Created in COMSOL Multiphysics 5.5

Loudspeaker Driver — Frequency-Domain Analysis

Introduction

This example shows how to model a loudspeaker driver of the dynamic cone type, common for low and medium frequencies. The analysis is carried out in the frequency domain and thus represents the linear behavior of the driver. The instructions walk you through modeling its electromagnetic, structural, and acoustic properties. The output from the model includes the total electric impedance and the sensitivity (the on-axis sound pressure level at a nominal driving voltage) as functions of the frequency. The spatial characteristics of the speaker are depicted in a directivity plot.

When performing the acoustic measurements in this model, the driver is set up in an infinite baffle — a wide reflective surface acting to shut out the sound produced on the backside of the cone. An extended 3D version of the model, Loudspeaker Driver in a Vented Enclosure, uses the electromechanical properties modeled here and adds a vented enclosure.

The model is set up with a combination of the Magnetic Fields interface from the AC/DC Module, the Acoustic-Structure Interaction multiphysics interface from the Acoustics Module. The Lorentz Coupling multiphysics feature is used for handling the electromagnetic forces and induced currents over the voice coil. A first, optional analysis solves only the electromagnetic part of the problem, with the driver in stand-still. From here, a driving force factor and the blocked voice coil impedance can be extracted and exported. The second and final analysis is of the full model, including the relevant multiphysics interactions all the way from the driving voltage to the computed sound pressure level.

Note: This model requires the both the Acoustics Module and the AC/DC Module.

Model Definition

Figure 1 shows the geometry of the baffled driver with its functional parts. The field from the *magnet* is supported and focused by the iron *pole piece* and *top plate* to the thin gap where the *voice coil* is wound around a former extending from the apex of the *cone*. Although the voice coil consists of many turns of wire, it is for simplicity drawn and modeled as a homogenized domain. When a driving AC voltage is applied to the voice coil, the resulting force causes it to vibrate, and the cone to create sound.

The *dust cap* protects the magnetic motor. In this design, it is made of the same stiff and light composite material as the cone and also contributes to the sound. A centered hole in the pole piece counteracts pressure buildup beneath the dust cap. The *suspension*, consisting of the *surround*, made of a light foam material, and the *spider*, a flexible cloth, keep the cone in place and provide damping and spring forces.

The outer perimeters of the magnet and suspension are normally attached to a *basket*, a hollow supporting metal structure. The basket is not included in this model, but the magnet assembly and outer rims of the spider and surround are considered to be fixed. The omission of the basket means that the considered geometry is rotationally symmetric and can be modeled in the *rz*-plane.

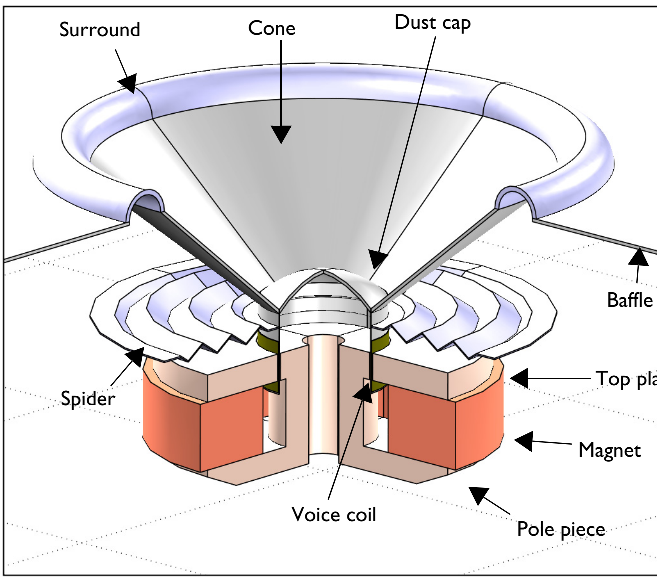

Figure 1: Geometry of the modeled loudspeaker driver.

The loudspeaker is driven by a time-harmonic voltage, $V = V_0\exp(i\omega t)$, applied to the voice coil. The following theory section first describes the electromagnetic analysis of the current in the voice coil and the driving force that this current gives rise to. The relation between the driving voltage and the force on the voice coil as well as the so-called back EMF are easily set up in COMSOL using built-in functionality. This force is then applied in an acoustic-structure interaction analysis to compute the sound generation. More details are given in the section Electromagnetic Interactions.

The structural equation is solved in the moving parts of the driver, and a pressure acoustics equation in the surrounding air. The pressure acoustics equation is automatically excited by the structural vibrations, and feeds back the pressure load onto the structure, using the built-in *Multiphysics* coupling.

The air domains and the baffle should ideally extend to infinity. To avoid unphysical reflections where you truncate the geometry, you use a perfectly matched layer (PML), as seen in Figure 2. For more information about PMLs in acoustics, see the section *Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces)* in the *Acoustics Module User's Guide*.

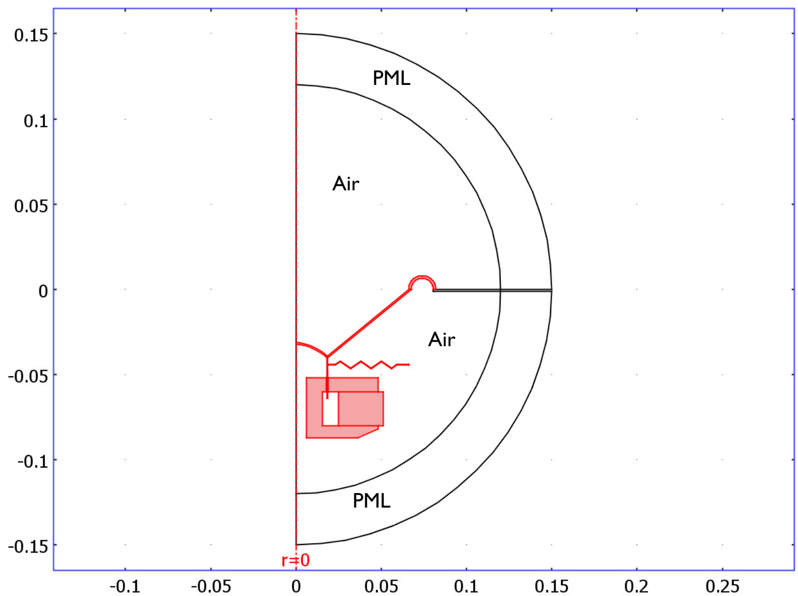

Figure 2: Overview of the model geometry with the structures in pink.

Although the modeled air domain has a radius of only 0.12 m, the local acoustic pressure and phase can be extracted anywhere outside the computational domain by using the exterior-field pressure computations. The sensitivity is calculated as the sound pressure level on the axis at a radius of 1 m, for the applied voltage V_0 = 4 V. This functionality also allows the computation of the Directivity plot for the speaker, again evaluated at 1 m.

ELECTROMAGNETIC INTERACTIONS

The Lorentz force on a wire of length L and with the current $\mathbf{I}$ in an externally generated magnetic flux density $\mathbf{B}$ perpendicular to the wire is given by $F = L\mathbf{I} \times \mathbf{B}$. The voice coil consists of a single copper wire making $N_0 = 100$ turns. The coil is homogenized so that

$$N_0 I = \int_A J_\varphi dA$$

where J_φ is the azimuthally directed current density through a cross-section of the coil, and the integral is taken over its area in the rz-plane. The total driving force on the coil hence becomes

$$F_\mathrm{e} = -\int_V J_\phi B_r \mathrm{d}V \qquad (1)$$

with B_r being the r-component of the magnetic flux density, and the integral evaluated over the volume occupied by the coil domain.

The Lorentz force is applied to the voice coil through The Lorentz Coupling multiphysics feature.

The current through the voice coil relates to the applied voltage as

$$I = (V_0 + V_\mathrm{be})/Z_\mathrm{b} \qquad (2)$$

where Z_b is the *blocked electric impedance* (the electric impedance of the voice coil measured while the speaker's moving parts are stationary) and $-V_\mathrm{be}$ denotes the *back EMF* (the voltage induced in the coil due to its motion through the permanent magnetic field in the gap).

The back EMF is automatically added to the voice coil using the the Lorentz Coupling multiphysics feature available in the AC/DC Module.

EXPORT TO LOUDSPEAKER DRIVER IN A VENTED ENCLOSURE

If you write Equation 1 in terms of the coil current I rather than the cross-sectional current density, you get

$$F_e = -\frac{2\pi I N_0}{A}\int r B_r \mathrm{dA} \qquad (3)$$

as it is assumed that $J_\phi = I{\cdot}N_0/A$ and is constant in the coil cross-section of area A. The common factor in F_e and V_be is known in the loudspeaker community as the force factor, BL:

$$\mathrm{BL} = -\frac{2\pi N_0}{A}\int rB_r \mathrm{dA} \tag{4}$$

Note that if $A \to 0$, the integral becomes equal to a magnetic flux density times the length of the coil; hence the name.

With knowledge of BL and the frequency-dependent Z_b, Equation 2, Equation 3, and Equation 4 can be rearranged to form a relationship between V_0 and F_e:

$$F_\mathrm{e} = \frac{\mathrm{BL}V_0}{Z_\mathrm{b}} - v\frac{(\mathrm{BL})^2}{Z_\mathrm{b}}$$

This is the expression used for the driving force in the Loudspeaker Driver in a Vented Enclosure model. Note the dependence on the velocity of the moving coil, which is unknown prior to the acoustic-structure interaction computation.

Results and Discussion

The magnetic field in and around the magnetic motor appears in Figure 3. The maximum field in the air arises in the gap between the pole piece and the top plate. Performing the integral in Equation 4 over the voice coil domain gives a force factor BL = 7.63 N/A.

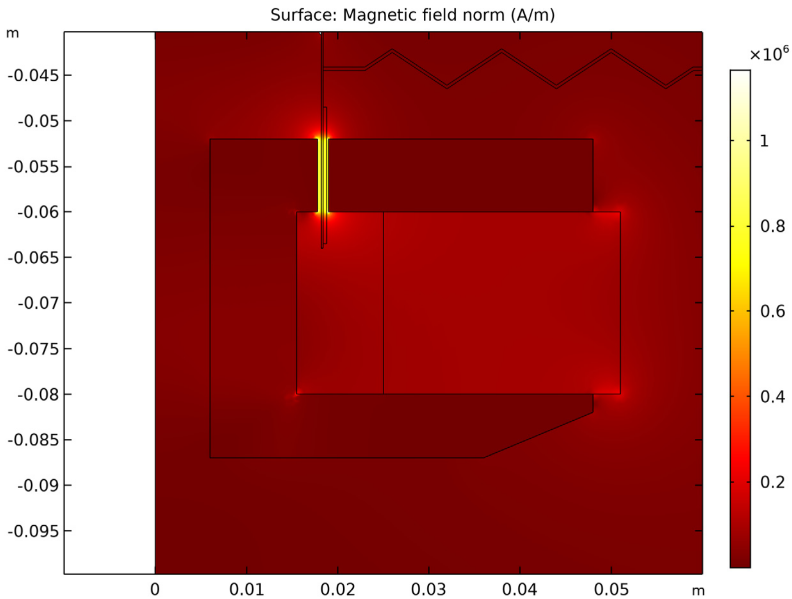

Figure 3: Magnetic field in and around the magnetic motor.

The iron in the pole piece and top plate is modeled as a nonlinear magnetic material, with the relationship between the B and H fields described by interpolation from measured data. Figure 4 shows the local effective relative permeability $\mu_r = B/(\mu_0 H)$. The plot shows that the iron is close to saturation in the center of the pole piece, but remains in the linear regime above and below the magnet. This indicates that if you want to use less material, you can likely decrease the radius of the pole piece and top plate with very little effect on the magnetic field in the gap.

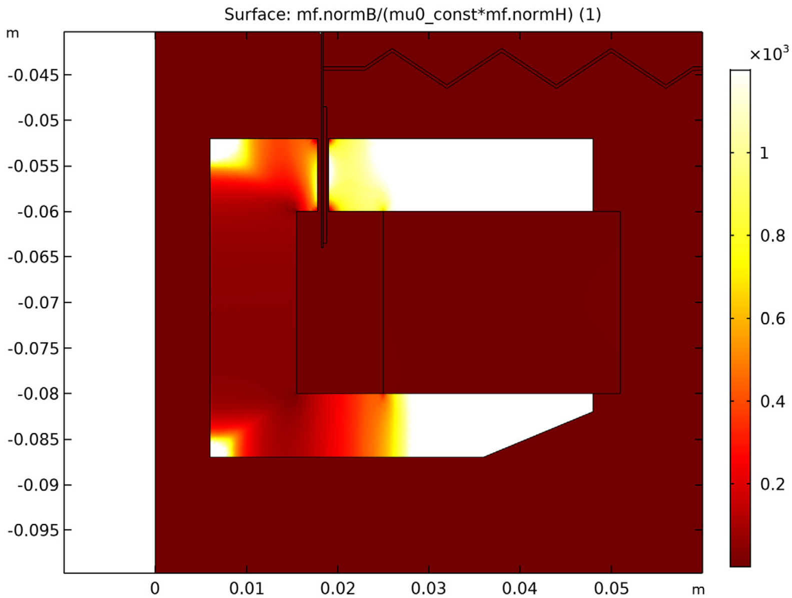

Figure 4: The local relative permeability in the pole piece and top plate, when subjected to the field from the magnet.

In computing the blocked coil impedance, the AC equation is linearized around the local permeability resulting from the static solution. Figure 5 shows the induced currents at two different frequencies.

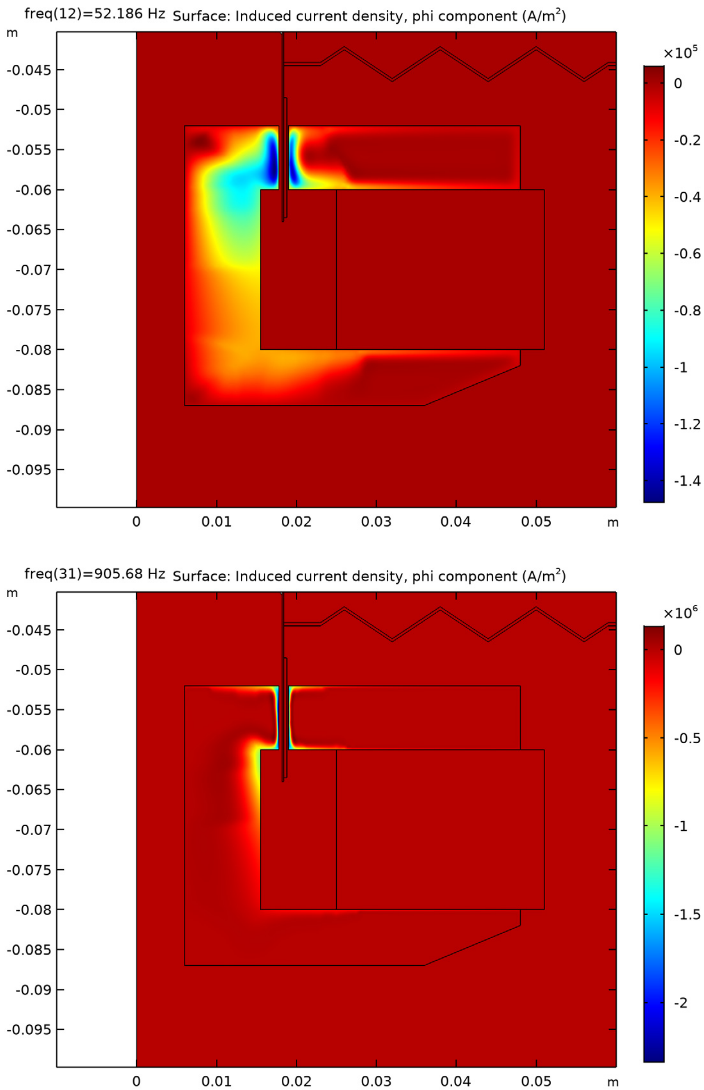

Figure 5: Induced currents in the pole piece and top plate at 52 Hz (top) and 905 Hz (bottom).

At the higher frequency, the skin effect brings the currents closer to the surfaces. This causes the inductance as well as the resistive part of the impedance to change with the frequency. Figure 6 shows a plot of the blocked coil inductance versus frequency.

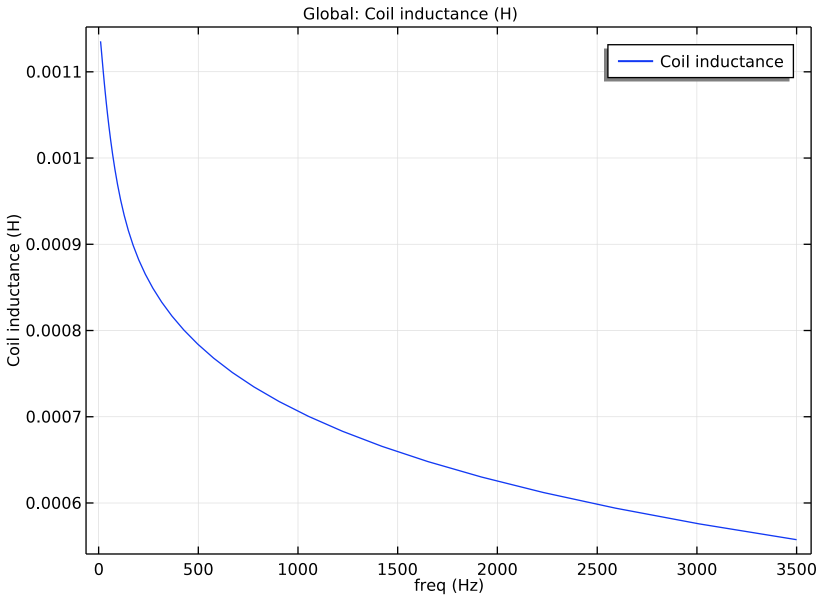

Figure 6: The inductance of the blocked coil as a function of frequency.

From the acoustic-structure interaction analysis, Figure 7 shows the sound pressure level distribution at 3500 Hz. A minimum has formed in a direction about 45 degrees above the baffle. At lower frequencies, the sound pressure level is rather evenly distributed but peaks in the on-axis direction.

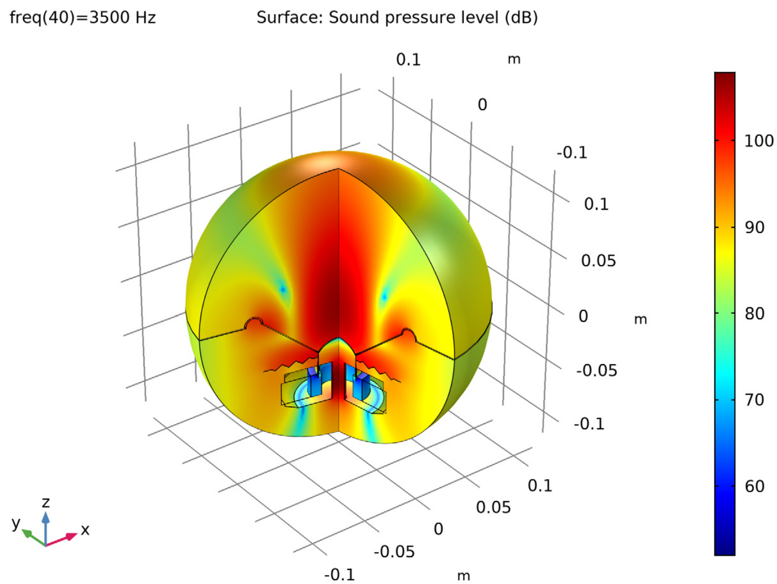

Figure 7: Sound pressure level distribution in dB at 3500 Hz.

Figure 8 presents the loudspeaker's sensitivity depicted both in 1/3 octave bands and as a continuous curve. The plot is realized using the specialized *Octave Band* plot available in the Acoustics Module. The preferred operating range is where the response is rather even, that is, roughly in the range 100 Hz–1500 Hz. A vented enclosure can extend the range to lower frequencies, as shown in the model Loudspeaker Driver in a Vented Enclosure.

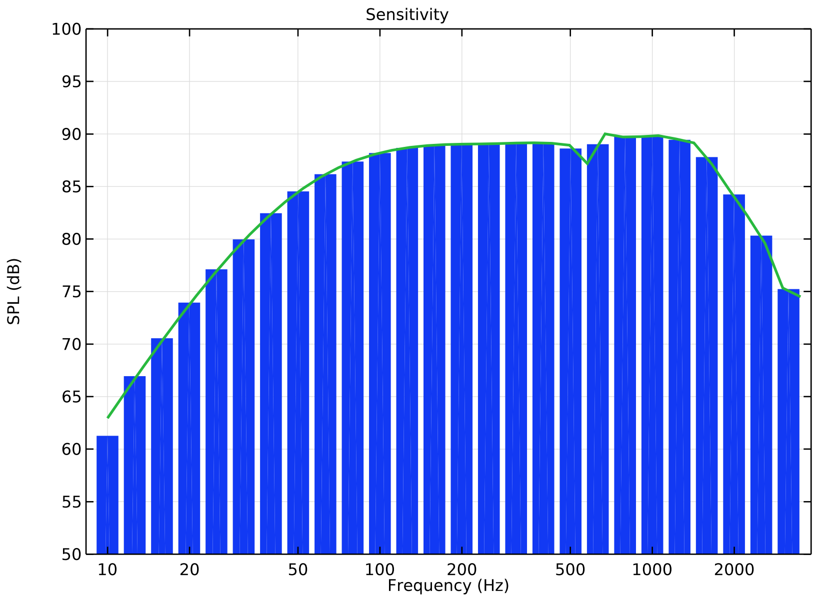

Figure 8: Loudspeaker sensitivity, measured as the on-axis sound pressure level (dB) at a distance of 1 m from the unit. The pressure is evaluated using an input signal of 4 V, or 2.83 V RMS, which corresponds to a power of 1 W at an 8 Ω. nominal impedance. Note the logarithmic frequency scale.

The total electric impedance, defined as $Z = V_0/I$, appears in Figure 9 (absolute, real, and imaginary parts are plotted). The features of this plot are very characteristic of loudspeaker drivers. The peak at approximately 40 Hz coincides with the mechanical resonance; at this frequency the reactive part of the impedance switches sign from inductive to capacitive. In most of the operational range the impedance is largely resistive. Between 100 Hz and 1 kHz it varies only between 6.1 Ω and 8.3 Ω. These are typical values for speakers with a nominal impedance of 8 Ω, as the nominal impedance is usually taken to represent a mean value over the usable frequency range, which for this driver extends between somewhat below 100 Hz and above 1 kHz. At frequencies higher than 1 kHz, the impedance continues to increase as the inductance of the voice coil starts playing a more important part.

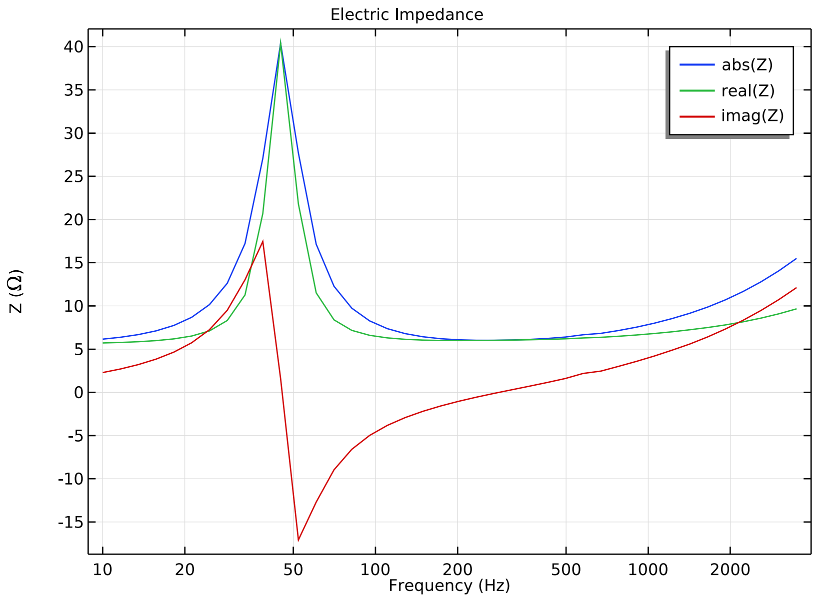

Figure 9: Electric impedance (Ω) of the loudspeaker as a function of frequency (Hz).

The deformation of the speaker cone, surround, spiders, and voice coil is depicted in Figure 10. The figure is evaluated at 3500 Hz and shows, for example, that the motion of the spider is out of phase with the motion of the cone. Typically, this kind of plot can be used to detect and pinpoint where breakups will occur. In the present case only rotationally symmetric breakups can be modeled (the speaker is analyzed in 3D in the Loudspeaker Driver in a Vented Enclosure model). Using an eigenfrequency study can pinpoint at which frequencies the breakup occurs.

Figure 11, finally, shows a directivity plot of the spatial speaker response. This is created using the dedicated Directivity plot available with the Acoustics Module. The plot shows a contour representation of the spatial response (measured on a half sphere in front of the speaker) versus the frequency. Directivity plots help analyze when side lobes occur and how they fall off. Several options, for example for the normalization and for switching the axes, are included. The plot in Figure 11 is normalized with respect to the level at 0^o.

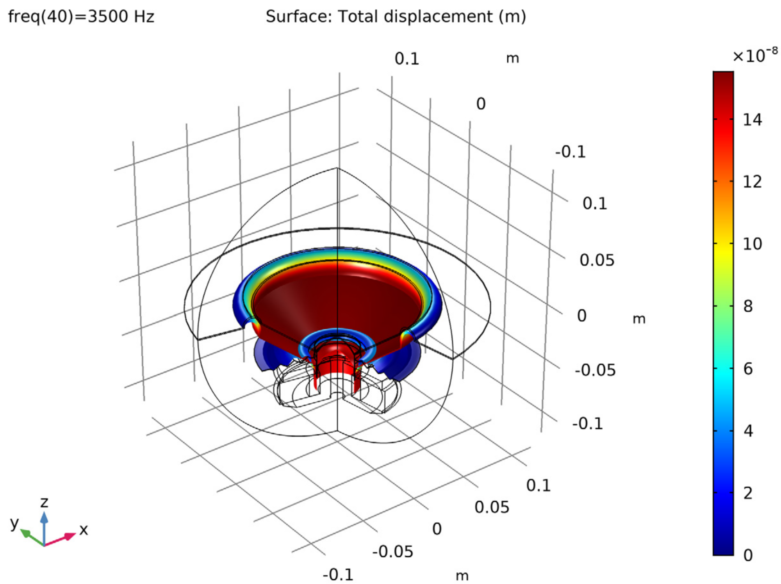

Figure 10: Deformation of the speaker cone, surround, spiders and voice-coil.

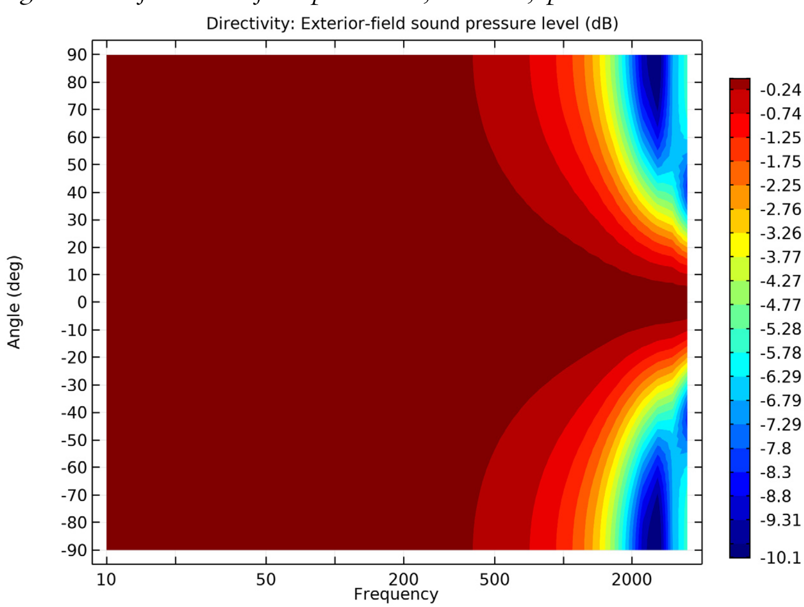

Figure 11: Directivity plot of the spatial speaker response.

Notes About the COMSOL Implementation

The step-by-step instructions take you through the following steps:

- Import the geometry and enter model parameters.
- Apply material settings.
- Set up the physics.
- Create a study computing first the static magnetic field from the permanent magnet, then the time-harmonic fields created by the voice coil over a range of frequencies.
- (Optional) Run the study to extract the force factor, BL, and the blocked coil impedance, Z_b.
- Copy and extend the study to include the acoustic-structure interaction.
- Solve to compute the sound pressure level and the total electric impedance of the driver over the same frequency range.

The iron used in the pole piece and top plate is a nonlinear magnetic material, with interpolation data describing the relationship between the $\boldsymbol{B}$ and $\boldsymbol{H}$ fields. Among other output, the static solution provides the local permeability, as shown in Figure 4.

The studies in this model have a Stationary study step followed by a Frequency Domain, Perturbation step. This automatically makes the stationary solution the linearization point for the subsequent frequency domain solution. This means that the Magnetic Fields interface derives and uses a differential permeability inherited from the one computed by the stationary study. For the frequency domain assumption to be strictly valid, the applied AC voltage must be so small that the resulting current creates a magnetic field which does not significantly alter this permeability. Even though this is not quite the situation here, linearizing around a local biased permeability should still be a better approximation than assuming a constant permeability. The most accurate way to compute the impedance would be in a fully transient analysis, which is outside the scope of this model.

The Acoustic-Structure Interaction multiphysics interface sets up the pressure acoustics and the solid mechanics interfaces together with the Acoustic-Structure Boundary multiphysics coupling. The multiphysics coupling (under the **Multiphysics** node) automatically provides and assigns the boundary conditions for the two-way acoustic-structural coupling between the air and the structures. The acoustic-structure interaction is solved for only in the Frequency Domain, Perturbation step.

The Lorentz Coupling multiphysics feature automatically provides and assigns the domain loads for the two-way electromagnetic-structural coupling in the coil domain.

The coil is driven with a voltage set to `linper(V0)`. The `linper()` operator ensures that the driving voltage `V0` is applied only in the Frequency Domain, Perturbation study step.

In most loudspeaker specifications, the suspension is characterized by a mechanical compliance C_s and resistance R_s. In order to keep the resistance constant over a range of frequencies, the material needs to have a damping factor that increases linearly with the frequency or, equivalently, Rayleigh damping with $\alpha_{\delta M} = 0$ and a constant $\beta_{dK} = \eta_0/\omega_0$, where η_0 is the loss factor measured at the angular frequency ω_0. In this model, the frequency where the loss factor is measured is chosen to be near the lowest mechanical resonance of the driver.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver`

Note: This application also requires the file `Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver_materials` as it contains the material definitions for Materials.

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 **2D Axisymmetric**.
2 In the **Select Physics** tree, select **AC/DC>Electromagnetic Fields>Magnetic Fields (mf)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Solid Interaction, Frequency Domain**.
5 Click **Add**.

6 Click **Study**.

The Model Wizard lets you select the first one of the study steps you plan to use in the model. Select a stationary study used for solving the static magnetic fields.

7 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.

8 Click **Done**.

GEOMETRY 1

When working with your own modeling project of an acoustic driver, you will typically either draw the geometry in COMSOL Multiphysics, or import a CAD file of the driver itself and add the surrounding air and PML domains. Here, the entire geometry is imported as a sequence from the geometry file. The instructions to the geometry can be found in the appendix at the end of this document.

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `loudspeaker_driver_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

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

GLOBAL DEFINITIONS

Parameters 1

Enter the model parameters or load them from the file `loudspeaker_driver_parameters.txt`. Here, as well as in all following sections, the Description field helps you keep track of what you are doing, but is completely optional.

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
N0	100	100	Number of turns in coil
V0	4[V]	4 V	Peak driving voltage
f0	40[Hz]	40 Hz	Frequency at which loss factor is given
omega0	2*pi*f0	251.33 Hz	Angular frequency at which loss factor is given

The loss factor frequency definition will be used when setting up the structural properties.

DEFINITIONS

Create a selection for the coil, this will simplify setting up the physics.

Explicit 1

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Coil` in the **Label** text field.
3. Select Domain 8 only.

Perfectly Matched Layer 1 (pml1)

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

   Use the Perfectly Matched Layers (PMLs) to model a nonreflecting condition-like behavior and avoid unphysical reflections (spurious reflections) where the sound leaves the model. This feature is also set up under **Definitions**. This makes them available for any physics interface that needs them.
2. Select Domains 1 and 5 only.

MATERIALS

While the material properties used in this model are partly made up, they resemble those used in a real driver. The diaphragm and dust cap both consist of a HexaCone®-like material; a light and very stiff composite. The apex has properties representative of glass fiber materials. The spider, acting as a spring, is made of a phenolic cloth with a much lower stiffness. The material used in the coil is taken to be lighter than copper, as the wire is insulated and does not completely fill the coil domain. The surround, finally, is a light resistive foam.

Except for air and soft Iron, the materials you will use all come from a material library created especially for this model (to be loaded from the file `loudspeaker_driver_materials.mph`). You may notice that some of the materials will report missing properties. For example, the composite does not include any electromagnetic properties. This is fine, as you will not model the magnetic fields in the domains where the composite is used.

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.

5 In the tree, select **AC/DC>Soft Iron (With Losses)**.

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

MATERIALS

Air (mat1)

First, add air which will be present everywhere in your geometry. Next, switch to using nonlinear Iron in the pole piece and top plate.

Soft Iron (With Losses) (mat2)

1 In the **Model Builder** window, click **Soft Iron (With Losses) (mat2)**.

2 Select Domains 6 and 11 only.

3 Right-click **Soft Iron (With Losses) (mat2)** and choose **Browse Materials**.

MATERIAL BROWSER

1 In the **Material Browser** window, In the ribbon make sure to select the **Materials** tab and then click the **Browse Materials** icon.

   The **Import Material Library** functionality is activated by clicking the small icon at the bottom right, below the Material Browser tree.

2 Click **Import Material Library**.

3 Browse to the model's Application Libraries folder and double-click the file `loudspeaker_driver_materials.mph`.

4 Click **Done**.

ADD MATERIAL

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

2 In the tree, select **loudspeaker driver materials>Composite**.

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

4 In the tree, select **loudspeaker driver materials>Cloth**.

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

6 In the tree, select **loudspeaker driver materials>Foam**.

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

8 In the tree, select **loudspeaker driver materials>Coil**.

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

10 In the tree, select **loudspeaker driver materials>Glass Fiber**.

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

12 In the tree, select **loudspeaker driver materials>Generic Ferrite**.

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

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

MATERIALS

Composite (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Composite (mat3)**.

2 Select Domains 3 and 10 only.

Cloth (mat4)

1 In the **Model Builder** window, click **Cloth (mat4)**.

2 Select Domain 9 only.

Foam (mat5)

1 In the **Model Builder** window, click **Foam (mat5)**.

2 Select Domain 13 only.

Coil (mat6)

1 In the **Model Builder** window, click **Coil (mat6)**.

2 Select Domain 8 only.

Glass Fiber (mat7)

1 In the **Model Builder** window, click **Glass Fiber (mat7)**.

2 Select Domain 7 only.

Generic Ferrite (mat8)

1 In the **Model Builder** window, click **Generic Ferrite (mat8)**.

2 Select Domain 12 only.

MAGNETIC FIELDS (MF)

The Magnetic Fields equation needs to be solved in and around the magnetic motor. To reduce simulation time, make this physics interface active only where it is needed. You can remove all domains where you expect the magnetic field to be negligible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

2 Select Domains 2, 6–9, 11, and 12 only.

Ampère's Law is per default solved in all domains where the physics interface is active. Add a second instance of it to apply to the permanent magnet, where you need a different constitutive relation.

Ampère's Law 2

1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 12 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Constitutive Relation B-H** section.

4 From the **Magnetization model** list, choose **Remanent flux density**.

5 Specify the **e** vector as

0	r
0	phi
1	z

This setting gives a static remanent flux density equal to 0.4 T in the z direction. This will create a static magnetic field distribution in the model, providing the linearization point for the frequency domain study.

Ampère's Law 3

1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domains 6 and 11 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Constitutive Relation B-H** section.

4 From the **Magnetization model** list, choose **B-H curve**.

The B-H curve is provided by the soft iron material.

Coil 1

1 In the **Physics** toolbar, click **Domains** and choose **Coil**.
2 In the **Settings** window for **Coil**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Coil**.
4 Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multi-turn**.
5 Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type `N0`.
6 In the a_{coil} text field, type `3.5e-8[m^2]`.

With `N0 = 100` turns, the total cross-sectional area covered by the wires will be 3.5e-6 m^2. The area of the coil domain is 6e-6 m^2, making the fill factor approximately 60 %.

7 Locate the **Coil** section. From the **Coil excitation** list, choose **Voltage**.
8 In the V_{coil} text field, type `linper(V0)`.

This is the driving voltage. Because the `linper()` operator is used it will kick in only in the **Frequency Domain, Perturbation** study.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Select the air domains above and under the speaker as well as the PML regions.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.
2 Select Domains 1, 2, 4, and 5 only.

Exterior Field Calculation 1

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

The exterior-field calculation requires a source boundary encompassing all local sound sources, and with a symmetry plane to account for the infinite baffle. After computing the solution, you can evaluate the pressure in any point (r,z) outside the domain by entering `pext(r,z)`.

2 Select Boundary 70 only.
3 In the **Settings** window for **Exterior Field Calculation**, locate the **Exterior Field Calculation** section.
4 From the **Condition in the z = z^0 plane** list, choose **Symmetric/ Infinite sound hard boundary**.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 3, 7–10, and 13 only.

With the above selection, you leave out the magnet, pole piece, and top plate. You will consider these domains as perfectly rigid, by using the default sound hard wall condition on their surfaces.

Add damping to some of the solid material.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 1

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

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

3 Click **Clear Selection**.

4 Select Domains 3, 7, and 10 only.

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

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 **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 9 only.

5 Locate the **Damping Settings** section. In the β_{dK} text field, type `0.14/omega0`.

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 **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 13 only.

5 Locate the **Damping Settings** section. In the β_{dK} text field, type `0.46/omega0`.

The spider and the surround are attached to the case.

Fixed Constraint 1

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

2 Select Boundaries 58 and 62 only.

Now is a good time to inspect **Acoustic-Structure Boundary 1** multiphysics coupling under the **Multiphysics** node. When using a predefined multiphysics interface the coupling is automatically applied to all acoustic-solid boundaries.

Now, the **Lorentz Coupling** multiphysics feature is added to handle Lorentz force on the coil (it represents the product of the time-harmonic current and the static magnetic field in which it is traveling). For details, see Notes About the COMSOL Implementation.

MULTIPHYSICS

Lorentz Coupling 1 (ltzc1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Lorentz Coupling**.

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

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

MESH 1

The mesh used in computing the impedance needs to resolve the induced eddy currents in the pole piece and the top plate. For the results to be accurate, the skin depth needs to be resolved by at least 1, preferably 2 quadratic elements.

With a conductivity of 1.12e7 S/m and a peak relative permeability of 1200, the skin depth in the iron at the maximum frequency of 3.5 kHz does not go below 0.07 mm. In practice, most of the induced currents will run in regions of the pole piece where the biased relative permeability is much less than 1200, which makes the skin depth greater. In this model, it is therefore sufficient to use a mesh size of 0.2 mm along the iron surfaces that are closest to the voice coil.

For the acoustic-structure interaction, the air domain and the thin moving structures also need to be well resolved. The Extra fine setting gives a maximum element size of 6 mm which is smaller than the suggested 6 elements per wavelength (this only requires a 16 mm mesh size). The PML is preferably meshed with mapped elements, use 8 elements for the default polynomial scaling.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

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

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

4 Select Domains 1 and 5 only.

Distribution 1

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

2 Select Boundaries 1 and 5 only.

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

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

Size

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

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

3 From the **Predefined** list, choose **Extra fine**.

Free Triangular 1

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

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

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

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

4 Select Domains 6 and 11 only.

5 Locate the **Element Size** section. Click the **Custom** button.

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

7 In the associated text field, type `1[mm]`.

Boundary Layers 1

1 In the **Model Builder** window, 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 Select Domains 6 and 11 only.

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 Selection** section.

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

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `3`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `0.2[mm]`.

7 Click **Build All**.

STUDY 1

Your Study node already contains the Stationary study that you picked from the Model Wizard. Disable the Pressure Acoustics and Solid Mechanics interfaces.

Add a Frequency-Domain, Perturbation study.

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: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain (acpr)** and **Solid Mechanics (solid)**.

Frequency Domain Perturbation

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain Perturbation**.

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

3 In the **Frequencies** text field, type `10^(range(1,(log10(3500)-1)/39, log10(3500)))`.

This generates 40 exponentially distributed frequencies between 10 Hz and 3.5 kHz.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain (acpr)** and **Solid Mechanics (solid)**.

NOTE: The first solution of this model is only of the electromagnetic part of the problem, with the coil assumed to be fixed in order to extract the BL factor and the blocked coil impedance. If you would like to skip ahead to the solution of the full electroacoustic problem, you can do so by proceeding from here to the instructions starting at the section **Component 1 (comp1)**, further below. Otherwise, continue by computing the solution.

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

RESULTS

First, create a 2D plot to view the magnetic field distribution from the permanent magnet.

2D Plot Group 1

1. In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2. In the **Settings** window for **2D Plot Group**, type `Static Magnetic Field` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

   Each step in a study creates its own data set. The numbering of the data sets begins from the last step. Hence in this model Solution 2 contains the stationary solution and Solution 1 the frequency domain perturbation.

Surface 1

1. Right-click **Static Magnetic Field** 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>Magnetic Fields>Magnetic>mf.normH - Magnetic field norm - A/m**.
3. Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.
4. In the **Static Magnetic Field** toolbar, click **Plot**.

   You are now looking at the magnetic field norm created by the permanent magnet. Note that it has a distinct maximum in the gap where the voice coil is moving. If you zoom in a little, the plot should look like Figure 3.

   Another interesting result is the permeability distribution in the iron. Begin by duplicating the existing plot.

Static Magnetic Field 1

1. In the **Model Builder** window, right-click **Static Magnetic Field** and choose **Duplicate**.
2. In the **Settings** window for **2D Plot Group**, type `Effective Permeability` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Results>Effective Permeability** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `mf.normB/(mu0_const*mf.normH)`.

4 In the **Effective Permeability** toolbar, click **Plot**.

This is the effective relative permeability at the linearization point, as given by the ratio of the flux density and the field multiplied by the permeability of vacuum. The plot should look like Figure 4.

Derived Values

Next, evaluate the BL force factor.

Surface Average 1

1. In the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.
2. In the **Settings** window for **Surface Average**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.
4. Locate the **Selection** section. From the **Selection** list, choose **Coil**.
5. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`-mf.Br*N0*2*pi*r`	`Wb/m`	

6. Locate the **Integration Settings** section. Clear the **Compute volume integral** check box.
7. Click **Evaluate**.

   The BL factor evaluates to 7.53 Wb/m.

   Next, study the induced current density at a few different frequencies.

2D Plot Group 3

1. In the **Results** toolbar, click **2D Plot Group**.
2. In the **Settings** window for **2D Plot Group**, type `Induced Current Density` in the **Label** text field.

Surface 1

1. Right-click **Induced Current Density** 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>Magnetic Fields> Currents and charge>Induced current density (spatial frame) - A/m²>mf.Jiphi - Induced current density, phi component**.

   The **Compute differential** check box, if turned on, wraps the expression in a `lindev()` operator. For more information, search for lindev in the COMSOL Multiphysics Documentation.

3 In the **Induced Current Density** toolbar, click **Plot**.

At 3500 Hz, the induced currents are highly localized to the surfaces of the top plate and pole piece.

Induced Current Density

1 In the **Model Builder** window, click **Induced Current Density**.

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

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

4 In the **Induced Current Density** toolbar, click **Plot**.

At 10 Hz, the induced currents distribute throughout the material. The distribution depends on the vicinity to the voice coil, but also very much on the local effective permeability from the static study. To reproduce Figure 5, try two frequencies in between.

5 From the **Parameter value (freq (Hz))** list, choose **905.68**.

Before proceeding to the full electroacoustic analysis, plot the blocked coil inductance as a function of the frequency.

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 `Blocked Coil Inductance` in the **Label** text field.

Global 1

1 Right-click **Blocked Coil Inductance** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields> Coil parameters>mf.LCoil_1 - Coil inductance - H**.

3 In the **Blocked Coil Inductance** toolbar, click **Plot**.

The result should look like Figure 6.

Having extracted the force factor and the voice coil inductance, you now have most of the electromagnetic data required for a separate model of the acoustic parts of a boxed loudspeaker driver. See the `Loudspeaker Driver in a Vented Enclosure` model also in the Application Library. What remains is the real part of the blocked coil impedance. This is available as `mf.RCoil_1`, the Coil Resistance. If you would like to extract this too, you can set it up in a Global plot just like the inductance, and export the results as described in the following steps.

Plot 1

1 Right-click **Global 1** and choose **Add Plot Data to Export**.

If you want to export the data, you can now enter a file name and click the **Export** button.

COMPONENT 1 (COMP1)

It is now time to compute and evaluate the solution of the entire model, including the acoustic-structure interaction. In order to this, you will set up a new study. Although you could technically reuse the static magnetic fields solution from the first study, it can be handy to include this step in the new study too. This allows you to make changes anywhere in the model, including such that affect the static magnetic fields, and run only the new study again to get correctly updated results.

ADD STUDY

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

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

Select an empty study so that you can copy and modify the study steps from your previous study.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

4 Click **Add Study** in the window toolbar.

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

STUDY 1

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

2 In the **Settings** window for **Study**, type `Magnetic Fields Study` in the **Label** text field.

Step 1: Stationary, Step 2: Frequency Domain Perturbation

1 In the **Model Builder** window, under **Magnetic Fields Study**, Ctrl-click to select **Step 1: Stationary** and **Step 2: Frequency Domain Perturbation**.

2 Right-click and choose **Copy**.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, right-click **Study 2** and choose **Paste Multiple Items**.

2 In the **Settings** window for **Study**, type `Complete Study` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 2: Frequency Domain Perturbation

Make the Frequency Domain, Perturbed study solve for the acoustics and the solid physics too.

1 In the **Model Builder** window, click **Step 2: Frequency Domain Perturbation**.

2 In the **Settings** window for **Frequency Domain Perturbation**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Pressure Acoustics, Frequency Domain (acpr)	√	Physics settings
Solid Mechanics (solid)	√	Physics settings

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

RESULTS

You will now have received two more data sets: Solution 4 containing the same static magnetic fields as Solution 2, and Solution 3 with the frequency domain electromagnetic and acoustic-structure interaction results. In order to get a good overview of the latter, begin by adding a selection of all domains except the PMLs. You will then create a Revolution data set in order to enable a 3D view of the model.

Selection

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

2 Right-click **Complete Study/Solution 3 (sol3)** and choose **Selection**.

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

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

5 From the **Selection** list, choose **All domains**.

6 Select Domains 2–4 and 6–13 only.

Revolution 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

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

3 From the **Dataset** list, choose **Complete Study/Solution 3 (sol3)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type -80.

5 In the **Revolution angle** text field, type `250`.

Create a 3D plot of the instantaneous pressure distribution at zero phase in and around the speaker.

3D Plot Group 5

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

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

Surface 1

1 Right-click **Instantaneous Pressure** and choose **Surface**.

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

5 Select the **Symmetrize color range** check box.

Contour 1

1 In the **Model Builder** window, right-click **Instantaneous Pressure** and choose **Contour**.

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

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

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

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

Next, reproduce Figure 7 with a plot of the local sound pressure level.

3D Plot Group 6

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

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

Surface 1

1 Right-click **Sound Pressure Level** and choose **Surface**.

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

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

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

Create a 1D plot of the sensitivity versus the frequency, as in Figure 8.

1D Plot Group 7

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 Locate the **Data** section. From the **Dataset** list, choose **Complete Study/Solution 3 (sol3)**.

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

5 In the **Title** text area, type `Sensitivity`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Frequency (Hz)`.

8 Select the **y-axis label** check box.

9 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**.

Use the Octave Band Plot to depict the sensitivity. Evaluate the pressure 1 m in front using `pext()` operator. The reference pressure is the default for an SPL evaluation. Plot the sensitivity both as a continuous curve and in 1/3 octave bands.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `pext(0,1)`.

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

6 Locate the **Plot** section. From the **Style** list, choose **1/3 octave bands**.

Octave Band 2

1 Right-click **Octave Band 1** and choose **Duplicate**.

2 In the **Settings** window for **Octave Band**, locate the **Plot** section.

3 From the **Style** list, choose **Continuous**.

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

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

Sensitivity

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

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **y minimum** text field, type `50`.

5 In the **y maximum** text field, type `100`.

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

Finally, plot the total electric impedance versus the frequency.

1D Plot Group 8

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

2 In the **Settings** window for **1D Plot Group**, type `Total Electric Impedance` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Complete Study/Solution 3 (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Electric Impedance`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Frequency (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Z (\Omega)`.

Global 1

1 Right-click **Total Electric Impedance** and choose **Global**.

The automatic computation of the coil impedance uses the entire applied voltage. Because this now includes a contribution from the motion of the coil, you need to define it manually as only the driving voltage V0 divided by the time-harmonic current through the coil. The results are depicted in Figure 9.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(V0/mf.ICoil_1)`		`abs(Z)`
`real(V0/mf.ICoil_1)`		`real(Z)`
`imag(V0/mf.ICoil_1)`		`imag(Z)`

4 In the **Total Electric Impedance** toolbar, click **Plot**.

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

Next create a plot of the deformation of the loudspeaker. This can in general be used to visualize breakups in the cone and surround.

3D Plot Group 9

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

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

Surface 1

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

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

3 In the **Expression** text field, type `solid.disp`.

Deformation 1

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

2 In the **Deformation** toolbar, click **Plot**.

The plot should look like the one in Figure 10.

Finally, visualize the directivity of the loudspeaker using the built in Directivity plot. This is an important plot used to visualize and analyze the spatial response of the speaker as function of the frequency.

First, create a copy of the relevant data set and switch to spatial coordinates. This data set is automatically created when the Generate default plot is not deselected. It is necessary for the Directivity plot and other plots using the exterior-field feature.

Complete Study/Solution 3 (5) (sol3)

1 In the **Model Builder** window, under **Results>Datasets** right-click **Complete Study/ Solution 3 (sol3)** and choose **Duplicate**.

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

3 From the **Frame** list, choose **Spatial (r, phi, z)**.

1D Plot Group 10

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.

Select the data set you just created.

3 Locate the **Data** section. From the **Dataset** list, choose **Complete Study/ Solution 3 (5) (sol3)**.

Directivity 1

1 In the **Directivity Plot** toolbar, click **More Plots** and choose **Directivity**.

Per default the reference direction is along the z-axis. This defines what 0 deg corresponds to. Change the angles to correspond to everything in front of the speaker, that is, from -90 to 90 deg.

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 **Directivity Plot** toolbar, click **Plot**.

The horizontal scale can be represented using a logaritmic scale.

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

The image should look like the one in Figure 11. If you are more familiar with having the frequency on the *y*-axis, you can just change that under the **Coloring and Style** tab and change the **Layout** option.

Appendix: Geometry Sequence Instructions

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D Axisymmetric**.

GEOMETRY 1

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `150[mm]`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	30[mm]

Circle 2 (c2)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `8[mm]`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **r** text field, type `74[mm]`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	1.5[mm]

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 On the object **c2**, select Boundaries 2–4 only.

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `70[mm]`.

4 In the **Height** text field, type `1[mm]`.

5 Locate the **Position** section. In the **r** text field, type `80.5[mm]`.

6 In the **z** text field, type `-1[mm]`.

Difference 1 (dif1)

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

2 Select the object **c1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Activate selection** toggle button.

5 Select the object **r1** only.

Rectangle 2 (r2)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `42[mm]`.

4 In the **Height** text field, type `35[mm]`.

5 Locate the **Position** section. In the **r** text field, type `6[mm]`.

6 In the **z** text field, type `-87[mm]`.

Rectangle 3 (r3)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `35.5[mm]`.

4 In the **Height** text field, type `20[mm]`.

5 Locate the **Position** section. In the **r** text field, type `15.5[mm]`.

6 In the **z** text field, type `-80[mm]`.

Rectangle 4 (r4)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.2[mm]`.

4 In the **Height** text field, type `8[mm]`.

5 Locate the **Position** section. In the **r** text field, type `17.8[mm]`.

6 In the **z** text field, type `-60[mm]`.

Rectangle 5 (r5)

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

2 In the **Settings** window for **Rectangle**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Locate the **Size and Shape** section. In the **Width** text field, type `26[mm]`.

5 In the **Height** text field, type `20[mm]`.

6 Locate the **Position** section. In the **r** text field, type `25[mm]`.

7 In the **z** text field, type `-80[mm]`.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **r** text field, type `48[mm] 36[mm] 36[mm] 48[mm]`.

5 In the **z** text field, type `-82[mm] -87[mm] -87[mm] -87[mm]`.

Difference 2 (dif2)

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

2 In the **Settings** window for **Difference**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Select the object **r2** only.

5 Locate the **Difference** section. Find the **Objects to subtract** subsection. Select the **Activate selection** toggle button.

6 Select the objects **pol1**, **r3**, and **r4** only.

Rectangle 6 (r6)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Selections of Resulting Entities** section.
3 Select the **Resulting objects selection** check box.
4 Locate the **Size and Shape** section. In the **Width** text field, type `0.2[mm]`.
5 In the **Height** text field, type `25[mm]`.
6 Locate the **Position** section. In the **r** text field, type `18.2[mm]`.
7 In the **z** text field, type `-64[mm]`.

Rectangle 7 (r7)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Selections of Resulting Entities** section.
3 Select the **Resulting objects selection** check box.
4 Locate the **Size and Shape** section. In the **Width** text field, type `0.4[mm]`.
5 In the **Height** text field, type `15[mm]`.
6 Locate the **Position** section. In the **r** text field, type `18.4[mm]`.
7 In the **z** text field, type `-63.5[mm]`.

Rectangle 8 (r8)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `4.6[mm]`.
4 In the **Height** text field, type `0.4[mm]`.
5 Locate the **Position** section. In the **r** text field, type `18.4[mm]`.
6 In the **z** text field, type `-44.5[mm]`.

Rectangle 9 (r9)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `7[mm]`.
4 In the **Height** text field, type `0.4[mm]`.
5 Locate the **Position** section. In the **r** text field, type `59[mm]`.
6 In the **z** text field, type `-44.5[mm]`.

Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **r** text field, type `23[mm] 26[mm] 26[mm] 32[mm] 32[mm] 38[mm] 38[mm] 44[mm] 44[mm] 50[mm] 50[mm] 56[mm] 56[mm] 59[mm] 59[mm] 59[mm] 59[mm] 56[mm] 56[mm] 50[mm] 50[mm] 44[mm] 44[mm] 38[mm] 38[mm] 32[mm] 32[mm] 26[mm] 26[mm] 23[mm] 23[mm] 23[mm]`.

5 In the **z** text field, type `-44.1[mm] -42.1[mm] -42.1[mm] -46.1[mm] -46.1[mm] -42.1[mm] -42.1[mm] -46.1[mm] -46.1[mm] -42.1[mm] -42.1[mm] -46.1[mm] -46.1[mm] -44.1[mm] -44.1[mm] -44.5[mm] -44.5[mm] -46.5[mm] -46.5[mm] -42.5[mm] -42.5[mm] -46.5[mm] -46.5[mm] -42.5[mm] -42.5[mm] -46.5[mm] -46.5[mm] -42.5[mm] -42.5[mm] -44.5[mm] -44.5[mm] -44.1`.

Union 1 (uni1)

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

2 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Select the objects **pol2**, **r8**, and **r9** only.

5 Locate the **Union** section. Clear the **Keep interior boundaries** check box.

Polygon 3 (pol3)

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

2 In the **Settings** window for **Polygon**, locate the **Selections of Resulting Entities** section.

3 Find the **Cumulative selection** subsection. Click **New**.

4 In the **New Cumulative Selection** dialog box, type `Composite` in the **Name** text field.

5 Click **OK**.

6 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

7 From the **Data source** list, choose **Vectors**.

8 In the **r** text field, type `18.4[mm] 66[mm] 66[mm] 67.5[mm] 67.5[mm] 18.4[mm] 18.4[mm] 18.4[mm]`.

9 In the **z** text field, type `-39[mm] 0[mm] 0[mm] 0[mm] 0[mm] -40.26[mm] -40.26[mm] -39`.

Circle 3 (c3)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `23.5848[mm]`.

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

Layer name	Thickness (m)
Layer 1	1[mm]

5 Locate the **Position** section. In the **z** text field, type `-54[mm]`.

6 Locate the **Size and Shape** section. In the **Sector angle** text field, type `90`.

Delete Entities 2 (del2)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **c3**, select Boundaries 1 and 2 only.

Rectangle 10 (r10)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.2[mm]`.

4 In the **Height** text field, type `25[mm]`.

5 Locate the **Position** section. In the **r** text field, type `18.2[mm]`.

6 In the **z** text field, type `-64[mm]`.

Union 2 (uni2)

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

2 Select the objects **del2** and **r10** only.

Delete Entities 3 (del3)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **uni2**, select Domains 2–5 only.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Composite**.

6 In the **Geometry** toolbar, click **Build All**.