Created in COMSOL Multiphysics 6.1

Generic 711 Coupler — An Occluded Ear-Canal Simulator

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of a 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 mold) 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.

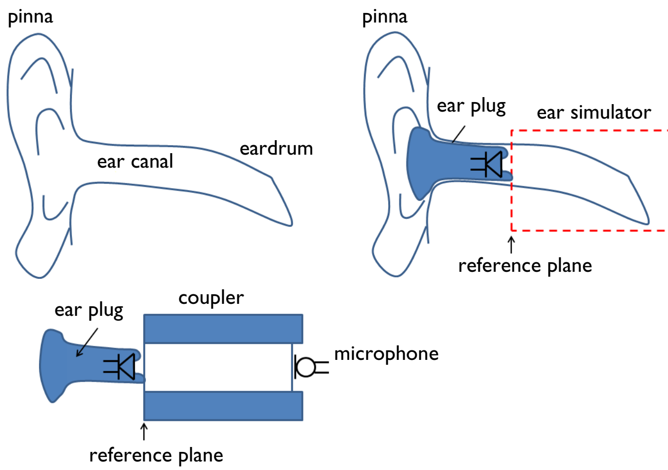


Figure 1: (upper left) sketch of the ear with pinna, ear canal, and eardrum, (upper right) ear canal occluded by an earplug with a loudspeaker, the occluded-ear canal is the part that the coupler is intended to model, (bottom) ear plug placed at the reference plane of the acoustic coupler including the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

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

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.

average human ear it does not mimic the large acoustic performance variations that exist between individual ears (see for example, Ref. 2).

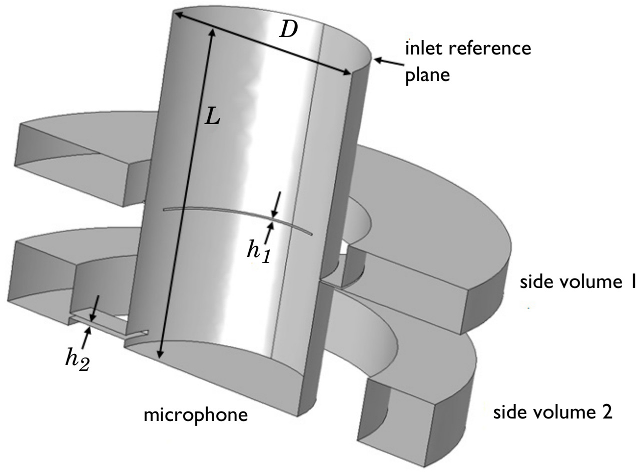


Figure 2: Sketch of the geometry used in the model (cut in half due to symmetry). The main volume is a cylinder of length L and diameter D. The two side volumes are attached to the main volume via slender slits of height h_1 and h_2.

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.

THERMOVISCOUS ACOUSTICS

As the thermal and viscous losses are important in the slits the finite element model is set up in COMSOL using two different methods; a model using *Thermoviscous Acoustics, Frequency Domain* interface and a model with *Pressure Acoustics, Frequency Domain* interface and the **Narrow Region Acoustics** feature to account for the losses in the slits. 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. Using air properties both layers will have roughly the same length for a given frequency. 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. The pressure acoustics and the thermoviscous acoustics domains are coupled using the built in Acoustics-Thermoviscous Acoustics Boundary multiphysics coupling found under the Multiphysics node in the model tree.

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_0 (for a constant volume displacement source):

$$\begin{aligned} Z_{trans} &= \frac{p_{mic}}{Q_{in}} \\ L_0 &= 10\log\left(\frac{\langle p_{mic}\rangle^2}{p_{ref}^2}\right) = 10\log(\langle p_{mic}\rangle^2) - L_{ref} \end{aligned} \tag{2}$$

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. Note that it is important to retain the sign of the velocity when computing Q_{in}, in this case such that you look "into" the coupler from the outside. The quantity exists as a predefined postprocessing variable on displacement, velocity, and acceleration boundary conditions in the pressure acoustics interface. The relevant variable `acpr.Zac` (the acoustic impedance) can be evaluated on surfaces. In systems with, for example, one symmetry its value needs to be divided with 2 (to account for the full area). The specific impedance on the boundary is named `acpr.Zi`.

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 displacement d_n of the boundary

$$d_n = d_0 \tag{4}$$

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

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

where $C_{\mathrm{mic}} = 0.62{\cdot}10^{-13}\ \mathrm{m^5/N}$ is the acoustic compliance, $R_{\mathrm{mic}} = 119{\cdot}10^{6}\ \mathrm{Ns/m^5}$ is the acoustic resistance, and $L_{\mathrm{mic}} = 710\ \mathrm{kg/m^4}$ is the acoustic mass. This is modeled using the built-in RCL option in the impedance boundary condition.

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

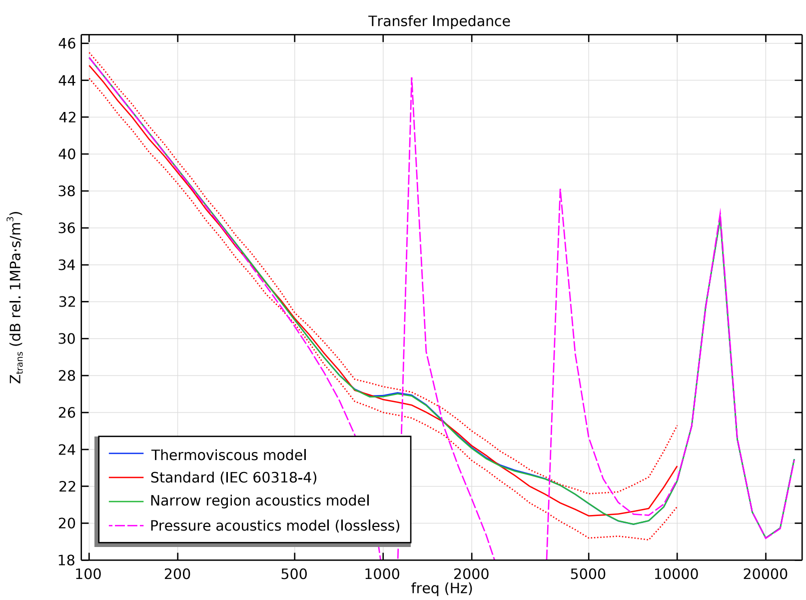


Figure 3: Transfer impedance as function of frequency for the thermoviscous model (blue line), the IEC standard curve (red line) including upper and lower tolerances (red dotted line), the system modeled through narrow region acoustics (green line) and the lossless pressure acoustics approach (dotted dark green line).

Also evident from the graph is that it is important to use thermoviscous acoustics to model such acoustic systems with small geometrical dimensions. The narrow region acoustics approach is in good agreement with the thermoviscous model results, showing that for some simple waveguide geometries (slits, rectangular and circular ducts of constant cross-section) the **Narrow Region Acoustics** feature can reduce substantially the complexity of the model while maintaining accuracy. 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 in the slits are much less pronounced and the only system losses are due to the RCL impedance condition representing the microphone.

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

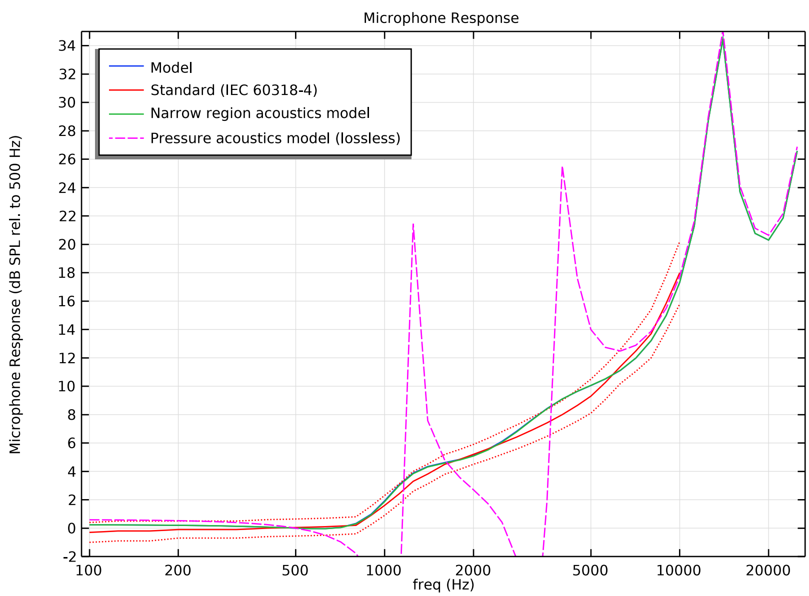


Figure 4: Microphone response as function of frequency for the thermoviscous model (blue line), the IEC standard curve (red line) including upper and lower tolerances (red dotted line), the system modeled through narrow region acoustics (green line) and the lossless pressure acoustics approach (dotted dark green line).

The input impedance of the system derived at the reference plane is shown in Figure 5 where the model results are compared to the results of the fully lossless model.

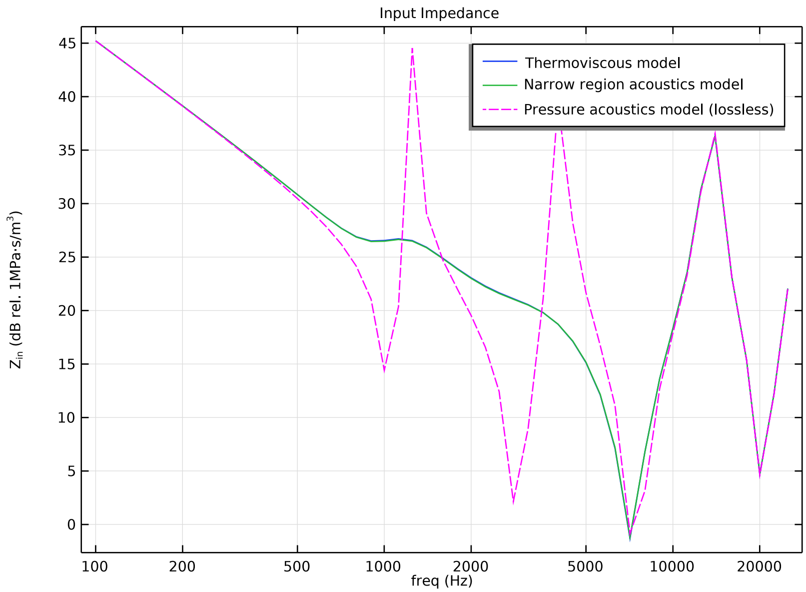


Figure 5: Input impedance as function of frequency for the thermoviscous model (blue line), the system modeled with narrow region acoustics (green line), and the lossless pressure acoustics approach (red line).

Figure 6, Figure 7, and Figure 8 depict the pressure distribution inside the 711 coupler at frequencies of 25 kHz, 14 kHz, and 900 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.

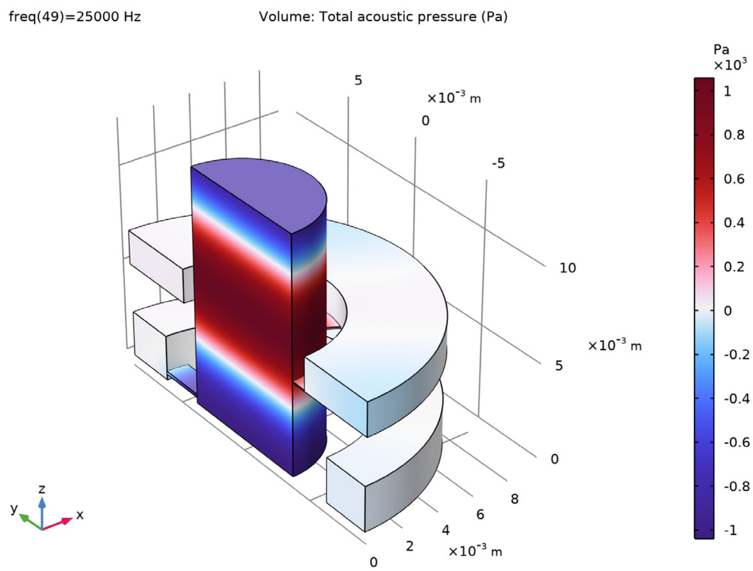


Figure 6: Instantaneous pressure distribution at f = 25 kHz.

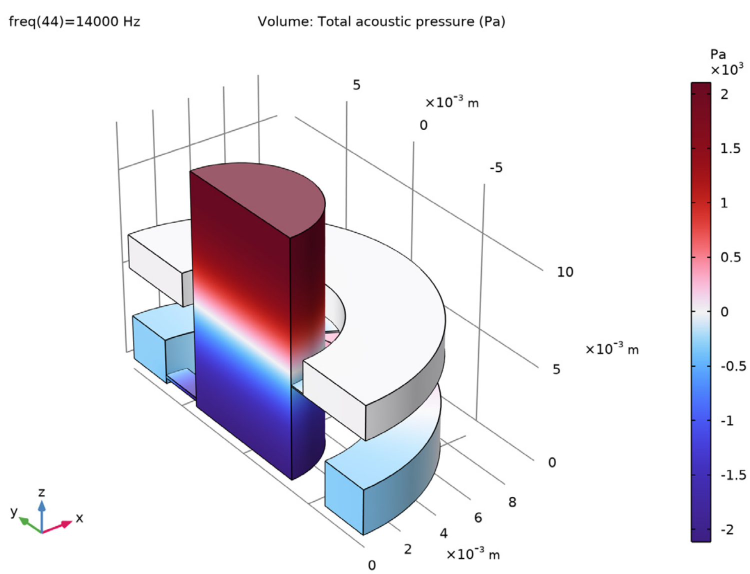


Figure 7: Instantaneous pressure distribution at f = 14 kHz.

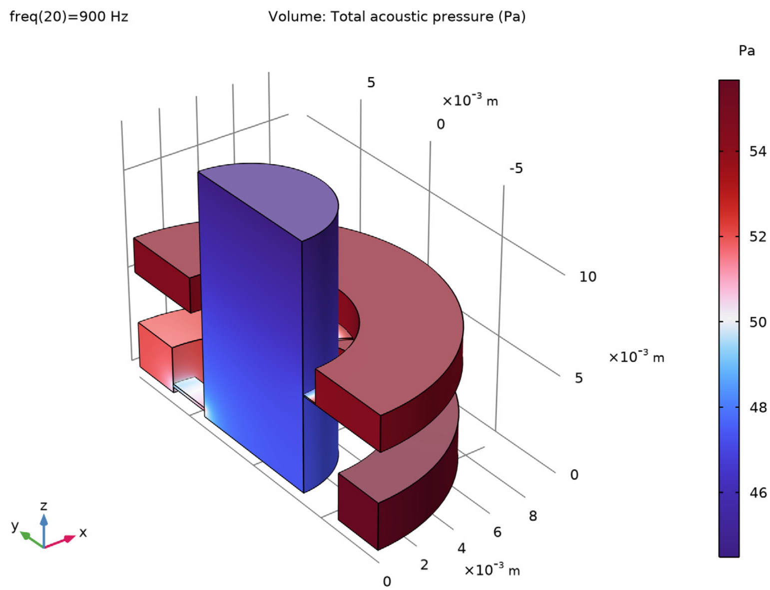


Figure 8: Instantaneous pressure distribution at f = 900Hz.

The losses in the thermoviscous model are shown in Figure 9 normalized to the total acoustic power in the system. The plot shows the effect of the different slits as the frequency increases, and how the microphone starts absorbing energy above 5 kHz.

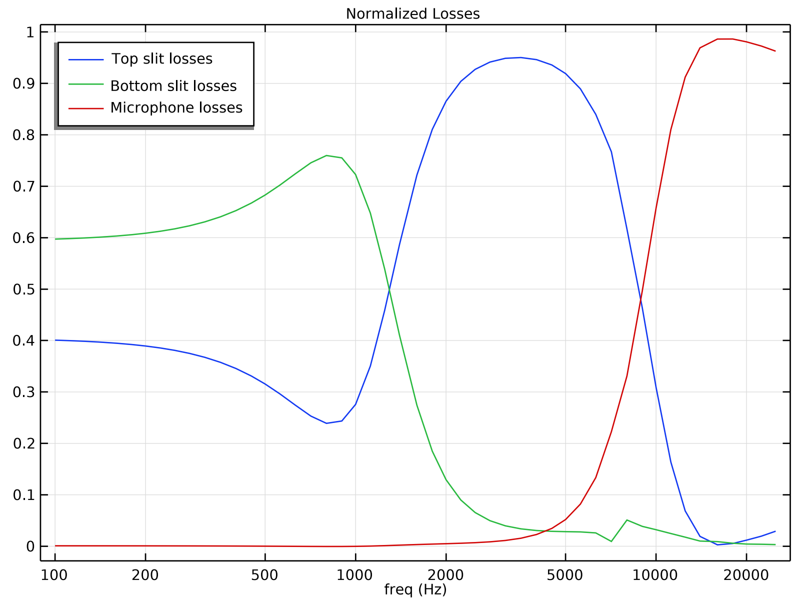


Figure 9: Normalized losses in the thermoviscous model.

The same conclusion could be drawn by plotting the streamline of acoustic intensity at the frequencies of the peak losses for each of the systems. Figure 10, Figure 11, and Figure 12 show the streamline of acoustic intensity at 800 Hz, 3,550 Hz and 18,000 Hz, matching the peak losses in the bottom slit, the top slit and the microphone respectively.

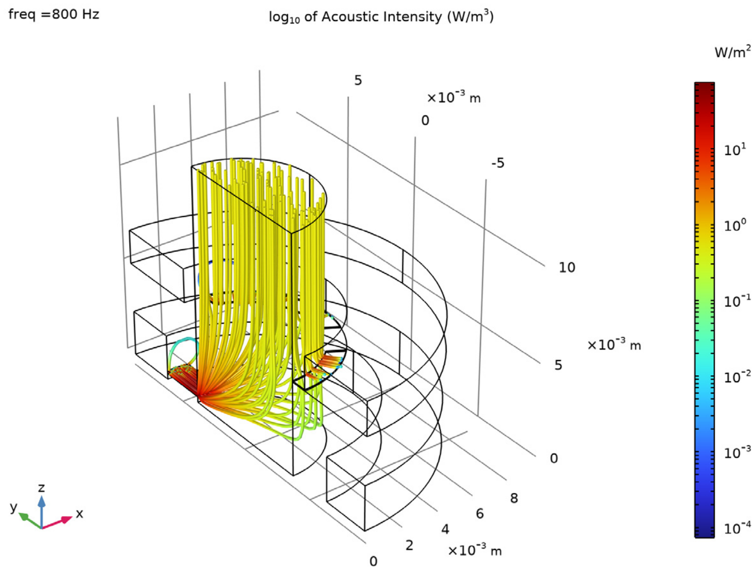


Figure 10: streamline of acoustic intensity at 800 Hz

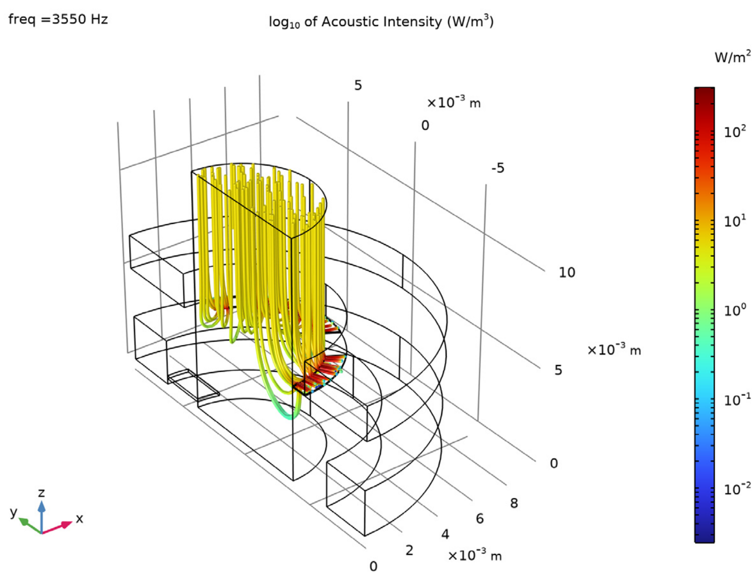


Figure 11: Streamline of acoustic intensity at 3,550 Hz

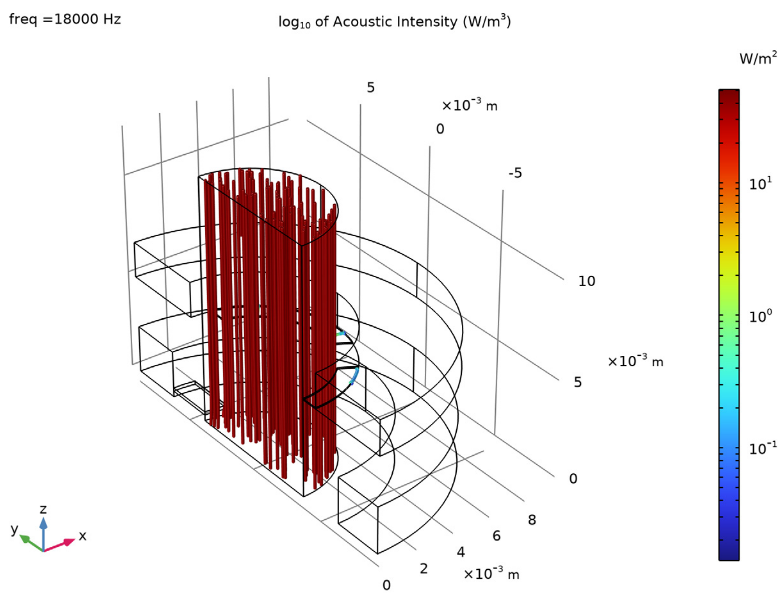


Figure 12: Streamline of acoustic intensity at 18,000 Hz

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, pp. 2492–2503, 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, no. 5, pp. 1596–1602, 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.

Application Library path: `Acoustics_Module/Tutorials,` `_Thermoviscous_Acoustics/generic_711_coupler`

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

3 Click **Add**.

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

5 Click **Add**.

6 Click **Done**.

GLOBAL DEFINITIONS

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

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 `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 (int1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

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

4 Click **Browse**.

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

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

7 Click **Import**.

8 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 (int2)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

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

4 Click **Browse**.

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

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

7 Click **Import**.

8 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

Import the model geometry from file by following these steps.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

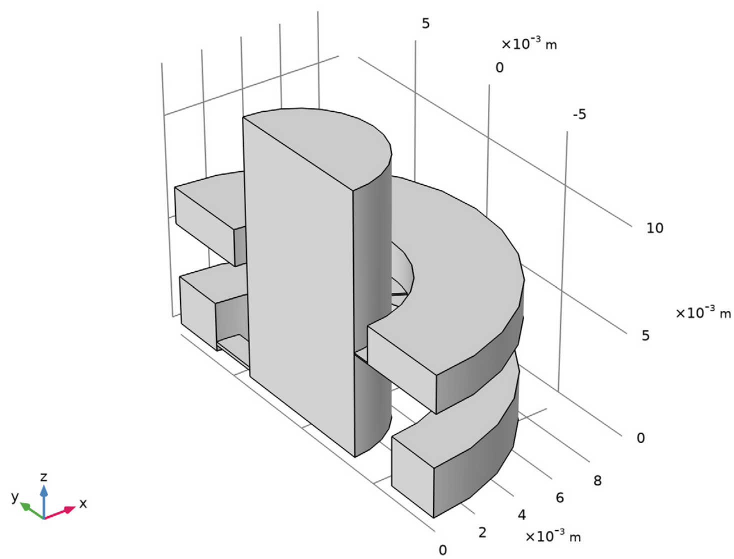
3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `generic_711_coupler.mphbin`.

5 Click **Import**.

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

The figure below shows the model geometry.



DEFINITIONS

Load a set of variables that define the microphone impedance (Equation 5), the acoustic transfer impedance (Equation 2), and the acoustic input impedance (Equation 4) from a file. Add operators to integrate values across the inlet plane, the microphone plane and the slit domains.

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 Click **Load from File**.

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

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, type `intop_in` in the **Operator name** text field.

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

4 Select Boundary 19 only.

Integration 2 (intop2)

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

2 In the **Settings** window for **Integration**, type `intop_mic` in the **Operator name** text field.

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

4 Select Boundary 17 only.

Integration 3 (intop3)

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

2 In the **Settings** window for **Integration**, type `intop_top_slit` in the **Operator name** text field.

3 Select Domains 3 and 6 only.

Integration 4 (intop4)

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

2 In the **Settings** window for **Integration**, type `intop_bottom_slit` in the **Operator name** text field.

3 Select Domain 5 only.

Next, make selections of different parts of the geometry to ease the subsequent application of boundary conditions. Then proceed and set up the acoustic model and the boundary conditions.

Pressure Acoustics (lossless)

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

2 Select Domains 1, 2, and 4 only.

3 In the **Settings** window for **Explicit**, type `Pressure Acoustics (lossless)` in the **Label** text field.

Thermoviscous Acoustics

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

2 Select Domains 3, 5, and 6 only.

3 In the **Settings** window for **Explicit**, type `Thermoviscous Acoustics` in the **Label** text field.

Symmetry

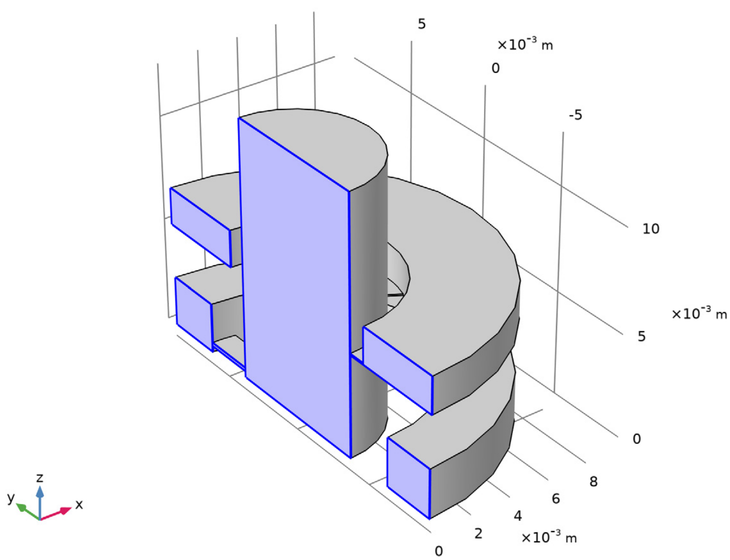
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 1, 5, 10, 15, 21, 25, and 27 only.

5 In the **Label** text field, type `Symmetry`.



ADD MATERIAL

1 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 Click **Add to Component** in the window toolbar.

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Pressure Acoustics (lossless)**.

Pressure Acoustics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)** click **Pressure Acoustics 1**.

2 In the **Settings** window for **Pressure Acoustics**, locate the **Model Input** section.

3 In the T text field, type `Tref`.

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**.

Normal Displacement 1

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

2 Select Boundary 19 only.

3 In the **Settings** window for **Normal Displacement**, locate the **Normal Displacement** section.

4 In the d_n text field, type `d0`.

Impedance 1

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

2 Select Boundary 17 only.

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

4 From the **Impedance model** list, choose **RCL**.

5 In the R_{ac} text field, type `Rmic`.

6 In the C_{ac} text field, type `Cmic`.

7 In the L_{ac} text field, type `Lmic`.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Thermoviscous Acoustics Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)** click **Thermoviscous Acoustics Model 1**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, locate the **Model Input** section.

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

4 In the **Model Builder** window, click **Thermoviscous Acoustics, Frequency Domain (ta)**.

5 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, locate the **Domain Selection** section.

6 From the **Selection** list, choose **Thermoviscous Acoustics**.

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**.

Next, create and set up the multiphysics coupling between the pressure acoustics domain and the thermoviscous acoustics domain.

MULTIPHYSICS

Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic-Thermoviscous Acoustic Boundary**.

2 In the **Settings** window for **Acoustic-Thermoviscous Acoustic Boundary**, locate the **Boundary Selection** section.

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

Add another Pressure Acoustics, Frequency Domain physics interface. You will use this to illustrate the effect of thermoviscous acoustics in the slit channels by comparing to a solution of the same problem using regular pressure acoustics and narrow region acoustics here.

ADD PHYSICS

1 In the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

4 Click **Add to Component 1** in the window toolbar.

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 2 (ACPR2)

1 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **Thermoviscous Acoustics**.

Pressure Acoustics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Frequency Domain 2 (acpr2)** click **Pressure Acoustics 1**.

2 In the **Settings** window for **Pressure Acoustics**, locate the **Model Input** section.

3 In the T text field, type `Tref`.

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**.

Couple the two pressure acoustics interfaces together. The simplest approach is to give the dependent variable, the pressure, the same name p. Note that this approach is only valid when coupling identical physics.

4 In the **Model Builder** window, click **Pressure Acoustics, Frequency Domain 2 (acpr2)**.

5 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, click to expand the **Dependent Variables** section.

6 In the **Pressure** text field, type `p`.

Now, add the narrow region acoustics features that will introduce losses in the narrow slits.

Narrow Region Acoustics 1

1 In the **Physics** toolbar, click **Domains** and choose **Narrow Region Acoustics**.

2 Select Domain 5 only.

3 In the **Settings** window for **Narrow Region Acoustics**, locate the **Duct Properties** section.

4 From the **Duct type** list, choose **Rectangular duct**.

5 In the W text field, type `2230 [um]`.

6 In the H text field, type `170 [um]`.

In this case, the slit is well approximated by a rectangular duct. The dimensions of the slit in the vertical direction and the total horizontal dimension are used to compute an equivalent loss instead of solving the set of thermoviscous equations.

Narrow Region Acoustics 2

1 In the **Physics** toolbar, click **Domains** and choose **Narrow Region Acoustics**.

2 Select Domains 3 and 6 only.

3 In the **Settings** window for **Narrow Region Acoustics**, locate the **Duct Properties** section.

4 From the **Duct type** list, choose **Slit**.

5 In the h text field, type `69[um]`.

As this slit has a very large aspect ratio, it is well approximated by a slit. In this case, only one of the dimensions of the slit is used to obtain the equivalent losses.

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

Start using a swept mapped mesh in the slits and then add a tetrahedral mesh in the remaining domains.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

2 Select Boundaries 14, 24, and 35 only.

Distribution 1

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

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

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

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

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 19 and 27 only.

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

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

Distribution 3

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

2 Select Edges 63 and 66 only.

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

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

Distribution 4

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

2 Select Edges 79 and 86 only.

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

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

Distribution 5

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

2 Select Edges 36 and 69 only.

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

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

Distribution 6

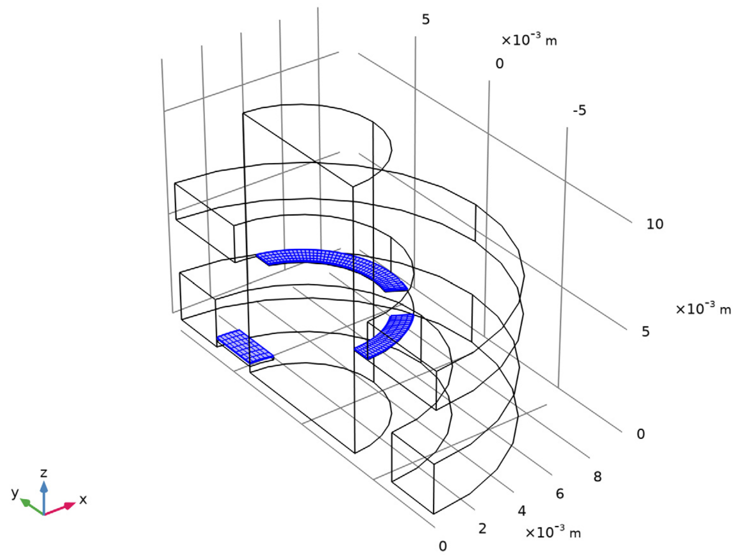
1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 37 and 50 only.

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

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

5 Click **Build Selected**.



The mapped mesh on the upper side of the slits looks like that in the figure above (switch to wireframe rendering).

Now, proceed to sweep the mesh in the slits.

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 `Lmax`.

5 In the **Minimum element size** text field, type `0.2[mm]`.

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 3, 5, and 6 only.

Distribution 1

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

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

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

4 Click **Build Selected**.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build Selected**.

Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size 1

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

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

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

4 Select Boundaries 17 and 19 only.

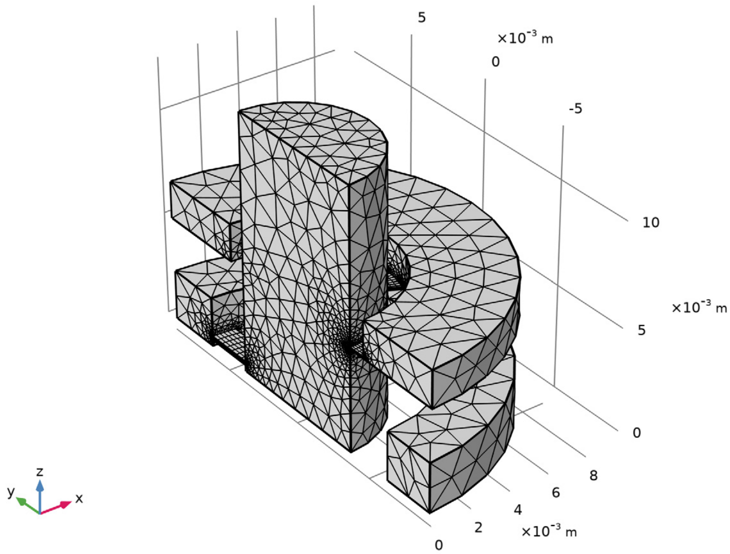
5 Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** check box. In the associated text field, type 1[mm].

8 Click **Build All**.

The mesh should look similar to that shown in the figure below.



ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Turn off the generation of the default plots and create your own. Leave this option on to have the default plot of Pressure Acoustics and Thermoviscous Acoustics generated.

STUDY 1 - THERMOVISCOUS MODEL

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

2 In the **Settings** window for **Study**, type `Study 1 - Thermoviscous Model` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - Thermoviscous Model** 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 box, choose **ISO preferred frequencies** from the **Entry method** list.
5 In the **Start frequency** text field, type 100.
6 In the **Stop frequency** text field, type 25000.
7 From the **Interval** list, choose **1/6 octave**.
8 Click **Replace**.
9 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.
10 Select the **Modify model configuration for study step** check box.
11 In the tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain 2 (acpr2)**.
12 Click **Disable in Model**.
13 In the **Home** toolbar, click **Compute**.

Note that the computation may take several minutes.

Add a second study to solve the model with narrow region acoustics. Disable thermoviscous acoustics and enable the second pressure acoustics model, in this way the system will use the narrow region acoustics. The results from the second study are stored in a separate dataset. Compare this second solution to the model that includes the complete set of equations.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Thermoviscous Acoustics, Frequency Domain (ta)**.
5 Click **Add Study** in the window toolbar.
6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2 - NARROW REGION ACOUSTICS

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

2 In the **Settings** window for **Study**, type `Study 2 - Narrow Region Acoustics` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Narrow Region Acoustics** 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 box, choose **ISO preferred frequencies** from the **Entry method** list.

5 In the **Start frequency** text field, type `100`.

6 In the **Stop frequency** text field, type `25000`.

7 From the **Interval** list, choose **1/6 octave**.

8 Click **Replace**.

Make some small changes to the default solver to use a fully coupled approach when solving the two pressure acoustics physics together.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2 - Narrow Region Acoustics>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.

4 Right-click **Study 2 - Narrow Region Acoustics>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** and choose **Fully Coupled**.

Select the direct solver that uses MUMPS, or change the solver in the selected Direct 1 solver.

5 In the **Settings** window for **Fully Coupled**, locate the **General** section.

6 From the **Linear solver** list, choose **Direct**.

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

Now create a last step where a pressure acoustics (Lossless) model is analyzed. To do this, you need to turn off the narrow region acoustics domains in the second pressure acoustics physics.

ADD STUDY

1 In the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Thermoviscous Acoustics, Frequency Domain (ta)**.

5 Find the **Multiphysics couplings in study** subsection. In the table, clear the **Solve** check box for **Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)**.

6 Click **Add Study** in the window toolbar.

7 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3 - PRESSURE ACOUSTICS

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3 - Pressure Acoustics` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

4 Clear the **Generate convergence plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3 - Pressure Acoustics** 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 box, choose **ISO preferred frequencies** from the **Entry method** list.

5 In the **Start frequency** text field, type `100`.

6 In the **Stop frequency** text field, type `25000`.

7 From the **Interval** list, choose **1/6 octave**.

8 Click **Replace**.

9 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

10 Select the **Modify model configuration for study step** check box.

11 In the tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain 2 (acpr2)>Narrow Region Acoustics 1** and **Component 1 (comp1)> Pressure Acoustics, Frequency Domain 2 (acpr2)>Narrow Region Acoustics 2**.

12 Click **Disable**.

With this, the narrow region acoustics will not be considered in the analysis, so the model becomes lossless.

Make some small changes to the default solver to use a fully coupled approach when solving the two pressure acoustics physics together.

13 Right-click **Study 3 - Pressure Acoustics>Step 1: Frequency Domain** and choose **Get Initial Value for Step**.

Solver Configurations

In the **Model Builder** window, expand the **Study 3 - Pressure Acoustics>Solver Configurations** node.

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Study 3 - Pressure Acoustics> Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node.

2 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

3 Right-click **Direct** and choose **Enable**.

4 In the **Settings** window for **Direct**, click **Compute**.

RESULTS

First, create plots of the pressure distribution inside the coupler. Then move on to plotting the transfer impedance, input impedance, and the microphone response.

Acoustic Pressure

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure` in the **Label** text field.

3 Locate the **Color Legend** section. Select the **Show units** check box.

Volume 1

1 Right-click **Acoustic Pressure** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Multiphysics> atb1.p_t - Total acoustic pressure - Pa**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Wave>Wave** in the tree.

5 Click **OK**.

6 In the **Acoustic Pressure** toolbar, click **Plot**.

The plot should look like Figure 6. To create Figure 7, and Figure 8 change the evaluation frequency to 14000 Hz and 900 Hz, respectively.

Next, generate a plot to show the streamline of the acoustic energy and investigate where the losses of the model are depending on the frequency.

Streamline - Acoustic Intensity

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 From the **Parameter value (freq (Hz))** list, choose **18000**.

4 In the **Label** text field, type `Streamline - Acoustic Intensity`.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `log<sub>10</sub> of Acoustic Intensity (W/m<sup>3</sup>)`.

7 In the **Parameter indicator** text field, type `freq =eval(acpr.freq) Hz`.

8 Locate the **Color Legend** section. Select the **Show units** check box.

Streamline 1

1 Right-click **Streamline - Acoustic Intensity** and choose **Streamline**.

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

3 In the **X-component** text field, type `if(isnan(acpr.p_t),ta.Ix,acpr.Ix)`.

4 In the **Y-component** text field, type `if(isnan(acpr.p_t),ta.Iy,acpr.Iy)`.

5 In the **Z-component** text field, type `if(isnan(acpr.p_t),ta.Iz,acpr.Iz)`.

6 Select Boundaries 11, 19, 29, 36, and 41 only.

7 Locate the **Streamline Positioning** section. In the **Number** text field, type `100`.

8 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `if(isnan(acpr.p_t),(ta.I_mag),(acpr.I_mag))`.

4 Locate the **Coloring and Style** section. From the **Scale** list, choose **Logarithmic**.

5 In the **Streamline - Acoustic Intensity** toolbar, click **Plot**.

The plot should look like Figure 10. To create Figure 11, and Figure 12 change the evaluation frequency to 3550 Hz and 800 Hz, respectively.

Plot the transfer impedance for the thermoviscous, the narrow region acoustics model, and the pressure acoustics (lossless) model as well as the curves given by the standard Figure 3.

Transfer Impedance

1 In the **Home** toolbar, click **Add Plot Group** and choose **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 **Label**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type Z_{trans} (dB rel. 1MPa·s/m³).

6 Locate the **Legend** section. From the **Position** list, choose **Lower 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
20*log10(abs(Ztrans/1e6[Pa*s/m^3]))		Thermoviscous model

Global 2

1 In the **Model Builder** window, 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
int_trans(freq)		Standard (IEC 60318-4)

4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Red**.

5 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Thermoviscous Model/ Solution 1 (sol1)**.

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

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

7 In the **Parameter values** list, choose values from 100 to 10000.

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

9 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

Global 3

1 Right-click **Global 2** and choose **Duplicate**.

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

3 In the table, enter the following settings:

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

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

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

Global 4

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

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

3 From the **Dataset** list, choose **Study 2 - Narrow Region Acoustics/Solution 2 (sol2)**.

4 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]))		Narrow region acoustics model

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

Global 5

1 Right-click **Global 4** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 3 - Pressure Acoustics/Solution 3 (sol3)**.

4 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]))`		`Pressure acoustics model (lossless)`

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Magenta**.

6 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

Transfer Impedance

1 In the **Model Builder** window, click **Transfer Impedance**.

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

3 Select the **Manual axis limits** check box.

4 In the **y minimum** text field, type `18`.

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

Microphone Response

1 Right-click **Transfer Impedance** and choose **Duplicate**.

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

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

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

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

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`L0`		`Model`

Global 2

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

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

3 In the table, enter the following settings:

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

Global 3

1 In the **Model Builder** window, click **Global 3**.

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

3 In the table, enter the following settings:

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

Global 4

1 In the **Model Builder** window, click **Global 4**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
LO		Narrow region acoustics model

Global 5

1 In the **Model Builder** window, click **Global 5**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
LO		Pressure acoustics model (lossless)

Microphone Response

1 In the **Model Builder** window, click **Microphone Response**.

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

3 In the **y minimum** text field, type -2.

4 In the **y maximum** text field, type 35.

5 In the **Microphone Response** toolbar, click **Plot**.

Now, plot the input impedance for the thermoviscous, the narrow region acoustics, and the pressure acoustics model as seen in Figure 5.

Input Impedance

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

2 In the **Settings** window for **1D Plot Group**, type Input 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 **y-axis label** check box. In the associated text field, type Z_{in} (dB rel. 1MPa·s/m³).

Global 1

1 Right-click **Input 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
20*log10(abs(Zin/1e6[Pa*s/m^3]))		Thermoviscous model

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2 - Narrow Region Acoustics/Solution 2 (sol2)**.

4 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]))		Narrow region acoustics model

Global 3

1 Right-click **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 3 - Pressure Acoustics/Solution 3 (sol3)**.

4 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]))		Pressure acoustics model (lossless)

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Magenta**.

6 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

8 In the **Input Impedance** toolbar, click **Plot**.

The plot should look like Figure 5.

Now, generate a plot showing the losses of the model as a function of the frequency.

Normalized Losses

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

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

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

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

Global 1

1 Right-click **Normalized Losses** 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
`-intop_top_slit(ta.diss_tot)/intop_in(down(acpr.Iz))`	1	`Top slit losses`
`-intop_bottom_slit(ta.diss_tot)/intop_in(down(acpr.Iz))`	1	`Bottom slit losses`
`intop_mic(down(acpr.Iz))/intop_in(down(acpr.Iz))`	1	`Microphone losses`

4 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

5 In the **Normalized Losses** toolbar, click **Plot**.

The plot should look like Figure 9.

In the same way as you have created the plot of the pressure distribution (the first 3D plot), by plotting `acpr.p_t` and `ta.p_t` together, other acoustic quantities can be depicted. For example, to plot the sound pressure level plot `acpr.Lp_t` and `ta.Lp_t` in the same plot. More details, resolved by thermoviscous acoustics, can be plotted inside the slits by, for example, plotting the acoustic temperature variations `ta.T_t`, the rms velocity `ta.v_rms`, or the instantaneous velocity `ta.v_inst`.