Created in COMSOL Multiphysics 6.1

Acoustic Scattering off an Ellipsoid

Introduction

This example studies the scattering of an incident plane acoustic wave off a rigid ellipsoid geometry. The model utilizes the scattered field formulation, which enables the separation of the incident (background) pressure field and the scattered field. Using the **Exterior Field Calculation** feature, the scattered field is determined at a given distance outside the computational domain. The results are presented as 3D cross-section plots and as polar plots of the scattered exterior pressure and sound pressure level.

Model Definition

Figure 1 shows a sketch of the modeled system. A rigid ellipsoid is hit by an incident plane wave (here named the background pressure field p_b). The scattered field off the ellipsoid is denoted p.

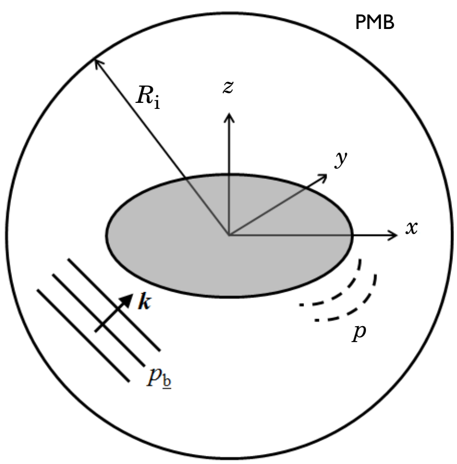


Figure 1: Sketch of the modeled system showing geometric scales, the computation domain bounded by the PMB, the incident background pressure field p_b, and the scattered field p.

The total acoustic field p_t is given by the sum of the scattered and the background pressure fields such that

$$p_t = p_b + p$$
$$p_b = p_0 e^{-i(\mathbf{k} \cdot \mathbf{x})} \quad (1)$$

The background pressure field is a plane wave of amplitude p_0 moving in the direction $\mathbf{k}$ with wave number $|\mathbf{k}| = 2\pi f_0/c_0$, where f_0 is the frequency and c_0 is the speed of sound. The governing equations are implemented as a scattered field formulation such that only the scattered field p is solved for. See the *Acoustics Module User's Guide* for information about the background pressure field feature.

The ellipsoid is located inside a computational domain of radius R_i, terminated by a *perfectly matched boundary* (PMB). The PMB sets up a perfectly matched layer without adding a domain to the geometry. It is used as a non-reflecting and absorbing boundary that mimics a domain stretching to infinity. For more information about PMBs in acoustics, see *The Pressure Acoustics, Frequency Domain Interface* section in the *Pressure Acoustics Module User's Guide*.

The surrounding fluid in this model is water. Approximate physical quantities (for water at 20°C) and dimensions, used in the model, are given in the table below.

TABLE 1: PHYSICAL QUANTITIES AND DIMENSIONS.

SYMBOL	VALUE	DESCRIPTION
R_i	1 m	Radius of inner modeled water region
R_{ext}	10 m	Distance at which the exterior field is evaluated
A	0.5 m	x-semiaxis of ellipsoid
B	0.25 m	y-semiaxis of ellipsoid
C	0.25 m	z-semiaxis of ellipsoid
f_0	1000 Hz	Driving frequency
c_0	1500 m/s	Speed of sound in water
λ_{min}	1.5 m	Wavelength in water at f0

MESH

When modeling a wave problem, the computational mesh has to provide sufficient resolution of the waves. In 3D acoustic models, it is necessary to have a minimum of 5 elements per wavelength when using second-order elements (this is the default element for pressure acoustics). In this model, 6 elements per wavelength are used. The mesh size as well as proper meshing of the boundary used for the exterior field calculation feature is automatically set up when using the *Physics-controlled mesh* functionality for pressure acoustics.

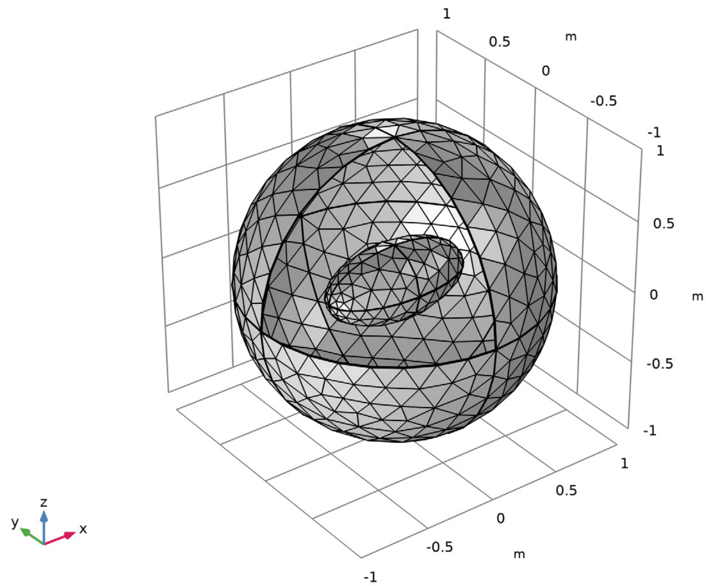


Figure 2: Illustration of the mesh on the boundary of the computational domain and the mesh on the ellipsoid surface.

EXTERIOR FIELD

After solving a pressure acoustics model, it is possible to determine the pressure outside the computational domain using the exterior field calculation feature. The exterior field calculation feature solves the Helmholtz-Kirchhoff (H-K) integral on the selected boundaries. The selected boundaries need to form a closed surface around all sources and scatterers. If the model has symmetries, these can be included. Note that two versions of the H-K integral exist, one that only determines the pressure at the infinity limit (an approximation to the H-K integral is then used) and one version that solves the full H-K integral. In this model, we use the full integral and can thus determine the exact exterior-field pressure (including phase) at any point and distance outside the computational domain.

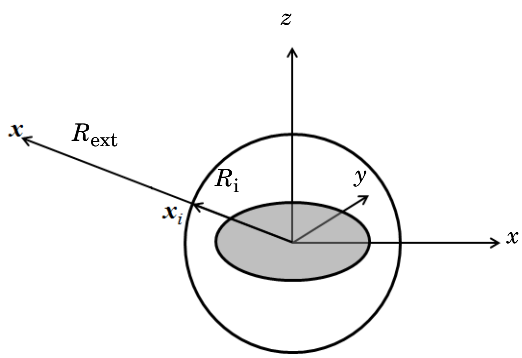


Figure 3: Relation between a coordinate defined in the exterior field boundary $\mathbf{x}_i$ *and the coordinate* $\mathbf{x}$ *at a distance* R_{ext}.

For plotting purposes, the exterior-field pressure variable `pext` is defined by COMSOL. This variable defines the pressure at any coordinates x, y, and z that are outside the boundary on which the exterior-field calculation is defined (for $|x| > R_i$ in Figure 3). The exterior-field pressure (variable name `acpr.efc1.pext`) and exterior-field sound pressure level (variable name `acpr.ffc1.Lp_pext`) are easily plotted and visualized using the radiation pattern plot types. They exist in 1D plot groups for plotting on, for example, a polar plot group, in 2D plot groups, and in 3D plot groups for creating 3D polar plots.

Finally, note that in order to get a precise evaluation of the exterior-field variable, the evaluation of the H-K integral must be accurate. This requires having a good numerical estimate of the normal derivative of the pressure on the exterior-field calculation surface. This is achieved using a single boundary-layer mesh. This is automatically handled by the physics-controlled mesh.

Results and Discussion

Figure 4 shows the total acoustic field p_t. It is the sum of the scattered field p and the incident background pressure field p_b, shown in Figure 5.

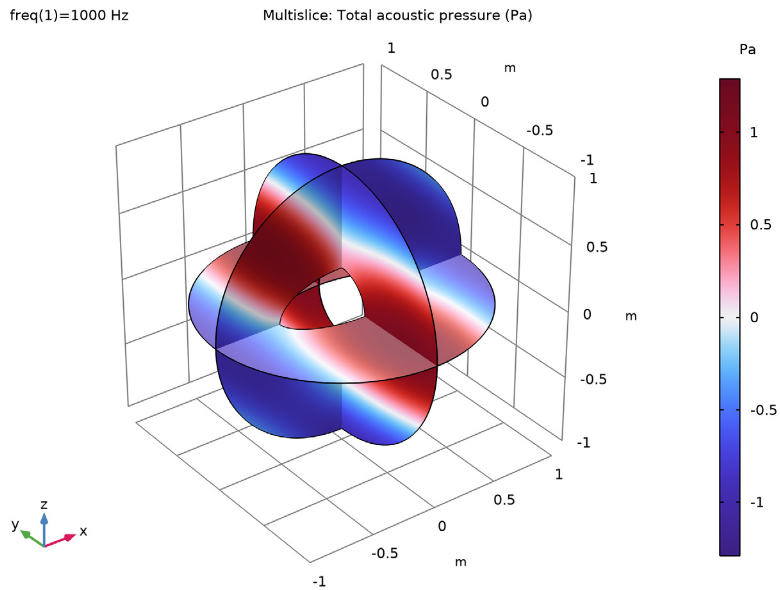


Figure 4: Total acoustic field at f = 1000 Hz.

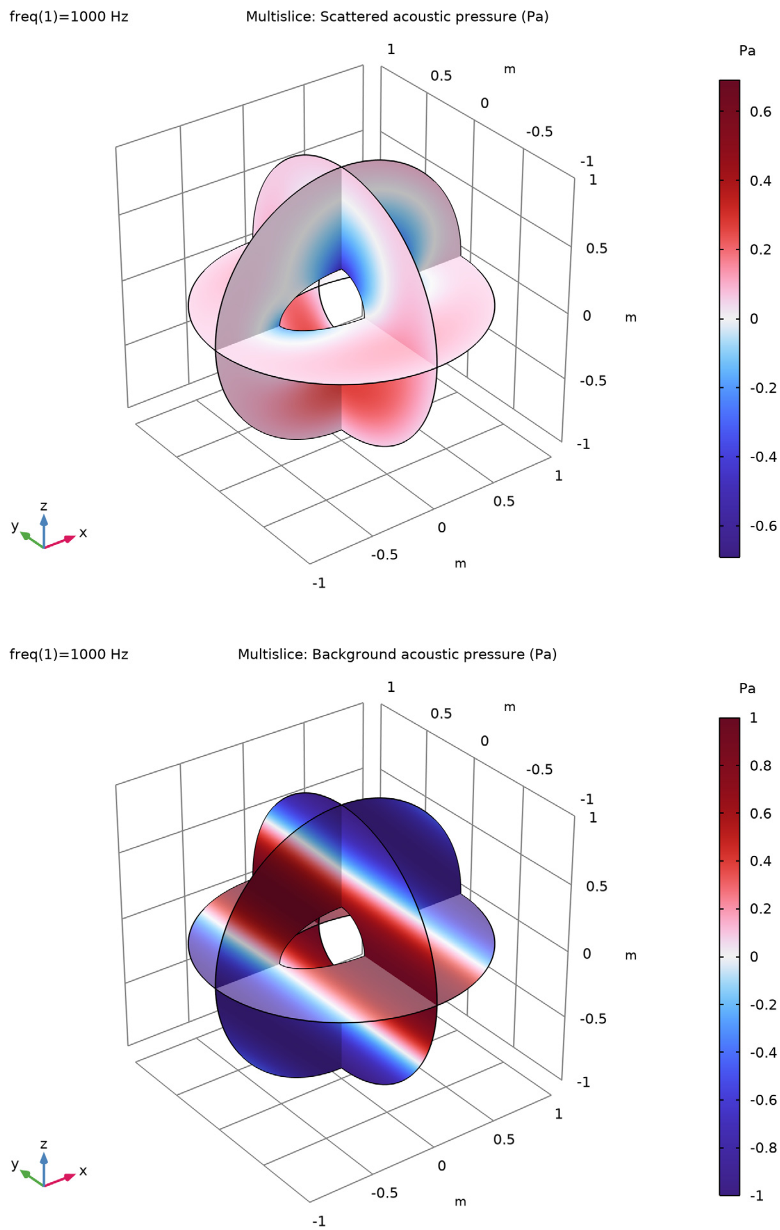


Figure 5: Scattered acoustic field (top) and incident plane-wave acoustic field (bottom).

Figure 6 plots the pressure in the exterior field at the distance R_{ext} = 10 m. The data is retrieved in the *xy*-plane and presented as a polar plot, with 0° corresponding to the positive *x* direction. The sound pressure level in the exterior field is likewise represented in a polar plot in Figure 7. It is easy to determine the pressure and sound pressure level at another distance; simply change the parameter value for `Rext` under the parameters and update the solution (press F5). The plots are then updated accordingly.

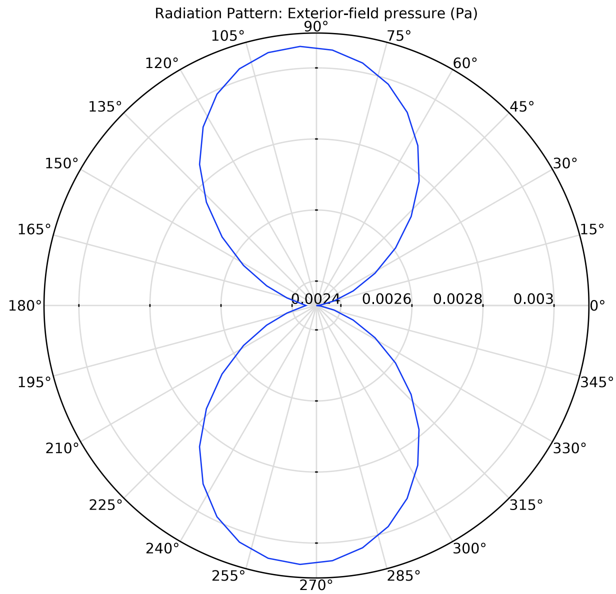


Figure 6: Polar plot of the pressure p at distance R_{ext} = 10 m from the origin. The plot represents data in the xy-plane.

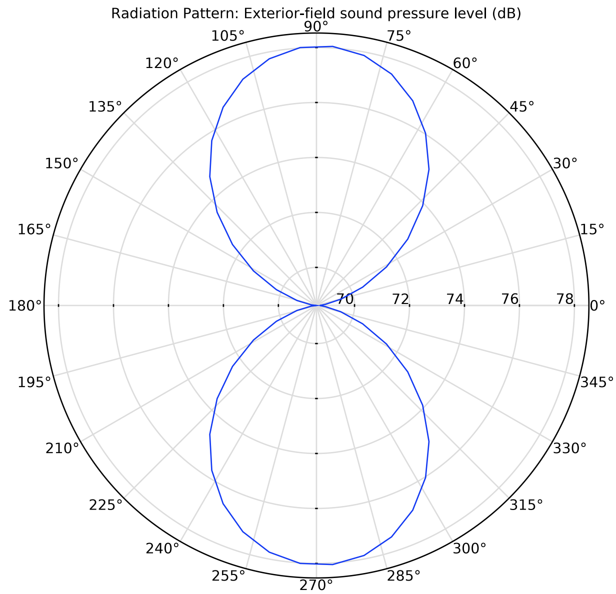


Figure 7: Polar plot of the exterior-field sound pressure level in the xy-plane.

The spatial response is visualized as a 3D radiation pattern plot in Figure 8. The plot represents the sound pressure level. The radial dB scale zero-point has been moved to 66 dB in order to enhance the visualization of the notches in the radiation pattern. The surface color scale is the actual sound pressure level.

Finally, in Figure 9 the pressure is plotted outside of the computational mesh using the Grid 3D dataset. Here, the scattered pressure `pext(x,y,z)` is shown. Alternatively, visualize the sound pressure level by plotting `acpr.efc1.Lp_pext`.

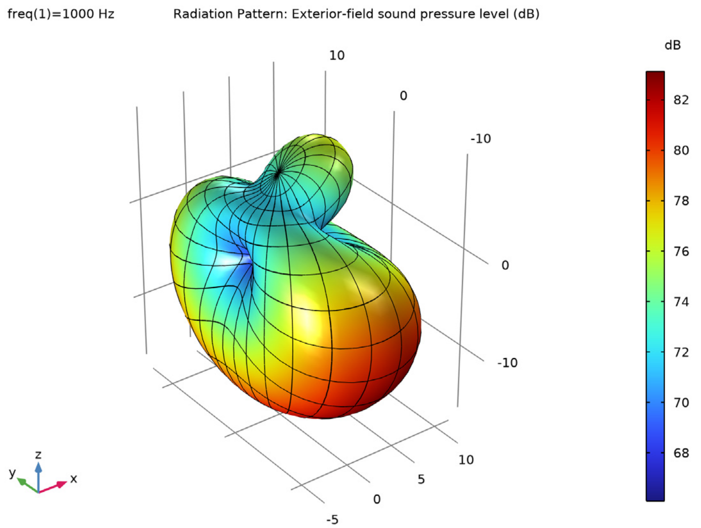


Figure 8: 3D radiation pattern plot of the sound pressure level. The surface color scale is the actual sound pressure level.

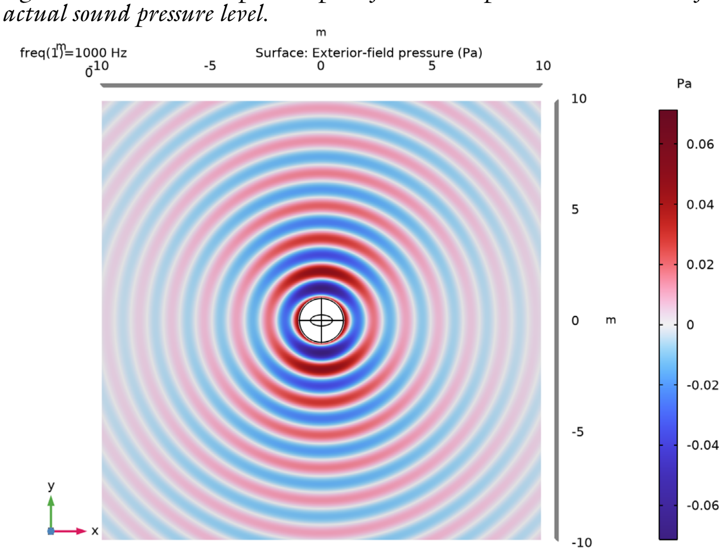


Figure 9: Scattered field outside the computational mesh plotted using the grid dataset and the exterior-field variable pext.

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

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

GLOBAL DEFINITIONS

Load the parameters defining the physical values and the geometric dimensions of the system from file (see Table 1).

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 `acoustic_scattering_parameters.txt`.

GEOMETRY 1

Ellipsoid 1 (elp1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Ellipsoid**.
2 In the **Settings** window for **Ellipsoid**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `A`.

4 In the **b-semiaxis** text field, type `B`.

5 In the **c-semiaxis** text field, type `C`.

Sphere 1 (sph1)

1 In the **Geometry** toolbar, click **Sphere**.

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

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

4 Click **Build Selected**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar to see the full geometry and then select wireframe rendering for easier visualization of the internal geometry. This makes selecting internal domains and boundaries much easier.

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

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **sph1** only.

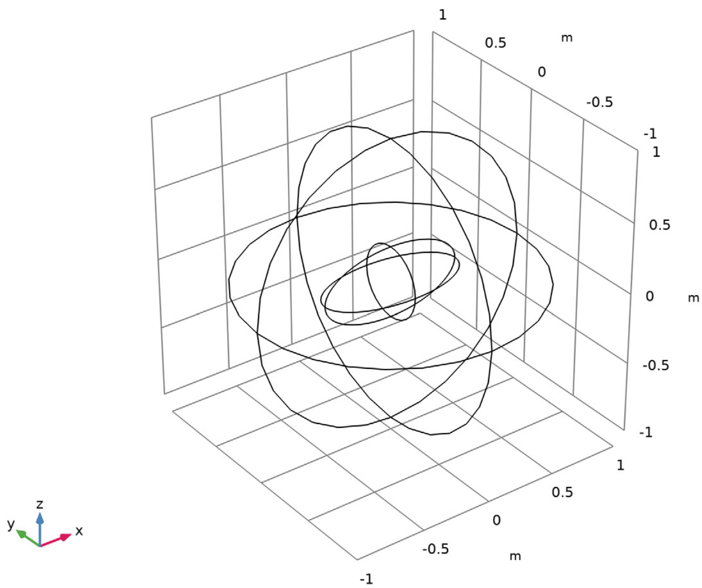
3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **elp1** only.

6 Click **Build All Objects**.

The geometry should look like that in the figure below.



Add a selection for the boundaries on which the exterior field is calculated. These boundaries must surround all scatterers, in this case the ellipsoid.

DEFINITIONS

Exterior Field

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 1–4, 9, 10, 13, and 16 only.

5 In the **Label** text field, type `Exterior Field`.

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>Water, liquid**.

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

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

Set the model reference pressure to 1 Pa, the default for water, and set the reference speed of sound to that in water. The latter option is used to determine the scaling used in the perfectly matched boundary.

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 **Sound Pressure Level Settings** section.

3 From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

The incident pressure field is defined as a **Background Pressure Field** domain contribution. In this model the incident wave has an amplitude of p_0 = 1 Pa and is traveling in the direction e_k = (1,0,1).

Background Pressure Field 1

1 In the **Physics** toolbar, click **Domains** and choose **Background Pressure Field**.

2 Select Domain 1 only.

3 In the **Settings** window for **Background Pressure Field**, locate the **Background Pressure Field** section.

4 In the p_0 text field, type 1.

5 From the c list, choose **From material**.

6 From the **Material** list, choose **Water, liquid (mat1)**.

7 Specify the $\mathbf{e}_k$ vector as

1	x
0	y
1	z

Now set up the exterior-field calculation.

Exterior Field Calculation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.

2 In the **Settings** window for **Exterior Field Calculation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior Field**.

As last step before meshing the model add the perfectly matched boundary.

Perfectly Matched Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfectly Matched Boundary**.

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

3 From the **Selection** list, choose **Exterior Field**.

MESH

Proceed and generate the mesh using the **Physics-controlled mesh** functionality. The frequency controlling the maximum element size is per default taken **From study**. Set the desired **Frequencies** in the study step. In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide*. In this model, we use 6 elements per wavelength; the default **Automatic** is to have 5.

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`.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Pressure Acoustics, Frequency Domain (acpr)** section.

3 From the **Number of mesh elements per wavelength** list, choose **User defined**.

4 In the text field, type `6`.

5 Click **Build All**.

Now inspect the generated mesh. Hide some domains and boundaries to get a better view of the interior parts of the mesh.

6 In the **Model Builder** window, click **Mesh 1**.

7 In the **Graphics** window toolbar, click ▾ next to **Select Boundaries**, then choose **Select Boundaries**.

8 Click the **Click and Hide** button in the **Graphics** toolbar. Clicking on a boundary now hides rather than selects it.

9 Select Boundary 2 only.

The mesh should look like the one depicted in Figure 2.

Now reset the hiding in order to see the full model when processing the results.

10 Click the **Reset Hiding** button in the **Graphics** toolbar.

11 Click the **Click and Hide** button in the **Graphics** toolbar to restore the default click behavior.

Now proceed and solve the model.

STUDY 1

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

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

3 Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click **Compute**.

RESULTS

Plot the total acoustic field (Equation 1).

Total Field

1 In the **Model Builder** window, expand the **Results** node.

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

3 In the **Settings** window for **3D Plot Group**, type `Total Field` in the **Label** text field.

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

Multislice 1

1 In the **Total Field** toolbar, click **More Plots** and choose **Multislice**.

2 In the **Settings** window for **Multislice**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

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

5 Click **OK**.

6 In the **Settings** window for **Multislice**, locate the **Coloring and Style** section.

7 From the **Scale** list, choose **Linear symmetric**.

8 In the **Total Field** toolbar, click **Plot**.

Total Field

The resulting plot should look like Figure 4.

Scattered Field

1 In the **Model Builder** window, right-click **Total Field** and choose **Duplicate**.

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

To reproduce the plot of the scattered acoustic field shown in Figure 5 (top), proceed as follows:

Multislice 1

1 In the **Model Builder** window, expand the **Scattered Field** node, then click **Multislice 1**.

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

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

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

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

6 Click **OK**.

7 In the **Settings** window for **Multislice**, locate the **Coloring and Style** section.

8 From the **Scale** list, choose **Linear symmetric**.

9 In the **Scattered Field** toolbar, click **Plot**.

Background Field

1 In the **Model Builder** window, right-click **Scattered Field** and choose **Duplicate**.

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

Multislice 1

1 In the **Model Builder** window, expand the **Background Field** node, then click **Multislice 1**.

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

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

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

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

6 Click **OK**.

7 In the **Settings** window for **Multislice**, locate the **Coloring and Style** section.

8 From the **Scale** list, choose **Linear symmetric**.

9 In the **Background Field** toolbar, click **Plot**.

Background Field

This plot represents the background or incident acoustic field, and should look like the plot in Figure 5 (bottom).

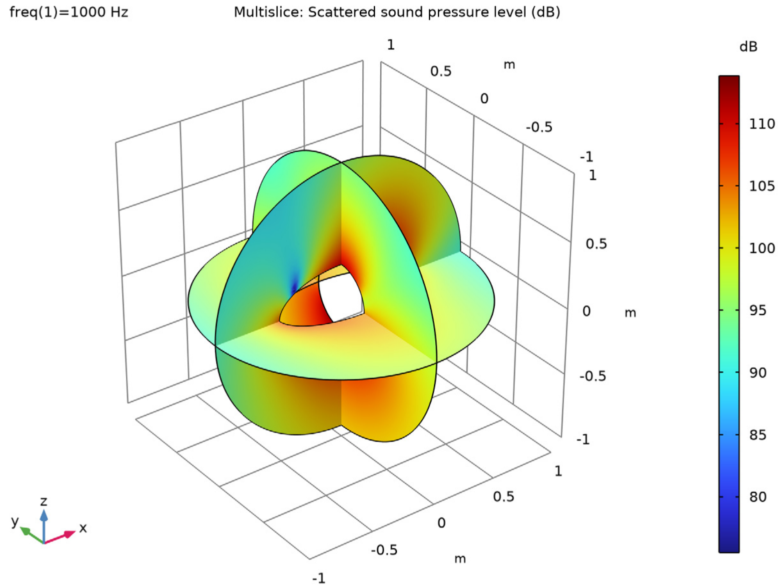
Sound Pressure Level

1 In the **Model Builder** window, right-click **Background Field** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level` in the **Label** text field.
3 Locate the **Color Legend** section. Select the **Show units** check box.

Multislice 1

1 In the **Model Builder** window, expand the **Sound Pressure Level** node, then click **Multislice 1**.
2 In the **Settings** window for **Multislice**, locate the **Expression** section.
3 In the **Expression** text field, type `acpr.Lp_s`.
4 Locate the **Coloring and Style** section. Click **Change Color Table**.
5 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.
6 Click **OK**.
7 In the **Settings** window for **Multislice**, locate the **Coloring and Style** section.
8 From the **Scale** list, choose **Linear**.

9 In the **Sound Pressure Level** toolbar, click **Plot**.



Continue with visualization of the scattered exterior-field pressure and sound pressure level at the distance R_{ext} = 10 m. Use the dedicated **Radiation Pattern** plots.

Reproduce the scattered pressure field (Figure 6) and the sound pressure level of the scattered pressure field (Figure 7) at a distance of 10 m from the center of the ellipsoid in the *xy*-plane as follows:

Exterior-Field Pressure, xy-Plane

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

2 In the **Settings** window for **Polar Plot Group**, type `Exterior-Field Pressure, xy-Plane` in the **Label** text field.

Radiation Pattern 1

1 In the **Exterior-Field Pressure, xy-Plane** 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 `acpr.efc1.pext`.

By default the real part of a variable is plotted. If you need the imaginary part, write `imag(acpr.efc1.pext)`, or if you need the absolute value, write `abs(acpr.efc1.pext)`.

4 Locate the **Evaluation** section. Find the **Evaluation distance** subsection. In the **Radius** text field, type `Rext`.

5 In the **Exterior-Field Pressure, xy-Plane** toolbar, click **Plot**.

Exterior-Field SPL, xy-Plane

1 In the **Model Builder** window, right-click **Exterior-Field Pressure, xy-Plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Exterior-Field SPL, xy-Plane` in the **Label** text field.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field SPL, xy-Plane** 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 `acpr.efc1.Lp_pext`.

4 In the **Exterior-Field SPL, xy-Plane** toolbar, click **Plot**.

Now, plot the scattered pressure field and the sound pressure level of the scattered pressure field at a distance of 10 m from the ellipsoid in the *yz*-plane. Note that the *yz*-plane has the normal in the *x* direction.

Exterior-Field Pressure, yz-Plane

1 In the **Model Builder** window, right-click **Exterior-Field Pressure, xy-Plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Exterior-Field Pressure, yz-Plane` in the **Label** text field.

Radiation Pattern 1

Set the reference direction. It defines the direction in space that corresponds to 0 deg. in the polar plot.

1 In the **Model Builder** window, expand the **Exterior-Field Pressure, yz-Plane** node, then click **Radiation Pattern 1**.

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

3 Find the **Reference direction** subsection. In the **x** text field, type `0`.

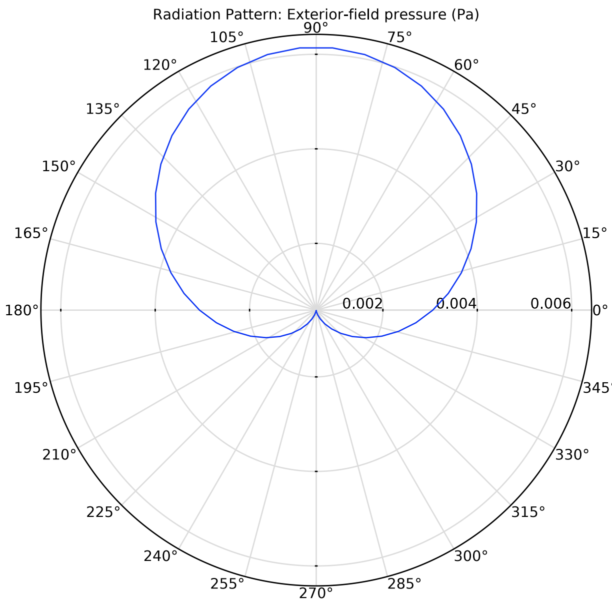
4 In the **y** text field, type `1`.

Set the normal to get the *yz*-plane.

5 Find the **Normal vector** subsection. In the **x** text field, type `1`.

6 In the **z** text field, type `0`.

7 In the **Exterior-Field Pressure, yz-Plane** toolbar, click **Plot**.



Exterior-Field SPL, yz-Plane

1 In the **Model Builder** window, right-click **Exterior-Field SPL, xy-Plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Exterior-Field SPL, yz-Plane` in the **Label** text field.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field SPL, yz-Plane** node, then click **Radiation Pattern 1**.

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

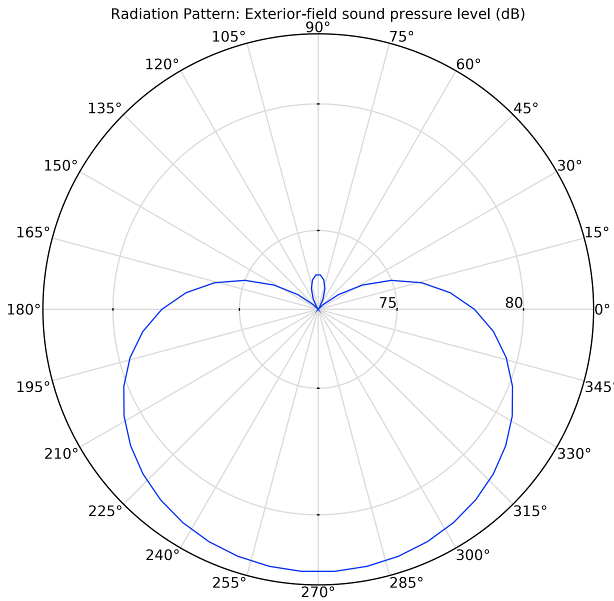
3 Find the **Reference direction** subsection. In the **x** text field, type `0`.

4 In the **y** text field, type `1`.

5 Find the **Normal vector** subsection. In the **x** text field, type `1`.

6 In the **z** text field, type 0.

7 In the **Exterior-Field SPL, yz-Plane** toolbar, click **Plot**.



Finally, plot the exterior-field sound pressure level as a 3D polar plot.

8 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Pressure Acoustics, Frequency Domain>Exterior-Field Sound Pressure Level (acpr)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Exterior-Field Sound Pressure Level (acpr)

Create a new view for this figure. This will keep the view and zoom settings on all the previous figures.

1 In the **Model Builder** window, under **Results** click **Exterior-Field Sound Pressure Level (acpr)**.

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

3 From the **View** list, choose **New view**.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field Sound Pressure Level (acpr)** node, then click **Radiation Pattern 1**.

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

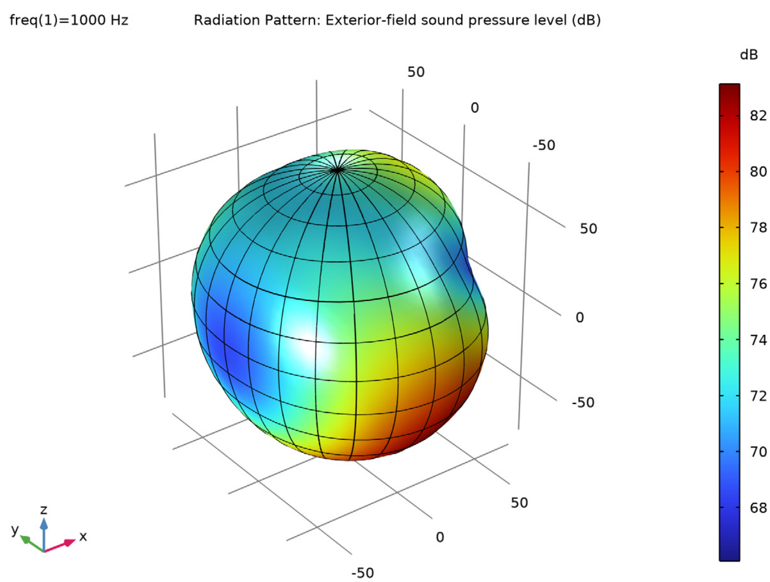
3 Find the **Sphere** subsection. From the **Sphere** list, choose **Manual**.

4 In the **Radius** text field, type `Rext`.

5 In the **Exterior-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

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

The figure should look like the one below. Click on the figure and rotate it to get a sense of the 3D spatial response of the sound pressure level of the scattered field.



In order to better visualize the spatial response, change the plot expression by subtracting 66 dB. This will move the dB scale zero point. The color scale on the surface still represents the sound pressure level. The plot should look like the one in Figure 8.

7 Locate the **Expression** section. In the **Expression** text field, type `acpr.efc1.Lp_pext-66`.

8 Select the **Description** check box. In the associated text field, type `Exterior-field sound pressure level`.

9 Clear the **Use as color expression** check box.

10 In the **Exterior-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

Finally, create and use a grid 3D dataset to plot the pressure field in the *xy*-plane outside of the computational mesh, reproducing Figure 9.

The settings for the grid dataset can be modified to show other cross sections. The plot can also be modified to, for example, plot the sound pressure level `acpr.efc1.Lp_pext`.

Grid 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Grid>Grid 3D**.

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 `-10`.

4 In the **Maximum** text field, type `10`.

5 Find the **Second parameter** subsection. In the **Minimum** text field, type `-10`.

6 In the **Maximum** text field, type `10`.

7 Find the **Third parameter** subsection. In the **Maximum** text field, type `0`.

8 Click to expand the **Grid** section. In the **x resolution** text field, type `300`.

9 In the **y resolution** text field, type `300`.

10 In the **z resolution** text field, type `2`.

Exterior Pressure Field

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

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

3 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 3**.

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

Surface 1

1 Right-click **Exterior Pressure Field** and choose **Surface**.

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

3 From the **Dataset** list, choose **Grid 3D 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `acpr.efc1.pext`.

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

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

7 Click **OK**.

8 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

9 From the **Scale** list, choose **Linear symmetric**.

Filter 1

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

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

3 In the **Logical expression for inclusion** text field, type `sqrt(x^2+y^2+z^2)>Ri*1.05`.

4 In the **Exterior Pressure Field** toolbar, click **Plot**.