line), the IEC standard curve (red line) including upper and lower tolerances (red dotted line), and the system modeled without losses (green line).

Also evident from the graph is that it is important to use thermoacoustics to model such acoustic systems with small geometrical dimensions. In the frequency range where the acoustic boundary-layer thickness is comparable to the small slit heights, the lossless model is completely off because all the resonances are undamped. Above about 10 kHz the thermal and viscous losses on the acoustic boundary are much less pronounced and the system is effectively lossless.

Figure 4 depicts the microphone response measured for a constant displacement source, again comparing model results to the IEC standard and the lossless model.
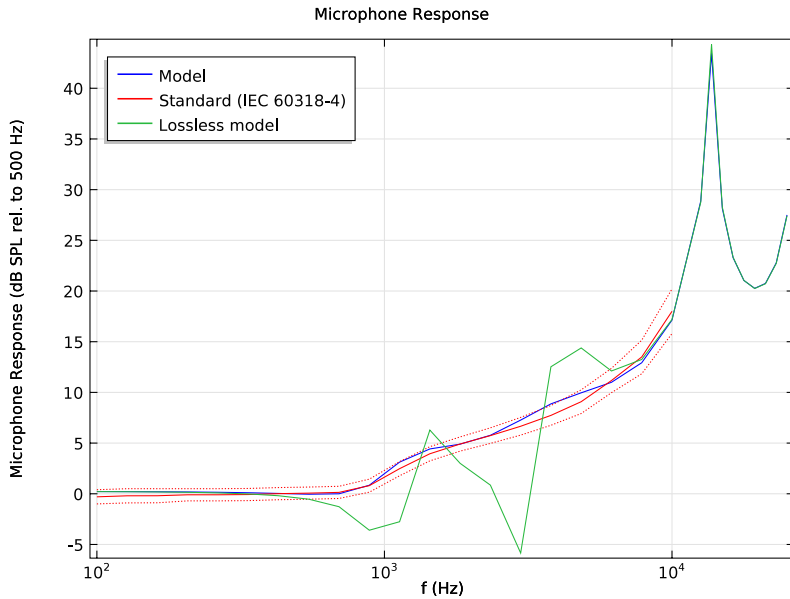

Figure 4: Microphone response as function of frequency for the model (blue line), the IEC standard curve (red line) including upper and lower tolerances (red dotted line), and the system modelled without losses (green line).

The input impedance of the system derived at the reference plane is depicted in Figure 5 where the model results are compared to the results of the fully lossless model.
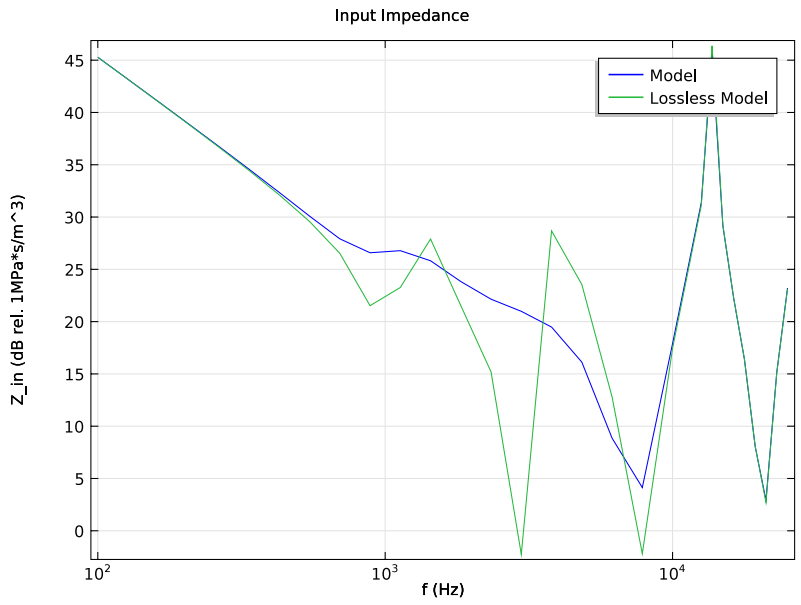
Figure 5: Input impedance as function of frequency for the model (blue line) and the system modelled without losses (green line).

Figure 6, Figure 7, and Figure 8 depict the pressure distribution inside the 711 coupler at frequencies of 25.1 kHz, 13.7 kHz, and 885 Hz, respectively. The first figure represents the standing wave mode inside the inner tube and the second figure the half wave standing mode. The last figure represents a Helmholtz-like resonance in the lower side volume.

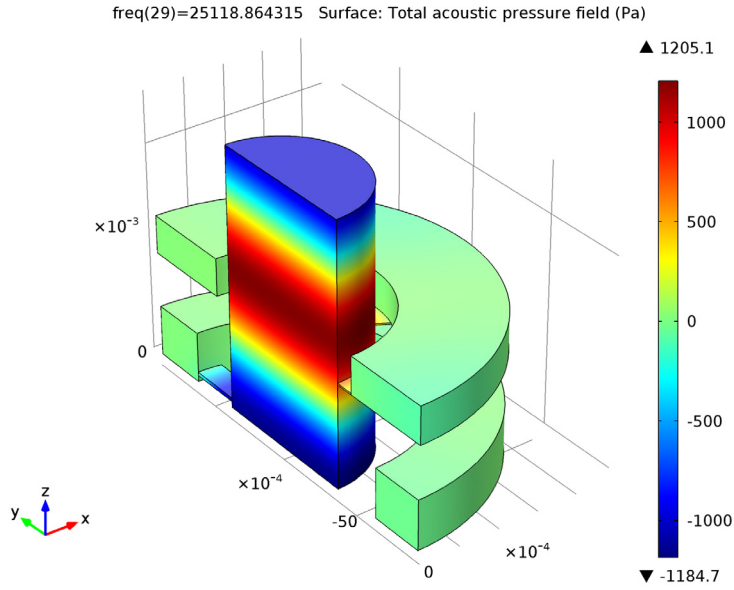freq(29)=25118.864315   Surface: Total acoustic pressure field (Pa)

Figure 6: Instantaneous pressure distribution at f = 25.1 kHz.

freq(22)=13724.60961   Surface: Total acoustic pressure field (Pa)
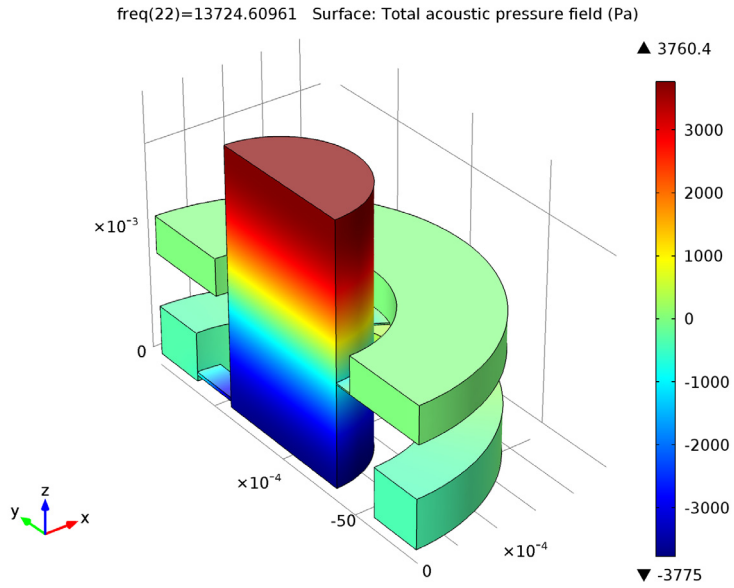
Figure 7: Instantaneous pressure distribution at f = 13.7 kHz.

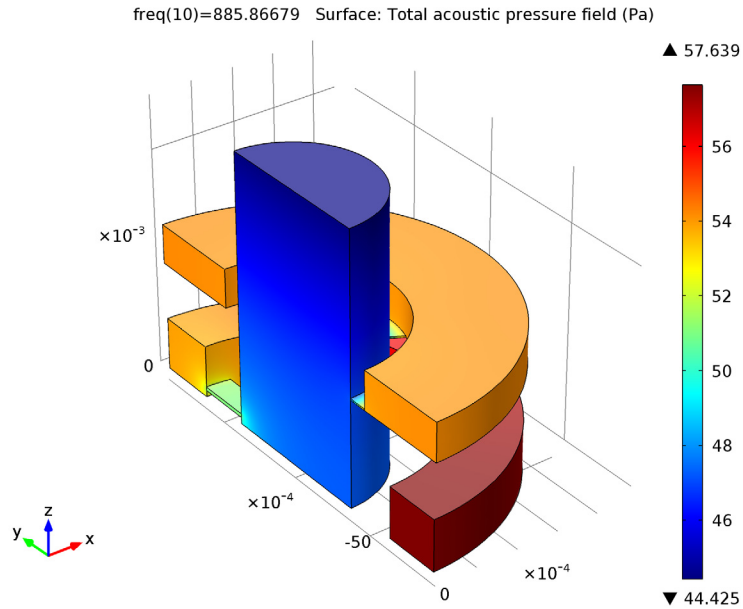freq(10)=885.86679   Surface: Total acoustic pressure field (Pa)

Figure 8: Instantaneous pressure distribution at f = 885 Hz.

References

1. IEC 60318-4, *Electroacoustics—Simulators of human head and ear—Part 4: Occluded-ear simulator for the measurement of earphones coupled to the ear by means of ear inserts*, Edition 1.0, 2010.

2. M.R. Stinson and B.W. Lawton, "Specification of the geometry of the human ear canal for the prediction of sound-pressure level distribution," *J. Acoust. Soc. Am.*, vol. 85, p. 2492, 1989.

3. M.R. Stinson, "The spatial distribution of sound pressure within scaled replicas of the human ear canal," *J. Acoust. Soc. Am.*, vol. 78, p. 21596, 1985.

4. Brüel and Kjær, *Microphone Handbook: For the Falcon Range Microphone Products*, Technical Documentation, 1995.

5. B.L. Zhang, S. Jønsson, A. Schuhmacher, and L.B. Nielsen, *A combined BEM/ FEM Acoustic Model of an Occluded Ear Simulator*, InterNoise 2004, Prague, Czech Republic, 2004.

Model Library path: `Acoustics_Module/Industrial_Models/`
`generic_711_coupler`

Modeling Instructions

MODEL WIZARD

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

2 Click **Next**.

3 In the **Add physics** tree, select **Acoustics>Thermoacoustics>Thermoacoustics, Frequency Domain (ta)**.

4 Click **Add Selected**.

5 Click **Next**.

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

7 Click **Finish**.

GLOBAL DEFINITIONS

Load the parameters for the model. The list of parameters include the maximal mesh size, the microphone impedance parameters, and other reference values.

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 `generic_711_coupler_parameters.txt`.

To make the data for the transfer impedance and microphone response available in the model, create six interpolation functions. The data are located in two text files and comprise the nominal values as well as upper and lower tolerances as defined in the IEC 60318-4 standard (Table 1 and Table B.1 in Ref. 1).

Interpolation 1

1 Right-click **Global Definitions** and choose **Functions>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Click the **Browse** button.

5 Browse to the model's Model Library folder and double-click the file
`generic_711_coupler_transfer_impedance.txt`.

6 In the **Number of arguments** edit field, type 1.

7 Find the **Functions** subsection. In the table, enter the following settings:

| Function name	Position in file
int_trans	1
int_trans_upper	2
int_trans_lower	3

Interpolation 2

1 Right-click **Global Definitions** and choose **Functions>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Click the **Browse** button.

5 Browse to the model's Model Library folder and double-click the file
`generic_711_coupler_mic_response.txt`.

6 In the **Number of arguments** edit field, type 1.

7 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
int_mic	1
int_mic_upper	2
int_mic_lower	3

GEOMETRY 1

The figure below shows the model geometry.
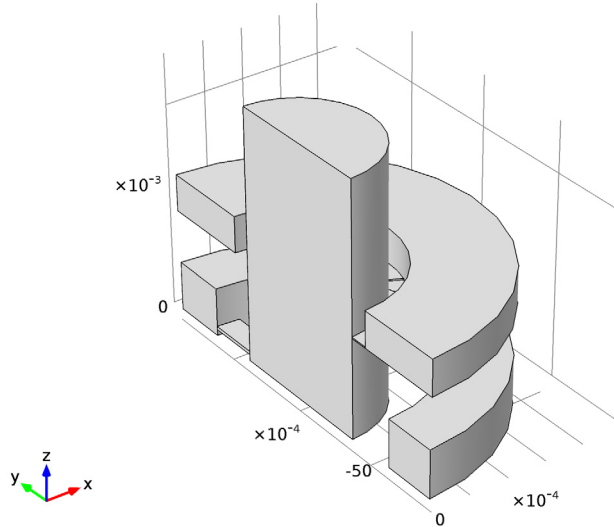
Import the model geometry from file by following these steps.

Import 1

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

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

3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file
`generic_711_coupler_geom.mphbin`.

5 Click the **Import** button.

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

DEFINITIONS

Load a set of variables that define the microphone impedance (Equation 5), the transfer impedance (Equation 2), and the input impedance (Equation 3) from a file. Add operators to integrate values across the reference plane (the inlet) and the microphone plane.

Variables 1

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

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

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `generic_711_coupler_variables.txt`.

Integration 1

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

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

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

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

5 Select Boundary 19 only.

Integration 2

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

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

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

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

5 Select Boundary 17 only.

Next, set up the acoustic model and the boundary conditions.

THERMOACOUSTICS, FREQUENCY DOMAIN

Thermoacoustics Model 1

1 In the **Model Builder** window, expand the **Thermoacoustics, Frequency Domain** node, then click **Thermoacoustics Model 1**.

2 In the **Thermoacoustics Model** settings window, locate the **Model Inputs** section.

3 In the T_0 edit field, type `Tref`.

4 Locate the **Thermoacoustics Model** section. From the $\rho_0(p_0, T_0)$ list, choose **From material**.

Pressure Acoustics Model 1

1 In the **Model Builder** window, under **Model 1>Thermoacoustics, Frequency Domain** click **Pressure Acoustics Model 1**.

2 Select Domains 1, 2, and 4 only.

3 Click the **Wireframe Rendering** button on the Graphics toolbar, to simplify the selection of the faces and edges in the geometry.

Symmetry 1

1 In the **Model Builder** window, right-click **Thermoacoustics, Frequency Domain** and choose **Symmetry**.

2 Select Boundaries 1, 5, 10, 15, 21, 25, and 27 only.
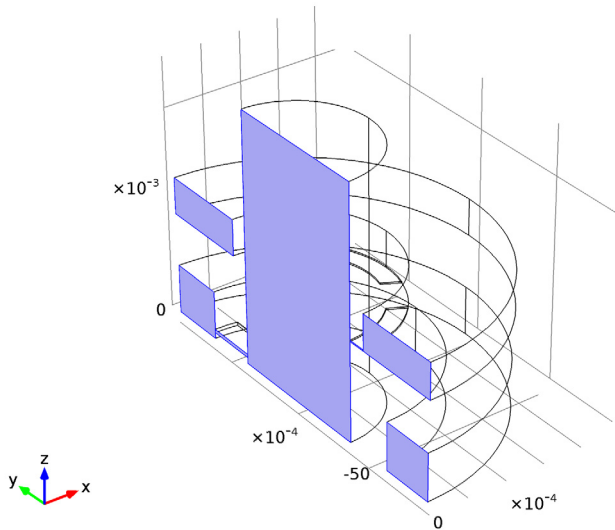
Normal Acceleration 1

1 Right-click **Thermoacoustics, Frequency Domain** and choose the boundary condition **Pressure Acoustics, Frequency Domain>Normal Acceleration**.

2 Select Boundary 19 only.

3 In the **Normal Acceleration** settings window, locate the **Normal Acceleration** section.

4 In the a_n edit field, type (ta.iomega)^2*d0.

The expression for the normal acceleration is defined in Equation 4.

Impedance 1

1 Right-click **Thermoacoustics, Frequency Domain** and choose the boundary condition **Pressure Acoustics, Frequency Domain>Impedance**.

2 Select Boundary 17 only.

3 In the **Impedance** settings window, locate the **Impedance** section.

4 In the Z_i edit field, type Zspmic.

Note that the impedance specified is a specific impedance, that is, the impedance multiplied by the area.

Set up a second pressure acoustics model defined in the thin slit domains. Disable this node in the full study (Study 1) (which includes thermoacoustic losses), re-enable the domain when simulating the effects of using a fully lossless model (in Study 2).

Pressure Acoustics Model 2

1 Right-click **Thermoacoustics, Frequency Domain** and choose **Pressure Acoustics Model**.

2 Select Domains 3, 5, and 6 only.

3 Right-click **Model 1>Thermoacoustics, Frequency Domain>Pressure Acoustics Model 2** and choose **Disable**.

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** window, locate the **Materials** section.

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

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

Air

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

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

3 In the table, enter the following settings:

Property	Name	Value
Bulk viscosity	muB	muB0

Now, build the mesh using an extruded mapped mesh in the slits and a tetrahedral mesh in the remaining domains.

MESH 1

Mapped 1

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

2 Select Boundaries 14 and 35 only.

Distribution 1

1 Right-click **Model 1>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 18, 62, 73, and 80 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type 4.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 19 and 27 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type 25.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 63 and 66 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

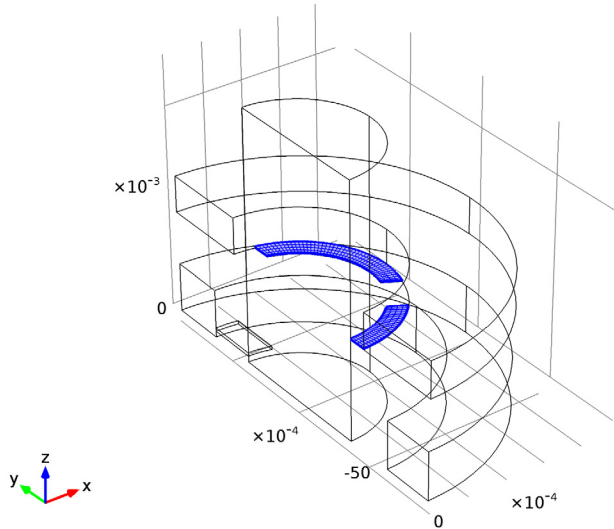**4** In the **Number of elements** edit field, type 36.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 79 and 86 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

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

5 Click the **Build Selected** button.

The mapped mesh on the upper side of the upper slits looks like that in the figure below.

Now, proceed to mesh the upper side of the lower slit and then extrude the mesh in the slits.

Mapped 2

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

2 Select Boundary 24 only.

Distribution 1

1 Right-click **Model 1>Mesh 1>Mapped 2** and choose **Distribution**.

2 Select Edges 36 and 69 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type 7.

Distribution 2

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

2 Select Edges 37 and 50 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type 6.

5 Click the **Build Selected** button.

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 Lmax.

Swept 1

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

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

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

4 Select Domains 3, 5, and 6 only.

Distribution 1

1 Right-click **Model 1>Mesh 1>Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** edit field, type 3.

4 Click the **Build Selected** button.

Convert 1

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

2 In the **Convert** settings window, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 11, 18, 22, 29, 33, 36, 39, and 41 only.

5 Click the **Build Selected** button.

Free Tetrahedral 1

Right-click **Mesh 1** and choose **Free Tetrahedral**.

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

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

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

4 Select Boundaries 17 and 19 only.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type 1[mm].

8 Click the **Build All** button.

The mesh should look similar to that shown in the figure below. The mesh sequence generates a warning message that has no influence on the quality of the mesh built here. The message is intended for helping find problem regions in the mesh when building it.
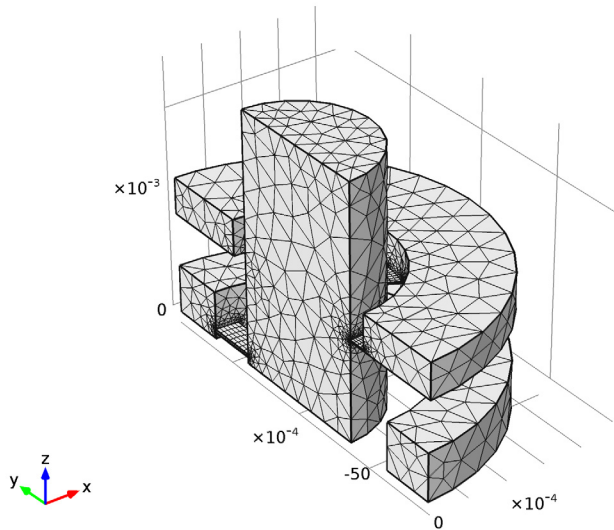
STUDY 1

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 `10^{range(2,2/19,4)} 10^{range(4.1,0.3/8,4.4)}`.

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

Note that, the computation may take several minutes.

Add a second study to solve the model neglecting the thermal and viscous losses in the slits. The pressure acoustics model will override the thermoacoustics model in the slits and the system will be lossless. The results from the second study are stored in a separate data set. Compare this second solution to the model that includes the thermal and viscous losses.

ROOT

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.

4 In the **Model Builder** window, click **Step 1: Frequency Domain**.

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

6 In the **Frequencies** edit field, type `10^{range(2,2/19,4)} 10^{range(4.1,0.3/8,4.4)}`.

THERMOACOUSTICS, FREQUENCY DOMAIN

Pressure Acoustics Model 2
In the **Model Builder** window, under **Model 1>Thermoacoustics, Frequency Domain** right-click **Pressure Acoustics Model 2** and choose **Enable**.
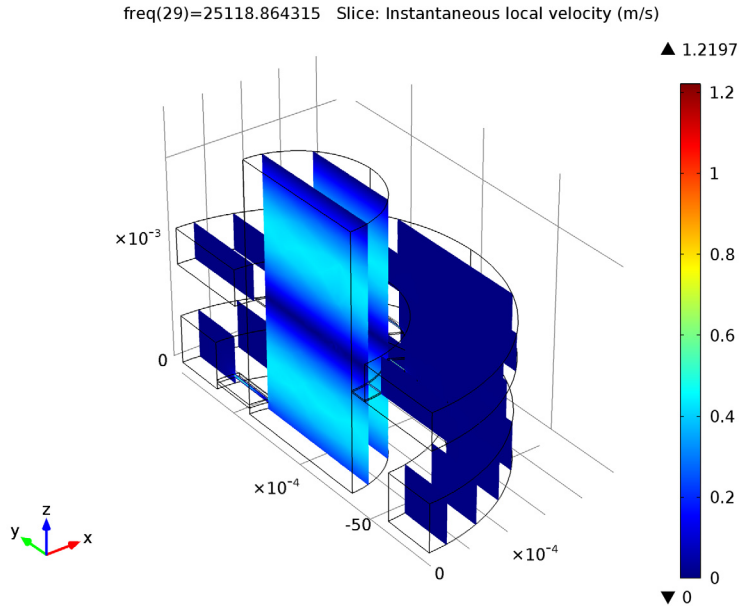
STUDY 2
In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Velocity (ta)

1 In the **Model Builder** window, under **Results** right-click **Velocity (ta)** and choose **Plot**.

The first default plot depicts the instantaneous velocity at 25 kHz and looks like the figure below.

freq(29)=25118.864315   Slice: Instantaneous local velocity (m/s)

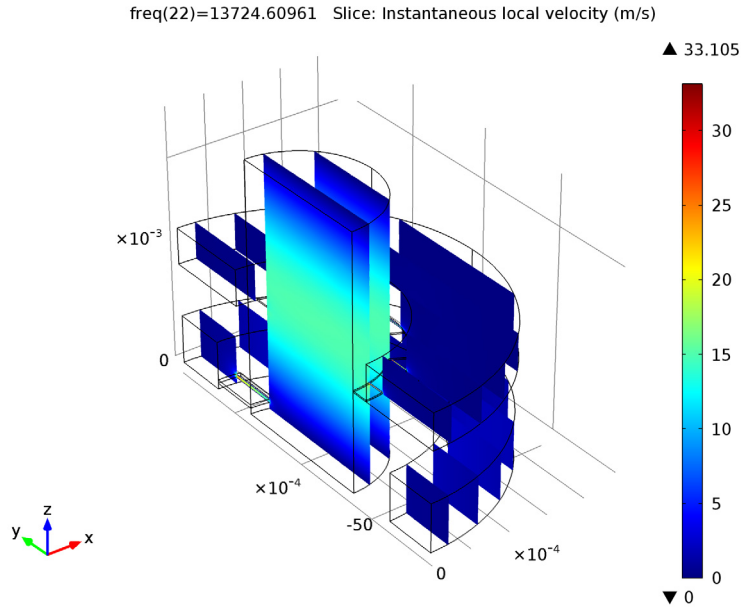**2** In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Parameter value (freq)** list, choose **13724.60961**.

4 Click the **Plot** button.

The velocity at the first longitudinal resonance at 13.8 kHz looks like the figure below.

freq(22)=13724.60961   Slice: Instantaneous local velocity (m/s)

Temperature (ta)

1 In the **Model Builder** window, under **Results** right-click **Temperature (ta)** and choose **Plot**.

The second default plot shows the temperature field at 25 kHz. The acoustic temperature variations are only plotted in the slits where the thermoacoustic model is used. Use the zoom functionality to get at detailed view of the temperature.

freq(29)=25118.864315   Surface: Temperature variation (K)

Acoustic Pressure (ta)

Modify the third default plot to show the instantaneous pressure distribution at 25, 14, and 890 kHz (Figure 6, Figure 7, and Figure 8).

1 In the **Model Builder** window, expand the **Acoustic Pressure (ta)** node.

2 Right-click **Multislice 1** and choose **Delete**.

3 Click **Yes** to confirm.

4 Right-click **Acoustic Pressure (ta)** and choose **Surface**.

5 Right-click **Results>Acoustic Pressure (ta)>Surface 1** and choose **Plot**.

6 In the **Model Builder** window, click **Acoustic Pressure (ta)**.

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

8 From the **Parameter value (freq)** list, choose **13724.60961**.
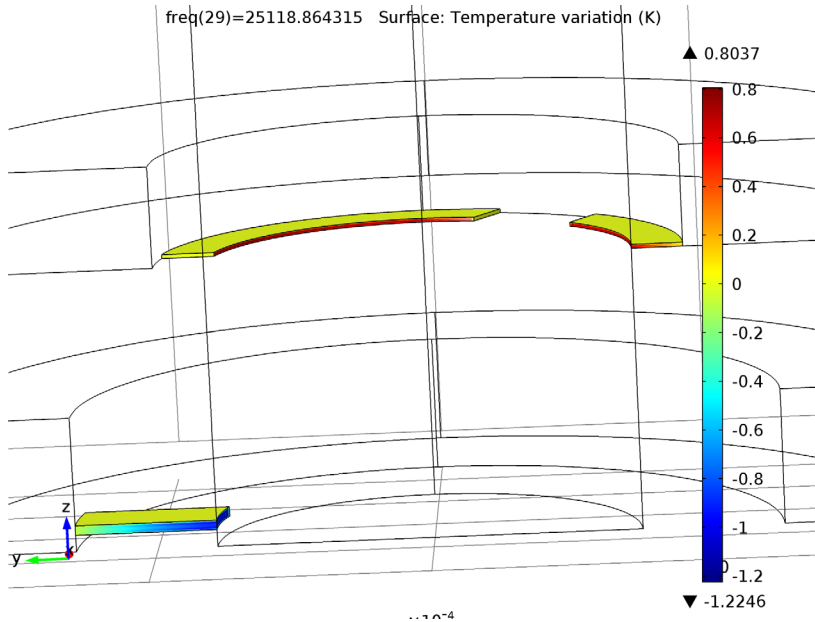
9 Click the **Plot** button.

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

11 From the **Parameter value (freq)** list, choose **885.86679**.

12 Click the **Plot** button.

Data Sets

Plot the sound pressure level distribution at 25 kHz and 885 kHz. First, extend the solution to the whole axisymmetric geometry.

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **More Data Sets>Mirror 3D**.
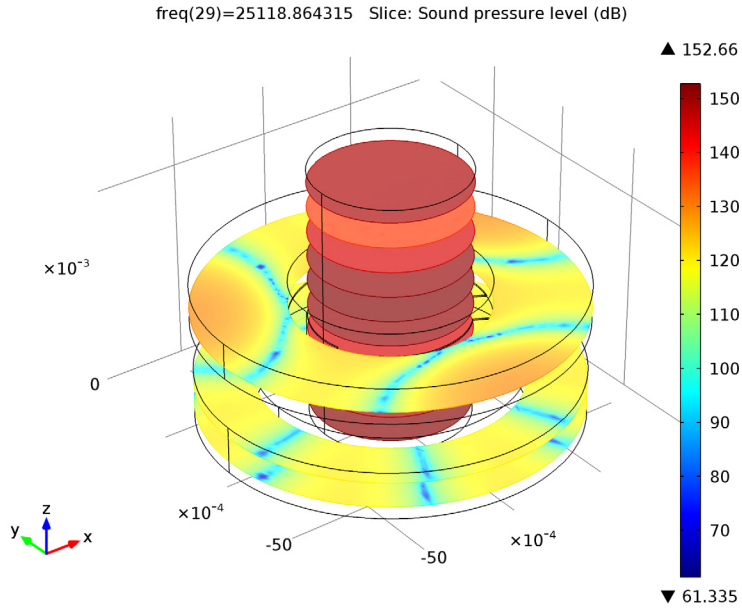
3D Plot Group 4

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

2 In the **Model Builder** window, under **Results** right-click **3D Plot Group 4** and choose **Rename**.

3 Go to the **Rename 3D Plot Group** dialog box and type Sound Pressure Level in the **New name** edit field.

4 Click **OK**.

Sound Pressure Level

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

2 From the **Data set** list, choose **Mirror 3D 1**.

3 Right-click **Results>Sound Pressure Level** and choose **Slice**.

4 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Thermoacoustics, Frequency Domain>Sound pressure level (ta.Lp)**.

5 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

6 In the **Planes** edit field, type 10.

7 Click the **Plot** button.

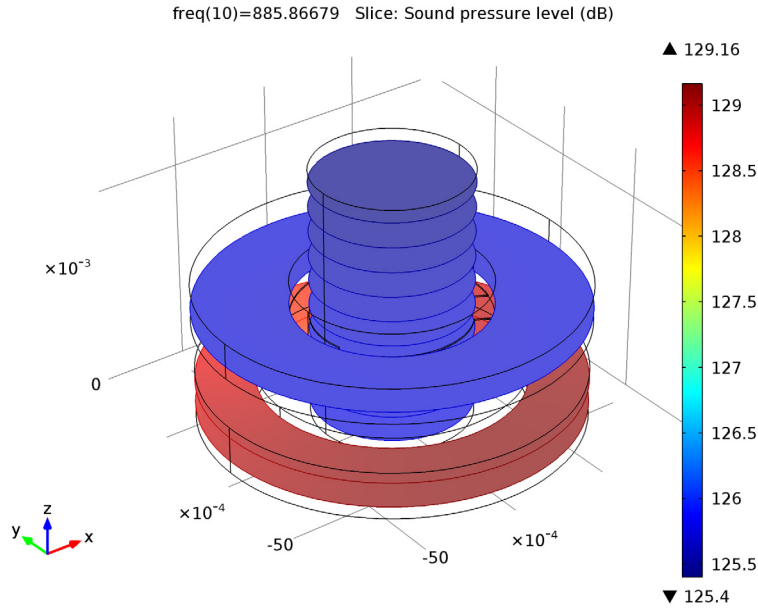freq(29)=25118.864315   Slice: Sound pressure level (dB)

8 In the **Model Builder** window, click **Sound Pressure Level**.

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

10 From the **Parameter value (freq)** list, choose **885.86679**.

11 Click the **Plot** button.

freq(10)=885.86679   Slice: Sound pressure level (dB)

Plot the transfer impedance for the lossy and the lossless model, as well as the curves given by the standard Figure 3.

1D Plot Group 5

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

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

3 Go to the **Rename 1D Plot Group** dialog box and type `Transfer Impedance` in the **New name** edit field.

4 Click **OK**.

Transfer Impedance

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

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Transfer Impedance`.

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

5 In the associated edit field, type `f (Hz)`.

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

7 In the associated edit field, type `Z_trans (dB rel. 1MPa*s/m^3)`.

8 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

9 Right-click **Results>Transfer Impedance** and choose **Global**.

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

11 In the table, enter the following settings:

Expression	Unit	Description
20*log10(abs(Ztrans/1e6[Pa*s/m^3]))		Model

12 In the **Model Builder** window, right-click **Transfer Impedance** and choose **Global**.

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

14 In the table, enter the following settings:

Expression	Unit	Description
int_trans(freq)		Standard (IEC 60318-4)

15 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Red**.

16 Locate the **Data** section. From the **Data set** list, choose **Solution 1**.

Only plot the curves given by the IEC standard in the range from 100 to 10 kHz, where they are defined.

17 From the **Parameter selection (freq)** list, choose **From list**.

18 In the **Parameter values (freq)** list, choose values from **100**, to **10000**.

19 Click the **Plot** button.

20 Click the **x-Axis Log Scale** button on the Graphics toolbar.

21 Right-click **Results>Transfer Impedance>Global 2** and choose **Duplicate**.

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

23 In the table, enter the following settings:

Expression	Unit	Description
int_trans_upper(freq)		Upper tolerance
int_trans_lower(freq)		Lower tolerance

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

25 Click to expand the **Legends** section. Clear the **Show legends** check box.

26 In the **Model Builder** window, under **Results>Transfer Impedance** right-click **Global 1** and choose **Duplicate**.

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

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

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

Expression	Unit	Description
20*log10(abs(Ztrans/1e6[Pa*s/m^3]))		Lossless model

30 Click the **Plot** button.

31 In the **Model Builder** window, right-click **Transfer Impedance** and choose **Duplicate**.

Proceed to reproduce the plot of the microphone response Figure 4.

Transfer Impedance 1

1 In the **Model Builder** window, under **Results** right-click **Transfer Impedance 1** and choose **Rename**.

2 Go to the **Rename 1D Plot Group** dialog box and type Microphone Response in the **New name** edit field.

3 Click **OK**.

Microphone Response

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

2 In the **Title** text area, type Microphone Response.

3 Locate the **Plot Settings** section. In the **y-axis label** edit field, type Microphone Response (dB SPL rel. to 500 Hz).

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 In the **Model Builder** window, expand the **Microphone Response** node, then click **Global 1**.

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

7 In the table, enter the following settings:

Expression	Unit	Description
20*log10(abs(intop_mic(p)/intop_mic(1)))-33.94		Model

8 In the **Model Builder** window, under **Results>Microphone Response** click **Global 2**.

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

10 In the table, enter the following settings:

Expression	Unit	Description
int_mic(freq)		Standard (IEC 60318-4)

11 In the **Model Builder** window, under **Results>Microphone Response** click **Global 3**.

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

13 In the table, enter the following settings:

Expression	Unit	Description
int_mic_upper(freq)		Upper tolerance
int_mic_lower(freq)		Lower tolerance

14 In the **Model Builder** window, under **Results>Microphone Response** click **Global 4**.

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

16 In the table, enter the following settings:

Expression	Unit	Description
20*log10(abs(intop_mic(p)/ intop_mic(1)))-33.94		Lossless model

17 Click the **Plot** button.

Finally, plot the input impedance for the lossy and the lossless model (Figure 5).

1D Plot Group 7

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

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type Input Impedance.

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

6 In the associated edit field, type f (Hz).

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

8 In the associated edit field, type Z_in (dB rel. 1MPa*s/m^3).

9 Right-click **Results>1D Plot Group 7** and choose **Global**.

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

11 In the table, enter the following settings:

Expression	Unit	Description
20*log10(abs(Zin/1e6[Pa*s/m^3]))		Model

12 In the **Model Builder** window, under **Results>1D Plot Group 7** right-click **Global 1** and choose **Duplicate**.

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

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

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

Expression	Unit	Description
20*log10(abs(Zin/1e6[Pa*s/m^3]))		Lossless Model

16 Click the **x-Axis Log Scale** button on the Graphics toolbar.

17 Click the **Plot** button.

18 Right-click **Results>1D Plot Group 7>Global 2** and choose **Rename**.

19 Go to the **Rename Global** dialog box and type Input Impedance in the **New name** edit field.

20 Click **OK**.