Model created in COMSOL Multiphysics 6.4

Bessel Panel

Introduction

The Bessel panel (patented by Philips, see Ref. 1) is a way to arrange a number of loudspeakers so that the angular sound distribution resembles that of a single speaker. This benchmark model is a study of the near and far sound fields created by 25 loudspeakers arranged as an array. The solution is compared with analytical results.

The sound field outside the computational domain is determined by using the built-in Acoustic BEM-FEM Boundary multiphysics coupling between *Pressure Acoustics, Frequency Domain* and *Pressure Acoustics, Boundary Elements*. The FEM interface is used to model the acoustics in the vicinity of the point sources and the BEM interface models the rest of the radiation problem.

Model Definition

A Bessel panel consists of a number of loudspeakers placed equidistantly in a row. The speakers are driven with different signals, some of them in counterphase. For a system of five speakers, the input (voltage and current) is weighted by the factors 1, 2, 2, -2, and 1. This results in an approximately homogeneous polar far-field distribution.

This model combines five Bessel panels in the same pattern to approximate a purely radial sound field. Figure 1 is a sketch of this assembly and the input to each speaker.

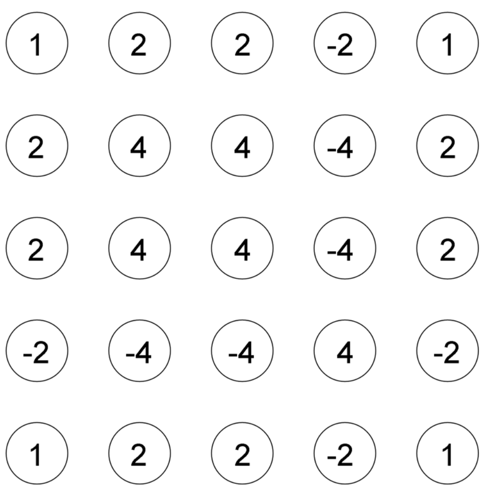


Figure 1: The Bessel panel combination used in the model. The circles represent the speakers and the numbers represent their input. Each row and each column is a Bessel panel in itself.

MODEL EQUATION

The *Pressure Acoustics, Frequency Domain* interface is used to model the pressure field near the point sources in this model. The interface solves the Helmholtz equation

$$\nabla \cdot \left(-\frac{1}{\rho}\nabla p\right) - \frac{\omega^2 p}{\rho c^2} = \sum_L Q_L \tag{1}$$

where p is the pressure, ρ is the density of the medium, $\omega = 2\pi f$ is the angular frequency, c is the speed of sound, and Q_L (SI unit: $1/\mathrm{s}^2$) is a monopole source representing a loudspeaker. In this model, a frequency of $f = 200$ Hz is used. Each loudspeaker L is represented by a monopole point source emitting a flow of strength $S_L = 10^{-2}\mathrm{m}^3/\mathrm{s}\; n_L$, where n_L is the weight factor shown in Figure 1. The Helmholtz equation is also solved in the *Pressure Acoustics, Boundary Elements* interface but with all right-hand-side sources equal to zero.

ANALYTICAL SOLUTION

Each monopole point source is described mathematically as

$$Q_L(\mathbf{x}) = \omega S_L\, \delta(\mathbf{x} - \mathbf{x}_L)$$

where δ refers to the 3D Dirac delta function, $\mathbf{x}_L$ is the location of the speaker L, and $\mathbf{x}$ is the spatial coordinate. Let G be the Green's function for the Helmholtz equation, that is, the solution to the equation

$$(\nabla^2 + k^2)G(\mathbf{x}) = -\delta(\mathbf{x}), \qquad \mathbf{x} \in \mathbf{R}^3$$

where k is the wave number given by ω/c. The equation has an outgoing spherical wave solution, given by

$$G(\mathbf{x}) = \frac{e^{-ik|\mathbf{x}|}}{4\pi|\mathbf{x}|}$$

The corresponding analytical solution to Equation 1 (for a constant density) is given by the convolution of ρG and $\sum Q_L$, that is

$$p(\mathbf{x}) = \int d^3x' \rho G(\mathbf{x} - \mathbf{x}')\sum_L Q_L(\mathbf{x}') = \sum_L (\omega S_L \rho)\frac{e^{-ik|\mathbf{x} - \mathbf{x}_L|}}{4\pi|\mathbf{x} - \mathbf{x}_L|}. \tag{2}$$

This analytical solution is set up as a COMSOL Multiphysics variable and compared with the simulation result (see Figure 6).

MODEL GEOMETRY AND SETUP

The model geometry is shown in Figure 2. The distance between two neighboring loudspeakers is 0.5 m. A box extends 0.2 m around the array of points. The void domain exterior of the solid box is modeled with the BEM interface.

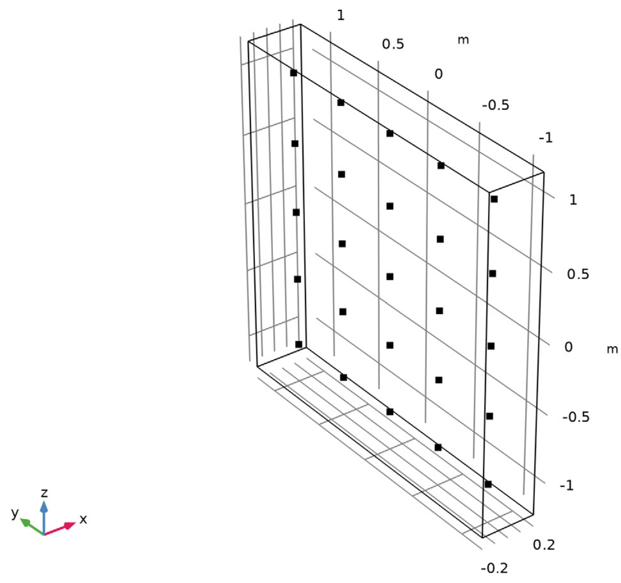


Figure 2: The model geometry.

Results and Discussion

Figure 3 shows the pressure distribution in a slice at $x = 0.2$ m close to the loudspeaker plane (located at $x = 0$ m). In this immediate vicinity of the sources, the sound field is still very inhomogeneous. The sound pressure level is shown in a slice at $x = 0$ m in Figure 4. In both plots, the small white region can be removed by increasing the spatial resolution in the **Grid 3D** dataset used for the BEM evaluation.

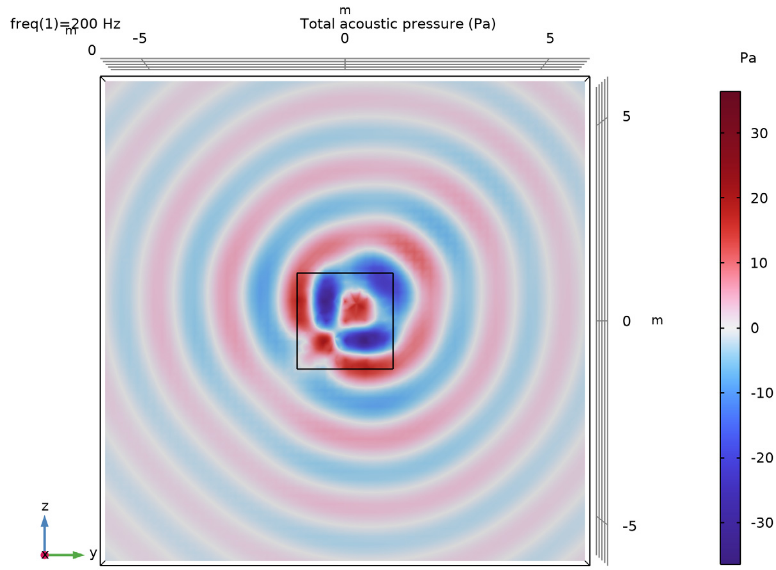


Figure 3: Slice plot of the pressure distribution at 200 Hz. The slice is parallel with the yz-plane and situated at x = 0.2 m.

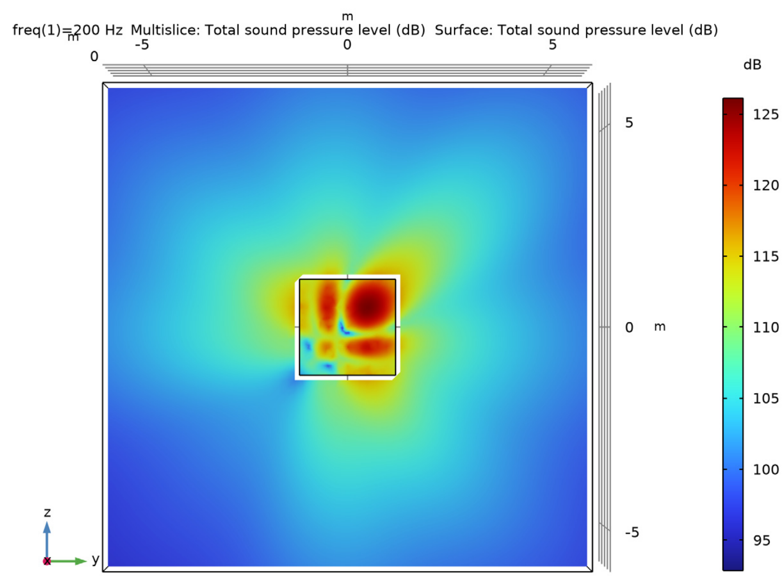


Figure 4: Slice plot of the sound pressure level distribution at 200 Hz. The slice is parallel with the yz-plane and situated at x = 0 m.

Figure 5 shows the exterior-field sound distribution at a distance of 100 m from the speakers. Note that the scale limits are equal to the global extremes of the sound pressure level. Hence the sound pressure level in any two given directions does not differ by more than 3 dB.

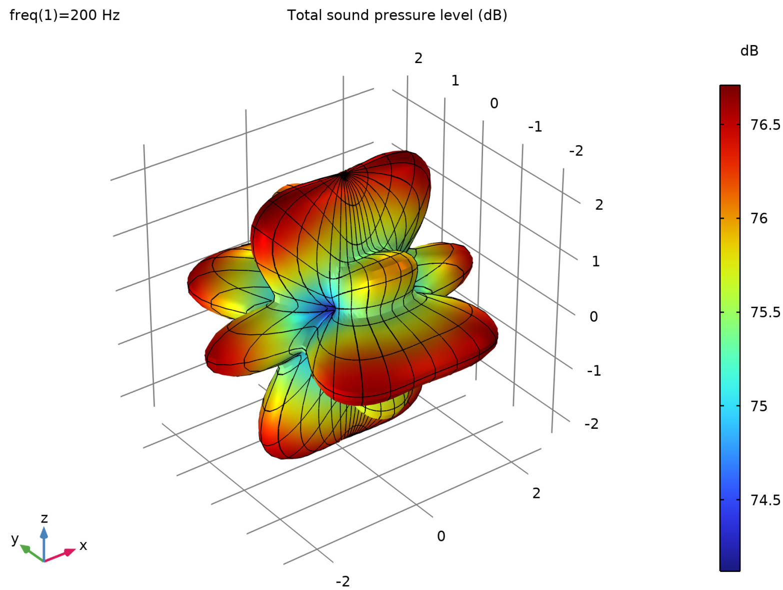


Figure 5: Sound pressure level (dB) at a distance of 100 m from the loudspeakers represented as a 3D radiation pattern plot.

Figure 6 plots the computed exterior-field pressure at a radial distance of 100 m versus polar angle in the positive *xz*-plane and compares it to the analytical solution. As the plot shows, the computed solution is very close to the analytical solution. The accuracy could probably be increased slightly by refining the mesh.

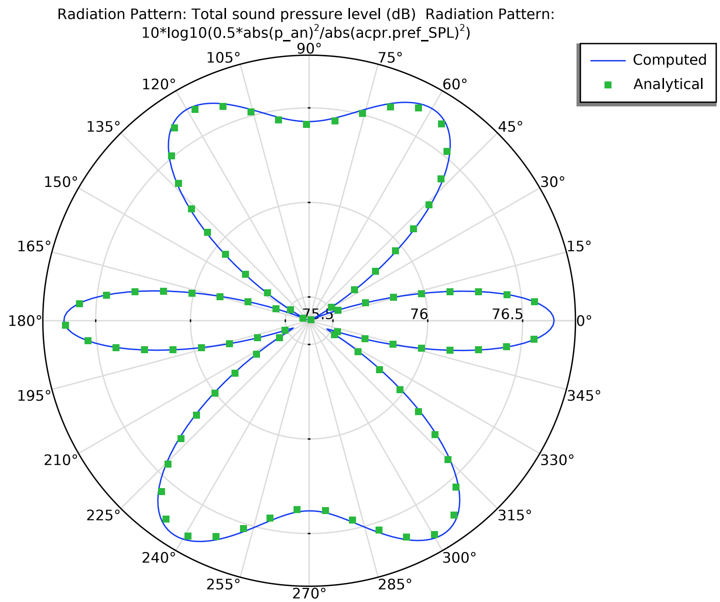


Figure 6: Sound pressure level (dB) at a radial distance of 100 m in the xz-plane (zero azimuthal angle) as a function of the polar angle from the xy-plane. The blue line represents the computed solution and the green line the analytical solution.

Reference

1. “Bessel Panels — High-power Speaker Systems with Radial Sound Distribution,” *Technical Publication 091*, Philips Export BV, 1983.

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

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Acoustics** > **Pressure Acoustics** > **Pressure Acoustics, Frequency Domain (acpr)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics** > **Pressure Acoustics** > **Pressure Acoustics, Boundary Elements (pabe)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies** > **Frequency Domain**.
8 Click **Done**.

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type `0.4`.
4 In the **Depth** text field, type `2.4`.
5 In the **Height** text field, type `2.4`.
6 Locate the **Position** section. From the **Base** list, choose **Center**.
7 Click **Build Selected**.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Point**.
2 In the **Settings** window for **Point**, locate the **Point** section.
3 In the **y** text field, type `-1`.
4 In the **z** text field, type `-1`.
5 Click **Build Selected**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.
2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.
3 Select the object **pt1** only.
4 In the **Settings** window for **Array**, locate the **Size** section.

5 In the **y size** text field, type 5.

6 In the **z size** text field, type 5.

7 Locate the **Displacement** section. In the **y** text field, type 0.5.

8 In the **z** text field, type 0.5.

9 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

10 From the **Show in physics** list, choose **Point selection**.

This makes the object **Array 1** available as a predefined point selection, for example when you later specify flow point sources.

11 Click **Build All Objects**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
S	0.01[m^3/s]	0.01 m³/s	Flow source
f0	200[Hz]	200 Hz	Frequency
c0	343[m/s]	343 m/s	Speed of sound

DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Qs	-S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type 5 9 25 29 in the **Selection** text field.

7 Click **OK**.

Variables 2

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Qs	2*S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `6 10 14 26` in the **Selection** text field.

7 Click **OK**.

Variables 3

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Qs	-2*S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `7 8 15 19 20 24 27 28` in the **Selection** text field.

7 Click **OK**.

Variables 4

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Qs	-4*S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type 11 17 18 22 23 in the **Selection** text field.

7 Click **OK**.

Variables 5

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Qs	4*S*i	m^3/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type 12 13 16 21 in the **Selection** text field.

7 Click **OK**.

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)

Add the material to all domains and the infinite void selection. This is all of the surroundings, where the BEM problem is solved.

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

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Monopole Point Source 1

1 In the **Physics** toolbar, click **Points** and choose **Monopole Point Source**.

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

3 From the **Selection** list, choose **Array 1**.

4 Locate the **Monopole Point Source** section. In the Q_S text field, type `Qs`.

PRESSURE ACOUSTICS, BOUNDARY ELEMENTS (PABE)

Apply the BEM interface to the surroundings by only selecting the voids, that is, not the solid where the FEM interface is used.

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**.

MULTIPHYSICS

Now, set up the multiphysics coupling between the BEM and the FEM pressure acoustics models.

Acoustic BEM–FEM Boundary 1 (abfb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary > Acoustic BEM–FEM Boundary**.

2 In the **Settings** window for **Acoustic BEM–FEM Boundary**, locate the **Boundary Selection** section.

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

DEFINITIONS

Now set up the analytical expression for the far field (see Equation 2). Call this variable `p_an`. You will use it later for comparison with the numerical results.

Integration 1 (intop1)

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

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

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

4 From the **Selection** list, choose **Array 1**.

Variables 6

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
r	sqrt((dest(x)-x)^2+ (dest(y)-y)^2 +(dest(z)-z)^2)	m	Distance between source and observation point
p_an	intop1(Qs*acpr.omega* acpr.rho*exp(-i*acpr.k* r)/(4*pi*r))	Pa	Analytic pressure at observation point

The operator `dest()` evaluates its argument on the destination side, independently of the integration. In other words, `p_an` is a variable defined on the modeling domain. When evaluated in the far field, it gives the analytical acoustic pressure in the point (x, y,z) to use in the radiation pattern plot. The variables are defined globally (entire model).

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, 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**.

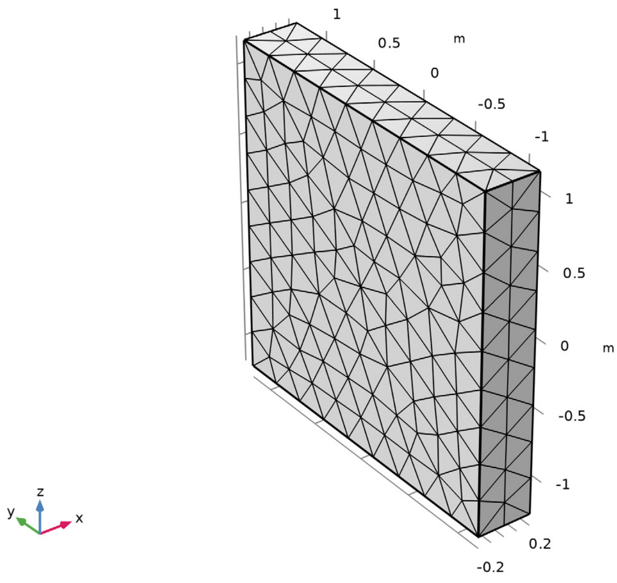
2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Coarse**.

4 Locate the **Pressure Acoustics, Frequency Domain (acpr)** section. From the **Number of mesh elements per wavelength** list, choose **User defined**.

5 In the text field, type 6.

6 Click **Build All**.



STUDY 1

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

RESULTS

Acoustic Pressure (acpr)

After solving the model, six default plots have been created. The first three stem from the FEM model (Pressure Acoustics, Frequency Domain) and the last three from the BEM model (Pressure Acoustics, Boundary Elements).

The FEM interface creates surface plots of the pressure and the sound pressure level, as well as an isosurface plot.

The BEM interface automatically creates a **Grid 3D** dataset where the BEM solution can be evaluated (using kernel evaluation). The first of the three BEM plots shows the pressure on the BEM boundary, in this case this is the boundary in common with the FEM interface. The last two plots depict the pressure and sound pressure levels on the BEM surface as well as in cross sections through the grid dataset.

Take a look at the default plots and then modify them as follows.

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 `-2`.
4. In the **Maximum** text field, type `2`.
5. Find the **Second parameter** subsection. In the **Minimum** text field, type `-6`.
6. In the **Maximum** text field, type `6`.
7. Find the **Third parameter** subsection. In the **Minimum** text field, type `-6`.
8. In the **Maximum** text field, type `6`.
9. Click to expand the **Grid** section. In the **y resolution** text field, type `60`.
10. In the **z resolution** text field, type `60`.

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 **x-planes** subsection. From the **Entry method** list, choose **Coordinates**.
4. In the **Coordinates** text field, type `0.2`.
5. Find the **y-planes** subsection. In the **Planes** text field, type `0`.
6. Find the **z-planes** subsection. In the **Planes** text field, type `0`.

Surface 1

In the **Model Builder** window, right-click **Surface 1** and choose **Disable**.

Multislice 2

1. In the **Model Builder** window, under **Results** > **Acoustic Pressure (pabe)** right-click **Multislice 1** and choose **Duplicate**.
2. In the **Settings** window for **Multislice**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4. Locate the **Expression** section. In the **Expression** text field, type `acpr.p_t`.
5. Click to expand the **Title** section. From the **Title type** list, choose **None**.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Multislice 1**.

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

8 Click the **Go to YZ View** button in the **Graphics** toolbar.

Compare the result with that in Figure 3.

Sound Pressure Level (pabe)

Compare the result with that in Figure 4.

Go back to the default 3D view and then create plots of the spatial response of the Bessel panel.

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

3D Spatial Response

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

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

3 Locate the **Color Legend** section. Select the **Show units** checkbox.

Radiation Pattern 1

1 In the **3D Spatial Response** 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 Select the **Threshold** checkbox. In the associated text field, type `74`.

The value used in the **Threshold** option is subtracted from the absolute level when representing the radiation pattern. This enhances visualizing the spatial characteristics.

5 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of elevation angles** text field, type `50`.

6 In the **Number of azimuth angles** text field, type `100`.

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

8 In the **Radius** text field, type `100`.

9 Locate the **Coloring and Style** section. From the **Grid** list, choose **Finer**.

10 In the **3D Spatial Response** toolbar, click **Plot**.

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

Compare the result with the plot in Figure 5. Note that you may need to click **Zoom Extents** when looking at other figures again.

Spatial Response in xz-plane

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

2 In the **Settings** window for **Polar Plot Group**, type `Spatial Response in xz-plane` in the **Label** text field.

Radiation Pattern 1

1 In the **Spatial Response in xz-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 `pabe.Lp_t`.

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 `100`.

6 Find the **Normal vector** subsection. In the **y** text field, type `1`.

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

The default reference direction (1,0,0) means that 0° in the polar plot corresponds to the positive x-axis direction.

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

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

10 In the table, enter the following settings:

Legends
Computed

Radiation Pattern 2

1 Right-click **Radiation Pattern 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type `10*log10(0.5*abs(p_an)^2/abs(acpr.pref_SPL)^2)`.

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

6 From the **Positioning** list, choose **Interpolated**.

7 In the **Number** text field, type `90`.

8 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Analytical`

9 In the **Spatial Response in xz-plane** toolbar, click **Plot**.

What you see now is the sound pressure level at a distance of 100 m from the panel as a function of the polar angle at zero azimuthal angle. This plot should resemble the one in Figure 6.