Model created in COMSOL Multiphysics 6.4

Loudspeaker Driver — Transient Analysis

Introduction

This tutorial presents a full transient analysis of a loudspeaker driver and is an extension of the frequency domain analysis carried out in the Loudspeaker Driver — Frequency-Domain Analysis model. The transient analysis allows for nonlinear characterization of the driver, for example, to determine the total harmonic distortion (THD) or the intermodulation distortion (IMD) of the acoustic signals produced but the system. Both of these quantities are important parts of the distortion measurements of loudspeakers, and this sort of analysis cannot be done in frequency domain simulations which are intrinsically linear. The step-by-step instructions will show you how to set up the transient analysis of the coupled electromagnetic, structural, and acoustic systems of the loudspeaker driver.

The model is set up with the **Magnetic Fields** interface from the AC/DC Module; and the **Pressure Acoustics, Transient**, the **Solid Mechanics**, and the **Thermoviscous Acoustics, Transient** interfaces available with the Acoustics Module. The **Magnetomechanics** multiphysics coupling is used for handling the electromagnetic forces and induced currents over the voice coil. The **Moving Mesh** functionality is used to handle the large displacements in the air domains.

The whole analysis is divided into two parts: a transient analysis of the loudspeaker subjected to a harmonic driving voltage and a frequency spectrum analysis of the acoustic pressure at the listening point. The first part is carried out using two study steps. First a stationary step solves only the electromagnetic part of the problem to evaluate the field of the permanent magnet, with the driver in stand-still. Then the full time dependent study step takes care of all the relevant multiphysics interactions of the moving speaker. The frequency spectrum analysis of the output signal is performed using the combination of the Time Dependent and the Time to Frequency FFT study steps solving an auxiliary algebraic 0D equation which is set up with the Global ODEs and DAEs interface.

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

Model Definition

The loudspeaker is a baffled driver similar to that studied in the Loudspeaker Driver — Frequency-Domain Analysis model. Figure 1 shows its geometry and 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*. A driving AC voltage applied to the voice coil 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, keeps 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 explicitly, but the magnet assembly and outer rims of the spider and surround are considered to be fixed. The absence of the basket means that the considered geometry is rotationally symmetric and can be modeled in the *rz*-plane.

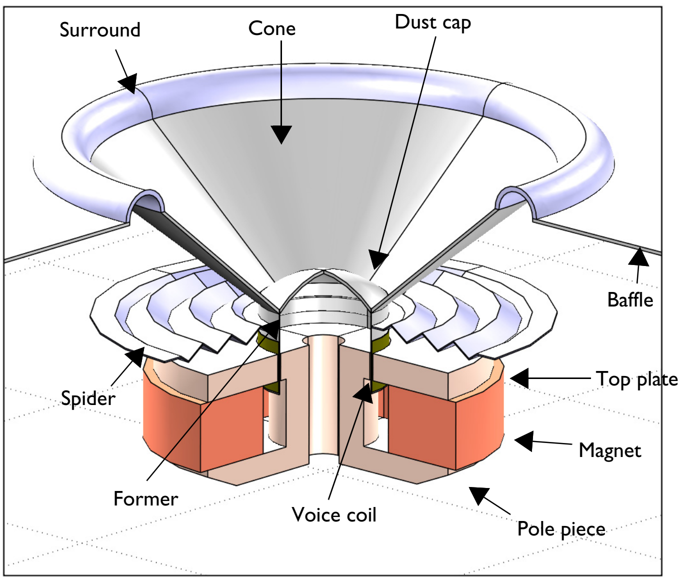


Figure 1: Geometry of the modeled loudspeaker driver.

INPUT SIGNALS FOR A DISTORTION ANALYSIS

Performing the transient analysis of the loudspeaker is possible for any time-dependent voltage signal $V(t)$ applied to the voice coil. The choice of a specific type of input signal depends on the characteristic under investigation. The step-by-step instruction below concerns the total harmonic distortion or THD analysis of the loudspeaker. The input is in this case a harmonic voltage signal

$$V(t) = V_0 \sin(2\pi f_0 t)\,. \tag{1}$$

The input signal frequency and the amplitude are chosen as f_0 = 70 Hz and V_0 = 10 V, respectively.

In the case of an intermodulation distortion or IMD analysis, the input signal is composed of two (or more) harmonic signals with different frequencies, for example,

$$V(t) = V_1 \sin(2\pi f_1 t) + V_2 \sin(2\pi f_2 t)\,.$$

An example of an IMD analysis of the same loudspeaker can be found in the COMSOL Application Gallery at www.comsol.com/model/loudspeaker-driver-transient-analysis-47151.

MULTIPHYSICS SETUP

The relation between the driving voltage and the electromagnetic force on the voice coil as well as the so-called back electromotive force (EMF) are easily set up in COMSOL using built-in functionality. More details are given in the section Electromagnetic Interactions. The force and the EMF generated by the displacement is fully coupled through an acoustic-structure interaction analysis to compute the sound generation.

The structural equations are solved in the moving parts of the driver and the acoustics equation in the surrounding air. The motion of the voice coil and the loudspeaker cone contributes to nonlinear behavior of the system as the topology changes. In the domains next to the moving structures the **Moving Mesh** feature (with **Deforming Domain**) is used to model the large deformations, such as the voice coil moving (partially) in and out of the magnetic gap. The structural equations are formulated in the so-called *material frame* while the acoustic and EM equations in the so-called *spatial frame*. Moving mesh is necessary to capture large displacements for the equations in the spatial frame.

It is important to note that the second order formulation (second order in time) used in the **Pressure Acoustics, Transient** physics is not compatible with **Moving Mesh**. The acoustics in the deforming domains are modeled using the **Thermoviscous Acoustics, Transient** physics, which is based on a first order formulation (first order in time). In most cases the best approach is to use the **Adiabatic** formulation of the equations as well as **Slip (ideal)** on walls. This is physically identical to solving pressure acoustics. As an added benefit, using thermoviscous acoustics makes it possible to easily include the viscous damping in the magnetic air-gap between the pole pieces and the voice coil. This effect is also included in the frequency domain version of the model, however it is here based on the Narrow Region Acoustics feature only available in the frequency domain.

The acoustics equation is automatically excited by the structural vibrations, and feeds back the pressure load onto the structure, using the built in **Thermoviscous Acoustic–Structure Boundary** multiphysics coupling. Pressure and thermoviscous acoustics are coupled using the **Acoustic–Thermoviscous Acoustic Boundary** multiphysics coupling.

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

Real life measurements of nonlinear distortions are performed in the near field of the loudspeaker. Therefore, the PML can be placed close to the driver. Here, the distance from the coordinate system origin to the PML is 0.12 m.

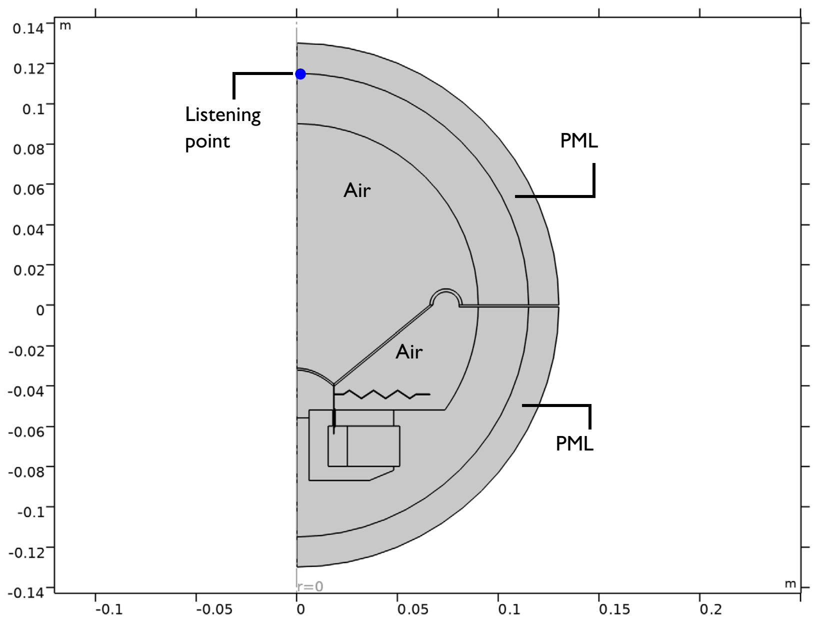


Figure 2: Overview of the model geometry.

ELECTROMAGNETIC INTERACTIONS

This theory section shortly describes the electromagnetic analysis of the current in the voice coil and the driving force that this current gives rise to.

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

$$F_e = -\int_V J_\varphi B_r \mathrm{d}V \qquad (2)$$

with B_r being the r-component of the magnetic flux density, and the integral evaluated over the volume occupied by the coil domain. If you write Equation 2 in terms of the coil current I rather than the cross-sectional current density taking the axial symmetry of the geometry into account, you get

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

where it is assumed that $J_\varphi = IN_0/A$ and is constant over the coil cross-section of area A. The factor Bl in Equation 3 is known in the loudspeaker community as the force factor:

$$\mathrm{Bl} = -\frac{2\pi N_0}{A}\int r B_r \mathrm{dA}$$

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

Results and Discussion

The magnetic field in and around the voice coil gap is depicted in Figure 3. The results correspond to the time steps t = 0.044 s (upper left), t = 0.048 s (upper right), and t = 0.052 s (lower left). The motion of the voice coil (in orange), the former, and the spider (both in pink) is clearly observable. You can also see how the moving mesh adapts the mesh to the changed topology of the system.

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 plots look similar to that in the Loudspeaker Driver — Frequency-Domain Analysis model. The relative permeability remains the same throughout the iron at different time steps. It changes slightly only in the areas close to the voice coil.

.

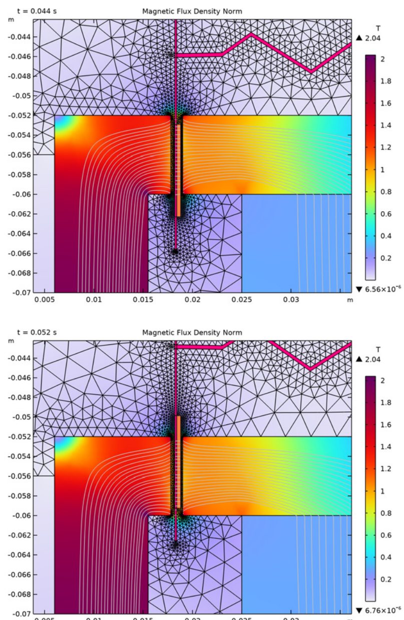


Figure 3: Magnetic field in and around the voice coil gap at three different time steps.

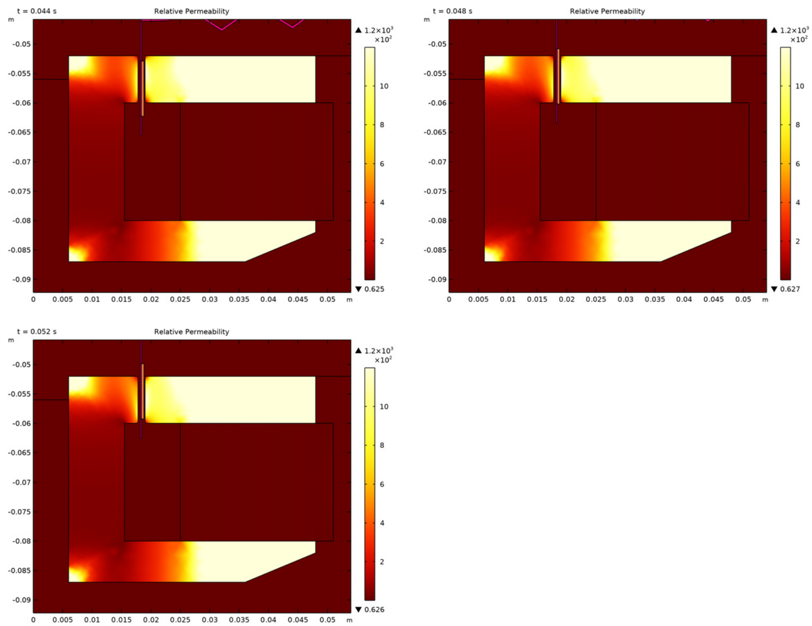


Figure 4: Local relative permeability in the pole piece and top plate at three different time steps.

Figure 5 shows the distribution of the acoustic pressure around the loudspeaker. The voice coil and the loudspeaker cone are at the lowermost position at t = 0.044 s (see Figure 7). At t = 0.048 s and t = 0.052 s, the cone is on its way up and down, respectively.

The acoustic particle velocity in the region of the magnetic air gap (between pole pieces and voice coil) is depicted in Figure 6. In this region a no-slip condition has been applied to the thermoviscous acoustic wall condition. This allows modeling of the damping occur here at resonances.

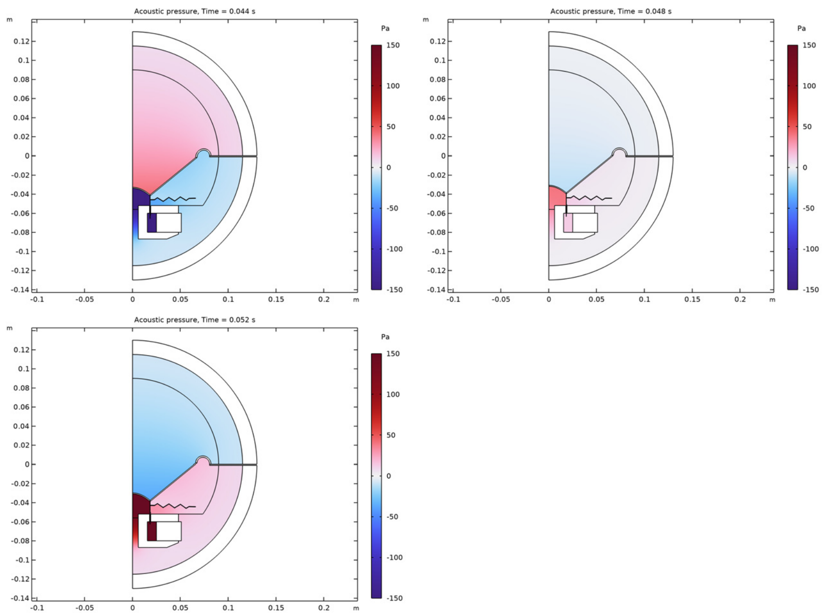


Figure 5: Acoustic pressure at three different time steps.

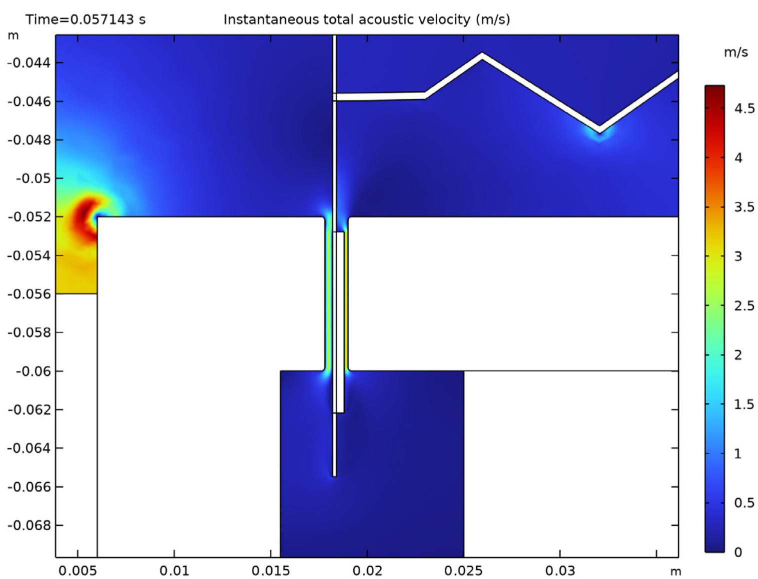


Figure 6: Acoustic particle velocity in the magnetic air gap and coil region.

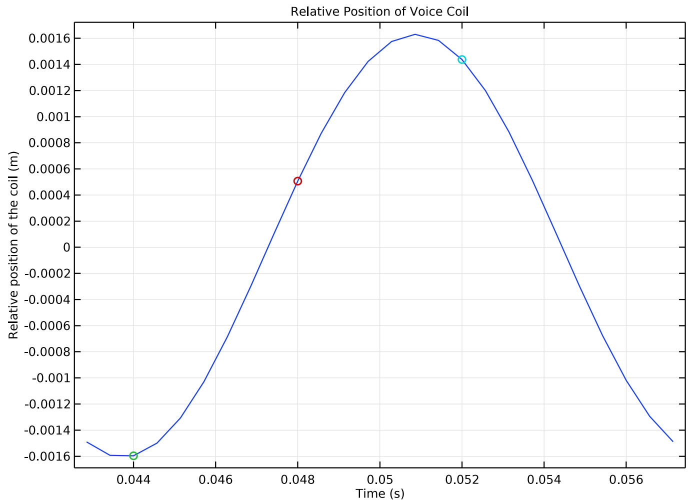


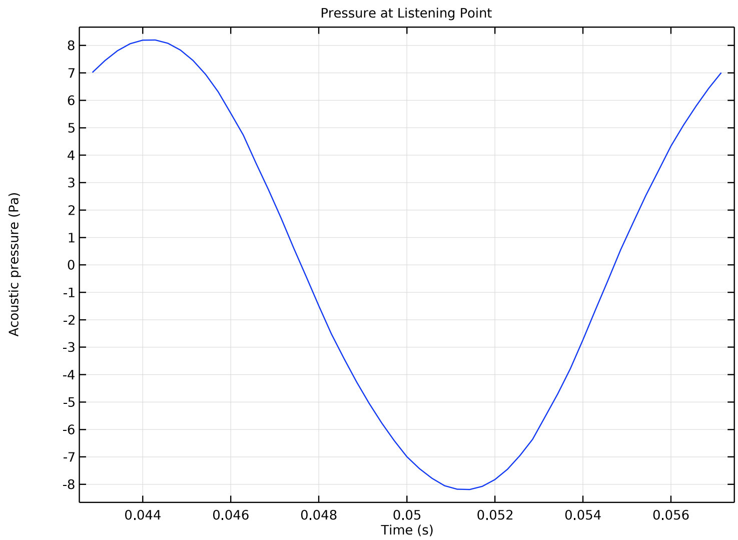
Figure 7: Relative position of the voice coil.

The acoustic pressure at the listening point is depicted in Figure 8. At the top, you can see the pressure signal as a function of time plotted from $t = 3T_0$ to $t = 4T_0$, where T_0 is the period given by the frequency of the input signal. Here, T_0 = 1/70 s. It is assumed that the loudspeaker reaches the steady state by the time $t = 3T_0$. This makes it possible to perform a periodic extension of the pressure signal over time. The extended signal is shown in Figure 8 at the bottom. You can see that the profile slightly differs from a perfect sinusoidal. This means that the output signal also contains frequency components other than f_0, that is, the value of THD calculated as

$$\mathrm{THD} = \frac{\sqrt{H_2^2 + H_3^2 + \ldots + H_N^2}}{H_1},$$

where H_N is the harmonic response of N$^{\mathrm{th}}$ harmonic and H_1 is the fundamental response, is different from 0.

.



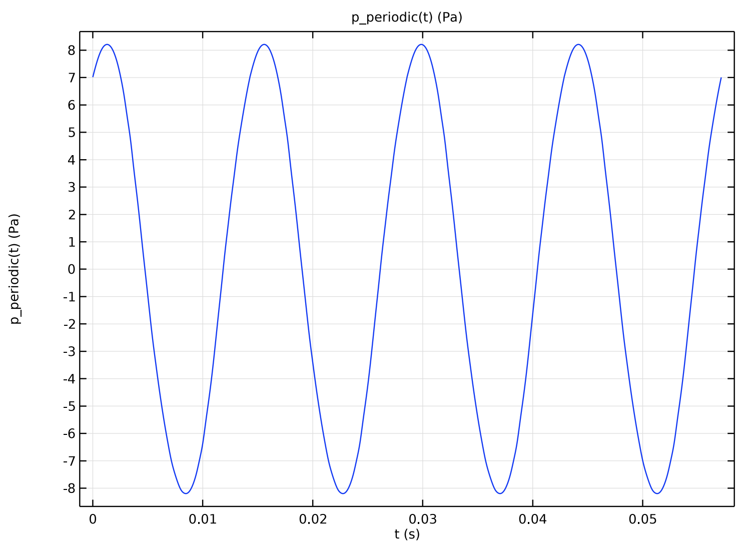


Figure 8: Acoustic pressure at the listening point (top) and its periodic extension (bottom).

Figure 9 shows the frequency spectrum of the acoustic signal at the steady state as the value of SPL over frequency. The highest peaks appear at the frequencies multiple of f_0 (red dots) and yields a $\text{THD} = 2.0\%$ (the value is computed in the Global Evaluation node

Derived Values > THD Evaluation). You can see that the SPL for the odd-order harmonics is higher than that for the even-order harmonics, which means the symmetrical system nonlinearities dominate the asymmetrical ones.

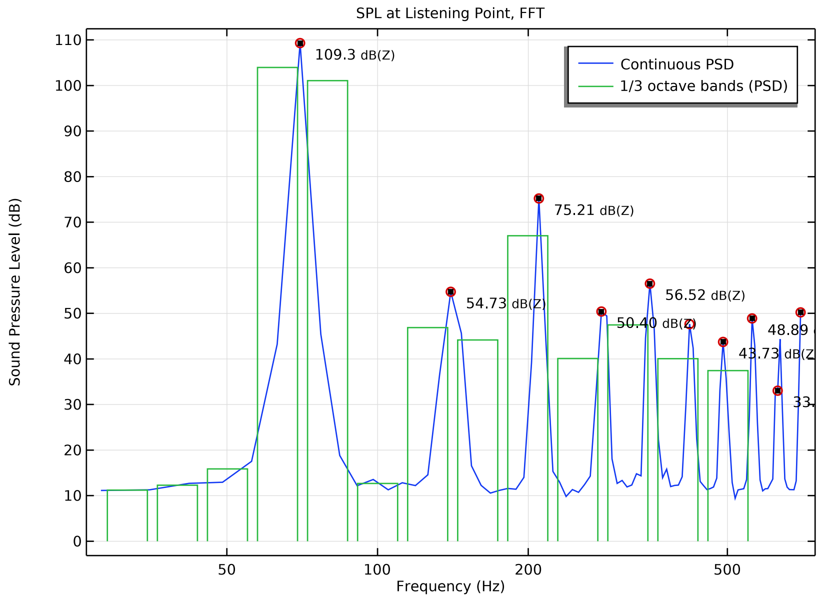


Figure 9: Sound pressure level distribution at the listening point.

The plots of the coil power and the dynamic force factor (BL) are depicted in Figure 10 and Figure 11. You can see that the coil power is negative over certain time intervals. This means that the power flows from the voice coil into the circuit, that is, the voice coil operates in the generator mode. The RMS value of the power is of course positive.

The BL curve has a typical shape for a loudspeaker configuration where the voice coil height is comparable to the magnetic pole piece gap depth. Obtaining an idealized BL factor curve, with a constant value, requires a configuration where the coil is much larger or much smaller than the gap height.

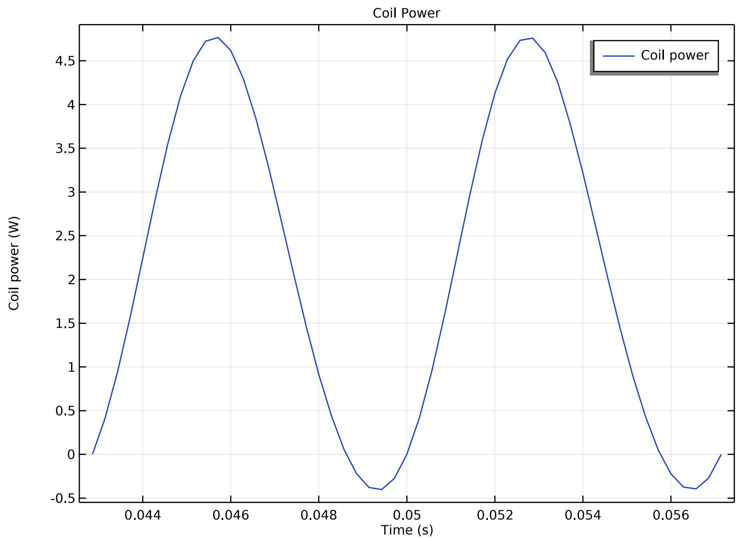


Figure 10: Coil power at the steady state.

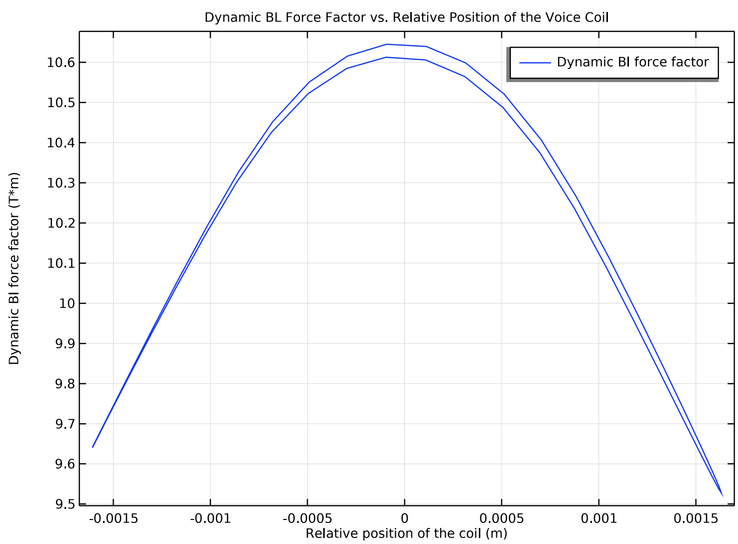


Figure 11: Dynamic Bl force factor versus relative position of the voice coil.

Notes About the COMSOL Implementation

The step-by-step instruction takes you through the following steps:

- Import the geometry and enter model parameters
- Apply material settings
- Set up the physics interfaces
- Set up the extra features: Moving Mesh and PML
- Create a study containing the Stationary and the Time Dependent study steps
- Set up an auxiliary interface and a study to perform a frequency spectrum analysis of a transient signal at a point of interest
- Work with built-in functionality provided in Functions: Interpolation, Analytic Extension

The first study in this model contains a Stationary and a Time Dependent study step. The first computes the stationary magnetic field from the permanent magnet. The magnetic field distribution is then used as the initial value for time $t = 0$ s in the second study step. This step computes the transient acoustic-structure interaction and electromagnetic behavior due to the applied voltage, movement, and geometry change. Both study steps account for nonlinear deformation of the structural parts of the loudspeaker (Include geometric nonlinearities checkbox). The Automatic remeshing option is enabled in the time-dependent study step to avoid using highly distorted meshes, which could lead to numerically ill-posed problems. In this model, a new mesh is created as soon as the distortion of the current mesh elements exceeds a certain level you specify in advance.

Note: It is important that the quality of the initial mesh is good. If the mesh has bad quality elements with sharp wedge-like corners, the remeshing may break down as the quality starts out worse than that specified in the **Condition for Remeshing** section.

The input voltage signal applied to the voice coil is defined by Equation 1. The signal is ramped over the first period, which yields a smooth transition of the voltage from 0 to $V(t)$. The Time Dependent study step solves for the period of time from 0 to $T_{\mathrm{end}} = 4T_0$. The full time interval is split into two subintervals in the study: from 0 to $3T_0$ and from $3T_0$ to T_{end}. The second one is of the most interest since it is assumed that the steady state takes place from the time $3T_0$. Therefore, the time stepping is finer over the second subinterval.

The steady-state output signal — the acoustic pressure at the listening point — is interpolated by a cubic spline and extrapolated from the interval $[3T_0, T_{\mathrm{end}}]$ to any time $t > 0$ through the Periodic Extension. The Global ODEs and DAEs interface and the second study take care of the computation of the frequency spectrum of the output signal. The study here consists of a Time Dependent and a Time to Frequency FFT step. The former picks up the values of the periodically extended signal on a specified time interval; the latter, computes the FFT of the signal. Note that the accuracy of the FFT computation becomes higher for longer time intervals. Here, the time interval is chosen to be equal to $10T_0$. Increasing the time interval from the initial T_0 to $10T_0$ (or even higher) is possible because of the Periodic Extension of the output signal.

The present model runs at a relatively low frequency (including the harmonics) and does not require a resolution of eddy currents (the skin depth) in the pole piece. Thus, compared to the frequency-domain analysis, the boundary layer mesh has been removed.

Reference

1. Brüel & Kjær, "Audio Distortion Measurements," *Application Note BO0385*, 1993.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver_transient`

Note: This model also requires the file `Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver_materials.mph`, which contains the required material definitions.

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** > **Pressure Acoustics** > **Pressure Acoustics, Transient (actd)**.
5. Click **Add**.
6. In the **Select Physics** tree, select **Acoustics** > **Elastic Waves** > **Solid Mechanics (Elastic Waves) (solid)**.
7. Click **Add**.
8. In the **Select Physics** tree, select **Acoustics** > **Thermoviscous Acoustics** > **Thermoviscous Acoustics, Transient (tatd)**.
9. Click **Add**.
10. Click **Study**.

    The **Model Wizard** lets you select the first of the study steps you plan to use in the model. Select a stationary study used to solve for the initial condition; the static (DC) magnetic fields.
11. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces** > **Stationary**.
12. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `loudspeaker_driver_transient_parameters.txt`.

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 are found in the appendix at the end of this document.

The geometry should look like that in Figure 2.

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

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

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

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

Add a **Ramp** function in order to model the transient regime of the input voltage before it reaches the steady state.

GLOBAL DEFINITIONS

Ramp 1 (rm1)

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

2 In the **Settings** window for **Ramp**, locate the **Parameters** section.

3 In the **Location** text field, type `0.1*T0`.

4 In the **Slope** text field, type `1/T0`.

5 Select the **Cutoff** checkbox.

6 Click to expand the **Smoothing** section.

7 Select the **Size of transition zone at start** checkbox. In the associated text field, type `0.2*T0`.

8 Select the **Size of transition zone at cutoff** checkbox. In the associated text field, type `0.2*T0`.

Create selections for the coil and the domains where you will specify the physics and also a selection for the boundary adjacent to the Solid Mechanics domains. Here, the selections are pasted to the text field to simplify the modeling. Normally, they are selected in the geometry window.

DEFINITIONS

Coil

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

2 In the **Settings** window for **Explicit**, type `Coil` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `15` in the **Selection** text field.

5 Click **OK**.

Solid Mechanics

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

2 In the **Settings** window for **Explicit**, type `Solid Mechanics` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `4, 9-17, 20` in the **Selection** text field.

5 Click **OK**.

Magnetic Fields

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

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `2, 3, 8-13, 15, 16, 18, 19` in the **Selection** text field.

5 Click **OK**.

Pressure Acoustic

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

2 In the **Settings** window for **Explicit**, type `Pressure Acoustic` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `1, 2, 6, 7` in the **Selection** text field.

5 Click **OK**.

Thermoviscous Acoustic and Moving Mesh

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

2 In the **Settings** window for **Explicit**, type `Thermoviscous Acoustic and Moving Mesh` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `3, 5` in the **Selection** text field.

5 Click **OK**.

Solid Mechanics Exterior Boundaries

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

2 In the **Settings** window for **Adjacent**, type `Solid Mechanics Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog, select **Solid Mechanics** in the **Input selections** list.

5 Click **OK**.

Define an **Average** operator over the coil domain.

Average 1 (aveop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, locate the **Source Selection** section.
3. From the **Selection** list, choose **Coil**.
4. Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** checkbox.

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 **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in** > **Air**.
4. Click the **Add to Component** button in the window toolbar.

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.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **AC/DC** > **Soft Iron (With Losses)**.

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

MATERIALS

Soft Iron (With Losses) (mat2)

1 Select Domains 8 and 18 only.

In the ribbon, on the **Materials** tab click **Browse Materials**.

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

MATERIAL BROWSER

1 In the **Material Browser** window, click **Import Material Library**.

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

3 Click **Done**.

ADD MATERIAL

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

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

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

MATERIALS

Composite (mat3)

Select Domains 4 and 17 only.

ADD MATERIAL

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

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

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

MATERIALS

Cloth (mat4)

Select Domain 16 only.

ADD MATERIAL

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

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

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

MATERIALS

Foam (mat5)
Select Domain 20 only.

ADD MATERIAL

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

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

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

MATERIALS

Coil (mat6)
Select Domain 15 only.

ADD MATERIAL

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

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

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

MATERIALS

Glass Fiber (mat7)
Select Domains 9–14 only.

ADD MATERIAL

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

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

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

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

MATERIALS

Generic Ferrite (mat8)
Select Domain 19 only.

Now it is time to set up the physics interfaces. Specify the selection where the Magnetic Fields equation needs to be solved, that is the magnetic motor domain and its surroundings. In the other domains, the magnetic field is assumed to be negligible.

MAGNETIC FIELDS (MF)

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

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

3 From the **Selection** list, choose **Magnetic Fields**.

Add an instance of the **Ampère's Law in Solids** domain feature in all Magnetic Fields domains, where the material is different from air.

Ampère's Law in Solids - Generic Ferrite

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

2 In the **Settings** window for **Ampère's Law in Solids**, type `Ampère's Law in Solids - Generic Ferrite` in the **Label** text field.

3 Select Domain 19 only.

4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.

5 Specify the **e** vector as

0	r
0	phi
1	z

Ampère's Law in Solids - Soft Iron

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

2 In the **Settings** window for **Ampère's Law in Solids**, type `Ampère's Law in Solids - Soft Iron` in the **Label** text field.

3 Select Domains 8 and 18 only.

4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **B-H curve**.

The BH curve is provided by the soft iron material.

Ampère's Law in Solids - Nonconductive Solids

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

2 In the **Settings** window for **Ampère's Law in Solids**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Solid Mechanics**.

4 In the **Label** text field, type `Ampère's Law in Solids - Nonconductive Solids`.

Domain Coil 1

1 In the **Physics** toolbar, click **Domains** and choose **Domain Coil**.

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

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

4 Locate the **Material Type** section. From the **Material type** list, choose **Solid**.

5 Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multiturn**.

6 From the **Coil excitation** list, choose **Voltage**.

7 In the V_{coil} text field, type `V0*sin(2*pi*f0*t)*rm1(t)`.

The harmonic voltage is multiplied by the previously defined ramp function to add the smooth build-up time to the signal.

8 Locate the **Homogenized Conductor** section. In the N text field, type `N0`.

9 From the list, choose **User defined**.

10 Find the **High-frequency effective loss** subsection. Clear the **Include harmonic loss** checkbox.

11 In the a text field, type `2.4e-8[m^2]`.

The area of the coil domain is $4{\cdot}10^{-6}$ m^2. The number of turns `N0 = 100` makes the total cross-sectional area covered by the wires equal to $2.4{\cdot}10^{-6}$ m^2, which will give the fill factor of 60%.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.

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

3 From the **Selection** list, choose **Pressure Acoustic**.

Modify the **Transient Solver Settings** according to the frequency of the input signal. This setting will adjust the time-dependent solver settings.

4 Locate the **Transient Solver Settings** section. In the **Maximum frequency to resolve** field enter `Nhar*f0`. It will give the maximal time step for the Transient Solver.

SOLID MECHANICS (SOLID)

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

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

3 From the **Selection** list, choose **Solid Mechanics**.

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 materials.

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 **Damping Settings** section.

3 In the β_{dK} text field, type `0.14/omega_d`.

4 Locate the **Domain Selection** section. Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `16` in the **Selection** text field.

7 Click **OK**.

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 **Damping Settings** section.

3 In the β_{dK} text field, type `0.46/omega_d`.

4 Locate the **Domain Selection** section. Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `20` in the **Selection** text field.

7 Click **OK**.

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 **Damping Settings** section.

3 From the **Damping type** list, choose **Viscous damping**.

4 In the η_b text field, type `eta_s_gf*K_gf/omega0`.

5 In the η_v text field, type `eta_s_gf*G_gf/omega0`.

6 Locate the **Domain Selection** section. Click **Clear Selection**.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog, type `9-14` in the **Selection** text field.

9 Click **OK**.

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 **Damping Settings** section.

3 From the **Damping type** list, choose **Viscous damping**.

4 In the η_b text field, type `eta_s_com*K_com/omega0`.

5 In the η_v text field, type `eta_s_com*G_com/omega0`.

6 Locate the **Domain Selection** section. Click **Clear Selection**.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog, type `4 17` in the **Selection** text field.

9 Click **OK**.

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 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `72 76` in the **Selection** text field.

5 Click **OK**.

Now proceed and set up the transient Thermoviscous Acoustics physics. It will be active in the Moving Mesh domain. For this large domain the Adiabatic formulation is ideal.

THERMOVISCOUS ACOUSTICS, TRANSIENT (TATD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Transient (tatd)**.

2 In the **Settings** window for **Thermoviscous Acoustics, Transient**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Thermoviscous Acoustic and Moving Mesh**.

4 Locate the **Transient Solver and Mesh Settings** section. In the $f_{\max}$ text field, type `Nhar*f0`.

5 Locate the **Thermoviscous Acoustics Equation Settings** section. Select the **Adiabatic formulation** checkbox.

Wall 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Thermoviscous Acoustics, Transient (tatd)** click **Wall 1**.

2 In the **Settings** window for **Wall**, locate the **Mechanical** section.

3 From the **Mechanical condition** list, choose **Slip (perfect)**.

Wall 2

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

2 Select Boundaries 16, 43, and 89–92 only.

Now, look into the multiphysics coupling under the **Multiphysics** node.

Now, the **Magnetomechanics** multiphysics feature is added to handle Lorentz force on the coil.

MULTIPHYSICS

Magnetomechanics, Solid 1 (mmcpl1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain** > **Magnetomechanics, Solid**.

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

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

4 Locate the **Lorentz Coupling** section. Select the **Only use Lorentz force** checkbox.

Acoustic–Thermoviscous Acoustic Boundary 1 (atb1)

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

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

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

Thermoviscous Acoustic–Structure Boundary 1 (tsb1)

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

2 In the **Settings** window for **Thermoviscous Acoustic–Structure Boundary**, locate the **Boundary Selection** section.

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

First select all the boundaries (giving all applicable), then go to **Manual** and de-select the two boundaries on the Coil (19 and 42). Here we want to model the viscous losses and will add a No Slip.

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

5 Locate the **Mechanical** section. From the **Mechanical condition** list, choose **Slip (perfect)**.

Thermoviscous Acoustic–Structure Boundary 2 (tsb2)

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

2 Select Boundaries 19 and 42 only.

Add a **Perfectly Matched Layer** to truncate the computational domain without introducing spurious reflections of the acoustic waves from the outer boundary.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 1 and 7 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 In the **PML scaling curvature parameter** text field, type 3.

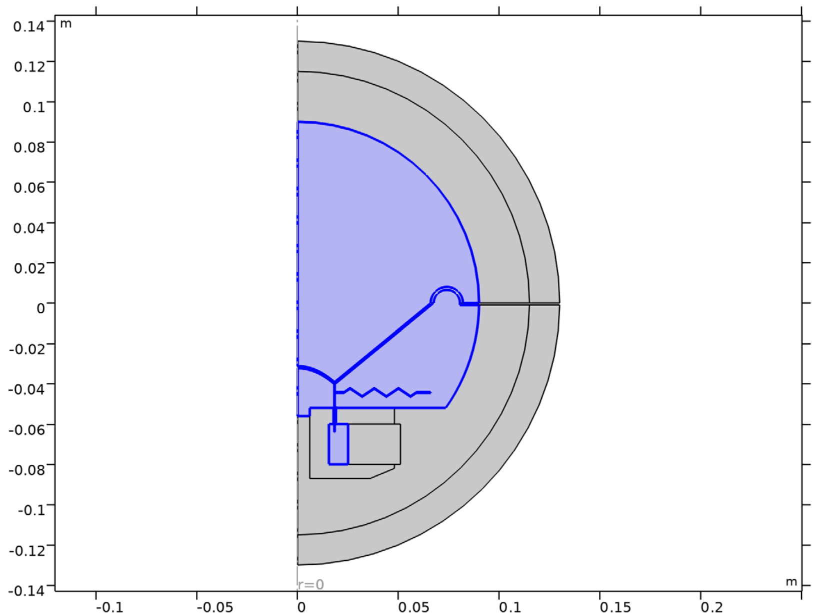
The **Solid Mechanics** domains will move and deform due to the motion of the voice coil caused by the Lorenz force. To account for the deformed configuration of the system during the calculation, add the **Moving Mesh** feature and specify the **Deforming Domain** above and below the moving parts. The movement of the mesh is automatically taken from the **Solid Mechanics** interface. The **Moving Mesh** is not compatible with the second order formulation of **Pressure Acoustics, Transient**. **Thermoviscous Acoustics, Transient** is used on the moving domains, this also allows to include the viscous damping in the magnetic air gap. On the axis-of-symmetry add the **Symmetry/Roller** conditions, as the mesh needs to slide where it is in contact with the moving dust cap. Not adding this condition will force the moving mesh to be fixed on the axis.

Deforming Domain 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Free Deformation**.

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

3 From the **Selection** list, choose **Thermoviscous Acoustic and Moving Mesh**.



4 Locate the **Smoothing** section. From the **Mesh smoothing type** list, choose **Laplace**.

Symmetry/Roller 1

1 In the **Moving Mesh** toolbar, click **Symmetry/Roller**.

2 Select Boundaries 3 and 6 only.

In the model `Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver`, it was important to have a finer mesh along the iron surfaces next to the voice coil. The mesh refinement took the skin depth into account, which resolved the eddy currents in the pole and the top plate at higher frequencies.

Here, the frequency of the driving voltage is `f0 = 70` Hz. This gives the skin depth of approximately 0.5 mm, which is comparable to the width of the voice coil. Therefore, the mesh refinement it is not necessary in this case.

For the acoustic-structure interaction, the air domain and the thin moving structures also need to be well resolved. In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details, see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide*. The **Extra fine** setting gives a maximum element size of 6 mm which is by orders of magnitude smaller than 6 elements per wavelength (80 cm here) slowing down the analysis without adding any relevant information. That is why the model uses a user defined mesh with a very fine mesh around the voice coil and a coarser mesh in the rest of the model. For the structural components, the model uses a **Mapped** mesh with 2 elements through the thickness. The PML is also preferably meshed with mapped elements; use 8 elements for the default polynomial scaling.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `15[mm]`.
5 In the **Minimum element size** text field, type `0.10[mm]`.
6 In the **Curvature factor** text field, type `0.25`.
7 Click **Build Selected**.

Mapped 1

1 In the **Mesh** toolbar, click **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 4, 9–17, and 20 only.
5 Click to expand the **Reduce Element Skewness** section. Select the **Adjust edge mesh** checkbox.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 20, 32, 35, and 39 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `2`.

Size 1

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. Click **Clear Selection**.
4. Select Domains 10 and 15 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** checkbox. In the associated text field, type `0.15[mm]`.

Size 2

1. Right-click **Mapped 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. Click **Clear Selection**.
4. Select Domains 13 and 16 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** checkbox. In the associated text field, type `0.7[mm]`.

Size 3

1. Right-click **Mapped 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. Click **Clear Selection**.
4. Select Domains 4, 17, and 20 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** checkbox. In the associated text field, type `1.0[mm]`.

Distribution 2

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundary 21 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. From the **Distribution type** list, choose **Predefined**.
5. In the **Number of elements** text field, type `20`.
6. In the **Element ratio** text field, type `3`.

Distribution 3

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

2 Select Boundary 17 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `10`.

6 In the **Element ratio** text field, type `3`.

7 Select the **Reverse direction** checkbox.

Free Triangular 1

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

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

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

4 Select Domains 2, 3, 5, 6, 8, 18, and 19 only.

Size 1

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

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

3 Click **Clear Selection**.

4 Select Domains 8, 18, and 19 only.

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

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `3[mm]`.

8 Select the **Minimum element size** checkbox. In the associated text field, type `0.5[mm]`.

Size 2

1 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 **Boundary**.

4 Select Boundary 12 only.

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

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `1[mm]`.

8 Click **Build All**.

Mapped 2

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Reduce Element Skewness** section.

3 Select the **Adjust edge mesh** checkbox.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 80 and 81 only.

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

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

5 Click **Build All**.

It is time to set up the study that will include the interaction between all physics interfaces present in the model. In the **Stationary** study step, clear all of the physics interfaces except for the Magnetic Fields.

STUDY 1

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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics (solid)** and **Moving Mesh**.

Add a Time Dependent study step which will account for the acoustics-structure interaction and the moving mesh.

Step 2: Time Dependent

1 In the **Study** toolbar, click **Time Dependent**.

Assume that the transient process reaches the steady state by the time `3*T0`. Split the time interval into two intervals: form `0` to `3*T0` and from `3*T0` to `T_end`. In the first one, use a coarser time stepping for the solution storage; in the second one, use a finer time stepping.

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

3 In the **Output times** text field, type `{range(0, T0/5, 14*T0/5) range(3*T0, T0/50, T_end)}`.

4 Click to expand the **Study Extensions** section. Select the **Automatic remeshing** checkbox.

The **Automatic Remeshing** option assures that the distortion of the mesh elements will not exceed a certain threshold. As soon as the threshold is reached, the domain will be remeshed.

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)** > **Dependent Variables 2** node, then click **Spatial Mesh Displacement (comp1.spatial.disp)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e-4`.

Change the Time Stepping Method to **BDF**, because it is more robust for detecting events necessary when Automatic Remeshing is used in the model.

6 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.

7 In the **Settings** window for **Time-Dependent Solver**, locate the **General** section.

8 From the **Times to store** list, choose **Output times by interpolation**.

9 Click to expand the **Time Stepping** section. From the **Method** list, choose **BDF**.

10 From the **Steps taken by solver** list, choose **Manual**.

11 In the **Initial step fraction** text field, type `0.5`.

12 In the **Initial step growth rate** text field, type `1.4`.

13 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** and choose **Fully Coupled**.

14 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

15 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

16 From the **Nonlinear method** list, choose **Automatic (Newton)**.

17 In the **Maximum number of iterations** text field, type `10`.

18 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** click **Direct**.

19 In the **Settings** window for **Direct**, locate the **General** section.

20 From the **Solver** list, choose **PARDISO**.

21 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** click **Automatic Remeshing**.

22 In the **Settings** window for **Automatic Remeshing**, locate the **Condition for Remeshing** section.

23 From the **Condition type** list, choose **Distortion**.

24 In the **Stop when distortion exceeds** text field, type `2.5`.

25 Locate the **Remesh** section. Clear the **Store solution when new meshes are created** checkbox.

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

27 In the **Settings** window for **Study**, type `Study 1 - Time Dependent Analysis` in the **Label** text field.

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

RESULTS

Plot the magnetic field in the voice coil air gap. It is clearly seen that the voice coil has moved down from its initial position. Use the **Zoom Box** to zoom in on the magnetic gap.

Magnetic Flux Density (mf)

1 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Magnetic Flux Density Norm`.

4 In the **Parameter indicator** text field, type `t = eval(t) s`.

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

6 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

7 In the **Magnetic Flux Density (mf)** toolbar, click **Plot**.

8 In the **Model Builder** window, expand the **Magnetic Flux Density (mf)** node.

Contour 1, Streamline 1

1 In the **Model Builder** window, under **Results** > **Magnetic Flux Density (mf)**, Ctrl-click to select **Streamline 1** and **Contour 1**.

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

Streamline 2

1 In the **Model Builder** window, right-click **Magnetic Flux Density (mf)** and choose **Streamline**.

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

3 Click **Paste Selection**.

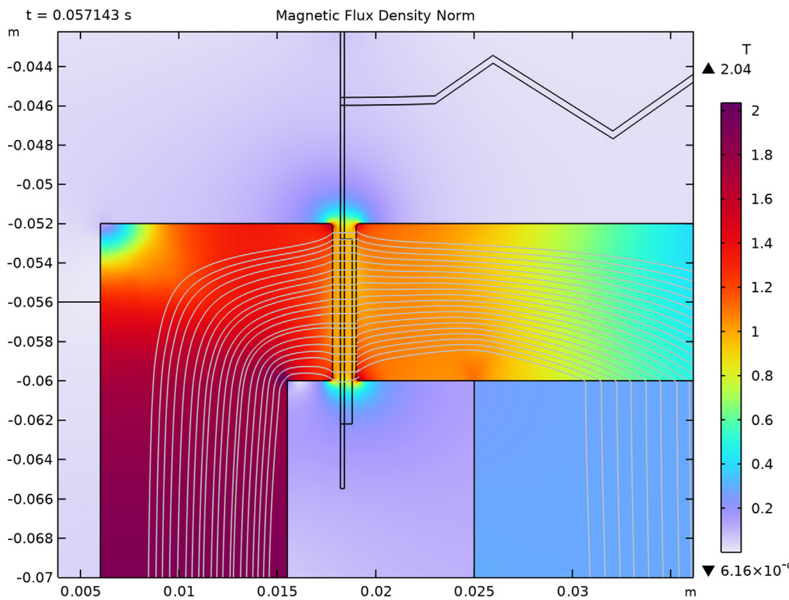
4 In the **Paste Selection** dialog, type 43 in the **Selection** text field.

5 Click **OK**.

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

7 Find the **Point style** subsection. From the **Color** list, choose **Gray**.

8 In the **Magnetic Flux Density (mf)** toolbar, click **Plot**.



Acoustic Pressure (actd)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (actd)**.

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Acoustic pressure, Time = eval(t) s`.

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

6 From the **Number format** list, choose **Automatic**.

7 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (r, phi, z)**.

Surface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (actd)** node, then click **Surface 1**.

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 (comp1) > Multiphysics > atb1.p_t - Total acoustic pressure - Pa**.

3 Click to expand the **Range** section. Select the **Manual color range** checkbox.

4 In the **Minimum** text field, type `-150`.

5 In the **Maximum** text field, type `150`.

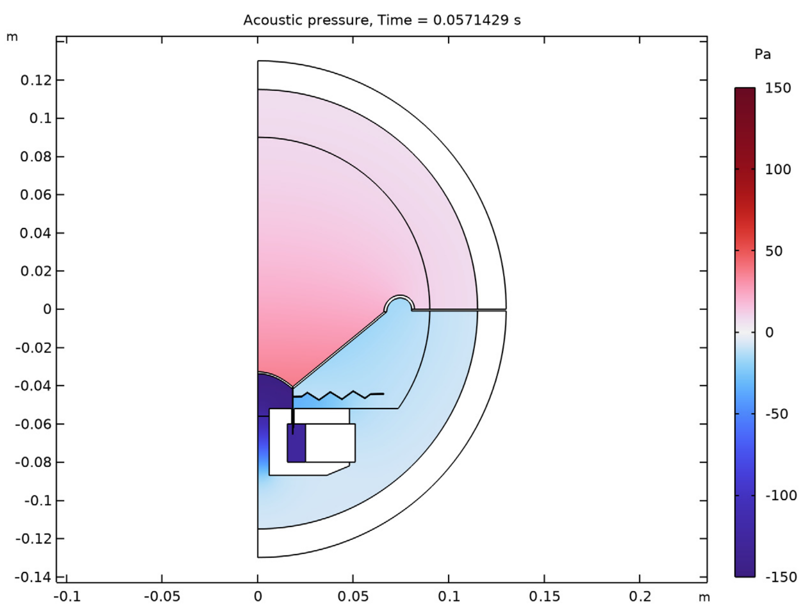
Selection 1

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

2 Select Domains 2–6 and 8–20 only.

Select all domains except the PML region where the solution is unphysical. Simply select all domains (you can use Ctrl+A) and then deselect the two PML domains (domains 1 and 6).

3 In the **Acoustic Pressure (actd)** toolbar, click **Plot**.



Plot the acoustic pressure at the listening point over the time interval `[3*T0, T_end]`.

Acoustic Velocity (tatd)

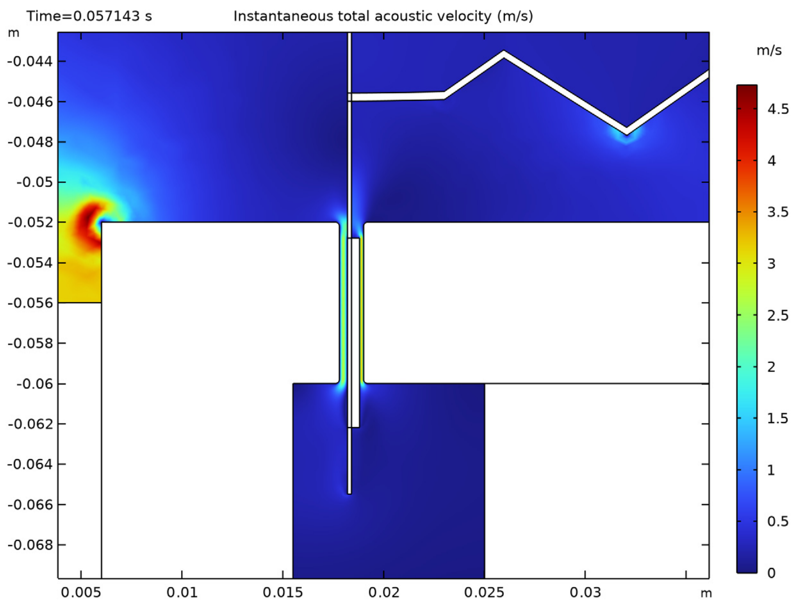
1 In the **Model Builder** window, under **Results** click **Acoustic Velocity (tatd)**.

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

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

4 In the **Acoustic Velocity (tatd)** toolbar, click **Plot**.

Look at the plot of the acoustic velocity, notice the no-slip behavior in the magnetic air gap.



Pressure at Listening Point

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

2 In the **Settings** window for **1D Plot Group**, type `Pressure at Listening Point` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Time Dependent Analysis/Remeshed Solution 1 (sol3)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(3*T0, T0/50, T_end)`.

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

Point Graph 1

1 Right-click **Pressure at Listening Point** and choose **Point Graph**.

2 Select Point 7 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `p`.

The pressure at the listening point should look like the one in Figure 8 at the top.

5 In the **Pressure at Listening Point** toolbar, click **Plot**.

It is seen that the shape of the signal slightly differs from a perfectly sinusoidal one. That is, its THD is different from 0. The instruction below will help you to do the frequency spectrum analysis of the signal and calculate its THD.

Pressure at Point

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Pressure at Point` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Time Dependent Analysis/Remeshed Solution 1 (sol3)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(3*T0, T0/50, T_end)`.

6 Select Point 7 only.

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

Expression	Unit	Description
p	Pa	Pressure

8 Click next to **Evaluate**, then choose **New Table**.

Pressure at Point

1 In the **Model Builder** window, expand the **Results** > **Tables** node, then click **Table 1**.

2 In the **Settings** window for **Table**, type `Pressure at Point` in the **Label** text field.

Make an Interpolation and a Periodic Extension of the pressure at the listening point.

GLOBAL DEFINITIONS

Interpolation 1 (int1)

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **Result table**.

4 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 1 and column number 1.

5 In the **Unit** text field, type `s`.

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

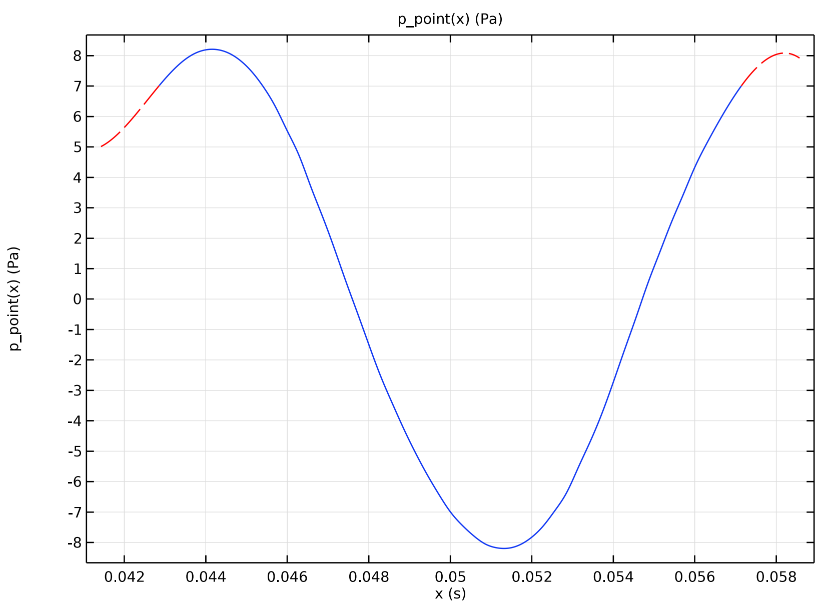
7 In the **Name** text field, type `p_point`.

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

9 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Cubic spline**.

10 From the **Extrapolation** list, choose **Nearest function**.

11 Click **Plot**.



Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `p_point(t)`.

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

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

6 In the **Lower limit** text field, type `3*T0`.

7 In the **Upper limit** text field, type `4*T0`.

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

Argument	Unit
t	s

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

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	t	0	4*T0	0	s

The Periodic Extension will reproduce the steady state of the signal as shown in Figure 8 at the bottom.

11 Click **Plot**.

Now, set up the components necessary to analyze the frequency spectrum and the THD of the acoustic pressure in steady state.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **0D**.

ADD PHYSICS

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

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

3 In the tree, select **Mathematics** > **ODE and DAE Interfaces** > **Global ODEs and DAEs (ge)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Study 1 - Time Dependent Analysis**.

5 Click the **Add to Component 2** button in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

GLOBAL ODES AND DAES (GE)

Global Equations 1 (ODE1)

1 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

2 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
P	P - p_periodic(t)	0	0	

3 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

4 In the **Physical Quantity** dialog, type `id:pressure` in the text field.

5 In the tree, select **General > Pressure (Pa)**.

6 Click **OK**.

7 In the **Settings** window for **Global Equations**, locate the **Units** section.

8 Click **Select Source Term Quantity**.

9 In the **Physical Quantity** dialog, type `id:pressure` in the text field.

10 In the tree, select **General > Pressure (Pa)**.

11 Click **OK**.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies > Time Dependent**.

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

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

STUDY 2

Step 1: Time Dependent

The Interpolation and the Periodic Extension performed earlier make it possible to increase the time interval and refine the time stepping for a more accurate frequency spectrum calculation.

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0, T0/200, 10*T0)`.

3 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

Step 2: Time to Frequency FFT

1 In the **Study** toolbar, click **More Study Steps** and choose **Frequency Domain > Time to Frequency FFT**.

2 In the **Settings** window for **Time to Frequency FFT**, locate the **Study Settings** section.

3 In the **End time** text field, type `10*T0`.

4 In the **Maximum output frequency** text field, type `10*f0`.

5 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, type `Study 2 - Periodic Signal Extraction and FFT` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Generate the default solver and then modify the **Time Stepping** to get minimal numerical damping. Manual time-stepping is used with a time step that resolves the wave nature of the problem.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Method** list, choose **Generalized alpha**.

5 From the **Steps taken by solver** list, choose **Manual**.

6 In the **Time step** text field, type `1/(6*f0)/60`.

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

Plot the sound pressure level at the listening point as a function of frequency to reproduce the result shown in Figure 9.

RESULTS

SPL at Listening Point, FFT

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

2 In the **Settings** window for **1D Plot Group**, type `SPL at Listening Point, FFT` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Periodic Signal Extraction and FFT/Solution 4 (4) (sol4)**.

4 From the **Parameter selection (freq)** list, choose **From list** and select all but the first four frequency parameters.

5 Locate the **Title** section. From the **Title type** list, choose **Label**.

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** checkbox. In the associated text field, type `Frequency (Hz)`.

8 Select the **y-axis label** checkbox. In the associated text field, type `Sound Pressure Level (dB)`.

9 Locate the **Axis** section. Select the **x-axis log scale** checkbox.

Octave Band 1

1 Right-click **SPL at Listening Point, FFT** and choose **More Plots** > **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 `comp2.P`.

5 Locate the **Plot** section. From the **Quantity** list, choose **Continuous power spectral density**.

6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

7 Select the **Show legends** checkbox.

8 In the table, enter the following settings:

Legends
`Continuous PSD`

SPL at Listening Point, FFT

In the **Model Builder** window, click **SPL at Listening Point, FFT**.

Octave Band 2

1 In the **SPL at Listening Point, FFT** 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 `comp2.P`.

5 Locate the **Plot** section. From the **Quantity** list, choose **Band average power spectral density**.

6 From the **Band type** list, choose **1/3 octave**.

7 Click to expand the **Coloring and Style** section. From the **Type** list, choose **Outline**.

8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

9 Select the **Show legends** checkbox.

10 In the table, enter the following settings:

Legends
`1/3 octave bands (PSD)`

Octave Band 3

1 In the **Model Builder** window, under **Results** > **SPL at Listening Point, FFT** right-click **Octave Band 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2 - Periodic Signal Extraction and FFT/ Solution 4 (4) (sol4)**.

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

5 In the **Parameter indices (1-101)** text field, type `range(11, 10, 101)`.

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

7 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

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

Graph Marker 1

1 Right-click **Octave Band 3** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Display** section.

3 From the **Scope** list, choose **Local**.

4 Locate the **Text Format** section. In the **Precision** text field, type `4`.

5 Select the **Include unit** checkbox.

6 In the **SPL at Listening Point, FFT** toolbar, click **Plot**.

For the THD calculation, pick up the pressure at the frequencies that are multiples of `f0`.

THD Evaluation

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `THD Evaluation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Periodic Signal Extraction and FFT/Solution 4 (4) (sol4)**.

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

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

Expression	Unit	Description
`sqrt(sum(with(11 + 10*k, abs(comp2.P)^2), k, 1, 9))/with(11, abs(comp2.P))`	1	THD

6 Click **Evaluate**.

Next, calculate the coil power and the dynamic BL force factor.

Coil Power

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

2 In the **Settings** window for **1D Plot Group**, type `Coil Power` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Time Dependent Analysis/Remeshed Solution 1 (sol3)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(3*T0, T0/50, T_end)`.

6 Locate the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1 Right-click **Coil Power** 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 (comp1)** > **Magnetic Fields** > **Coil parameters** > **mf.PCoil_1 - Coil power - W**.

The coil power plot should look like Figure 10.

3 In the **Coil Power** toolbar, click **Plot**.

This expression corresponds to the z-coordinate of the voice coil center relative to its position at the time $t = 0$.

Dynamic BL Force Factor

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

2 In the **Settings** window for **1D Plot Group**, type `Dynamic BL Force Factor` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Time Dependent Analysis/Remeshed Solution 1 (sol3)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(3*T0, T0/50, T_end)`.

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

7 In the **Title** text area, type `Dynamic BL Force Factor vs. Relative Position of the Voice Coil`.

Global 1

1 Right-click **Dynamic BL Force Factor** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`aveop1(-mf.Br*N0*2*pi*r)`	`T*m`	`Dynamic Bl force factor`

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `aveop1(z - Z)`.

6 Select the **Description** checkbox. In the associated text field, type `Relative position of the coil`.

The curve of the dynamic BL force factor is depicted in Figure 11.

7 In the **Dynamic BL Force Factor** toolbar, click **Plot**.

Appendix: Geometry Sequence Instructions

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

NEW

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

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 `130[mm]`.

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

Layer name	Thickness (m)
Layer 1	15[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 On the object **c2**, select Boundaries 2–4 only.

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

4 Select the **Resulting objects selection** checkbox.

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 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

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 **Size and Shape** section.

3 In the **Width** text field, type `26[mm]`.

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

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

6 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 Select the object **r2** only.

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

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

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

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

Rectangle 6 (r6)

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]`.

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

Layer name	Thickness (m)
Layer 1	1.26[mm]
Layer 2	3.84[mm]
Layer 3	0.4[mm]

8 Clear the **Layers on bottom** checkbox.

9 Select the **Layers on top** checkbox.

Rectangle 7 (r7)

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 `0.6[mm]`.

4 In the **Height** text field, type `9.4[mm]`.

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

6 In the **z** text field, type `-60.7[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 Select the objects **pol2**, **r8**, and **r9** only.

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

4 Select the **Resulting objects selection** checkbox.

5 Locate the **Union** section. Clear the **Keep interior boundaries** checkbox.

Polygon 3 (pol3)

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 `18.4[mm] 66[mm] 66[mm] 67.5[mm] 67.5[mm] 18.4[mm] 18.4[mm] 18.4[mm]`.

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

Quadratic Bézier 1 (qb1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `-18.2[mm]`.

4 In row **3**, set **r** to `18.2[mm]`.

5 In row **1**, set **z** to `-39[mm]`.

6 In row **2**, set **z** to `-23.5[mm]`.

7 In row **3**, set **z** to `-39[mm]`.

8 Locate the **Weights** section. In the **2** text field, type `1`.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

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

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

7 Locate the **Starting Point** section. In the **z** text field, type `-39[mm]`.

8 Locate the **Endpoint** section. In the **z** text field, type `-40.26[mm]`.

Quadratic Bézier 2 (qb2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `18.2[mm]`.

4 In row **3**, set **r** to `-18.2[mm]`.

5 In row **1**, set **z** to `-40.26[mm]`.

6 In row **2**, set **z** to `-24.26[mm]`.

7 In row **3**, set **z** to `-40.26[mm]`.

8 Locate the **Weights** section. In the **2** text field, type `1`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

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

6 Locate the **Endpoint** section. In the **r** text field, type `-18.2[mm]`.

7 Locate the **Starting Point** section. In the **z** text field, type `-40.26[mm]`.

8 Locate the **Endpoint** section. In the **z** text field, type `-39[mm]`.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the objects **ls1**, **ls2**, **qb1**, and **qb2** only.

Line Segment 3 (ls3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **z** text field, type `-52[mm]`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **r** text field, type `sqrt((115[mm])^2 - (52[mm])^2)`.

7 In the **z** text field, type `-52[mm]`.

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

Proceed by adding a domain where to apply a Moving Mesh and a Thermoviscous Acoustics, Transient interface.

Circle 3 (c3)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `90[mm]`.

Union 2 (uni2)

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

2 Select the objects **c3** and **ls3** only.

3 In the **Settings** window for **Union**, click **Build Selected**.

Delete Entities 2 (del2)

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

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

3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

Union 3 (uni3)

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

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

3 From the **Input objects** list, choose **All objects**.

4 Click **Build Selected**.

Delete Entities 3 (del3)

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

2 On the object **uni3**, select Boundaries 4, 13, 19, 33, and 45 only.

3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **del3**, select Points 14, 15, 35, and 36 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.2[mm]`.

5 Click **Build Selected**.

Line Segment 4 (ls4)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **r** text field, type `0[cm]`.

5 In the **z** text field, type `-5.6[cm]`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **r** text field, type `0.6[cm]`.

8 In the **z** text field, type `-5.6[cm]`.

9 Click **Build Selected**.

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

Ignore Vertices 1 (igv1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Vertices**.

2 On the object **fin**, select Points 20, 28, and 35 only.

3 In the **Settings** window for **Ignore Vertices**, click **Build Selected**.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **igv1**, select Boundary 98 only.

3 In the **Settings** window for **Ignore Edges**, click **Build Selected**.