Created in COMSOL Multiphysics 5.5

Loudspeaker Driver in a Vented Enclosure

This model is licensed under the COMSOL Software License Agreement 5.5.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example models the acoustic behavior of a loudspeaker driver mounted in a bass reflex enclosure.

Two of the most important design parameters for a loudspeaker driver is its sensitivity and the directivity of the system. The sensitivity is commonly defined as the on-axis sound pressure level, measured at a 1 m distance, as the driver is loaded by an AC voltage of 4 V. The directivity is assessed using the Directivity plot and represents the spatial sensitivity plotted against the frequency in a contour-like plot.

To isolate the driver's performance from that of the environment it usually operates in, the driver is often set directly in an infinite baffle. This approach is used in another example Loudspeaker Driver — Frequency-Domain Analysis model in the Acoustics Module Application Library. The model described here borrows the electromagnetic results from that example and shows how the enclosure affects the sensitivity.

The model uses the Acoustic-Shell Interaction, Frequency Domain multiphysics interface. This interface provides automatic coupling between the Shell equation for the moving parts and the Pressure Acoustics equation in the surrounding air.

Note: The model requires the Acoustics Module and the Structural Mechanics Module since it involves the use of the Shell interface.

Model Definition

Figure 1 shows the geometry of the considered driver in an infinite baffle, as modeled in the Loudspeaker Driver — Frequency-Domain Analysis example. In the model described here, the driver is set in a frame and placed in a bass reflex enclosure (Figure 2). The defining feature of this enclosure type is the vent, which in a properly designed enclosure acts to boost the sound at low frequencies.

Figure 3 shows the driver mounted in the enclosure. The infinite baffle is now flush with the front wall of the enclosure. The moving parts of the driver are drawn as surfaces rather than thin volumes. This lets you model them as shells, and vastly reduces the number of mesh elements required to resolve the model.

Figure 4 displays the complete model geometry, which includes a spherical domain for the air outside the enclosure. The concentric spherical mantle that envelopes the air domain is a perfectly matched layer (PML), acting to absorb the outgoing waves with a minimum of reflections.

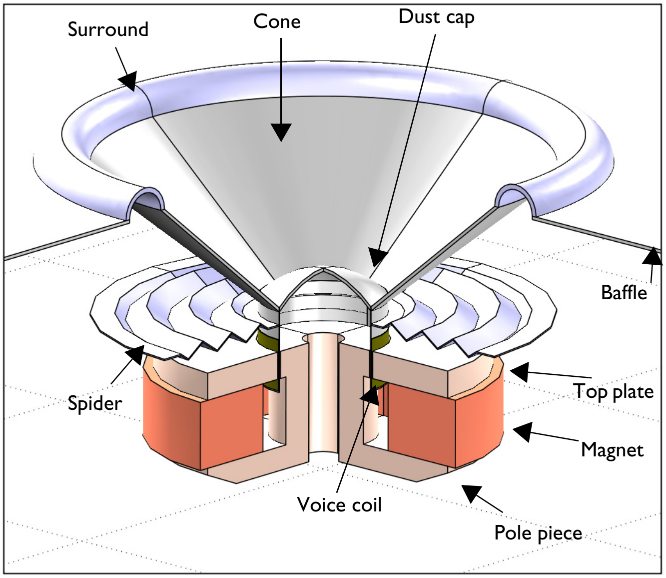


Figure 1: The driver, here set in an infinite baffle as in the Loudspeaker Driver model.

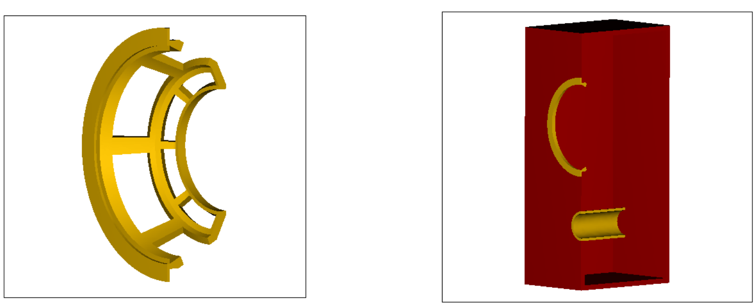


Figure 2: The frame and the vented enclosure.

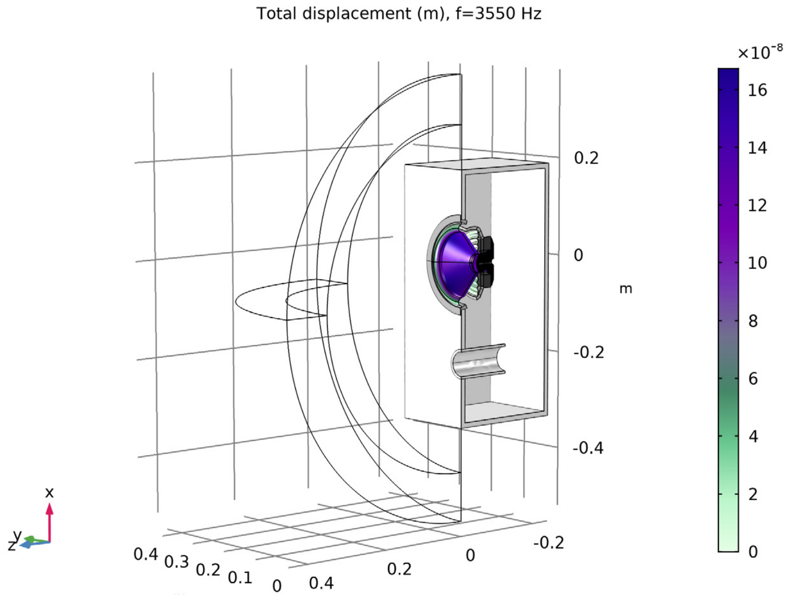


Figure 3: The geometry of the loudspeaker. The front of the box is set in an infinite baffle (not shown). The figure also shows the displacement of the moving parts at 3550 Hz.

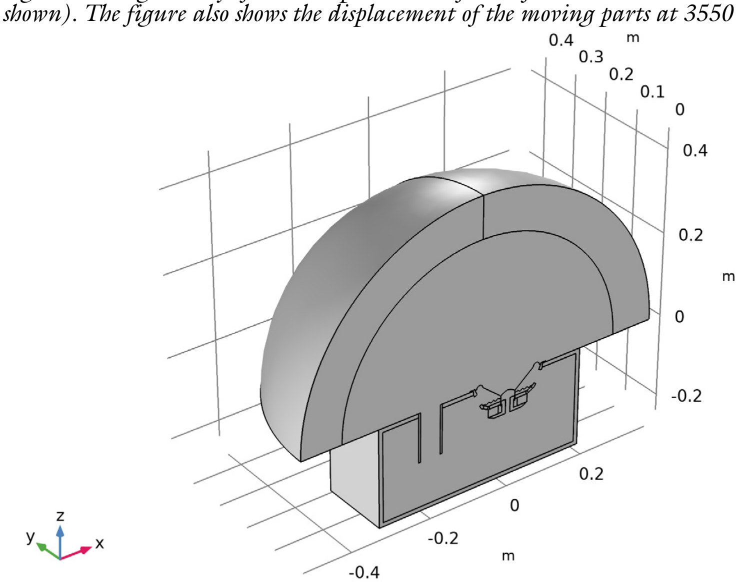


Figure 4: The complete model geometry. Thanks to the symmetry with respect to the xz-plane, the model consists of only one half of the speaker and the outside air.

The model is driven by an electromagnetic force acting on the voice coil:

$$F_e = \frac{\mathrm{BL}V_0}{Z_b} - v\frac{(\mathrm{BL})^2}{Z_b}$$

as derived in the documentation for the Loudspeaker Driver — Frequency-Domain Analysis model. BL is the so-called force factor of the voice coil and Z_b is its blocked impedance; the electric impedance as measured when the coil is at stand-still. While BL is a constant 7.55 N/A, Z_b is complex-valued function of the frequency. Both BL and Z_b are taken directly from the Loudspeaker Driver — Frequency-Domain Analysis, the latter through interpolation from text files with the resistive and inductive contributions listed versus the frequency.

V_0 is the applied driving voltage. The definition of sensitivity assumes a driving power that equals 1 W when the total impedance of the loudspeaker is at its nominal value. The modeled driver has a nominal impedance of 8 Ω, which translates to a driving voltage of $V = V_0 e^{i\omega t}$ with the amplitude V_0 = 4 V. The second term in the expression for the driving force contains the velocity of the voice coil, v, which is unknown prior to the computation.

The electromagnetic force is applied as a homogeneous force density over the boundaries constituting the voice coil. As the cone and the suspension move along and deform, their local normal acceleration is automatically coupled over to become a sound source in the Pressure Acoustics equation. The computed acoustic pressure acts back as a load on the shell.

For a discussion of the material and damping parameters used in the moving parts of the driver, see the Loudspeaker Driver — Frequency-Domain Analysis model documentation.

Results and Discussion

The model produces the local stresses and strains in all the moving parts, as well as the sound pressure distribution inside and outside the enclosure at all frequencies solved for. As an example of how you can visualize the deformations, Figure 3 includes them as a deformed surface plot.

One way of looking at the sound distribution is as in Figure 5, which displays the pressure at 1000 Hz as an isosurface plot. An alternative option is shown in Figure 6, where the sound pressure level in dB is plotted on a slice near the symmetry plane. In Figure 7 and Figure 8, the same plot types illustrate the solution at the highest frequency, 3550 Hz. At this frequency, there is a complex standing wave pattern inside the enclosure.

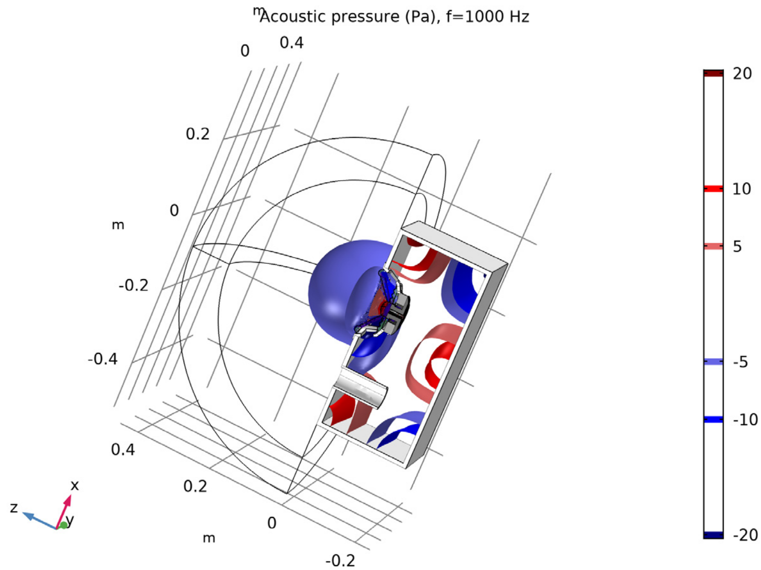


Figure 5: Isosurface plot of the sound pressure at 1000 Hz.

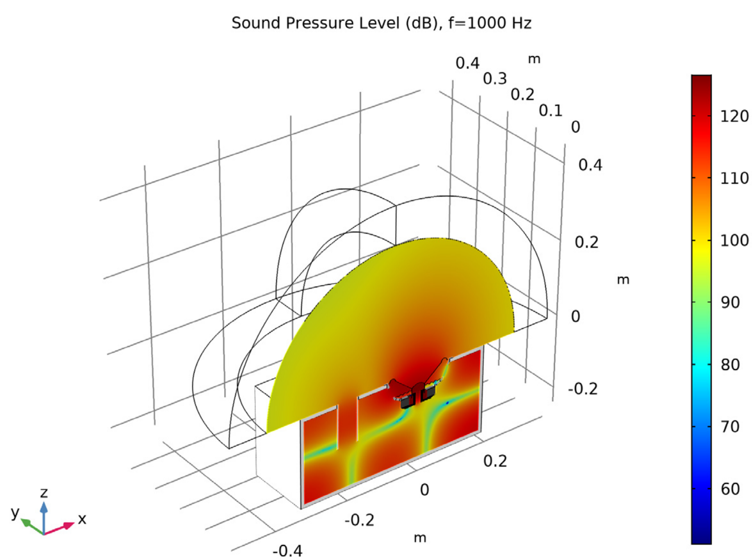


Figure 6: Slice plot of the sound pressure level at 1000 Hz.

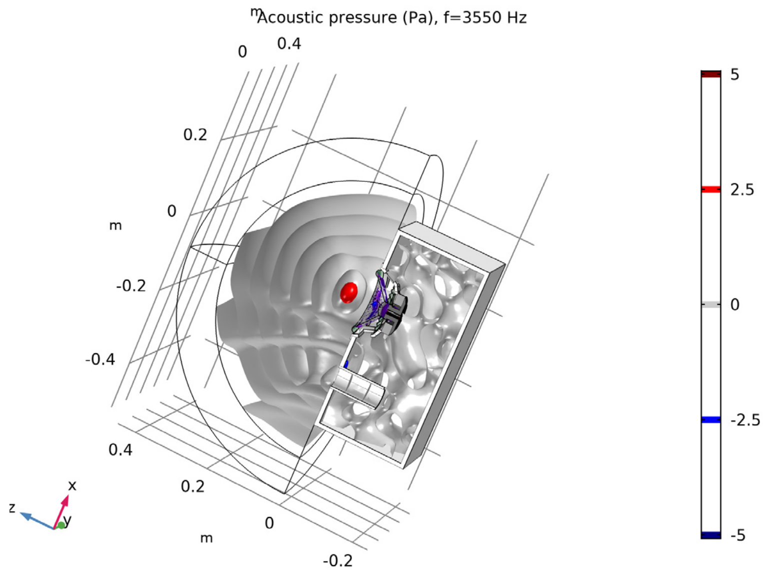


Figure 7: Isosurface plot of the sound pressure at 3550 Hz.

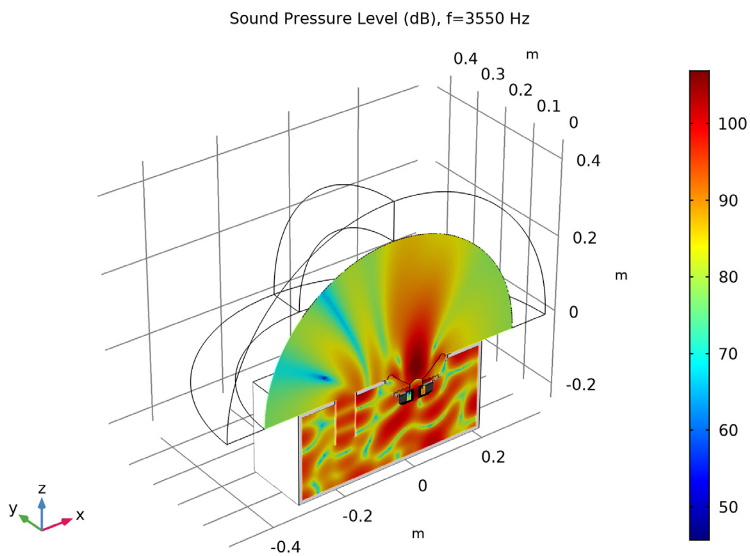


Figure 8: Slice plot of the sound pressure level at 3550 Hz.

Thanks to the full integral exterior-field evaluation available in COMSOL, you can evaluate the pressure not only inside the computational domain but also anywhere outside the domain. This makes it possible to, for instance, plot the sound pressure level at a given distance versus the elevation angle, or evaluate the directivity. The step-by-step instructions for this model show you how to plot the sensitivity (Figure 9) and the directivity (Figure 11).

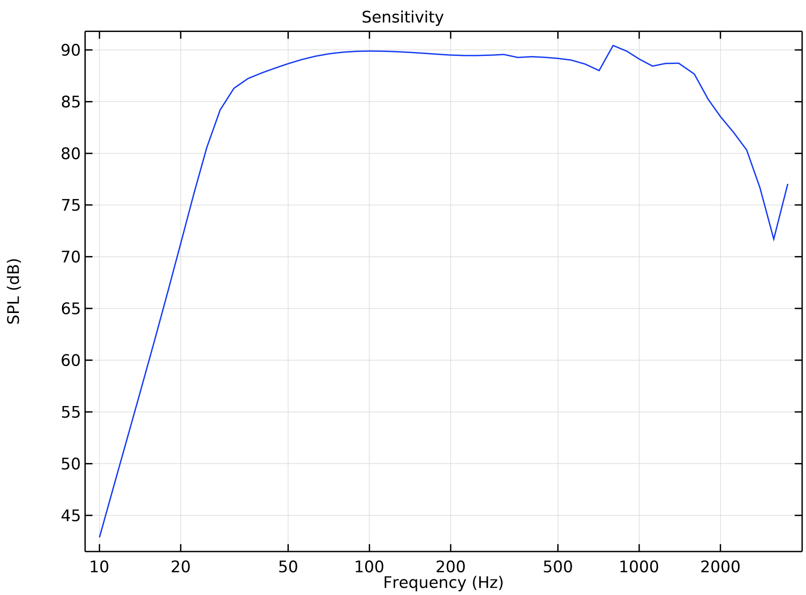


Figure 9: Loudspeaker sensitivity measured as the on-axis sound pressure level (dB) at a distance of 1 m from the unit. The pressure is evaluated using an RMS input signal of 2.83 V, corresponding to a power of 1 W at 8 Ω. Note the logarithmic scale for the frequency.

Compared to the sensitivity of the baffled driver alone (Figure 8 in the Loudspeaker Driver — Frequency-Domain Analysis model), adding the enclosure clearly results in a "boost" for the lower frequencies, roughly in the range between 30 Hz and 100 Hz. As in the other model, the dip at around 700 Hz is due to a Helmholtz resonance in the cavity beneath the voice coil. This resonance would be less pronounced if the model were to account for the thermoviscous losses in the thin air gap between the magnet and the pole piece.

When using the exterior field calculation feature it is also possible to use the dedicated radiation pattern plots to display the spatial response of the speaker. They exist as both 1D polar plots, as 2D plots, and as 3D polar plots. A slightly modified version of the default generated 3D radiation pattern plot is shown in Figure 10. The plot, evaluated for

$f = 1000$ Hz, clearly shows that the speaker has a notch in the horizontal plane and a boost upward and downward. The sensitivity curve in Figure 9 is evaluated at a fixed point 1 m in front of the speaker for all frequencies and, thus, does not provide any spatial information about the speaker performance. This spatial information is visualized in the 3D radiation pattern plot (one frequency at the time) and is of course an important component in speaker design and optimization.

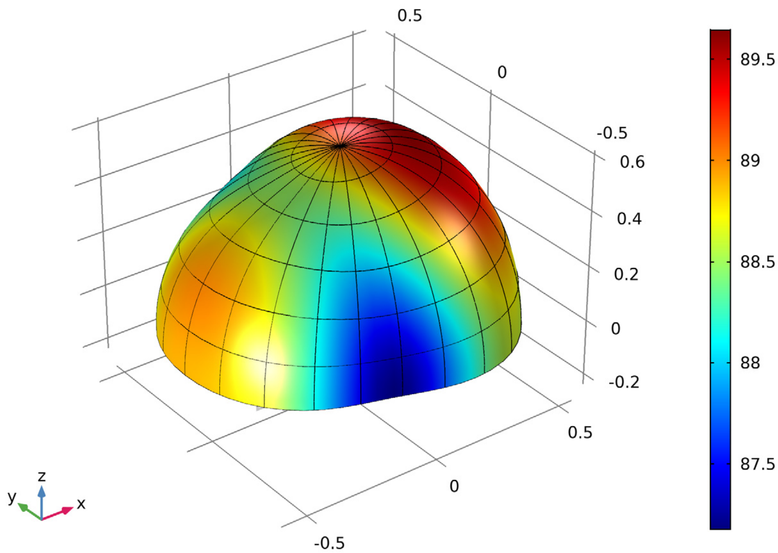


Figure 10: 3D radiation pattern plot of the speaker sensitivity at 1 m evaluated at 1000 Hz.

A plot that gives a mixed frequency and spatial information is the so-called directivity plot depicted in Figure 11. The response of the speaker is evaluated on a half circle 1 m in front of the speaker, the data is collected for all frequencies and plotted in this contour-like plot. The plot is predefined in the Acoustics Module and is simply called *Directivity*.

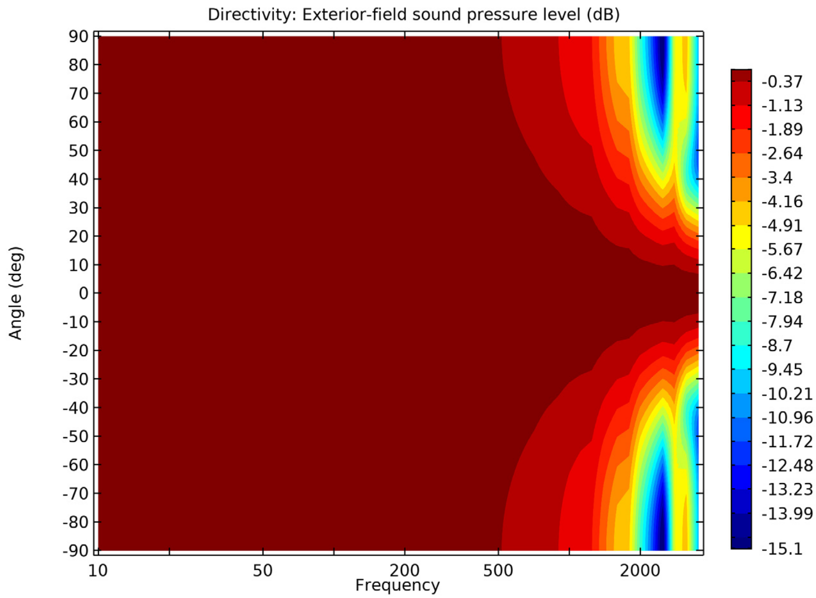


Figure 11: Directivity plot for the loudspeaker.

Notes About the COMSOL Implementation

This application requires the Acoustics Module and the Structural Mechanics Module.

The model instructions let you import the entire geometry, including the air and PML domains. In a similar real-life modeling situation it is usually straightforward to add air and PML domains to your own CAD drawing of the enclosure.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/vented_loudspeaker_enclosure`

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>Acoustic-Structure Interaction>Acoustic-Shell Interaction, Frequency Domain**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Frequency Domain**.
6 Click **Done**.

GEOMETRY 1

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 `vented_loudspeaker_enclosure.mphbin`.
5 Click **Import**.

DEFINITIONS

To make it easier to select some important sets of domains and boundaries in a complicated geometry such as the one you are looking at, it is good modeling practice to begin by defining selections. When working on your own model, these selections are most conveniently defined by clicking and selecting directly in the geometry. The instructions however refer to them by numbers. Whenever a selection is made, it is therefore recommended that you use the Paste Selection button. Enter the number or list of numbers in the text field that appears. Input such as 5, 5-8, 13, or 4, 7, and 9 is accepted.

Explicit 1

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Model Domain` in the **Label** text field.
3 Select Domains 1, 2, and 5 only.

   Because the magnetic engine and the walls of the enclosure will be considered rigid structures, there is no need to include their interior in the simulation.

Explicit 2

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Air Domain` in the **Label** text field.
3. Select Domain 2 only.

Explicit 3

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `PML Domain` in the **Label** text field.
3. Select Domains 1 and 5 only.

Explicit 4

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Cone` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Click the **Wireframe Rendering** button in the **Graphics** toolbar.

   With wireframe rendering, you can see through boundaries and get a better view of which ones you are selecting.
5. Select Boundaries 50, 89, 97, and 108 only.

Explicit 5

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Spider` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 49, 55, 56, 64, 74, 77, 80, 81, 85, 107, 112, 116, 117, 119, 120, 126, 130, and 131 only.

Explicit 6

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Surround` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 35, 37, 133, and 135 only.

Explicit 7

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Apex` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 87, 88, 105, and 106 only.

Union 1

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

2 In the **Settings** window for **Union**, type `Shell 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 **Cone**, **Spider**, **Surround**, and **Apex**.

6 Click **OK**.

Next, define interpolation functions to bring in the blocked resistance and inductance from the model of the driver.

Interpolation 1 (int1)

1 In the **Definitions** toolbar, click **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 `vented_loudspeaker_enclosure_Rb.txt`.

6 Click **Import**.

7 In the **Function name** text field, type `Rb`.

8 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

9 In the **Function** text field, type `ohm`.

10 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Nearest function**.

Set the extrapolation such that the results extend correctly just above the maximal frequency of 3500 Hz in the imported data. The simulation will run up to 3550 Hz.

Interpolation 2 (int2)

1 In the **Definitions** toolbar, click **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 `vented_loudspeaker_enclosure_Lb.txt`.

6 Click **Import**.

7 In the **Function name** text field, type `Lb`.

8 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

9 In the **Function** text field, type `H`.

10 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Nearest function**.

To enable extraction of the velocity and application of the electric force on the apex, define average and integration operators.

Integration 1 (intop1)

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

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

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

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

Average 1 (aveop1)

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

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

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

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

GLOBAL DEFINITIONS

Define parameters for the driving voltage, the BL factor from the loudspeaker driver model, the frequency at which the material losses are specified, and the wavelength at 3500 Hz (used to set the mesh size).

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
V0	4[V]	4 V	Driving Voltage
BL	7.55[N/A]	7.55 Wb/m	Force factor from loudspeaker driver model
f_loss	40[Hz]	40 Hz	Frequency at which loss factor is given
omega_loss	2*pi*f_loss	251.33 Hz	Angular frequency at which loss factor is given
lambda_min	343[m/s]/3500[Hz]	0.098 m	Wavelength at 3500 Hz (used in mesh)

DEFINITIONS

Next, create the expressions used in defining the electric driving force. The Description field is optional, but helps you keep track of what you are doing.

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
v0	av_apex(shell.u_tZ)	m/s	Cone velocity
vol_apex	2*int_apex(shell.d)	m³	Full coil volume
Zb	Rb(freq)+acpr.iomega*Lb(freq)	Ω	Blocked coil impedance
Fe	BL*V0/Zb-v0*BL^2/Zb	N·m/m	Electric driving force

Before creating the materials for use in this model, it is a good idea to specify where you want to solve the pressure acoustics equation and also which boundaries constitute moving parts and thus are to be modeled as shells. Using this information, the software can detect which material properties are needed.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

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

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

3 From the **Selection** list, choose **Model Domain**.

The selection you just made removes the rigid structures from the volume where the pressure acoustics equation will be solved.

SHELL (SHELL)

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

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

3 From the **Selection** list, choose **Shell Boundaries**.

ADD MATERIAL

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

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

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

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

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

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

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

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	`140[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.42`	1	Basic
Density	rho	`720`	kg/m^3	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

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

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

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

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	0.58[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	650	kg/m³	Basic

Material 4 (mat4)

1 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 **Geometric entity level** list, choose **Boundary**.

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

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	1.6[MPa]	Pa	Basic
Poisson's ratio	nu	0.4	1	Basic
Density	rho	67	kg/m³	Basic

ADD MATERIAL

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

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

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

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

MATERIALS

Copper (mat5)

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

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

3 From the **Selection** list, choose **Apex**.

NOTE: In the 2D model of the driver, the apex and the voice coil that is wound around it are two separate domains. In the model at hand, they are lumped together and treated as Copper. Because the apex is not expected to deform considerably, this is fine as long as the shell thickness is tuned so that the total mass becomes the same as in the 2D model.

DEFINITIONS

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 Domain**.

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

5 Find the **Center coordinate** subsection. In the table, enter the following settings:

Xm (m)	Ym (m)	Zm (m)
-0.07	0	0

The default **Polynomial** stretching is kept as it ensures better convergence of the iterative solver used in the study.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

With the materials defined, it is now time to set up the remaining physics of the model. Begin by specifying the symmetry condition in the acoustics domain.

Symmetry 1

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

2 Select Boundaries 1, 4, and 44 only.

The way the geometry is set up, there are two boundaries just outside the surround, separating the air inside and outside the enclosure. To avoid the air leaking through these boundaries, turn them into (interior) hard walls.

Interior Sound Hard Boundary (Wall) 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 Select Boundaries 33 and 136 only.

The acoustics physics is now ready, add an exterior-field calculation.

Exterior Field Calculation 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.
2. Select Boundaries 6 and 45 only.
3. In the **Settings** window for **Exterior Field Calculation**, locate the **Exterior Field Calculation** section.
4. From the **Condition in the y = y^0 plane** list, choose **Symmetric/ Infinite sound hard boundary**.
5. From the **Condition in the z = z^0 plane** list, choose **Symmetric/ Infinite sound hard boundary**.

   You have now supplied a source boundary encompassing all local sound sources and applied symmetry planes to account for the infinite baffle and the geometric symmetry. The Full integral setting lets you compute the pressure field (including phase) at any finite distance from speaker.

SHELL (SHELL)

Now set up the shell physics. Begin by specifying the individual thicknesses and damping properties of the moving parts of the driver.

Damping 1

1. In the **Model Builder** window, right-click **Linear Elastic Material 1** and choose **Damping**.
2. In the **Settings** window for **Damping**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Cone**.
4. Locate the **Damping Settings** section. From the **Damping type** list, choose **Isotropic loss factor**.
5. From the η_s list, choose **User defined**. In the associated text field, type `0.04`.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 2

1. In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2. In the **Settings** window for **Damping**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Spider**.
4. Locate the **Damping Settings** section. In the β_{dK} text field, type `0.14/omega_loss`.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 3

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 From the **Selection** list, choose **Surround**.

4 Locate the **Damping Settings** section. In the β_{dK} text field, type `0.46/omega_loss`.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 4

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 From the **Selection** list, choose **Apex**.

4 Locate the **Damping Settings** section. From the **Damping type** list, choose **Isotropic loss factor**.

5 From the η_s list, choose **User defined**. In the associated text field, type `0.04`.

Thickness and Offset 1

Change the thickness in the default node to `1[mm]`, this will be applied to the Cone of the speaker. Set the thickness of the other speaker parts.

1 In the **Model Builder** window, click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d text field, type `1[mm]`.

Thickness and Offset 2

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

2 In the **Settings** window for **Thickness and Offset**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Spider**.

4 Locate the **Thickness and Offset** section. In the d text field, type `0.35[mm]`.

Thickness and Offset 3

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

2 In the **Settings** window for **Thickness and Offset**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Surround**.

4 Locate the **Thickness and Offset** section. In the d text field, type `1.4[mm]`.

Thickness and Offset 4

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

2 In the **Settings** window for **Thickness and Offset**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Apex**.

4 Locate the **Thickness and Offset** section. In the d text field, type `0.166[mm]`.

Next, apply the electric load onto the apex. Because the load is entered as a volume force density, you need to divide the force by the volume of the apex.

Body Load 1

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

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

3 From the **Selection** list, choose **Apex**.

4 Locate the **Force** section. Specify the $\mathbf{F}_V$ vector as

0	x
0	y
Fe/vol_apex	z

Fix the outer rim of the spider and the surround.

Fixed Constraint 1

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

2 Select Edges 64, 94, 264, and 275 only.

Keep in mind that you can use the **Paste Selection** button for all selections. This is also good idea for the following edge setting, which serves to impose the mirror symmetry.

Symmetry 1

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Select Edges 63, 68, 93, 95, 107, 109, 124, 139, 147, 155, 157, 164, 169, 171, 173, 196, 311–314, 318, 322, 323, 332, 336, 345, 357, 358, 368, and 376 only.

3 In the **Settings** window for **Symmetry**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Global coordinate system**.

Note that the **Axis** to use as symmetry plane normal is set to 2. This means that the solution will be symmetric with respect to the xz-plane.

ADD PHYSICS

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

You have now set up all the relevant physics. This may be a good time to have a second look at all the features of the model, to verify that all settings make sense and apply to the right domains or boundaries. In particular, you can check the **Multiphysics>Acoustic-Structure Boundary 1** node, which has been set automatically.

MESH 1

The mesh needs to resolve the fine details of the geometry as well as the waves at all frequencies. Furthermore, the PML should have at least 5 elements across its thickness (for the rational scaling). To improve the exterior-field evaluation add a thin boundary layer, of thickness `lambda_min/5/10`, adjacent to the PML domain. Turn off the smooth transition option - the single layer is used to get a good normal gradient evaluation. These requirements are all met by the following settings.

Boundary Layers 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Boundary Layers**.
2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Air Domain**.
5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.
3 In the **Number of boundary layers** text field, type `1`.
4 Select Boundaries 6 and 45 only.
5 From the **Thickness of first layer** list, choose **Manual**.
6 In the **Thickness** text field, type `lambda_min/5/10`.

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.
3 In the **Maximum element size** text field, type `lambda_min/5`.

   This gives you a minimum of 5 elements per wavelength at the highest frequency, 3500 Hz. In practice, remembar that a mesh convergence analysis is always recommended.

4 In the **Minimum element size** text field, type `1[mm]`.

The shortest edges of the geometry have length 1 mm, which you will resolve with this setting. You can increase the **Minimum element size** setting to get a coarser mesh in the narrow region. This can trigger a warning as COMSOL checks if the minimum size is larger than a gometric entity. The model will solve when a warning is present. It is up to the user to decide if the warning is relevant for the quality of the simulation results.

Use a swept mesh for the PML (with the size settings you will get 6 elements in the thickness). For more details on meshing PMLs see the modeling sections in the Acoustics Module User's Guide.

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 **PML Domain**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `8`.

Use the recommended 8 mesh elements in the PML when using the polynomial stretching type.

Finally, mesh the loudspeaker box. The mesh here is only for postprocessing purposes to be able to plot the box as a solid structure. It is not used in the computation.

Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.
2 Click **Build All**.

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.

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 Click **Range**, the icon at the right of the **Frequencies** edit-filed.

4 In the **Range** dialog box, choose **ISO preferred frequencies** from the **Entry method** list.

5 In the **Start frequency** text field, type `10`.

6 In the **Stop frequency** text field, type `3550`.

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

8 Click **Replace**.

This gives you frequencies with a 1/6 octave resolution from 10 to 3550 Hz with values specificed by the ISO standard. The model takes about 5 GB of RAM to solve and will solve in around 40 min (depending on your hardware). If you are short on time you can, for example, select 1/3 octave or simply octave spacing.

Generate the default solver sequence, then expand and enable the suggested iterative solve for this acoustic-shell interaction problem. The iterative solver is both faster and more memory efficient than the default direct solver.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (GMRES with GMG and Direct Precon.) (asb1)** and choose **Enable**.

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

RESULTS

3D Plot Group 1

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

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

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

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

5 In the **Title** text area, type `Total displacement (m), f=eval(freq,Hz) Hz`.

6 Clear the **Parameter indicator** text field.

Surface 1

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

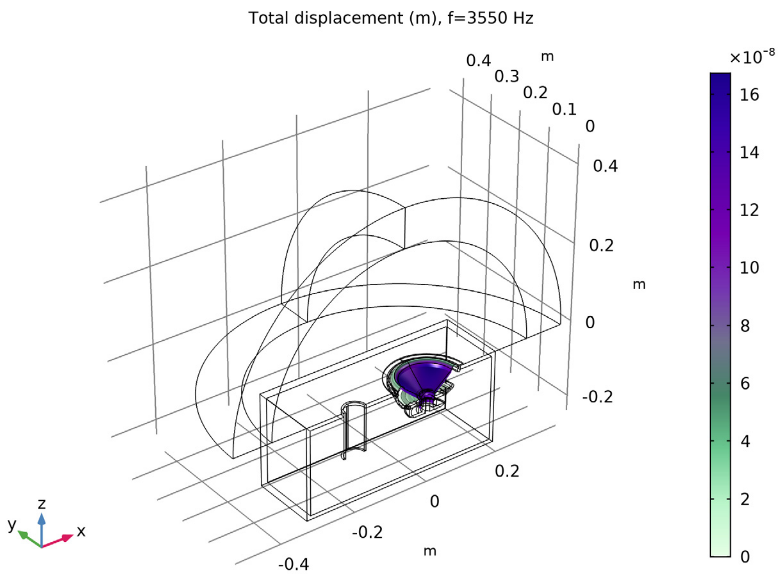
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Shell>Displacement> shell.disp - Total displacement - m**.

Deformation 1

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

Surface 1

1 In the **Displacement** toolbar, click **Plot**.
2 Click the **Go to Default View** button in the **Graphics** toolbar.
3 In the **Model Builder** window, click **Surface 1**.
4 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
5 From the **Color table** list, choose **AuroraBorealis**.



This first plot shows how the shell displaces. The colors represent the total displacement and the deformation shows the position of the membrane at zero phase. Try looking at a couple of different solutions to see how the displacement amplitude and phase change with the frequency.

6 In the **Displacement** toolbar, click **Plot**.

Displacement

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

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

3 From the **Parameter value (freq (Hz))** list, choose **10**.

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

5 From the **Parameter value (freq (Hz))** list, choose **3550**.

6 In the **Displacement** toolbar, click **Plot**.

Datasets

In order to display the box and reproduce Figure 3, create a new data set defined only in the solid parts of the model.

Study 1/Solution 1 (2) (sol1)

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

2 Right-click **Results>Datasets>Study 1/Solution 1 (sol1)** and choose **Duplicate**.

3 In the **Settings** window for **Solution**, type `Study 1/Solution 1 - Speaker box` in the **Label** text field.

Selection

1 Right-click **Study 1/Solution 1 - Speaker box** and choose **Selection**.

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

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

4 Select Domains 3, 4, 6, and 7 only.

Volume 1

1 In the **Model Builder** window, right-click **Displacement** and choose **Volume**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 - Speaker box (sol1)**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Geometry>dom - Entity index**.

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

6 Clear the **Color legend** check box.

7 Select the **Reverse color table** check box.

8 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

By plotting the domain number in grayscale, you have now added a black and white visualization of the enclosure and the solid parts of the driver. Lowering the resolution makes the visualization faster. If you rotate the plot a little, it should now resemble Figure 3.

Next, create a plot of the sound pressure level in a centered slice of the geometry. It is convenient to copy and paste the plot group from the first plot.

Displacement 1

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

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

3 Locate the **Title** section. In the **Title** text area, type `Sound Pressure Level (dB), f=eval(freq,Hz) Hz`.

Surface 1

1 In the **Model Builder** window, expand the **Results>Sound Pressure Level** node, then click **Surface 1**.

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

3 Clear the **Color legend** check box.

Slice 1

1 In the **Model Builder** window, right-click **Sound Pressure Level** and choose **Slice**.

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level - dB**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.

The smoothing algorithm will attempt to make the pressure continuous across the membrane. You get a sharper plot if you remove it.

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

The plot range is dominated by the large pressure drop in the PML. You can hide the PML domain by letting the solution data set be defined only in the air.

Study 1/Solution 1 (3) (sol1)

1 In the **Results** toolbar, click **More Datasets** and choose **Solution**.

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

Selection

1 Right-click **Study 1/Solution 1 - Air domain** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Air Domain**.

Sound Pressure Level

1 In the **Model Builder** window, click **Sound Pressure Level**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 - Air domain (sol1)**.

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

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

The plot should now like Figure 8. Switch to 1000 Hz to reproduce Figure 6.

6 From the **Parameter value (freq (Hz))** list, choose **1000**.

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

Create a third plot group to show the 3D exterior-field polar plot. It is the spatial sensitivity characteristic evaluated at a distance of 1 m and at 1000 Hz.

3D Plot Group 3

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

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

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **1000**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Radiation Pattern 1

1 In the **Exterior-Field Sound Pressure Level** 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.Lp_pext-77)/20`.

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

5 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of elevation angles** text field, type `30`.

6 In the **Number of azimuth angles** text field, type `60`.

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

8 In the θ **range** text field, type `90`.

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

The first expression defines the shape of the 3D polar plot. In this case it is modified to set the reference at 77 dB and then scaled by a factor of 20 in order to visualize it relative to the model geometry. Changing the reference to 77 dB enhances the visualization of the notches. The surface color (second expression, enabled with the check box) gives the true SPL at 1 m from the speaker. Change the evaluation frequency to see how the directivity of the speaker changes with frequency. Set the expression back to `acpr.efc1.Lp_pext` and use the **Zoom Extents** tool.

Volume 1

In the **Model Builder** window, right-click **Volume 1** and choose **Copy**.

Volume 1

1 In the **Model Builder** window, right-click **Exterior-Field Sound Pressure Level** and choose **Paste Volume**.
2 In the **Exterior-Field Sound Pressure Level** toolbar, click **Plot**.
3 Click the **Go to Default View** button in the **Graphics** toolbar.

Rotate the plot in order to reproduce Figure 10.

Now, plot the sensitivity versus the frequency and reproduce Figure 9 as follows.

1D Plot Group 4

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Sensitivity` 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 `Sensitivity`.
5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
6 In the associated text field, type `Frequency (Hz)`.
7 Select the **y-axis label** check box.
8 In the associated text field, type `SPL (dB)`.

Octave Band 1

1 In the **Sensitivity** toolbar, click **More Plots** and choose **Octave Band**.
2 In the **Settings** window for **Octave Band**, locate the **Selection** section.
3 From the **Geometric entity level** list, choose **Global**.
4 Locate the **y-Axis Data** section. In the **Expression** text field, type `pext(0,0,1)`.

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

This will plot the sound pressure level using the reference pressure, which defaults to the commonly used value 20 μPa. The `pext(x,y,z)` operator extracts the sound pressure in (x,y,z) = (0,0,1[m]).

You can change the style to 1/3 octave or octave to plot the sensitivity in bands if needed.

6 In the **Sensitivity** toolbar, click **Plot**.

The next set of instructions show you how to create an isosurface plot of the pressure distribution in and outside of the speaker. The last plot is the directivity plot for the speaker.

Displacement 1

1 In the **Model Builder** window, right-click **Displacement** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure` in the **Label** text field.
3 Locate the **Title** section. In the **Title** text area, type `Acoustic pressure (Pa), f=eval(freq,Hz) Hz`.
4 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 - Air domain (sol1)**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Acoustic Pressure** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3 Clear the **Color legend** check box.

Isosurface 1

1 In the **Model Builder** window, right-click **Acoustic Pressure** and choose **Isosurface**.
2 In the **Settings** window for **Isosurface**, locate the **Levels** section.
3 From the **Entry method** list, choose **Levels**.
4 In the **Levels** text field, type `-5 -2.5 0 2.5 5`.
5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.
6 In the **Acoustic Pressure** toolbar, click **Plot**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

The plot should resemble Figure 5. To reproduce Figure 7, change the frequency and the levels for the isosurface plot.

Acoustic Pressure

1. In the **Model Builder** window, click **Acoustic Pressure**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Parameter value (freq (Hz))** list, choose **1000**.

Isosurface 1

1. In the **Model Builder** window, click **Isosurface 1**.
2. In the **Settings** window for **Isosurface**, locate the **Levels** section.
3. In the **Levels** text field, type `-20 -10 -5 5 10 20`.
4. In the **Acoustic Pressure** toolbar, click **Plot**.
5. Click the **Zoom Extents** button in the **Graphics** toolbar.

Finally, create the directivity plot depicted in Figure 11 for the loudspeaker.

1D Plot Group 6

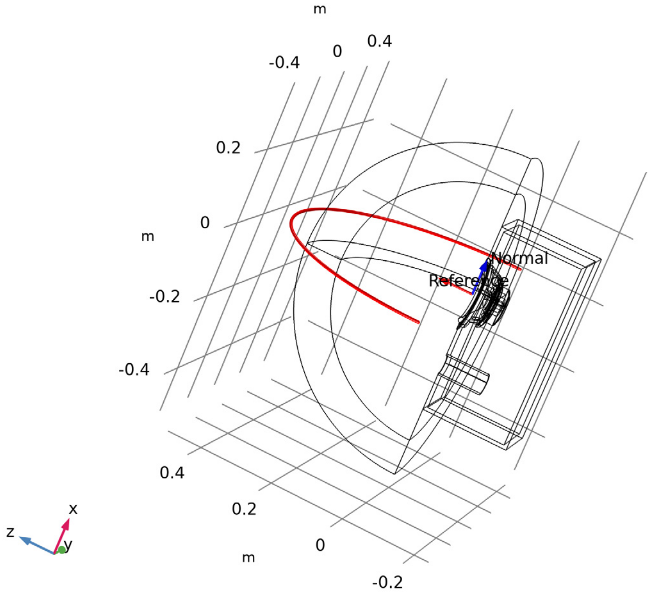
1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Directivity Plot` in the **Label** text field.

Directivity 1

1. In the **Directivity Plot** toolbar, click **More Plots** and choose **Directivity**.
2. In the **Settings** window for **Directivity**, locate the **Evaluation** section.
3. Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.
4. In the ϕ **start** text field, type `-90`.
5. In the ϕ **range** text field, type `180`.
6. In the **Number of angles** text field, type `90`.
7. Find the **Reference direction** subsection. In the **x** text field, type `0`.
8. In the **z** text field, type `1`.
9. Find the **Normal** subsection. In the **x** text field, type `1`.
10. In the **z** text field, type `0`.

    These settings give a directivity plot taken in the yz-plane where 0 deg. correspond to the z-axis direction. The evaluation plane can easily be visualized using the **Preview Evaluation Plane** feature. The radius of the red evaluation circle displayed is normalized relative to the geometry.

11 Click **Preview Evaluation Plane**.



12 In the **Directivity Plot** toolbar, click **Plot**.

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