Created in COMSOL Multiphysics 5.5

Headphone on an Artificial Ear

Introduction

In this tutorial a headphone is simulated in a typical measurement setup. As headphones are closely coupled to the ear, it is not representative to measure their sensitivity in an acoustic free-field in the classical setup used for loudspeakers. The measurement requires the use of artificial heads and ears to accurately represent the usage conditions. This model shows the coupling of a circumaural headphone to a generic artificial ear.

To model all components in a headphone, this tutorial uses several physics and features. The foam is modeled with the Poroelastic Waves interface and coupled to Pressure Acoustics, Frequency Domain interface for the air domains. The Interior Perforated Plate condition is used to model the perforated plates and meshes in the headphone casing. The artificial ear is coupled to a simplified ear canal and the impedance of the ear drum is specifically considered in the model. The dynamic speaker driver is modeled through a lumped approach following Ref. 1.

Lumped representations of drivers are well known and widely used in the industry. The parameters that characterize the low-frequency performance of a loudspeaker, commonly known as the Thiele-Small or the small-signal parameters, are obtained from Ref. 2. This lumped model is coupled to the 3D Pressure Acoustics model describing the surrounding air domain.

Note: Many of the working principles of the lumped speaker model are described in the Lumped Loudspeaker Driver model. Application Library path `Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver`.

Model Definition

GEOMETRY

A schematic section of the model is shown in Figure 2. The pinna (peach color) is obtained from a 3D scan of an actual human ear. The ear canal has been idealized as a cylinder of 7.5 mm diameter and 19.8 mm length. The pinna and ear canal have been rotated to maintain the headphone oriented in global coordinates. The acoustic domain is shown in three regions; the pressure chamber (blue) the external domain (light blue) and the perfectly matched layer domain (dark blue). The driver is included in the model as its lumped equivalent (Electrical Circuit interface) enforcing a velocity on the diaphragm (yellow line). The pressure drop across the diaphragm is coupled back to the circuit. The different chambers of the acoustic domain are connected through perforated plates (green lines). The headphone casing (gray) is considered as rigid (the model can be extended to model the casing as an elastic structure). The foam (red) is a Poroelastic Waves domain completely fixed on the boundaries attached to the skin and the headphone casing.

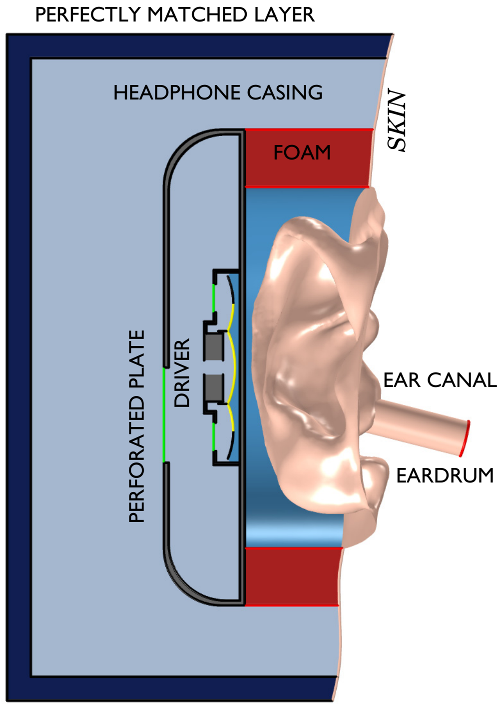


Figure 1: Schematic representation of the model.

PARAMETERS

The model parameters are given in the table below. The speeds defined in the table are used exclusively for the definition of the mesh size in the different domains. The value of 272 m/s is the speed of the fast pressure waves in the poroelastic waves (PELW) domain. The maximum mesh size h_{max} is in general given by

$$h_{\mathrm{max}} = \frac{1}{5}\frac{\min(c_{\mathrm{i}})}{f}$$

where c_{i} represents all the wave speed present in a model and f is the frequency. In a pure fluid there is just one speed of sound. By using the speed of the fast pressure waves in the PELW domain, we will be under-resolving the slow pressure waves and the shear waves in the PELW domain. This is done here to reduce the model size when solving the tutorial model. Ideally, the mesh should consider the minimum of the of the pressure waves speeds (variables `pelw.cp_fast` and `pelw.cp_slow`) and shear waves (variable `pelw.cs_poro`) in the PELW domain.

TABLE 1: MODEL PARAMETERS.

VARIABLE	VALUE	DESCRIPTION
f_{max}	20.0 kHz	Maximal frequency
c_{air}	343 m/s	Speed of sound in air
c_{poro}	272 m/s	Speed of wave to be resolved by the mesh (here the fast pressure wave speed in the PELW domain)

The model includes the driver of the headphone through a lumped equivalent. Thiele-Small parameters obtained from Ref. 2 are used in this model and listed in the table below.

TABLE 2: THIELE SMALL PARAMETERS.

VARIABLE	VALUE	DESCRIPTION
R_{g}	0.8 Ω	Cable resistance
n_{e}	0.7	Voice coil loss factor
R_{E}	124.3 Ω	Voice coil resistance
L_{E}	5.53 mH	Voice coil inductance (constant)
C_{MS}	$2.51\cdot10^{-3}$ m/N	Suspension compliance
R_{MS}	$12.9\cdot10^{-3}$ N·s/m	Suspension mechanical losses
M_{MD}	314.9 µg	Moving mass (voice coil and diaphragm)
BL	4.56 T·m	Force factor, flux density (B) times coil length (L)
V_0	$200\sqrt{2}$ mV	Driving voltage (peak)

The model includes a set of perforated plates connecting the different chambers of the headset. The perforated plate parameters used in the model are shown in the table below.

TABLE 3: PERFORATED PLATE PARAMETERS.

VARIABLE	VALUE	DESCRIPTION
$\mathrm{Rad}_{\mathrm{p1}}$	10 mm	Radius of the perforated plate 1
n_1	1	Number of circles defining the plate 1
d_{h1}	0.5 mm	Diameter of the holes in plate 1
t_{p1}	0.5 mm	Thickness of perforated plate 1
N_{h1}	150	Number of holes in the perforated plate 1

TABLE 3: PERFORATED PLATE PARAMETERS.

VARIABLE	VALUE	DESCRIPTION
$\mathrm{Rad}_{\mathrm{p2}}$	6 mm	Radius of the perforated plate 2
n_2	4	Number of circles defining the plate 2
d_{h2}	0.5 mm	Diameter of the holes in plate 2
t_{p2}	0.5 mm	Thickness of perforated plate 2
N_{h2}	200	Number of holes in the perforated plate 2
$\mathrm{Rad}_{\mathrm{p3}}$	6 mm	Radius of the perforated plate 3
n_3	4	Number of circles defining the plate 3
d_{h3}	0.5 mm	Diameter of the holes in plate 3
t_{p3}	0.5 mm	Thickness of perforated plate 3
N_{h3}	300	Number of holes in the perforated plate 3

Each of the Interior Perforated Plate condition uses an area porosity derived from the parameters listed previously.

The porous material parameters used are those for a generic foam with parameters taken from Ref. 3. The model does not include any compression or prestressing of the foam. Getting a general constitutive model, that predict how all porous properties change with local compression or deformation, is extremely difficult. To include the effects of prestress will typically rely on measuring the porous properties under various compression/deformation states to get local material values. This can, for example, be achieved in impedance tube measurements as shown in Ref. 4, where an optimization is used to fit the material parameters to the test data. In general it should be noticed that good material data is important for the quality of numerical simulations.

BOUNDARY CONDITIONS

The model makes use of two of the **Physiological** impedance models, described in the *Acoustics Module User's Guide,* to accurately represent the skin (**Human skin**) and the eardrum (**Human ear drum**).The boundaries of the model that included the skin impedance condition are shown in Figure 2.

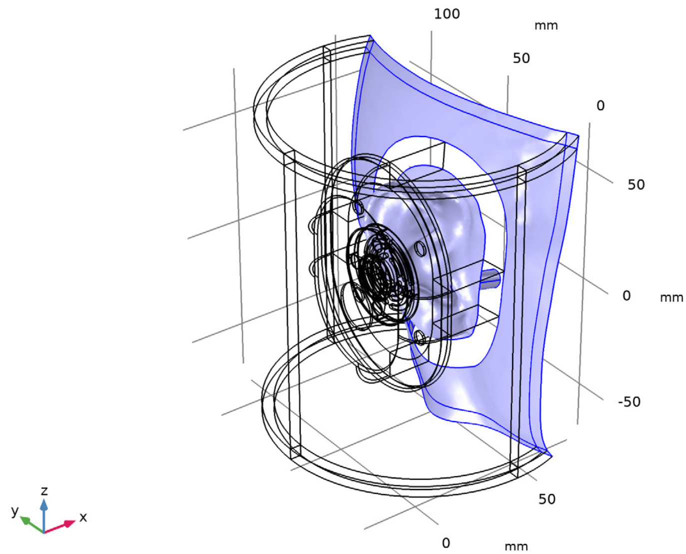


Figure 2: Boundaries of the acoustic domain with skin impedance.

Details about the lumped driver approach used in this model are found in the Lumped Loudspeaker Driver model and in the Modeling Instructions below. The perforated plates of the headset modeled through the Interior Perforated Plate condition, is described in detail in the *Acoustics Module User's Guide* in the *Theory For The Interior Impedance Models* section.

The sound pressure level on the skin (on and around the ear) at four different frequencies is shown in Figure 3. At the lowest frequencies the effect of the foam is clearly visible, where the large transition in SPL is seen.

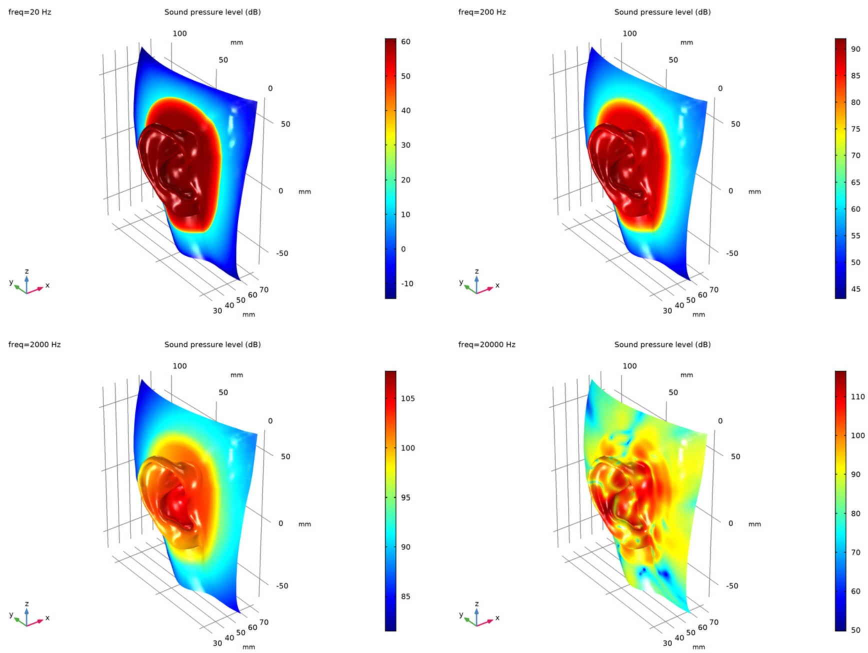


Figure 3: Sound pressure level at the skin at different frequencies.

The average sound pressure level at the eardrum is shown in Figure 4. This model has been solved on a high performance computer (HPC) with a finer mesh to capture the slow pressure waves and the shear waves. These results are imported and compared to the current model. The results of this finer meshed model show good agreement at low frequency (as all pressure waves are correctly resolved at low frequency in both models) but show significant differences as the frequency increases. The model requires about 28 GB of RAM to solve with the coarse mesh (used in the model) while it requires about 100 GB to solve with the default solver on the HPC system (single node). The model setup and the mesh can easily be modified to resolve all wave speed by modifying the parameter `cporo` from `272[m/s]` (fast pressure wave speed) to `96[m/s]` (shear wave speed, the slowest wave).

Although it is not shown in this tutorial, it is possible to explore the effect the perforates/ meshes have on the on the sound pressure level by modifying the Interior Perforated Plate parameters.

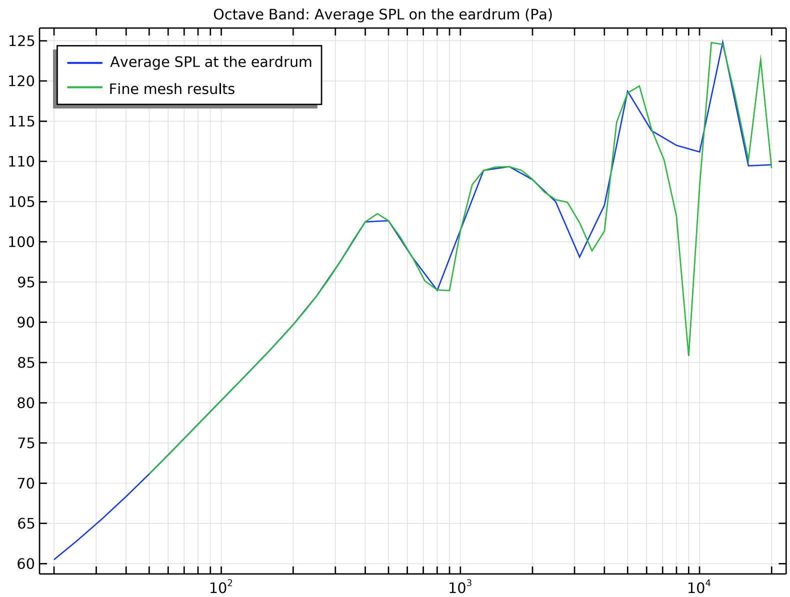


Figure 4: Sound pressure level at the eardrum.

Notes About the COMSOL Implementation

As described previously, and in the Modeling Instructions below, only the fast pressure waves have been considered while choosing the mesh size in the PELW domain. This has been done to limit the size of the model and make sure that the model can run on a computer with 32 GB of RAM. The solver setup in the model uses an iterative solver while the fine mesh model has been solved using a direct solver. It is very important to use a tight relative tolerance in the stationary solver (for this model `2e-7` is a good choice) when working with iterative solvers. It is good practice to do a convergence analysis on the relative tolerance until the results remain unaltered.

The PML in the model is set up using the **User defined** option for the **Geometry Type**. This is the case as the automatic detection fails when the PML is only part of a cylindrical layer (and is cut using a complex surface). Three PML region have been defined with different expressions for the **Distance function**. One definition for the top, one for the sides, and one for the corners. The distance function is a mathematical expression that describes the distance from the inner PML boundary to the outer boundary.

References

1. *Lumped Loudspeaker Driver Model Documentation*, from the COMSOL Application Library.

2. C. A. Poldy, "Headphones," in J. Borwick, *Loudspeaker and Headphone Handbook,* 3rd ed. Focal Press, 2001.

3. J. F. Allard and N. Atalla, *Propagation of Sound in Porous Media, Modeling Sound Absorbing Materials*, 2nd Edition, Wiley, 2009.

4. *Impedance Tube Parameter Estimation with Data Generation*, from the COMSOL Application Library.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/headphone_artificial_ear`

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 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Model parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

Parameters 2

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Perforated plates parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

Parameters 3

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Thiele-Small Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

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**.

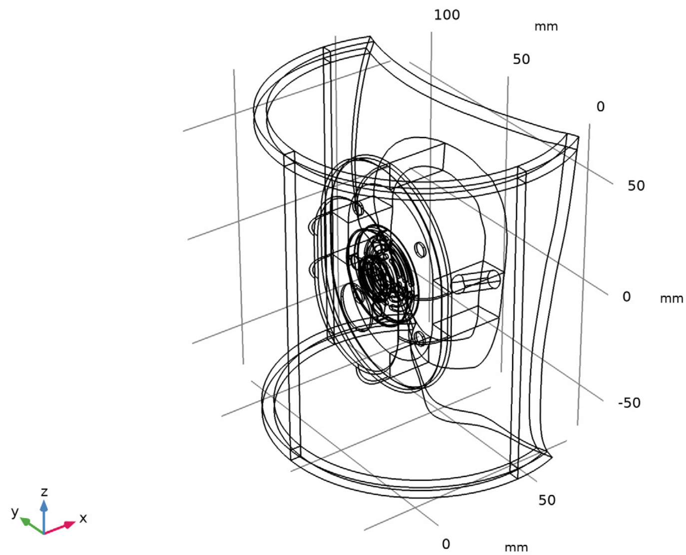
4 Browse to the model's Application Libraries folder and double-click the file `headphone_artificial_ear_geometry.mphbin`.

5 Click **Import**.

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

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

The figure below shows the model geometry.



In the following steps we will create the selections that will be used to define the model.

DEFINITIONS

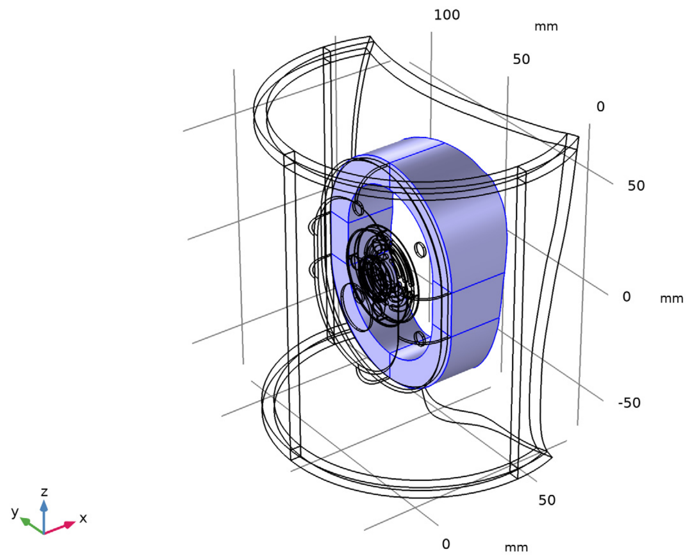
Explicit 1

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

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

3 Select Domains 15–20 only.

The selection should look like this.



Explicit 2

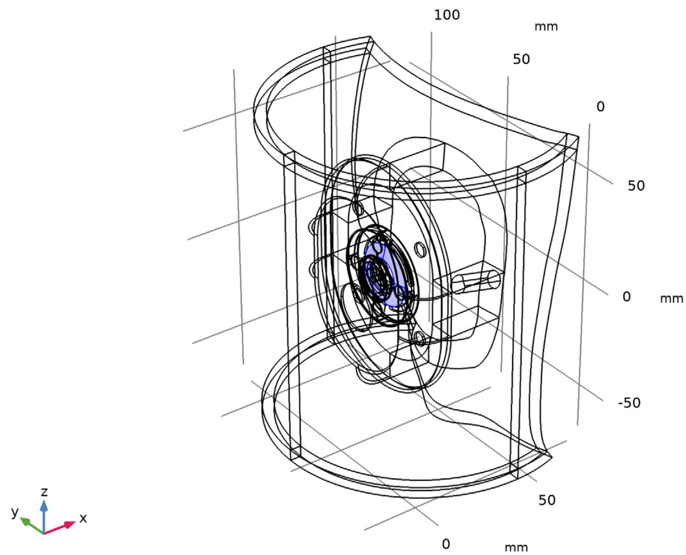
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 150, 152, 156, and 158 only.

The selection should look like this.



Explicit 3

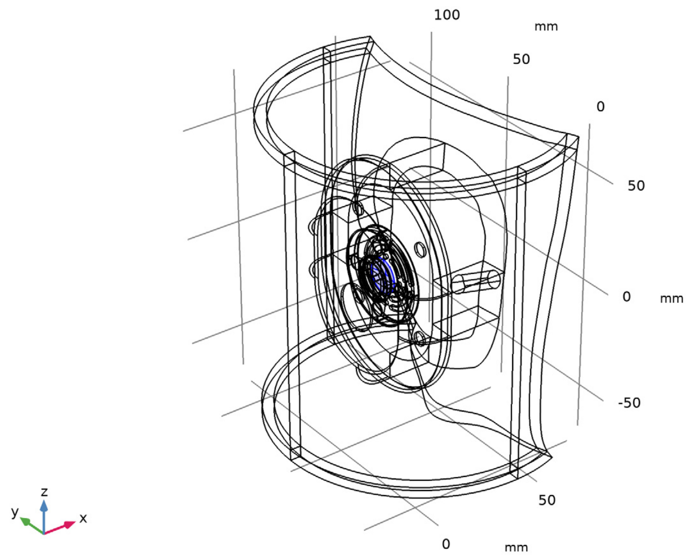
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 151, 153, 157, and 159 only.

The selection should look like this.



Explicit 4

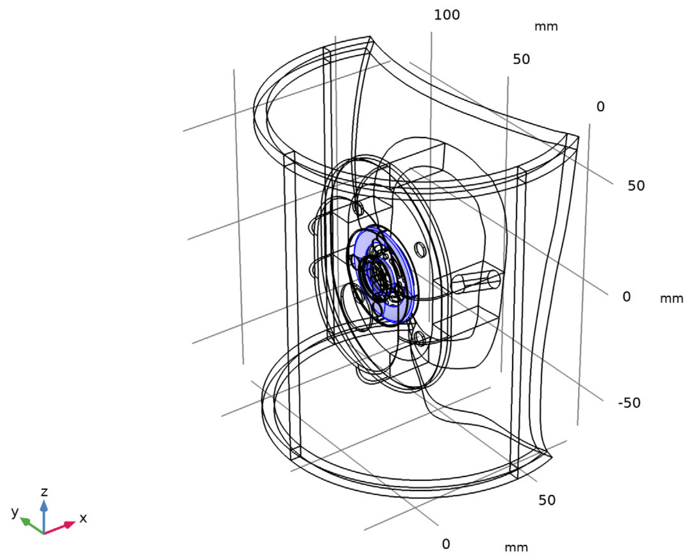
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Interior sound hard boundary` in the **Label** text field.

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

4 Select Boundaries 126–129, 146–149, 154, 155, 160, and 161 only.

The selection should look like this.



Explicit 5

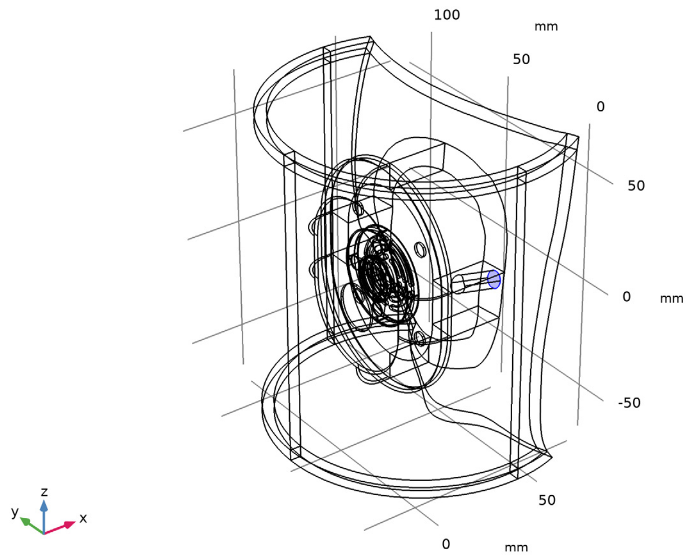
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundary 333 only.

The selection should look like this.



Explicit 6

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

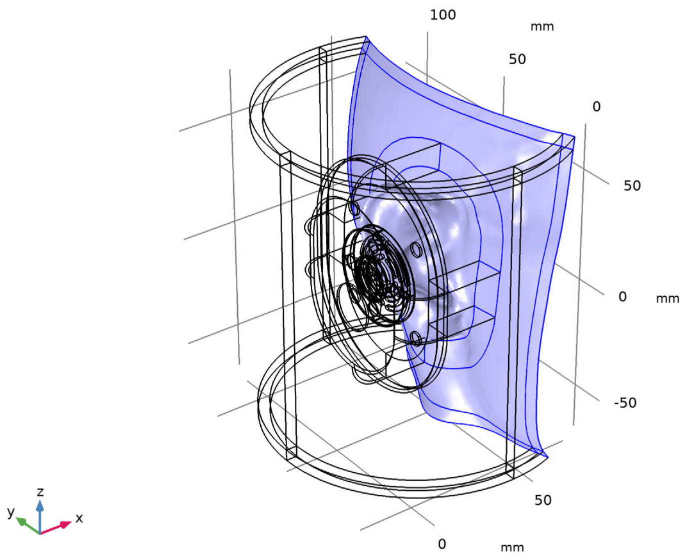
2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 In the **Label** text field, type `Skin with PML`.

5 Select Boundaries 289, 316–332, 334, and 335 only.

The selection should look like this.



Explicit 7

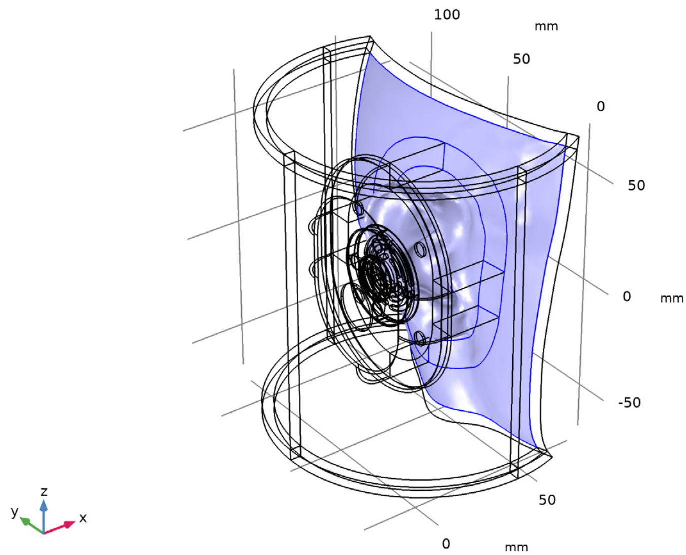
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Skin without PML` in the **Label** text field.

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

4 Select Boundaries 289, 317–322, and 324–328 only.

The selection should look like this.



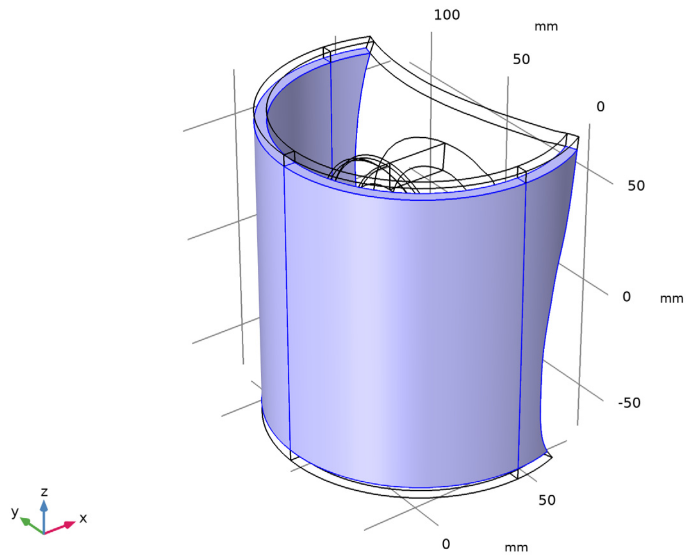
Explicit 8

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

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

3 Select Domains 3, 4, 22, and 25 only.

The selection should look like this.



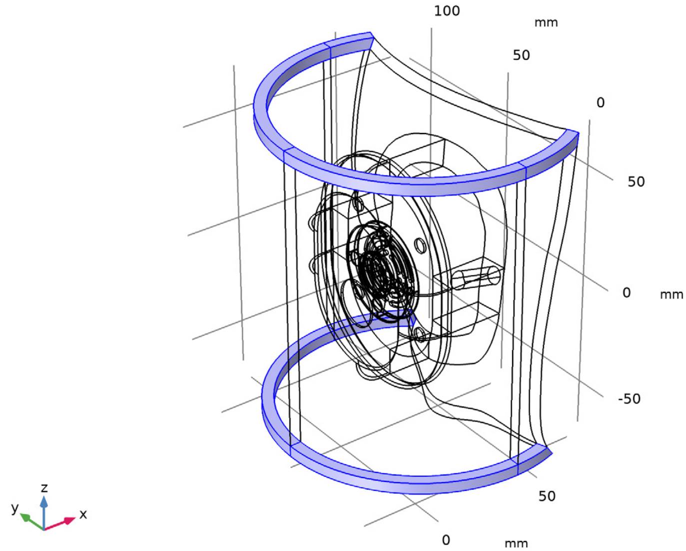
Explicit 9

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

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

3 Select Domains 1, 2, 5, 6, 21, 23, 24, and 26 only.

The selection should look like this.



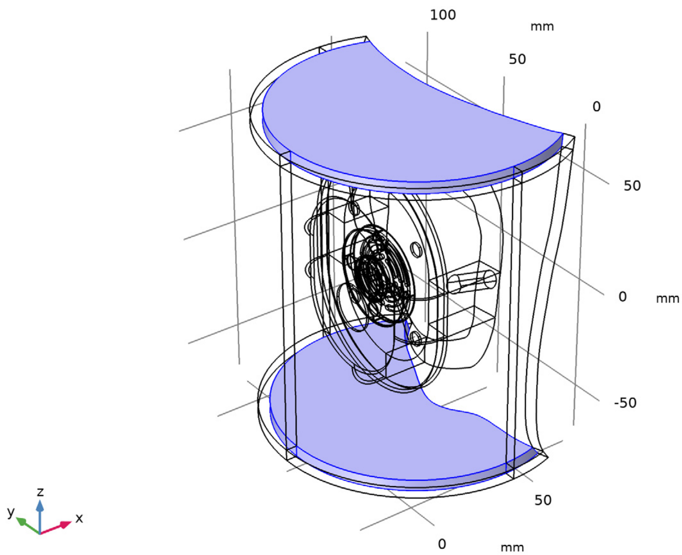
Explicit 10

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

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

3 Select Domains 7 and 9 only.

The selection should look like this.



Explicit 11

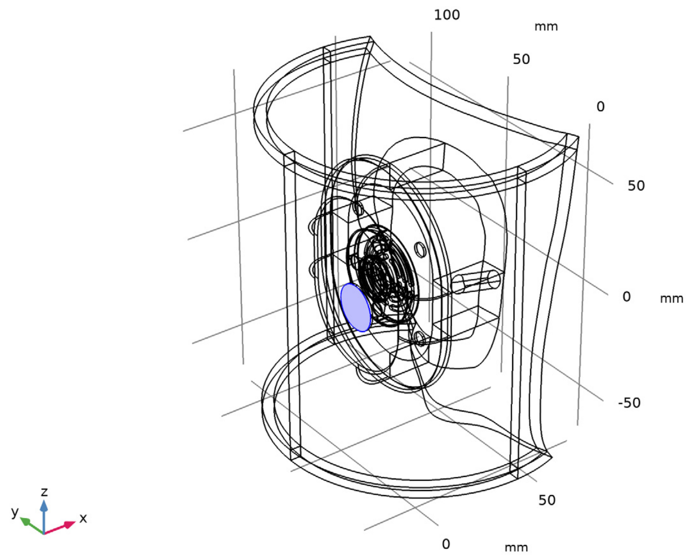
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Perforated plate 1` in the **Label** text field.

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

4 Select Boundary 32 only.

The selection should look like this.



Explicit 12

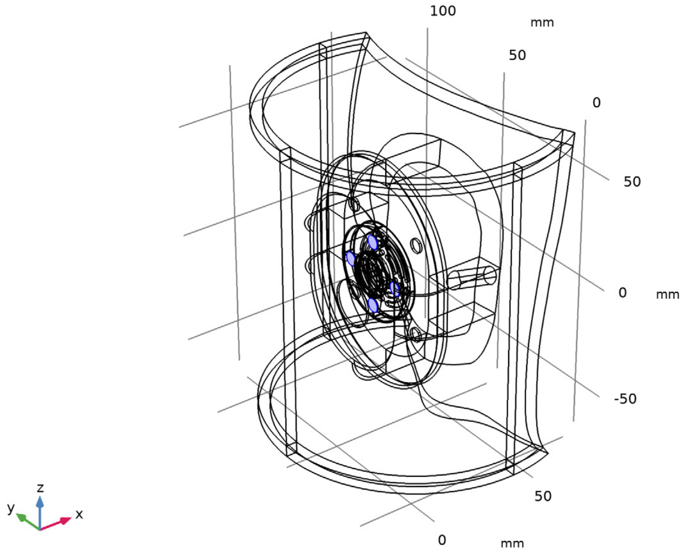
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 88, 95, 98, and 113 only.

The selection should look like this.



Explicit 13

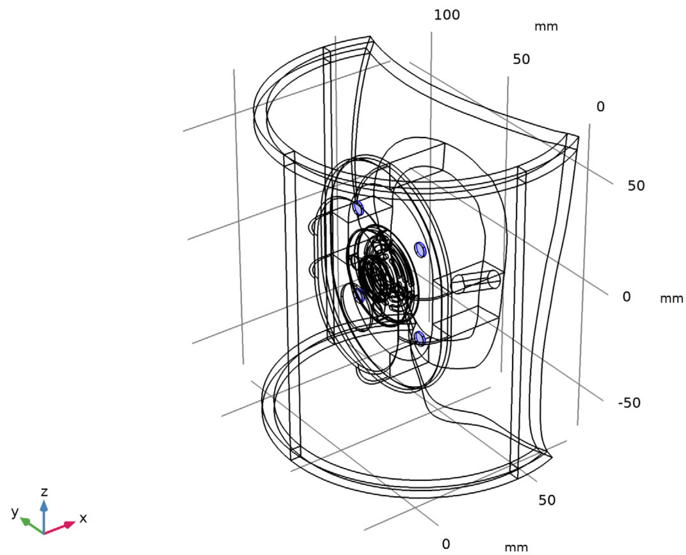
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Perforated plate 3` in the **Label** text field.

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

4 Select Boundaries 272, 273, 282, and 283 only.

The selection should look like this.



Explicit 14

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

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

3 Locate the **Input Entities** section. Select the **All domains** check box.

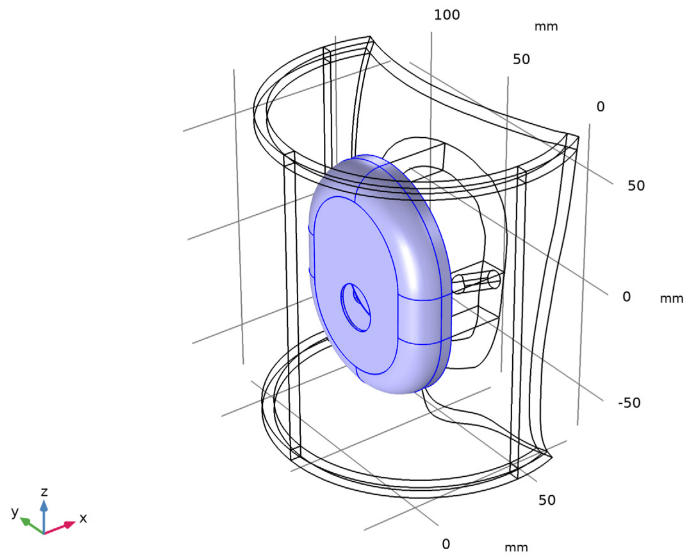
Explicit 15

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

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

3 Select Domains 10 and 12 only.

The selection should look like this.



Difference 1

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

2 In the **Settings** window for **Difference**, type `Air with PML` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **All Domains** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click **Add**.

8 In the **Add** dialog box, in the **Selections to subtract** list, choose **Foam** and **Plastic casing**.

9 Click **OK**.

Difference 2

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

2 In the **Settings** window for **Difference**, type `Air without PML` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **Air with PML** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click **Add**.

8 In the **Add** dialog box, in the **Selections to subtract** list, choose **PML sides**, **PML corners**, and **PML caps**.

9 Click **OK**.

Union 1

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

2 In the **Settings** window for **Union**, type `Air boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Eardrum**, **Skin with PML**, **Perforated plate 1**, **Perforated plate 2**, and **Perforated plate 3**.

6 Click **OK**.

Union 2

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

2 In the **Settings** window for **Union**, type `Moving membrane` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Moving membrane positive** and **Moving membrane negative**.

6 Click **OK**.

Difference 3

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

2 In the **Settings** window for **Difference**, type `Meshed domains without PML and foam` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **All Domains** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click **Add**.

8 In the **Add** dialog box, in the **Selections to subtract** list, choose **Foam**, **PML sides**, **PML corners**, **PML caps**, and **Plastic casing**.

9 Click **OK**.

Union 3

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

2 In the **Settings** window for **Union**, type `PML` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **PML sides**, **PML corners**, and **PML caps**.

5 Click **OK**.

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, type `Integration on the moving membrane` in the **Label** text field.

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

4 From the **Selection** list, choose **Moving membrane**.

Variables 1

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

2 In the **Settings** window for **Variables**, type `Model variables` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

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

ADD PHYSICS

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

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

3 In the tree, select **AC/DC>Electrical Circuit (cir)**.

4 Click **Add to Selection** in the window toolbar.

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

ELECTRICAL CIRCUIT (CIR)

Voltage Source V1

1 Right-click **Component 1 (comp1)>Electrical Circuit (cir)** and choose **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type `V0`.

Resistor R1

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `R_g`.

Resistor R2

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	3

4 Locate the **Device Parameters** section. In the R text field, type `R_E`.

Inductor L1

1 In the **Electrical Circuit** toolbar, click **Inductor**.

2 In the **Settings** window for **Inductor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the *L* text field, type `L_E`.

Resistor R3

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the *R* text field, type `Rp_E`.

Inductor L2

1 In the **Electrical Circuit** toolbar, click **Inductor**.

2 In the **Settings** window for **Inductor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	6
n	7

4 Locate the **Device Parameters** section. In the *L* text field, type `M_MD[H/kg]`.

Current-Controlled Voltage Source H1

1 In the **Electrical Circuit** toolbar, click **Current-Controlled Voltage Source**.

2 In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	4
n	0

4 Locate the **Device Parameters** section. From the **Measure current for device** list, choose **Inductor L2**.

Current-Controlled Voltage Source H2

1 In the **Electrical Circuit** toolbar, click **Current-Controlled Voltage Source**.

2 In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	6
n	0

4 Locate the **Device Parameters** section. In the **Gain** text field, type `BL[m/Wb*ohm]`.

5 From the **Measure current for device** list, choose **Resistor R2**.

Resistor R4

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	7

4 Locate the **Device Parameters** section. In the R text field, type `R_MS[ohm/kg*s]`.

Capacitor C1

1 In the **Electrical Circuit** toolbar, click **Capacitor**.

2 In the **Settings** window for **Capacitor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	8

4 Locate the **Device Parameters** section. In the C text field, type `C_MS[F*N/m]`.

Voltage Source V2

1 In the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	**Node names**
p	9
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type `-F_D[V/N]`.

COMPONENT 1 (COMP1)

In the **Home** toolbar, click **Windows** and choose **Add Physics**.

ADD PHYSICS

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

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

3 Click **Add to Selection** in the window toolbar.

4 In the **Model Builder** window, click **Component 1 (comp1)**.

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

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

2 From the **Selection** list, choose **Air with PML**.

Impedance 1

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

2 In the **Settings** window for **Impedance**, type `Eardrum Impedance` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Eardrum**.

4 Locate the **Impedance** section. From the **Impedance model** list, choose **Physiological**.

5 From the list, choose **Human ear drum**.

Impedance 2

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

2 In the **Settings** window for **Impedance**, type `Skin impedance` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Skin with PML**.

4 Locate the **Impedance** section. From the **Impedance model** list, choose **Physiological**.

The selection should look like Figure 2.

Interior Sound Hard Boundary (Wall) 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.
2 In the **Settings** window for **Interior Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Interior sound hard boundary**.

Interior Perforated Plate 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Perforated Plate**.
2 In the **Settings** window for **Interior Perforated Plate**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Perforated plate 1**.
4 Locate the **Interior Perforated Plate** section. In the d_h text field, type `dh1`.
5 In the t_p text field, type `tp1`.
6 In the σ text field, type `sigma1`.

Interior Perforated Plate 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Perforated Plate**.
2 In the **Settings** window for **Interior Perforated Plate**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Perforated plate 2**.
4 Locate the **Interior Perforated Plate** section. In the d_h text field, type `dh2`.
5 In the t_p text field, type `tp2`.
6 In the σ text field, type `sigma2`.

Interior Perforated Plate 3

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Perforated Plate**.
2 In the **Settings** window for **Interior Perforated Plate**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Perforated plate 3**.
4 Locate the **Interior Perforated Plate** section. In the d_h text field, type `dh3`.
5 In the t_p text field, type `tp3`.
6 In the σ text field, type `sigma3`.

Interior Normal Velocity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Normal Velocity**.
2 In the **Settings** window for **Interior Normal Velocity**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Moving membrane**.

4 Locate the **Interior Normal Velocity** section. Specify the $\mathbf{v}_0$ vector as

-u_D	x
0	y
0	z

COMPONENT 1 (COMP1)

In the **Home** toolbar, click **Windows** and choose **Add Physics**.

ADD PHYSICS

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

2 In the tree, select **Acoustics>Elastic Waves>Poroelastic Waves (pelw)**.

3 Click **Add to Selection** in the window toolbar.

4 In the **Model Builder** window, click **Component 1 (comp1)**.

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

POROELASTIC WAVES (PELW)

1 In the **Settings** window for **Poroelastic Waves**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **Foam**.

Fixed Constraint 1

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

2 Select Boundaries 260, 262, 265, 274, 278, 284, 318–322, and 326 only.

Poroelastic Material 1

1 In the **Model Builder** window, click **Poroelastic Material 1**.

2 In the **Settings** window for **Poroelastic Material**, locate the **Poroelastic Model** section.

3 From the **Model** list, choose **Biot-Allard (thermal and viscous losses)**.

MULTIPHYSICS

Acoustic-Porous Boundary 1 (apb1)

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

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

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

The following steps define the PML used in the model. The number of stretching directions and the distance function are defined manually to make sure that the PML works as intended.

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **PML sides**.

4 Locate the **Geometry** section. From the **Type** list, choose **User defined**.

5 In the table, enter the following settings:

	Distance function (m)	Thickness (m)
Direction 1	sqrt((x-40[mm])^2+(y-50[mm])^2)-60[mm]	5[mm]

Perfectly Matched Layer 2 (pml2)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **PML caps**.

4 Locate the **Geometry** section. From the **Type** list, choose **User defined**.

5 In the table, enter the following settings:

	Distance function (m)	Thickness (m)
Direction 1	abs(z)-65 [mm]	5[mm]

Perfectly Matched Layer 3 (pml3)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **PML corners**.

4 Locate the **Geometry** section. From the **Type** list, choose **User defined**.

5 From the **Number of stretching directions** list, choose **2**.

6 In the table, enter the following settings:

	Distance function (m)	Thickness (m)
Direction 1	`sqrt((x-40[mm])^2+(y-50[mm])^2)-60[mm]`	`5[mm]`
Direction 2	`abs(z)-65 [mm]`	`5[mm]`

MATERIALS

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Air**.

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

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, type `Air - Domains` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Air with PML**.

POROELASTIC WAVES (PELW)

Poroelastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Poroelastic Waves (pelw)** click **Poroelastic Material 1**.

2 In the **Settings** window for **Poroelastic Material**, locate the **Fluid Properties** section.

3 From the **Fluid material** list, choose **Air - Domains (mat1)**.

MATERIALS

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Air**.

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

MATERIALS

Air (mat2)

1 In the **Settings** window for **Material**, type `Air - Boundaries` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

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

Material 3 (mat3)

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

2 In the **Settings** window for **Material**, type `Foam` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Foam**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	0.3	1	Basic
Shear modulus	G	110[kPa]	N/m²	Bulk modulus and shear modulus
Density	rho	41[kg/m^3]	kg/m³	Basic
Porosity	epsilon	0.85	1	Basic
Tortuosity factor	tau	1.18	1	Poroacoustics model
Flow resistivity	Rf	34000[N*s/m^4]	Pa·s/m²	Poroacoustics model
Viscous characteristic length	Lv	60[um]	m	Poroacoustics model
Thermal characteristic length	Lth	87[um]	m	Poroacoustics model
Isotropic structural loss factor	eta_s	0.015	1	Basic

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

Now proceed to generate the mesh. Change the meshing method to manual to make sure that the model is correctly resolved in the frequency range and reduce the running time of the model.

MESH 1

1 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

2 From the **Sequence type** list, choose **User-controlled mesh**.

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 `lambda_air/5`.

5 In the **Minimum element size** text field, type `0.25 [mm]`.

6 In the **Maximum element growth rate** text field, type `1.4`.

7 In the **Curvature factor** text field, type `0.5`.

8 In the **Resolution of narrow regions** text field, type `1`.

9 Click **Build Selected**.

Mapped 1

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

2 Select Boundaries 261, 264, 267, 276, 281, and 288 only.

Size 1

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

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

3 Click the **Custom** button.

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

5 In the associated text field, type `lambda_poro/5`.

Ideally, all the waves propagating in the PELW domain should be considered for the determination of the mesh size. In order to fit the model in a 32 GB RAM computer, the model consider only the fast pressure waves and will be therefore under resolve the slow pressure waves or the shear waves. To get all waves correctly solved, change the parameter `cporo` to the slowest wave speed in the PELW, in this case the shear waves with speed of `96 [m/s]`.

Swept 1

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

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

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

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

Size 1

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

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

3 Click the **Custom** button.

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

5 In the associated text field, type `lambda_poro/5`.

Ideally, all the waves propagating in the PELW domain should be considered for the determination of the mesh size. In order to fit the model in a 32 GB RAM computer, the model consider only the fast pressure waves and will be therefore under resolve the slow pressure waves or the shear waves. To get all waves correctly solved, change the parameter `cporo` to the slowest wave speed in the PELW, in this case the shear waves with speed of `96 [m/s]`.

Free Tetrahedral 1

1 In the **Model Builder** window, click **Free Tetrahedral 1**.

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

3 From the **Selection** list, choose **Meshed domains without PML and foam**.

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 From the **Selection** list, choose **Interior sound hard boundary**.

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 text field, type `2.0[mm]`.

8 Click **Build All**.

Swept 2

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

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

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

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

Distribution 1

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

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

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

4 Click **Build All**.

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.

STUDY 1

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

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

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - Frequency domain** 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 `20`.

6 In the **Stop frequency** text field, type `20000`.

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

8 Click **Replace**.

9 In the **Study** toolbar, click **Get Initial Value**.

STUDY 1 - FREQUENCY DOMAIN

Solver Configurations

In the **Model Builder** window, expand the **Study 1 - Frequency domain>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1 - Frequency domain>Solver Configurations>Solution 1 (sol1)** node.

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

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `2e-7`.

   It is very important to use a tight relative tolerance in the stationary solver when working with iterative solvers

5 Right-click **Stationary Solver 1** and choose **Iterative**.

6 In the **Model Builder** window, expand the **Study 1 - Frequency domain>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** node.

7 Right-click **Iterative 1** and choose **Multigrid**.

8 In the **Settings** window for **Multigrid**, click to expand the **Hybridization** section.

9 From the **Use as** list, choose **Multi preconditioner**.

10 In the **Preconditioner variables** list, choose **Pressure (comp1.p2)**, **Displacement field (comp1.u)**, **comp1.currents**, **comp1.voltages**, and **comp1.current_time**.

11 Under **Preconditioner variables**, click **Delete**.

12 Right-click **Iterative 1** and choose **Direct Preconditioner**.

13 In the **Settings** window for **Direct Preconditioner**, locate the **General** section.

14 From the **Solver** list, choose **PARDISO**.

15 Click to expand the **Hybridization** section. In the **Preconditioner variables** list, select **Pressure (comp1.p)**.

16 Under **Preconditioner variables**, click **Delete**.

17 Click **Compute**.

RESULTS

Acoustic Pressure (acpr)

Click the **Go to Default View** button in the **Graphics** toolbar.

Table 1

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Fine_mesh_solution` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

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

3D Plot Group 5

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

2 In the **Settings** window for **3D Plot Group**, type `SPL on Mannikin Surface` in the **Label** text field.

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

4 In the **Title** text area, type `Sound pressure level (dB)`.

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

6 Select the **Allow evaluation of expressions** check box.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **SPL on Mannikin Surface** and choose **Surface**.

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

3 In the **Expression** text field, type `acpr.Lp`.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Skin without PML**.

Surface 2

1 In the **Model Builder** window, right-click **SPL on Mannikin Surface** and choose **Surface**.

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

3 In the **Expression** text field, type `pelw.Lp`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 From the **Selection** list, choose **Skin without PML**.

4 In the **SPL on Mannikin Surface** toolbar, click **Plot**.

The image should look like Figure 3.

1D Plot Group 6

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

2 In the **Settings** window for **1D Plot Group**, type `Average SPL at the Eardrum` in the **Label** text field.

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

4 In the **Title** text area, type `Octave Band: Average SPL on the eardrum (Pa)`.

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

Octave Band 1

1 In the **Average SPL at the Eardrum** toolbar, click **More Plots** and choose **Octave Band**.

2 In the **Settings** window for **Octave Band**, type `Average SPL at the eardrum` in the **Label** text field.

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

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

5 Locate the **Plot** section. From the **Style** list, choose **Continuous**.

6 In the **Average SPL at the Eardrum** toolbar, click **Plot**.

7 Click to expand 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
`Average SPL at the eardrum`

Table Graph 1

1 In the **Model Builder** window, right-click **Average SPL at the Eardrum** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
`Fine mesh results`

6 In the **Average SPL at the Eardrum** toolbar, click **Plot**.

The image should look like Figure 4.