Model created in COMSOL Multiphysics 6.4

Head and Torso HRTF Computation

Introduction

This tutorial model shows how to import a 3D scanned geometry of a human head and torso and compute the head related transfer function (HRTF). The scan is imported as a stl file and converted into a COMSOL geometry. The HRTF is computed using the reciprocity principle, locating the source at the ear canal entrance; this approach greatly reduces the computational cost to get a full 3D response. The acoustics are modeled using the Pressure Acoustics, Boundary Element interface of the Acoustics Module. The simulated results are compared with measured data from the actual subject and show good agreement.

The HRTF gives a complete description of the way the head and torso of an individual distorts incident sound fields. The HRTF is an important component of spatial hearing. The HRTF includes both so-called monaural and binaural cues. Binaural cues include the interaural time differences (ITD) and interaural level differences (ILD), whereas the monaural cues represent a spectral distortion of the sound that is identical for both ears, see Ref. 1. The HRTF is defined as the sound pressure level (SPL) measured at the eardrum (or the ear canal entrance as in this model) relative to the SPL when no head is present.

When virtual sound is used (or acoustic virtual reality), the HRTF is important in order to make the test subject experience a virtual sound scene. The HRTF can be measured, which can be a tedious task, or it can be simulated based on a scan of the individual. This model presents the latter approach on a scanned head geometry provided by the, Institute for Hearing Technology and Acoustics (IHTA), RWTH Aachen University, Germany, Ref. 2. The scan is of an actual individual where the facial features have been removed, while all the details of the ear geometry have been retained.

Note: The scanned geometry (stl mesh) and measured data is with courtesy of the Teaching and Research Area of Medical Acoustics, Institute of Technical Acoustics, RWTH Aachen University, Germany. The stl mesh is licensed under the Creative Commons Attribution 4.0 International License and is provided "as is" with all warranties disclaimed, as stated in that license. See Ref. 7 and Ref. 8 for details.

Model Definition

A common approach when simulating the HRTF is to use the reciprocity principle; the source and receiver locations are reverted, Ref. 3. This means that, in the model, the source is located at the entrance of the ear canal and the evaluation is performed along a circle (or on a sphere for the full bubble) with its center in the middle of the head, between the ears. In this way, the HRTF can be deduced for all spatial directions for each frequency with just one simulation. Not using reciprocity requires solving one problem per incidence direction per frequency, which is not practical. Reciprocity is used for similar applications in Ref. 4 and Ref. 5.

The acoustic problem is modeled using the boundary element method (BEM) with the Pressure Acoustics, Boundary Element interface. This is especially efficient since the present model represents a pure radiation problem.

The imported stl mesh is depicted in Figure 1 and the COMSOL geometry generated from the stl mesh is depicted in Figure 2. Notice that the geometry has been moved and rotated (in Figure 2) to align the coordinate axis with the commonly used directions for directivity assessment. The evaluation circle for the HRTF that is used in the model, is represented in Figure 2. The evaluation is performed using a Radiation Pattern plot.

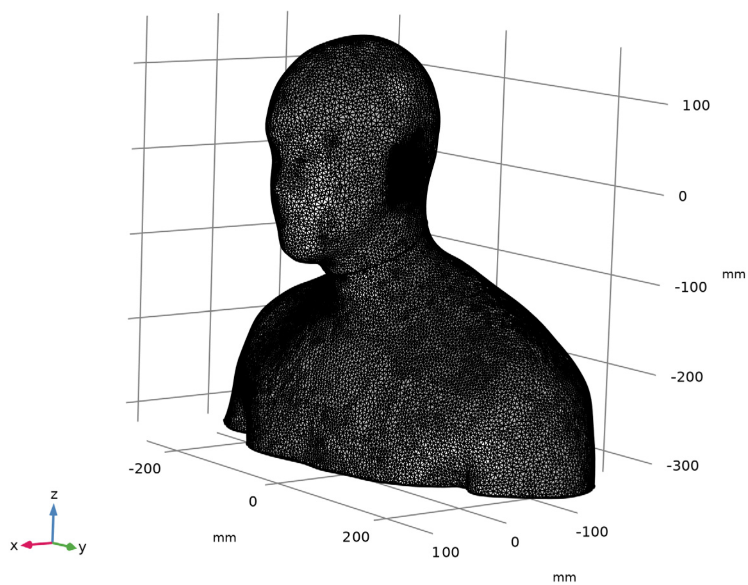


Figure 1: The imported stl mesh. The imported stl mesh is courtesy of the Teaching and Research Area of Medical Acoustics, Institute of Technical Acoustics, RWTH Aachen University, Germany. The stl mesh is licensed under the Creative Commons Attribution 4.0 International License *and is provided "as is" with all warranties disclaimed, as stated in that license. See* Ref. 7 *and* Ref. 8.

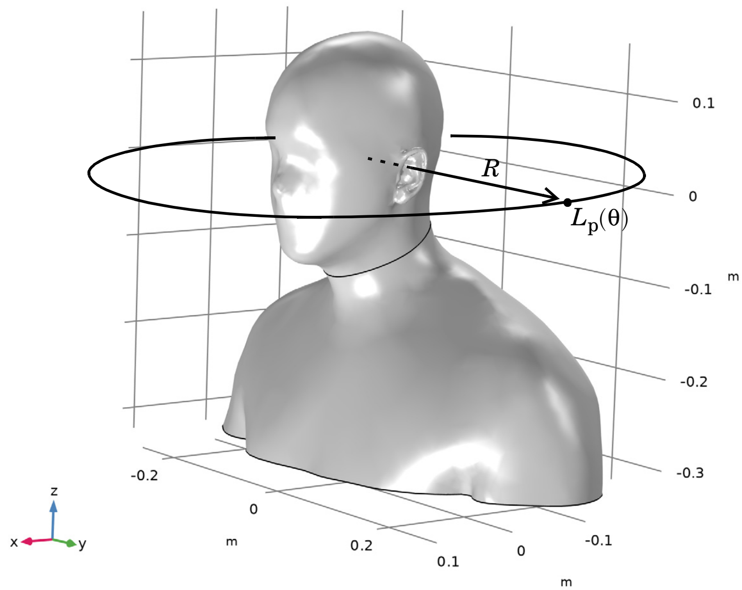


Figure 2: The generated COMSOL geometry from the stl mesh. Representation of the location for the HRTF evaluation on a circle in the horizontal plane.

Results and Discussion

The model is solved for the three frequencies where measurement data is provided (f = 1033.6 Hz, 2067.5 Hz, and 3962.1 Hz) as well as two additional higher frequencies (f = 6000 Hz and 8000 Hz).

The pressure field generated from the excitation at the ear canal entrance is depicted in Figure 3 for the frequencies f = 1033.6 Hz, 2067.5 Hz, and 3962.1 Hz. These have been selected as they coincide with the measurement data (octave band center frequencies). A unit normal velocity is assigned to the ear canal entrance.

The acoustic pressure is also depicted in a cross-section plane in Figure 4 and the corresponding sound pressure level (SPL) is depicted in Figure 5. Both are evaluated at the 4 kHz octave band center frequency. The SPL plot clearly shows the presence of notches (cancellations) for certain directions. These are more evident at the higher frequencies.

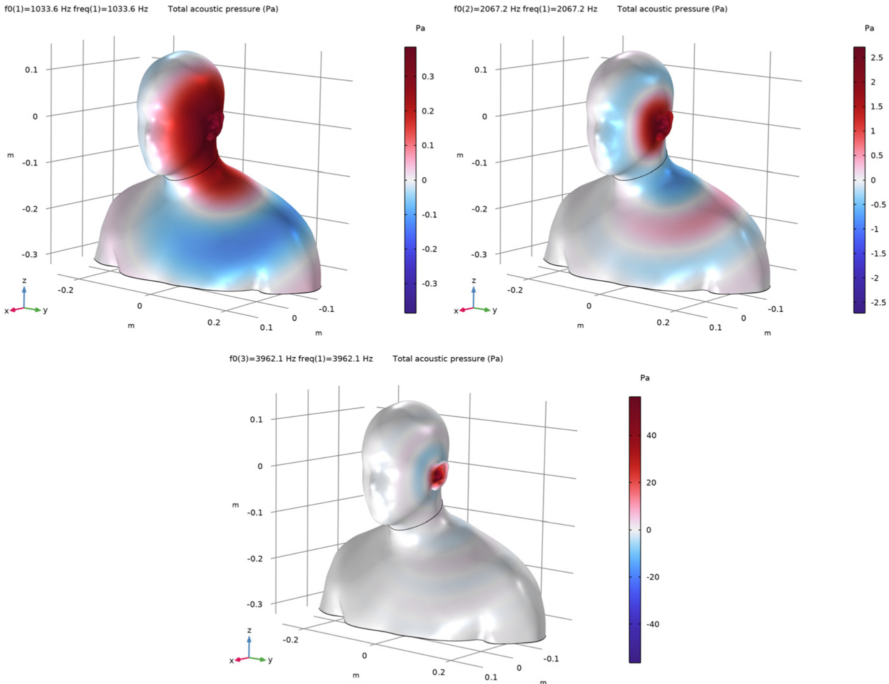


Figure 3: Acoustic pressure at the surface of the head and torso evaluated at three frequencies.

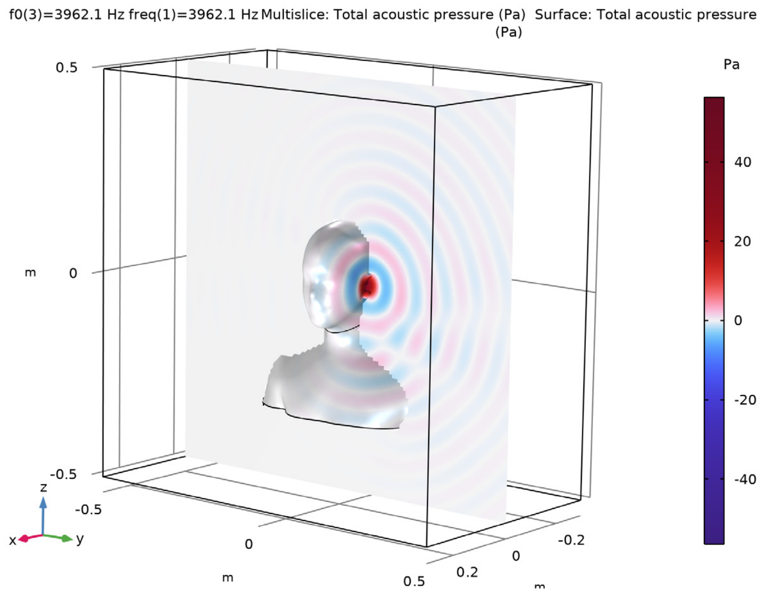


Figure 4: Acoustic pressure on the head and torso and in a cut plane.

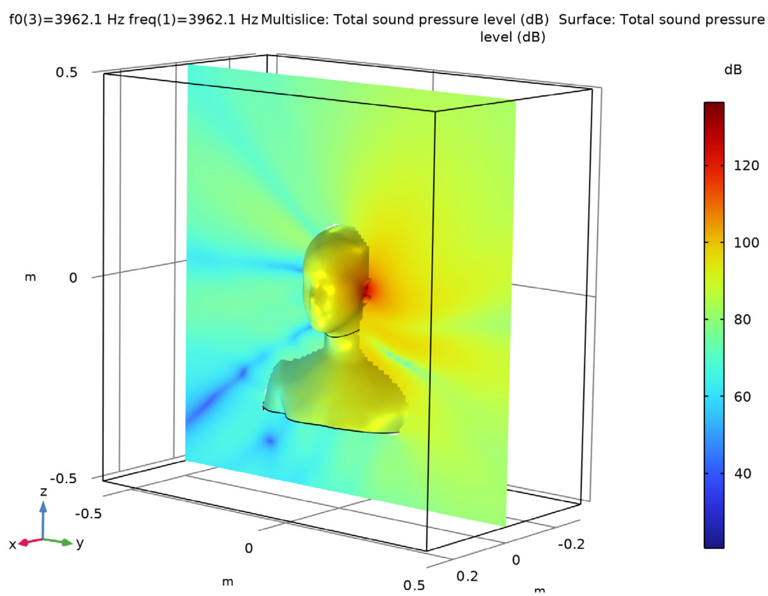


Figure 5: Sound pressure level on the surface of the head and torso and in a cut plane.

The HRTF in the horizontal plane (the *xy*-plane) is depicted for the five evaluation frequencies in Figure 6. In the plot, the HRTF is normalized to 0 dB toward the front (polar angle $\theta = 0°$). In the following three plots — Figure 7, Figure 8, and Figure 9 — the computed HRTFs are compared to measurement data; see Ref. 6 and Ref. 8. In the plots, the HRTF data has been rotated by $\theta_0 = 4.5°$ to make the location of the notches match (defined by the parameter `theta0`).

Notice that the model assumes that the head and torso is located in free space. This is consistent with the measured data where the floor reflections have been removed. A time-windowed approach is used, so that the reflections will not affect the data. In the measurements, there is a maximum length of the impulse response of about 330 samples at a 44,100 Hz sampling rate.

The COMSOL models results agree well with the measurement data. There are some general small discrepancies; these can be due to head movement during the measurements. In the current measurement setup at RWTH, head movement is tracked and compensated for. A larger discrepancy can be seen in the 1 kHz plot toward 30° (in Figure 7). Shoulder reflections are typically seen at around 1.5 kHz so a slight under- or overestimate of the shoulder size, when generating the head scan, could introduce this error.

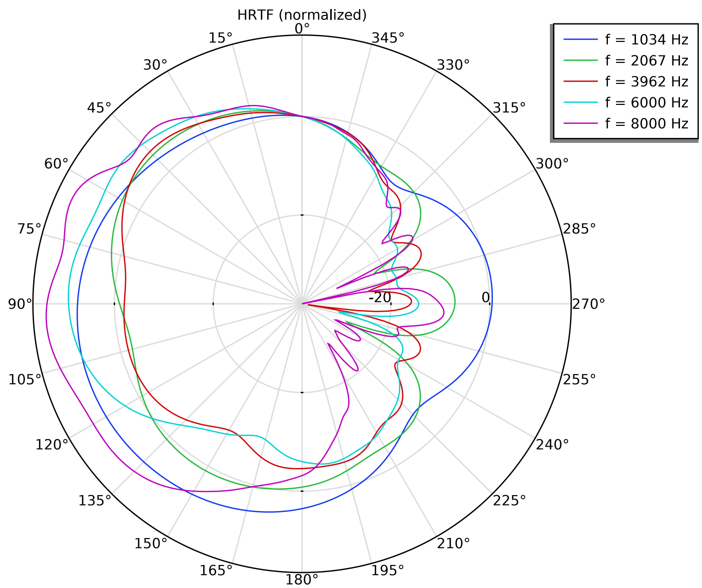


Figure 6: Comparison of the normalized HRTF evaluated at the three frequencies.

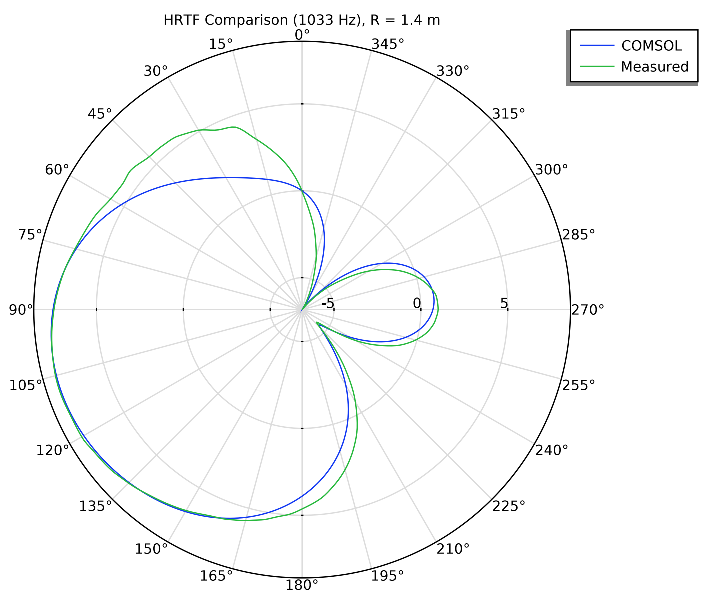


Figure 7: Comparison of the simulated HRTF with measured data at 1033 Hz.

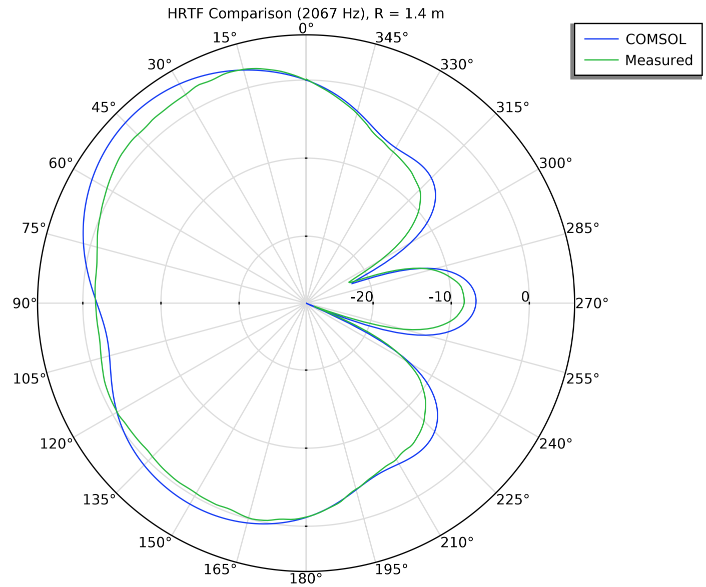


Figure 8: Comparison of the simulated HRTF with measured data at 2066 Hz.

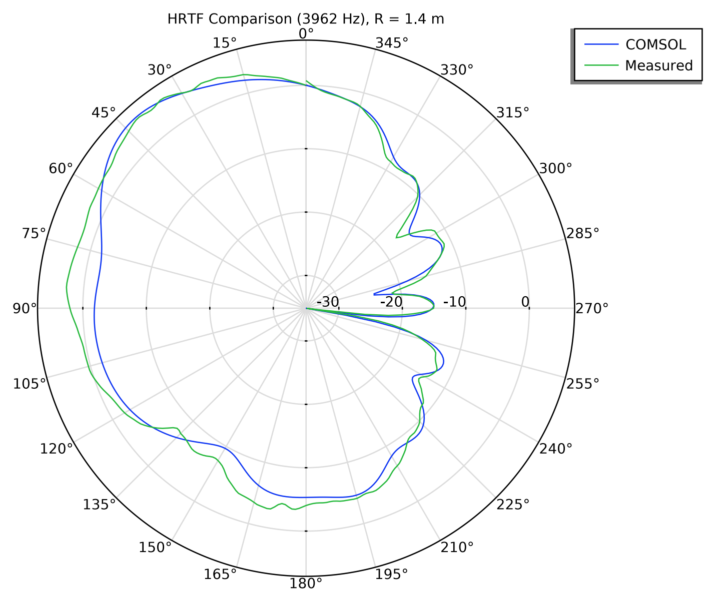


Figure 9: Comparison of the simulated HRTF with measured data at 3962 Hz.

References

1. M. Vorländer, *Auralization*, Springer, 2008.

2. Web link: www.akustik.rwth-aachen.de/

3. A.D. Pierce, *Acoustics: An Introduction to Its Physical Principles and Applications*, Acoustical Society of America, 1991.

4. Z. Conrad, “Hats Off to the Boundary Element Method,” *IEEE Spectrum Multiphysics Simulation*, October 2018, p. 30, web link: www.comsol.com/offers/multiphysics-simulation-2018.

5. M.H. Jensen, “Improving the Performance of Hearing Aids Using Acoustic Simulations,” COMSOL Conference 2009, web link: www.comsol.com/paper/improving-the-performance-of-hearing-aids-using-acoustic-simulations-7227.

6. Web link: www.akustik.rwth-aachen.de/cms/Technische-Akustik/Forschung/~lxfd/Downloads/lidx/1/.

7. H.S. Braren and J. Fels, “A High-Resolution Individual 3D Adult Head and Torso Model for HRTF Simulation and Validation: 3D Data,” web link (DOI): doi.org/10.18154/RWTH-2020-06760

8. H.S. Braren and J. Fels, “A High-Resolution Individual 3D Adult Head and Torso Model for HRTF Simulation and Validation: HRTF Measurement,” web link (DOI): doi.org/10.18154/RWTH-2020-06761

Application Library path: `Acoustics_Module/Tutorials,_Pressure_Acoustics/head_torso_hrtf`

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, Boundary Elements (pabe)**.

3 Click **Add**.

4 Click **Study**.

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

6 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**, locate the **Parameters** section.

3 Click **Load from File**.

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

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 `head_torso_hrtf_measured.txt`.

6 Locate the **Data Column Settings** section. In the table, enter the following settings:

Columns	Type	Settings
Column 2	Function values	Function name=int1a
Column 3	Function values	Function name=int1b
Column 4	Function values	Function name=int1
Column 5	Function values	Function name=int1c
Column 6	Function values	Function name=int1d
Column 7	Function values	Function name=int1e

7 In the table, click to select the cell at row number 1 and column number 1.

8 In the **Unit** text field, type `rad`.

9 In the table, click to select the cell at row number 2 and column number 1.

10 In the **Name** text field, type `HRTF_1033_real`.

11 In the **Unit** text field, type `Pa`.

12 In the table, click to select the cell at row number 3 and column number 1.

13 In the **Name** text field, type `HRTF_1033_imag`.

14 In the **Unit** text field, type `Pa`.

15 In the table, click to select the cell at row number 4 and column number 1.

16 In the **Name** text field, type `HRTF_2067_real`.

17 In the **Unit** text field, type `Pa`.

18 In the table, click to select the cell at row number 5 and column number 1.

19 In the **Name** text field, type `HRTF_2067_imag`.

20 In the **Unit** text field, type `Pa`.

21 In the table, click to select the cell at row number 6 and column number 1.

22 In the **Name** text field, type `HRTF_3962_real`.

23 In the **Unit** text field, type `Pa`.

24 In the table, click to select the cell at row number 7 and column number 1.

25 In the **Name** text field, type `HRTF_3962_imag`.

26 In the **Unit** text field, type `Pa`.

27 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

28 From the **Extrapolation** list, choose **Linear**.

29 Locate the **Definition** section. Click **Import**.

Analytic 1 (an1)

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

2 In the **Settings** window for **Analytic**, type `p1033` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `HRTF_1033_real(theta+theta0)+i*HRTF_1033_imag(theta+theta0)`.

4 In the **Arguments** text field, type `theta`.

5 Click to expand the **Periodic Extension** section. Select the **Make periodic** checkbox.

6 In the **Upper limit** text field, type `2*pi`.

7 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
theta	`rad`

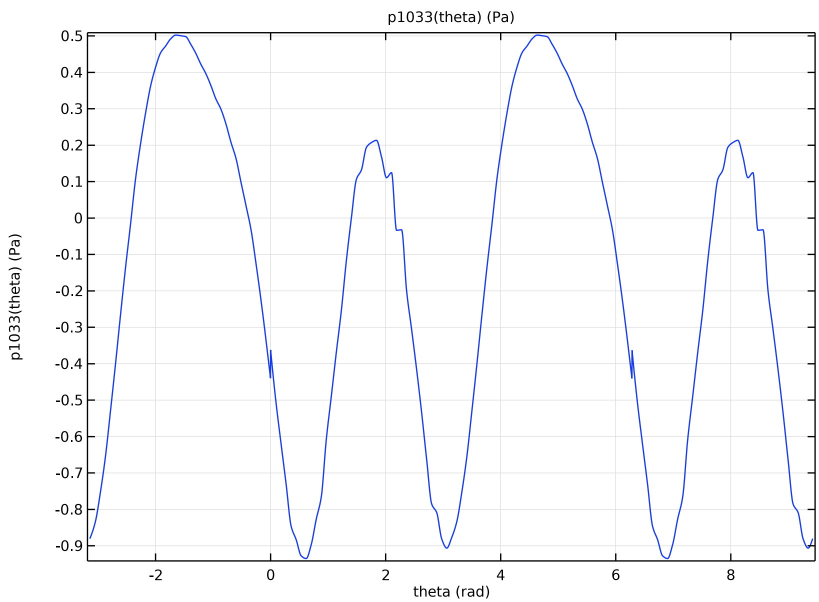
8 In the **Function** text field, type `Pa`.

9 Click to expand the **Advanced** section. Select the **May produce complex output for real arguments** checkbox.

10 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	theta	`-pi`	`3*pi`	0	rad

11 Click **Plot**.



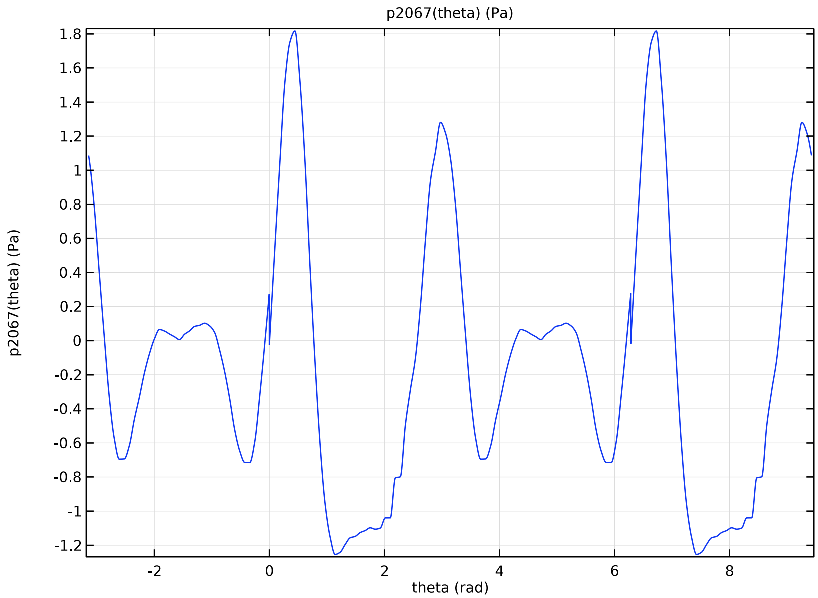
Analytic 2 (p2)

1 Right-click **Analytic 1 (p1033)** and choose **Duplicate**.

2 In the **Settings** window for **Analytic**, type `p2067` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `HRTF_2067_real(theta+theta0)+i*HRTF_2067_imag(theta+theta0)`.

4 Click **Plot**.



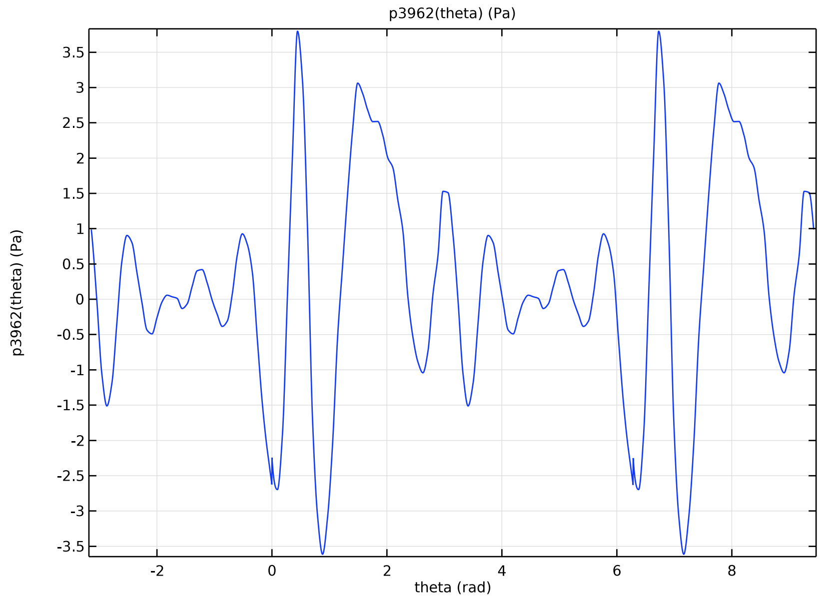
Analytic 3 (p3)

1 Right-click **Analytic 2 (p2067)** and choose **Duplicate**.

2 In the **Settings** window for **Analytic**, type `p3962` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `HRTF_3962_real(theta+theta0)+i*HRTF_3962_imag(theta+theta0)`.

4 Click **Plot**.



5 In the **Model Builder** window, right-click **Global Definitions** and choose **Mesh Parts** > **3D Part**.

MESH PART 1

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

2 Select the **Use units** checkbox.

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

Import 1

1 In the **Model Builder** window, under **Global Definitions** > **Mesh Parts** > **Mesh Part 1** click **Import 1**.

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 `head_torso_hrtf_scan.stl`.

5 From the **Boundary partitioning** list, choose **Detect boundaries**.

6 Locate the **Detect Faces** section. In the **Maximum neighbor angle** text field, type 180.

Add a **Transform** attribute to mirror the STL mesh in the xy-plane.

Transform 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Transform**.

2 In the **Settings** window for **Transform**, locate the **Scale** section.

3 From the **Scaling** list, choose **Anisotropic**.

4 In the **z scale** text field, type `-1`.

5 Click **Build Selected**.

Now, cut the surface mesh (using a cylinder) in order to create selections for the entrance of the ear canal.

Partition with Cylinder 1

1 In the **Mesh** toolbar, click **Booleans and Partitions** and choose **Partition with Cylinder**.

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

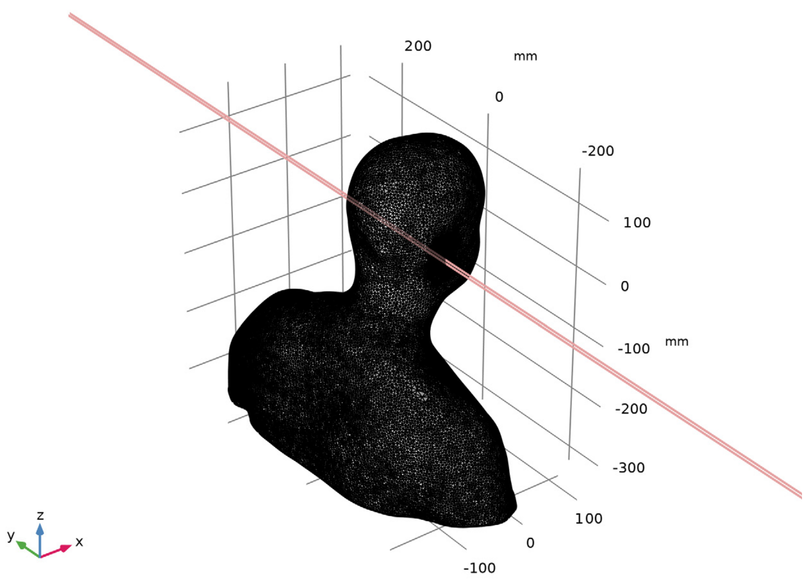
3 In the **Radius** text field, type `2.8`.

4 Locate the **Position** section. In the **x** text field, type `-1.3`.

5 In the **z** text field, type `0.6`.

6 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

7 Click **Build Selected**.



Intersect with Plane 1

1 In the **Mesh** toolbar, click **Booleans and Partitions** and choose **Intersect with Plane**.

2 In the **Settings** window for **Intersect with Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Coordinates**.

4 In row **Point 1**, set **z** to `-95`.

5 In row **Point 2**, set **x** to `10`.

6 In row **Point 2**, set **z** to `-100`.

7 In row **Point 3**, set **z** to `-95`.

Leave all other components at their default zero value.

8 Click **Build Selected**.

Finalize

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

The final mesh part created from the imported `.stl` file, of the scanned head and torso, should look like the image in Figure 1. Use the mouse to rotate, zoom, and move the geometry in the graphics window.

GEOMETRY 1

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

Import 1 (imp1)

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

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

3 From the **Source** list, choose **Mesh or 3D printing file (STL, 3MF, PLY)**.

4 From the **Mesh** list, choose **Mesh Part 1**.

5 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Remove Details 1 (rmd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Remove Details**.

2 In the **Settings** window for **Remove Details**, click **Build Selected**.

In the **Information** section the number of details removed can be seen.

Use the mouse to rotate, zoom, and move the geometry to see it from the front. The geometry should look like the image in Figure 2.

ADD MATERIAL

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

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

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

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

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

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **All voids**.

PRESSURE ACOUSTICS, BOUNDARY ELEMENTS (PABE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Boundary Elements (pabe)**.

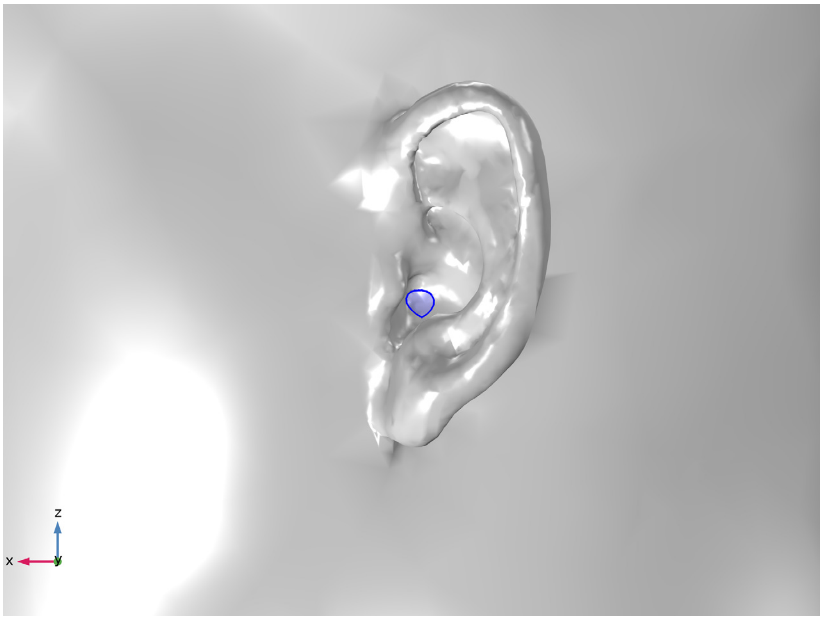
2 In the **Settings** window for **Pressure Acoustics, Boundary Elements**, locate the **Domain Selection** section.

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

Normal Velocity 1

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

2 Select Boundary 6 only.



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

4 In the v_n text field, type `1`.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

Size

1 In the **Model Builder** window, 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 `min(20[mm],lam0/4)`.

5 In the **Minimum element size** text field, type `3[mm]`.

6 In the **Resolution of narrow regions** text field, type `2`.

Free Triangular 1

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

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

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

Size 1

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

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** checkbox. In the associated text field, type `lam0/4`.

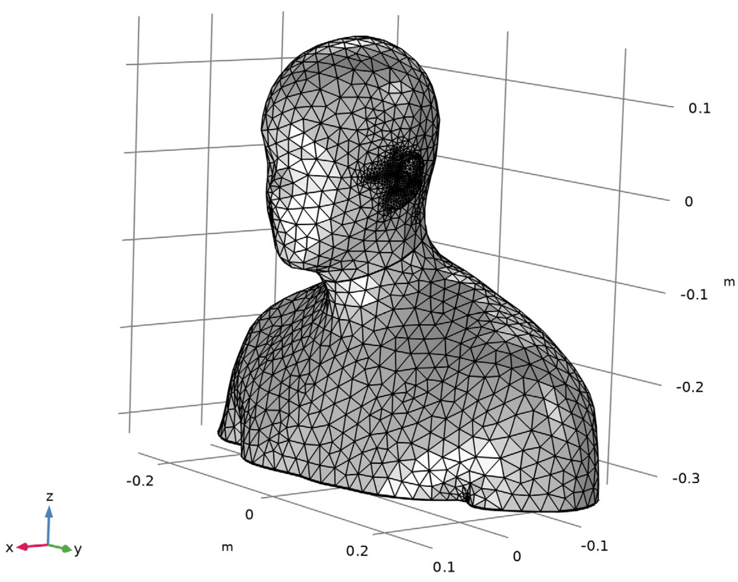
6 Locate the **Geometric Entity Selection** section. In the list box, select **1**.

7 Click **Clear Selection**.

8 Select Boundaries 1 and 2 only.

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

The mesh should look like the image below, here meshed to resolve a frequency of 4000 Hz. You can change the parameter `f0` and build the mesh again, to see how it looks at different frequencies.



DEFINITIONS (COMP1)

Before solving the model, add a variable `theta` that defines the horizontal polar angle. The variable is used when postprocessing the measured HRTF data.

Variables 1

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

2 Right-click **Component 1 (comp1)** > **Definitions** and choose **Variables**.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
theta	atan2(y,x)	rad	Horizontal polar angle

STUDY 1

Step 1: Frequency Domain

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

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `f0`.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

Solve the model for the three frequencies used for comparison to the measurements and two additional higher frequencies.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Frequency)	{1033.6, 2067.2, 3962.1, 6000, 8000}	Hz

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

RESULTS

Grid 3D 1

1 In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Grid 3D 1**.

2 In the **Settings** window for **Grid 3D**, locate the **Parameter Bounds** section.

3 Find the **First parameter** subsection. In the **Minimum** text field, type `-0.3`.

4 In the **Maximum** text field, type `0.3`.

5 Find the **Second parameter** subsection. In the **Minimum** text field, type `-0.5`.

6 In the **Maximum** text field, type `0.5`.

7 Find the **Third parameter** subsection. In the **Minimum** text field, type `-0.5`.

8 In the **Maximum** text field, type `0.5`.

9 Click to expand the **Grid** section. Now, set the resolution of the evaluation grid. This resolution should be adequate to resolve the wave pattern being visualized (the wavelength). If the resolution is too poor the results will appear wrong.

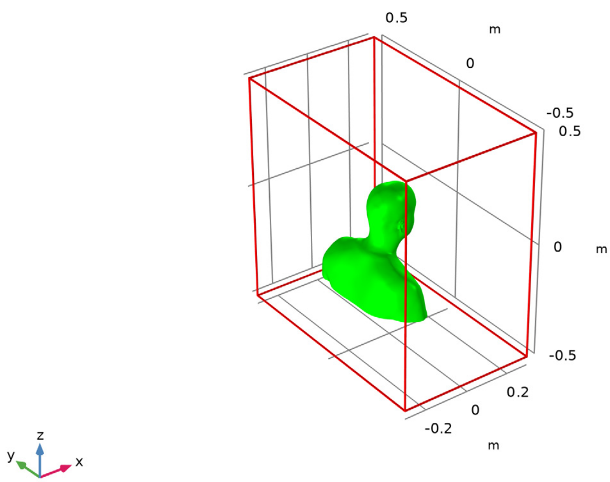
10 In the **x resolution** text field, type `40`.

11 In the **y resolution** text field, type `120`.

12 In the **z resolution** text field, type `120`.

To visualize the extent of the grid dataset, where the BEM solution is visualized, plot the dataset.

13 Click **Plot**.



SHOW MORE OPTIONS

In preparation for setting up the plots, enable custom result views.

1 Click the **Show More Options** button in the **Model Builder** toolbar.
2 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Results** > **Views**.
3 Click **OK**.

RESULTS

Acoustic Pressure, Boundaries (pabe)

The first default plot shows the pressure on the surface of the head and torso.

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Boundaries (pabe)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
3 From the **View** list, choose **New view**. This allows you to set up and use a dedicated view for this plot group.
4 Locate the **Data** section. From the **Parameter value (f0 (Hz))** list, choose **1033.6**.
5 In the **Acoustic Pressure, Boundaries (pabe)** toolbar, click **Plot**.

   Use the mouse and the **Graphics** window toolbar buttons to rotate and zoom the geometry so that the left side of the head and torso is clearly visible.

   Change the frequency parameter `f0` as needed. The three solved frequencies are shown in Figure 3.

Before turning the attention to the second default plot, lock the view for this one.

Acoustic Pressure (pabe)

1 In the **Model Builder** window, expand the **Results** > **Views** node, then click **Results** > **Acoustic Pressure (pabe)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Parameter value (f0 (Hz))** list, choose **3962.1**.

   This plot shows the pressure on the surface of the head and torso and in slices through the grid dataset. Add a color table transformation to enhance visualization of the pressure field.

Multislice 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (pabe)** node, then click **Multislice 1**.
2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
3 Find the **y-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **z-planes** subsection. In the **Planes** text field, type 0.

Turn on the nonlinear symmetric color transformation and set the color calibration parameter to, for example, a value of -1.

5 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Nonlinear symmetric**.

6 Set the **Color calibration parameter** value to **-1**.

7 In the **Acoustic Pressure (pabe)** toolbar, click **Plot**.

Zoom out to get a better view of the space around the head and torso. The image should look like that in Figure 4 at 3962 Hz.

Sound Pressure Level (pabe)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (pabe)**.

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

3 From the **Parameter value (f0 (Hz))** list, choose **3962.1**.

Multislice 1

1 In the **Model Builder** window, expand the **Sound Pressure Level (pabe)** node, then click **Multislice 1**.

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

3 Find the **y-planes** subsection. In the **Planes** text field, type 0.

4 Find the **z-planes** subsection. In the **Planes** text field, type 0.

5 In the **Sound Pressure Level (pabe)** toolbar, click **Plot**.

The image should look like that in Figure 5.

HRTF

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

2 In the **Settings** window for **Polar Plot Group**, type HRTF in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Radiation Pattern 1

1 In the **HRTF** toolbar, click **More Plots** and choose **Radiation Pattern**.

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

3 In the **Expression** text field, type pabe.Lp_t.

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of angles** text field, type `360`.

Next, inspect the location of the evaluation plane/circle.

5 Click **Preview Evaluation Plane**.

Then add a suitable label.

6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

7 From the **Legends** list, choose **Evaluated**.

8 In the **Legend** text field, type `f = eval(f0,Hz,4) Hz`.

HRTF

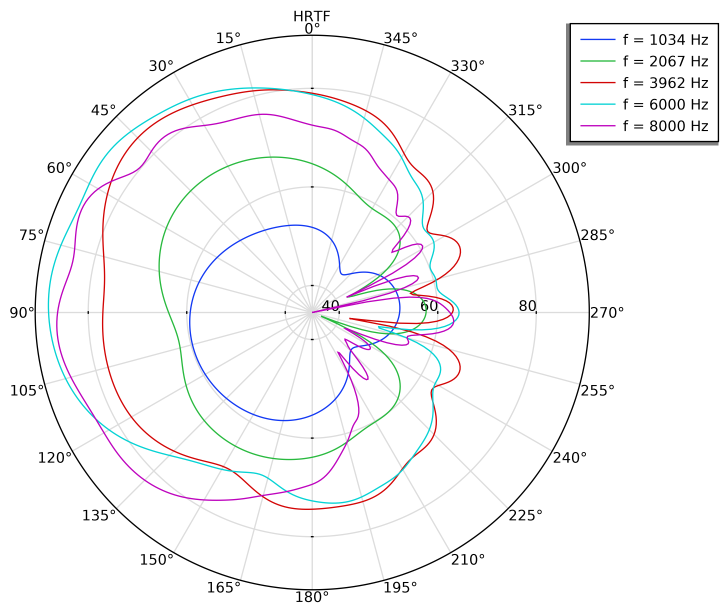
1 In the **Model Builder** window, click **HRTF**.

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

3 From the **Zero angle** list, choose **Up**.

4 In the **HRTF** toolbar, click **Plot**.

First, plot the HRTF (without normalization) using the Radiation Pattern plot and then add a second plot where it is normalized with reference to the front.



HRTF (normalized)

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

2 In the **Settings** window for **Polar Plot Group**, type `HRTF (normalized)` in the **Label** text field.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **HRTF (normalized)** node, then click **Radiation Pattern 1**.

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

3 In the **Expression** text field, type `pabe.Lp_t-at3_spatial(1[m],0,0,pabe.Lp_t,'minc')`.

4 In the **HRTF (normalized)** toolbar, click **Plot**.

The plot should look like that in Figure 6.

HRTF Comparison (1033 Hz), R = 1.4 m

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

2 In the **Settings** window for **Polar Plot Group**, type `HRTF Comparison (1033 Hz), R = 1.4 m` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (f0)** list, choose **From list**.

5 In the **Parameter values (f0 (Hz))** list box, select **1033.6**.

6 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

7 Locate the **Axis** section. From the **Zero angle** list, choose **Up**.

Radiation Pattern 1

1 In the **HRTF Comparison (1033 Hz), R = 1.4 m** toolbar, click **More Plots** and choose **Radiation Pattern**.

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

3 In the **Expression** text field, type `pabe.Lp_t-at3_spatial(1.4[m],0,0,pabe.Lp_t,'minc')`.

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of angles** text field, type `360`.

5 Find the **Evaluation distance** subsection. In the **Radius** text field, type `1.4`.

6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

8 In the table, enter the following settings:

Legends
COMSOL

HRTF Comparison (1033 Hz), R = 1.4 m

In the **Model Builder** window, click **HRTF Comparison (1033 Hz), R = 1.4 m**.

Radiation Pattern 2

1 In the **HRTF Comparison (1033 Hz), R = 1.4 m** toolbar, click **More Plots** and choose **Radiation Pattern**.

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

3 In the **Expression** text field, type `20*log10(abs(p1033(theta)/p1033(0)))`.

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of angles** text field, type `360`.

5 Find the **Evaluation distance** subsection. In the **Radius** text field, type `1.4`.

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

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

8 In the table, enter the following settings:

Legends
Measured

9 In the **HRTF Comparison (1033 Hz), R = 1.4 m** toolbar, click **Plot**.

The plot should look like that in Figure 7.

HRTF Comparison (2067 Hz), R = 1.4 m

1 Right-click **HRTF Comparison (1033 Hz), R = 1.4 m** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `HRTF Comparison (2067 Hz), R = 1.4 m` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (f0 (Hz))** list box, select **2067.2**.

Radiation Pattern 2

1 In the **Model Builder** window, expand the **HRTF Comparison (2067 Hz), R = 1.4 m** node, then click **Radiation Pattern 2**.

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

3 In the **Expression** text field, type `20*log10(abs(p2067(theta)/p2067(0)))`.

4 In the **HRTF Comparison (2067 Hz), R = 1.4 m** toolbar, click **Plot**.

The plot should look like that in Figure 8.

HRTF Comparison (3962 Hz), R = 1.4 m

1 In the **Model Builder** window, right-click **HRTF Comparison (2067 Hz), R = 1.4 m** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `HRTF Comparison (3962 Hz), R = 1.4 m` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (f0 (Hz))** list box, select **3962.1**.

Radiation Pattern 2

1 In the **Model Builder** window, expand the **HRTF Comparison (3962 Hz), R = 1.4 m** node, then click **Radiation Pattern 2**.

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

3 In the **Expression** text field, type `20*log10(abs(p3962(theta)/p3962(0)))`.

4 In the **HRTF Comparison (3962 Hz), R = 1.4 m** toolbar, click **Plot**.

The plot should look like that in Figure 9.