Model created in COMSOL Multiphysics 6.4

Lumped Loudspeaker Driver

Introduction

This is a model of a moving-coil loudspeaker where a lumped parameter analogy represents the behavior of the electrical and mechanical speaker components. This lumped model is coupled to a 2D axisymmetric pressure acoustics model describing the surrounding air domain. The coupling is performed using the built-in Interior Lumped Speaker Boundary feature. The model is solved with two different configurations, first with an open back-volume and then with a closed back-volume. The main part of the analysis focuses on the open back-volume configuration.

Electrical circuit representations of transducers are well known and widely used. In the loudspeaker industry such models have been employed for a long time and with great success. The parameters that characterize the low-frequency performance of a loudspeaker are commonly known as the Thiele–Small or the small-signal parameters. It is common to use these parameters to design and simulate a desired speaker performance in terms of, for example, the on-axis speaker response, the electric impedance, and the diaphragm velocity.

In the low-frequency regime, the motion of the speaker cone can be approximated by the motion of a rigid piston in an infinite baffle. Analytical expressions exist for the sound field radiated from a piston. In the model at hand, the simple piston geometry is replaced by a more realistic speaker cone shape, and the sound field is solved using the finite element method. Of course, this is still an approximation as the motion of the speaker cone is still assumed to be rigid; however, the spatial response of the speaker is more realistic. The methodology of lumping certain parts of a complex system is a general and powerful approach that can be applied to other systems. A first extension of the current model can be to only lump the electrical components and model the mechanical components fully.

The output from the model includes, among many things, the total electric impedance, the on-axis sound pressure level at a nominal driving voltage, and the mean speaker cone velocity. The results are compared with an analytical solution based on the flat piston approximation. Finally, it is also shown how to create a so-called directivity plot of the speaker.

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

Model Definition

A schematic representation of a moving coil loudspeaker is given in Figure 1. The figure shows a cross section of a loudspeaker. The speaker driver is placed in an infinite baffle with free space in front and on the back of the speaker. This will be referred to as the open back-volume configuration. In the closed back-volume configuration, a closed box is located behind the driver (see Figure 3). The speaker cone consists of the outer suspension, the diaphragm, and a dust cap (not marked in the figure). The mechanical and electrical components of the speaker that are lumped are visualized inside the dotted box. On the electrical side it includes the voice-coil and magnetic system (permanent magnet and pole pieces), and on the mechanical side it includes the moving mass of the voice coil and speaker cone, the spring effect of the spider and outer suspension, as well as possible losses due to damping in these suspensions. In the discussion below, the lumped model of the driver itself is not concerned with the open or closed configuration of the back-volume. The effect of where the driver is located or how it is mounted in a system is modeled in the finite element domain.

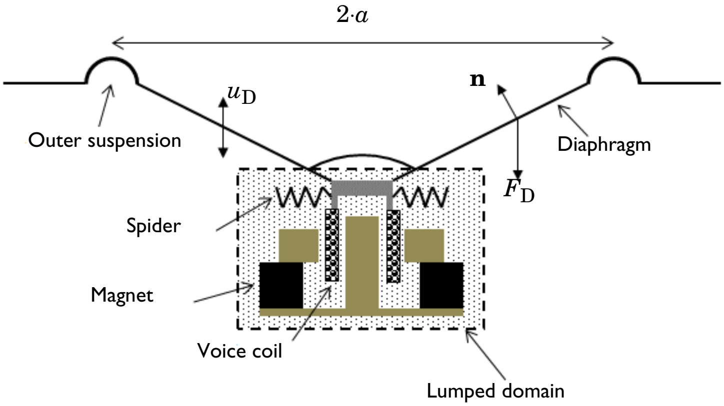


Figure 1: Schematic representation of a moving coil speaker unit.

ELECTROACOUSTIC ANALOGY

In an electroacoustic analogy, mechanical and acoustic physical properties such as force, velocity, pressure, and volume velocity are equated to voltages and currents in an analogous electrical circuit. In this model, the impedance analogy is used for the mechanical components only. This means that the current in the circuit represents the velocity of a moving part (SI unit: m/s) and the voltage represents a force (SI unit: N). In contrast, the acoustics is not represented by a circuit in this model. In general, however, in an acoustic analogous circuit, a voltage represents pressure (SI unit: Pa) and a current represents volume velocity (SI unit: m^3/s).

The passive elements in a circuit — that is, resistors, inductors, and capacitors — represent different processes, namely resistance, mass movement, and compliance, respectively. The components and their analogs are listed in Table 1 (see also Ref. 1 for further details).

TABLE 1: PASSIVE ELEMENTS IN MECHANICAL AND ACOUSTIC ANALOGIES.

ELECTRICAL COMPONENT	MECHANICAL (IMPEDANCE ANALOGY)	ACOUSTIC
• Resistor • SI unit: Ω	• Mechanical resistance • SI unit: N·s/m • Losses due to friction, as in a car suspension.	• Acoustic resistance • SI unit: kg/(m^4·s) • Dissipative losses due to viscosity and thermal conduction in the fluid.
• Inductor • SI unit: H	• Mass • SI unit: kg • Inertial force, acceleration of a mass.	• Acoustic mass • SI unit: kg/m^4 • Internal force of a volume of air that is accelerated but not compressed.
• Capacitor • SI unit: F	• Compliance • SI unit: m/N • The inverse of the spring constant.	• Acoustic compliance • SI unit: m^5/N • Compressibility effect of a volume of air that is not accelerated.

The active components in a circuit represent sources; they are external forces (pressures) or applied velocities (volume velocities). The sources are also used for couplings between the electrical, mechanical, and acoustic domains.

Note: To make the units fit in COMSOL, it is necessary to make unit transforms to fit the electrical units. When, for example, inserting a capacitor representing a mechanical compliance, C_{MS}, in the physics interface type: `C_MS[F*N/m]`

THE ANALOGOUS CIRCUIT

The analogous circuit for the electrical and mechanical parts of the system sketched in Figure 1 is shown in Figure 2. The upper figure represents the voice coil electrical system and the lower figure the mechanical analogue of the speaker cone, suspensions, and mass of the voice coil. In both figures, the node numbers are also shown — they are very useful when setting up the circuit model in COMSOL.

In Figure 2 (top) the external voltage source is denoted by V_0 and the generator output resistance is R_g, in this model $R_g = 0\ \Omega$. The voice coil resistance is R_E, and the voice coil inductance is $L_E(\omega)$, which is frequency dependent. The losses in the magnetic circuit are modeled through the frequency dependent resistance $R'_E(\omega)$. The current controlled voltage source $BL{\cdot}u_D$, represents the back induced electromagnetic voltage generated when the voice coil (of length L) moves with velocity u_D in the magnetic field B. Here BL is the product of the magnetic field strength and the voice coil length L (see also Ref. 2 on how this can be modeled). In the electrical circuit, the current is denoted i_c. The current example uses one model topology to describe the electric impedance of the coil. It is a version of the so-called Leach model (see Ref. 1). Other models like a simple LR, the LR-2, the LR-3, or the so-called Wright model can be set up by modifying the electrical circuit.

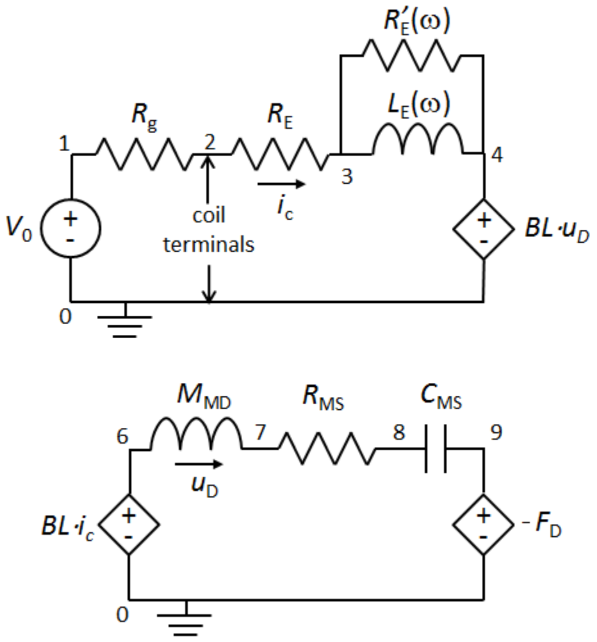


Figure 2: Analogous circuits for the electrical (top) and mechanical (bottom) properties of the speaker driver.

In the mechanical system given in Figure 2 (bottom), the current in the circuit is the velocity of the voice coil and speaker cone in the axial direction (see Figure 1). The force acting on the diaphragm is given by $BL{\cdot}i_\mathrm{c}$. This is the Lorentz force on a voice coil of length L with current i_c, where B is the magnetic flux density. The force acts on a system of mass M_MD (voice coil and diaphragm assembly). The resistance R_MS models the damping and C_MS the mechanical compliance in the speaker suspensions (both spider and outer suspension). Finally, the voltage source $-F_\mathrm{D}$ represents the pressure force acting on the speaker diaphragm (in the axial direction). Notice the minus sign that indicates that the force acts against the movement of the diaphragm (see Ref. 1). The force is given by

$$F_D = \int(\Delta p \cdot n_z)dA \tag{1}$$

where Δp is the pressure drop across the diaphragm and n_z is the axial component of the surface normal $\mathbf{n}$ (see Figure 1). This expression gives the couplings from the acoustic finite element model to the lumped circuit model. On the other hand, the coupling from the circuit model to the finite element model comes from specifying the velocity $\mathbf{v}$ on the surface of the diaphragm, which is given by

$$\mathbf{v} = u_\mathrm{D}\mathbf{e}_\mathrm{z} \tag{2}$$

where $\mathbf{e}_\mathrm{z}$ is the unit vector in the axial direction (see Figure 3) and u_D is the current in the mechanical circuit analog. The coupling introduced through Equation 1 and Equation 2 is automatically performed when using the built-in **Interior Lumped Speaker Boundary** feature (when modeling the air on both sides) or the **Lumped Speaker Boundary** features. The first is used when setting up the model. Dedicated postprocessing variables can be found in the plot menu for the diaphragm velocity, axial pressure, radiated power and more.

Note: A version of this model using the *Lumped Mechanical System* interface also exists as a tutorial: Lumped Loudspeaker Driver Using a Lumped Mechanical System. In that model, the mechanical part of the system uses the mobility analogy where the present model is based entirely on the impedance analogy. The model additionally requires the Multibody Dynamics Module.

SMALL-SIGNAL PARAMETERS

The fundamental small-signal parameters of the system (or Thiele–Small parameters) are the physical parameters of the loudspeaker driver. They are all constants, as they are given in the low-frequency limit of the model. See Table 2 below.

TABLE 2: FUNDAMENTAL SMALL-SIGNAL PARAMETERS.

SYMBOL	VALUE	DESCRIPTION
M_{MD}	33.4 g	Moving mass (voice coil and diaphragm)
C_{MS}	$1.18\cdot 10^{-3}$ m/N	Suspension compliance
R_{MS}	1.85 Ns/m	Suspension mechanical losses (damping)
L_{E}	6.89 mH	Voice coil inductance
R_{E}	7 Ω	DC resistance of voice coil
BL	11.4 T·m	Force factor
S_{D}	$a^2\pi$	Driver equivalent area
a	12 cm	Piston radius of driver (equivalent)

In Table 2 the radius a is the piston radius of the driver. It is typically taken as half the diaphragm aperture diameter, measured half-way into the outer suspension.

Note: The constants used in this model are all taken from Example 22 in Ref. 1.

Other small-signal parameters may be determined on basis of the fundamental small-signal parameters. Actually, these parameters are the ones measured when characterizing a driver; the fundamental parameters are inferred hereafter. The measured small-signal parameters are given in Table 3. In the table, the speed of sound is denoted c_0 and the density of air is ρ_0. The values of these parameters are determined in the model using the Parameters feature under Global Definitions.

TABLE 3: SMALL-SIGNAL PARAMETERS (MEASURED).

Symbol	Expression	Description
M_{MS}	$M_{\mathrm{MD}} + 2S_{\mathrm{D}}^2\dfrac{8\rho_0}{3\pi^2 a}$	Moving mass including acoustic load (low-frequency approximation).
F_{s}	$\dfrac{1}{2\pi\sqrt{C_{\mathrm{MS}}M_{\mathrm{MS}}}}$	Fundamental resonant frequency.

TABLE 3: SMALL-SIGNAL PARAMETERS (MEASURED).

Symbol	**Expression**	**Description**
Q_{ES}	$\dfrac{2\pi F_s M_{\mathrm{MS}} R_{\mathrm{E}}}{(BL)^2}$	Electrical Q factor at F_s
Q_{MS}	$\dfrac{2\pi F_s M_{\mathrm{MS}}}{R_{\mathrm{MS}}}$	Mechanical Q factor at F_s
Q_{TS}	$\dfrac{Q_{\mathrm{MS}} Q_{\mathrm{ES}}}{Q_{\mathrm{MS}} + Q_{\mathrm{ES}}}$	Total Q factor at F_s
V_{AS}	$\rho_0 c_0^2 S_{\mathrm{D}}^2 C_{\mathrm{MS}}$	Equivalent volume compliance (air volume having the same compliance as the suspension).
η_0	$\dfrac{4\pi^2 F_s^3 V_{\mathrm{AS}}}{c_0^3 Q_{\mathrm{ES}}}$	Reference efficiency of the driver.

OTHER PARAMETERS

The model also uses other parameters than the small-signal parameters presented above. Two of them are the frequency dependent voice coil inductance $L_{\mathrm{E}}(\omega)$ and the resistance $R'_{\mathrm{E}}(\omega)$ associated with the losses in the magnetic system. A model for this is given in Ref. 1, for the higher audio frequencies, defining

$$L_{\mathrm{E}}(\omega) = \left[\frac{L_{\mathrm{E}}}{\sin\left(n_{\mathrm{e}}\frac{\pi}{2}\right)}\right]\omega^{(n_{\mathrm{e}}-1)} \qquad R'_{\mathrm{E}}(\omega) = \left[\frac{L_{\mathrm{E}}}{\cos\left(n_{\mathrm{e}}\frac{\pi}{2}\right)}\right]\omega^{n_{\mathrm{e}}}$$

where n_{e} is the so-called voice coil loss factor. For $n_{\mathrm{e}} = 1$ and for in the low-frequency limit the loss-less behavior is recovered where R'_{E} is an open circuit and L_{E} is constant. In this model $n_{\mathrm{e}} = 0.7$.

Another expression used in the model is the acoustic radiated power which is given by

$$P_{\mathrm{AR}} = \int(\mathbf{n}\cdot\mathbf{I})dA \tag{3}$$

where $\mathbf{I}$ is the intensity vector. A variable is predefined by the lumped speaker feature and called `acpr.ilsb1.P_front` (front side of the speaker) and `acpr.ilsb1.P_front` (back side of the speaker). The integral is taken over the front and back side of the speaker diaphragm, respectively. The electric input power (RMS value) is defined as

$$P_{\mathrm{E}} = 0.5\mathrm{Re}(V_0 \cdot i_{\mathrm{c}}^{*}) \tag{4}$$

where * is the complex conjugate operator, and, finally, the speaker efficiency comparing the input electric power to the radiated acoustic power is

$$\eta = \frac{P_{\mathrm{AR}}}{P_{\mathrm{E}}}$$

A number of the analytical results derived in Ref. 1, which are based on the piston approximation, are also used for comparison with the current more realistic speaker cone shape. They are the low-frequency and high-frequency approximations to the speaker velocity, the on-axis pressure at 1 m in front of the piston, and the acoustic radiated power from a piston in an infinite baffle. Expressions for these are given in the reference and under the variable node **Component 1** > **Definitions** > **Analytical approximations** in the model.

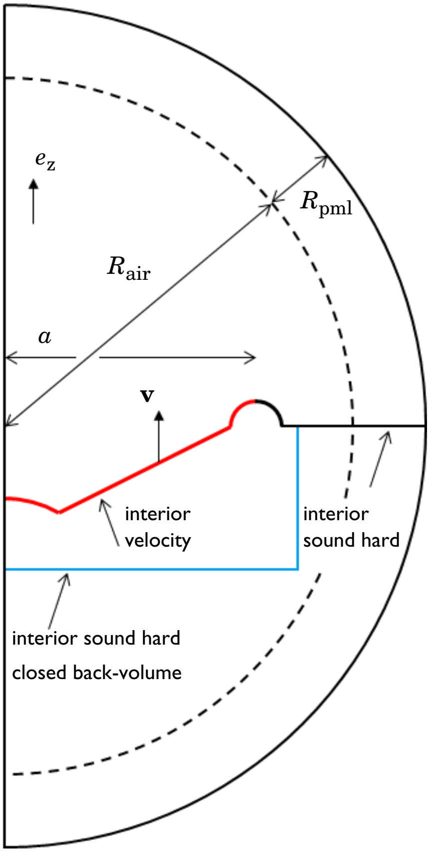


Figure 3: Computational domain and boundary conditions.

FINITE ELEMENT DOMAIN

The computational domain where the pressure acoustics model is solved is sketched in Figure 3. It represents the speaker cone, dust cap, and outer suspension, in an infinite baffle in a 2D axisymmetric model. On the speaker (red line), the **Interior Lumped Speaker Boundary** feature is applied and the rest of the baffle is an **Interior Sound Hard Boundary (Wall)** condition. In the closed back-volume configuration, the blue line is also modeled an interior sound hard wall. In the open configuration, no condition is applied to that boundary (it is transparent). The domain is truncated with a perfectly matched layer (PML) to mimic an infinite open domain. Note that the interior sound hard boundary of the baffle should be applied inside the PML. This is to avoid erroneous energy leaks between the two sides of the infinite baffle. The air domain has a radius of R_{air} while the thickness of the PML layer is R_{pml}. For more information about PMLs in acoustics, see the *Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces)* section in the *Pressure Acoustics Module User's Guide*.

Results and Discussion

In this section, the results of the open back-volume configuration are first discussed in detail. Then selected results are presented for the closed back-volume configuration.

OPEN BACK-VOLUME CONFIGURATION

The generated pressure field is shown in Figure 4 for $1\ \mathrm{kHz}$ and 5 kHz. This plot shows the directive characteristic of the speaker cone at increasing frequencies; this nature is discussed more at the end of this section when discussing the directivity plot Figure 11.

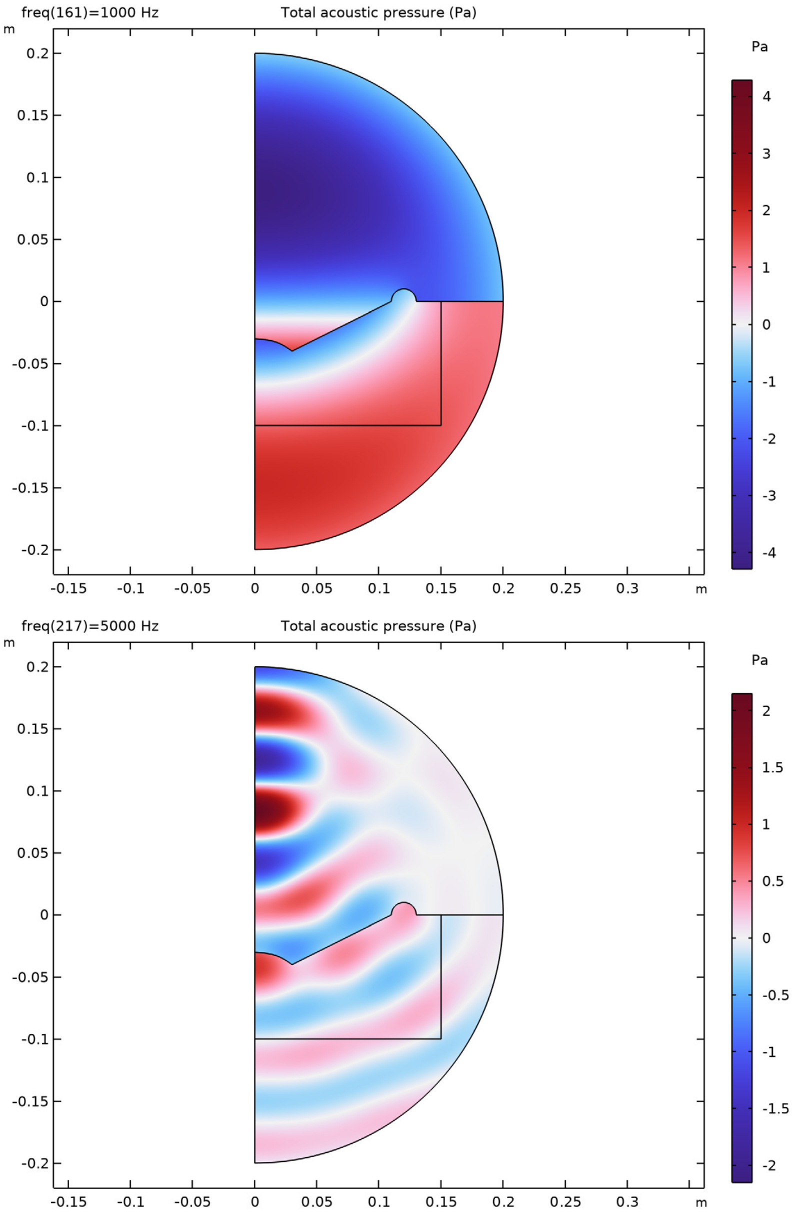


Figure 4: Acoustic pressure for a frequency of 1 kHz (top) and 5 kHz (bottom).

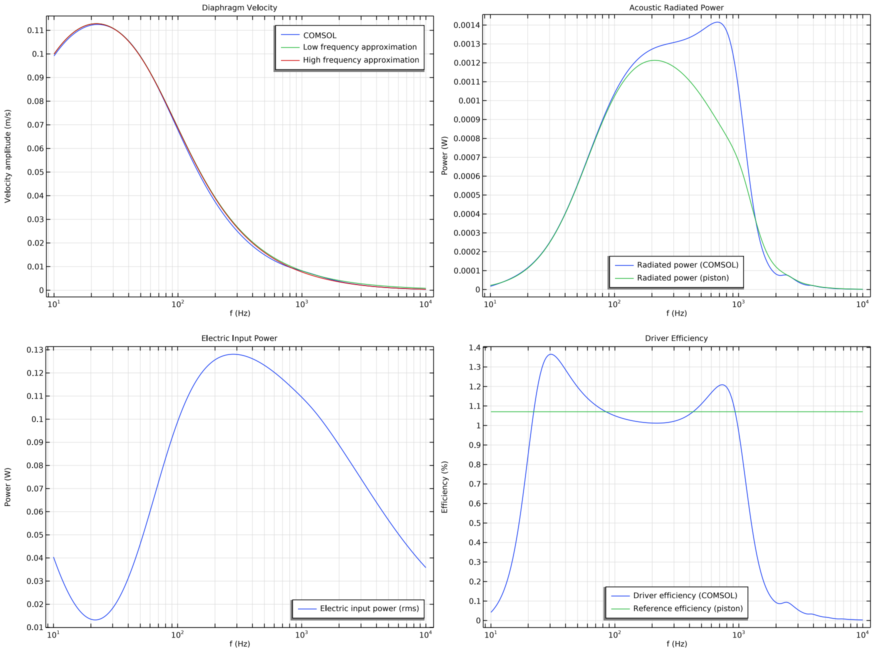


Figure 5: The frequency-dependent diaphragm velocity amplitude (upper left), acoustic radiated power (upper right), electric input power (lower left), and efficiency (lower right).

In Figure 5 and Figure 6, a series of characteristic loudspeaker driver curves are shown. In some figures the results obtained with the hybrid lumped and finite element of COMSOL are compared with theoretical curves that are based on the piston approximation.

In the upper left of Figure 5, the calculated speaker cone axial velocity u_{D} is shown together with two theoretical curves representing the approximate high-frequency and low-frequency behavior of a piston driver (see Ref. 1). The general trend is that the low-frequency curve fits well for most of the frequency range. The high-frequency curve is seen to converge toward the higher frequencies. If you increase the frequency range of the model, the trend is that the high-frequency approximation is a better fit as expected.

The next two graphs in Figure 5 (top right and bottom left) represent the acoustically radiated power P_{AR} (see Equation 3) and the electric input power P_{E} (see Equation 4). At the low frequencies, the behavior is just as the piston model, as expected. The transition to nonmatching models occurs when the speaker diameter becomes comparable to half the wavelength at around 700 Hz. The last graph of Figure 5 represents the driver efficiency given in percent (%), that is, the ratio of the input electric power to the acoustic radiated power. The actual efficiency is seen to match well with the predicted reference efficiency η_0 of about 1%.

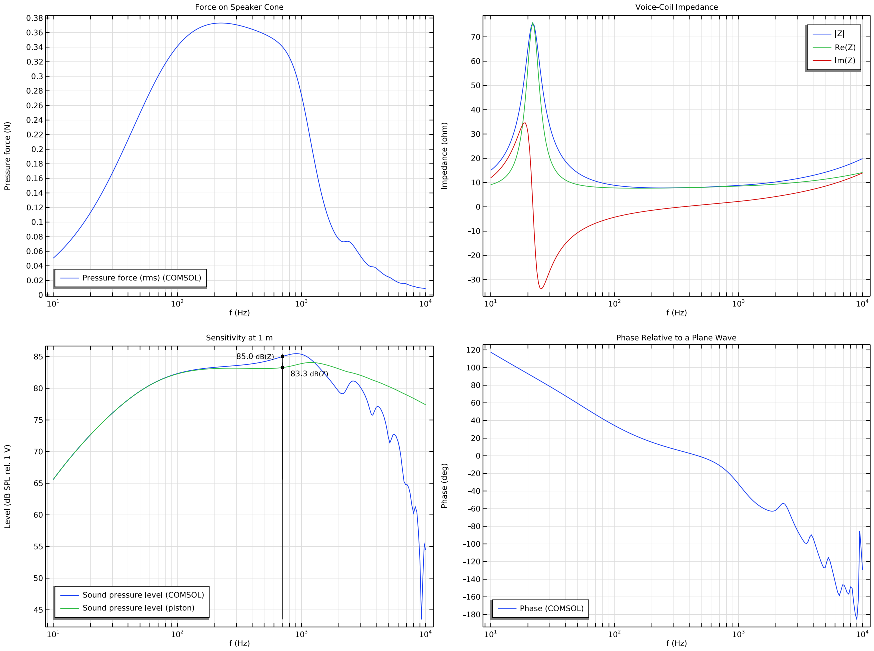


Figure 6: The frequency-dependent pressure force on the speaker cone (rms value) (upper left), voice coil impedance (upper right), speaker sensitivity (lower left), and phase response (lower right).

In Figure 6 (upper left), the RMS value of the acoustic pressure force F_D on the membrane (see Equation 1) is shown as function of frequency. The RMS value is obtained by the usual formula

$$\sqrt{0.5 \cdot F_D \cdot F_D^*}$$

where the * represents the complex conjugate.

Figure 6 (top right) represents the voice coil impedance (absolute value, real, and imaginary part) calculated as V_i/i_c. The resonance in the electric system is seen to coincide with the fundamental resonance frequency F_s = 22.0 Hz (see Table 3 and the **Global Definitions** > **Parameters** list in the model).

The two last figures in Figure 6 (bottom left and right) represent the speaker response (sound pressure level) measured 1 m in front and driven at 1 V RMS and the relative phase measured in the same point. The sensitivity is seen to match the piston model at the low frequencies as expected and it has realistic values for a speaker unit. The phase is represented as the phase of the pressure $p(0{,}1\text{ m})$ relative to the phase a plane wave would have in the same point $\exp(-ik_0z)$ with $z = 1$ m. In this way, the pure distance (phase lag) component of the phase has been removed.

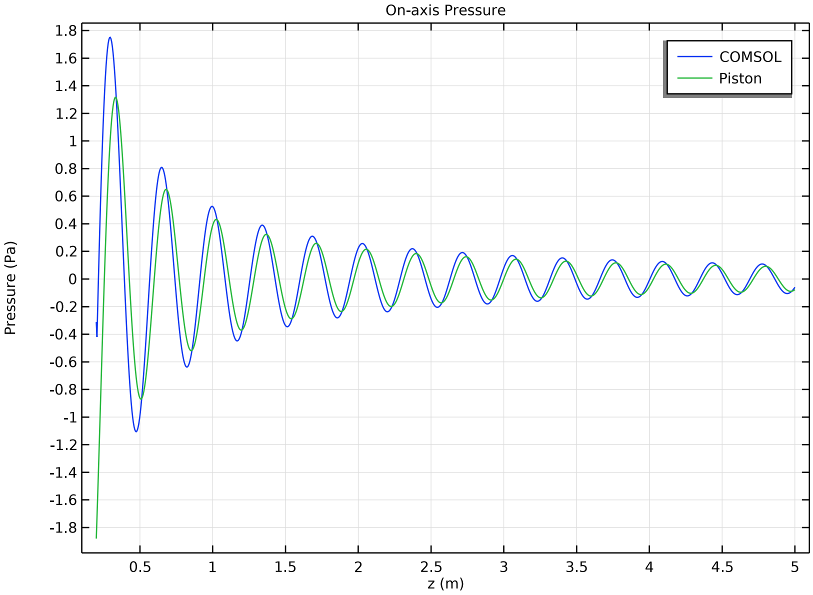


Figure 7: The pressure along the center z-axis from just outside the computational domain to a distance of 5 m. Evaluated for a frequency of 1000 Hz. The pressure is calculated using the exterior field calculation feature.

The pressure field along the z-axis is shown in Figure 7 from $z = R_{\text{air}}$ to $z = 5$ m, evaluated at a frequency of 1000 Hz. In the model, a *Parametric Curve 2D* is used to evaluate the exterior-field pressure outside the computational mesh. The figure compares the modeled pressure with the analytical on axis pressure from a piston. The agreement is seen to be quite good at this frequency. For lower frequencies, the agreement becomes much better while the two curves diverge at higher frequencies, as expected (change the evaluation frequency in the plot to see this trend).

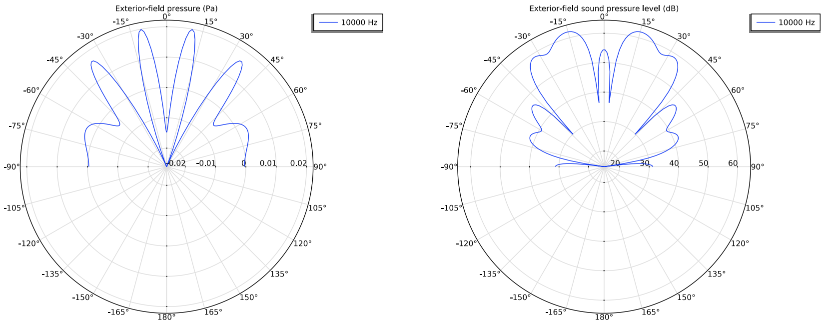


Figure 8: Exterior-field pressure and sound pressure level evaluated at a distance of 1 m the half sphere in front of the speaker and at 10 kHz.

The pressure and the sound pressure level evaluated at 1 m, using the dedicated radiation pattern plots, are shown in Figure 8, here evaluated at 10 kHz. In the figure, the 0° mark corresponds to the axial z direction. Both figures show a very strong directive pattern as expected at this high frequency.

In Figure 9 and Figure 10, the radiated intensity is illustrated and evaluated around the four frequencies 100 Hz, 1000 Hz, 5 kHz, and 10 kHz. The color plot represents the magnitude of the intensity vector $\mathbf{I}$, the domain vector field represents the components of the intensity vector, and finally the vectors plotted on the edges represent the surface normals. The normals are useful when setting up, for example, the expression for the total radiated power (Equation 3). In this case, it is necessary to use the `up()` operator to get the intensity on the upper side of the speaker cone (seen relative to the normal direction). The four plots clearly show how the acoustic energy is focused for increasing frequencies.

The so-called directivity plot of the speaker unit is shown in Figure 11. This plot represents a contour plot of the sound pressure level L_p evaluated along a half circle in front of the speaker as function of the angle and the frequency, that is, $L_\mathrm{p}(\theta, f)$. The x-axis represents the angle and runs from –90° to 90°. The y-axis is a logarithmic frequency axis running from 10^1 Hz to 10^4 Hz. The plot illustrates how the spatial response goes from a nearly omnidirectional constant value at the low frequencies, through a single lobe response at intermediate frequencies, and ends up as a complex directive pattern at high frequencies. This type of plot is very often used, in industry, to characterize speakers and speaker units.

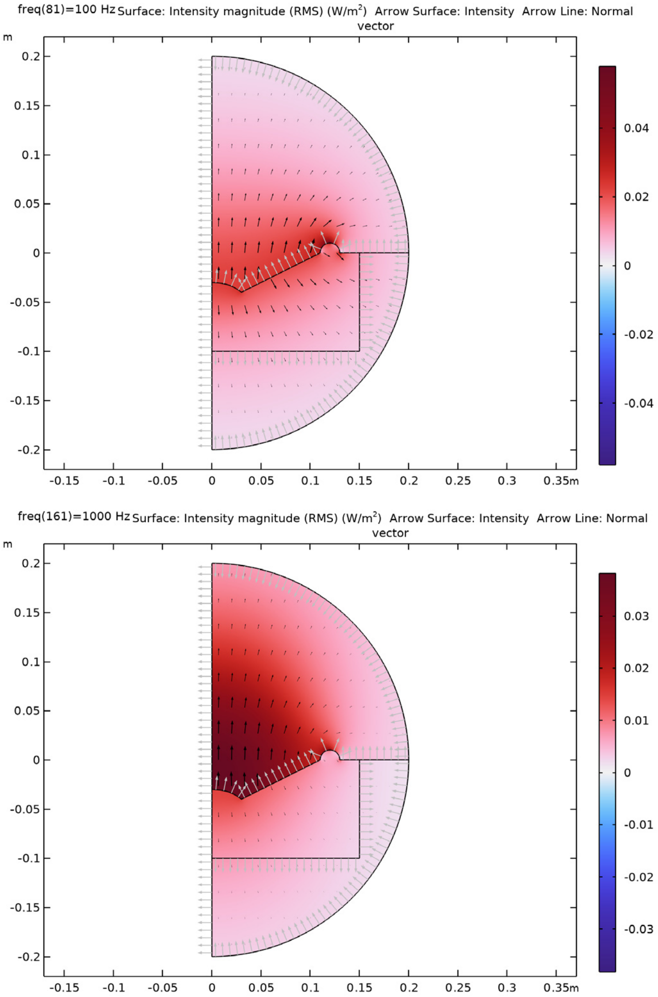


Figure 9: Intensity magnitude (color plot), intensity vector field (domain arrows), and surface normals (edge arrows) for 100 Hz and 1000 Hz.

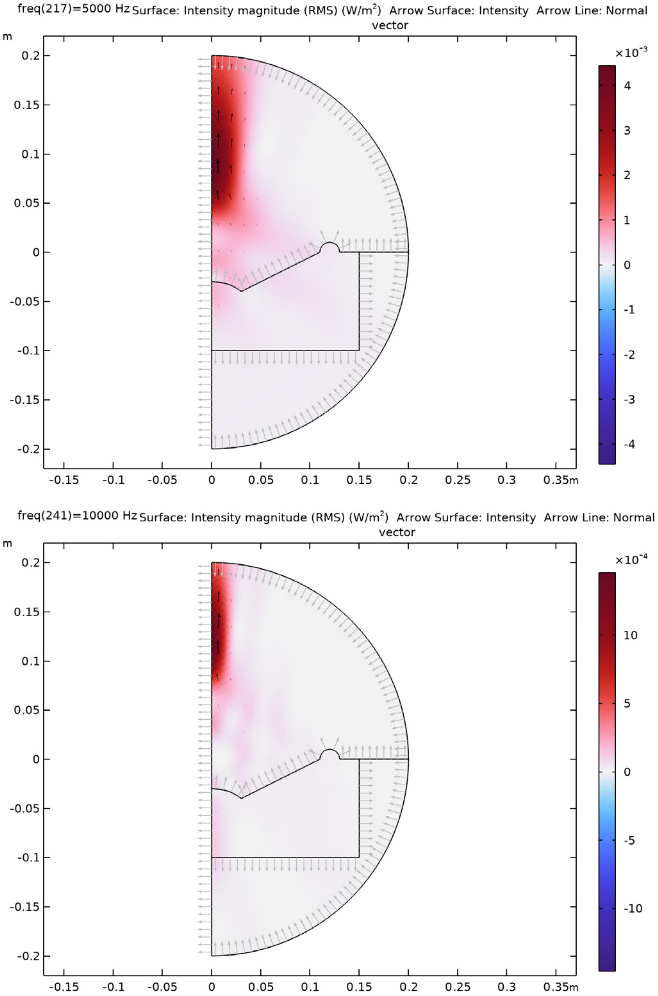


Figure 10: Intensity magnitude (color plot), intensity vector field (domain arrows), and surface normals (edge arrows) for 5 kHz and for10 kHz.

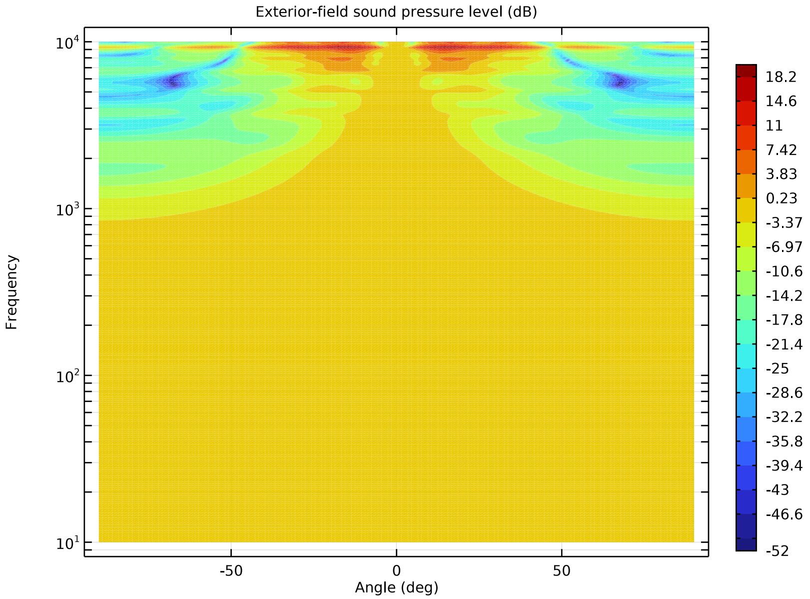


Figure 11: Directivity plot for the speaker. The x-axis is a scaled azimuthal angle that runs from -90° to 90° and the y axis is a logarithmic frequency axis that runs from 10^1 *Hz to* 10^4 *Hz = 10 kHz.*

CLOSED BACK-VOLUME CONFIGURATION

When the model is solved with the closed back-volume configuration, the response of the system changes. This is easily seen in the voice coil impedance curves in Figure 12. The location of the fundamental resonance frequency F_s is seen to increase. This is because the presence of the closed space behind the speaker diaphragm corresponds to an increased mechanical stiffness. The sensitivity of the speaker is depicted in Figure 13. The presence of the closed volume is seen to heavily influence the low-frequency performance of the speaker. This is typically also why speaker cabinets have a ventilation port, that is, to reduce this low-frequency roll-off.

Testing the model with a closed back-volume also has the advantage of testing the entire model setup (the electrical circuit to FEM coupling). For example, a sign error in the applied pressure force on the diaphragm would give more or less correct results for the open configuration, but would not predict the expected resonance shift. This kind of sanity tests are important when setting up numerical models.

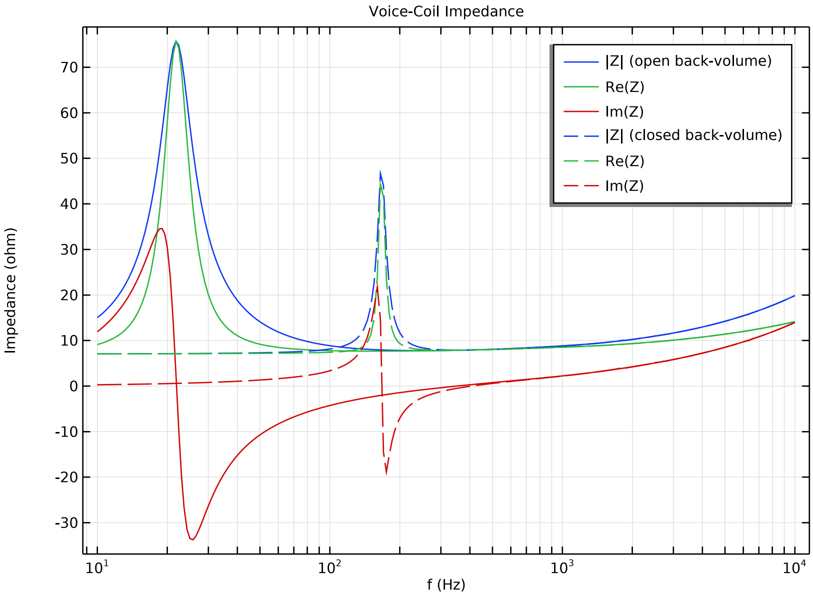


Figure 12: Voice coil impedance curves for both the open (solid) and closed (dashed) back-volume configurations.

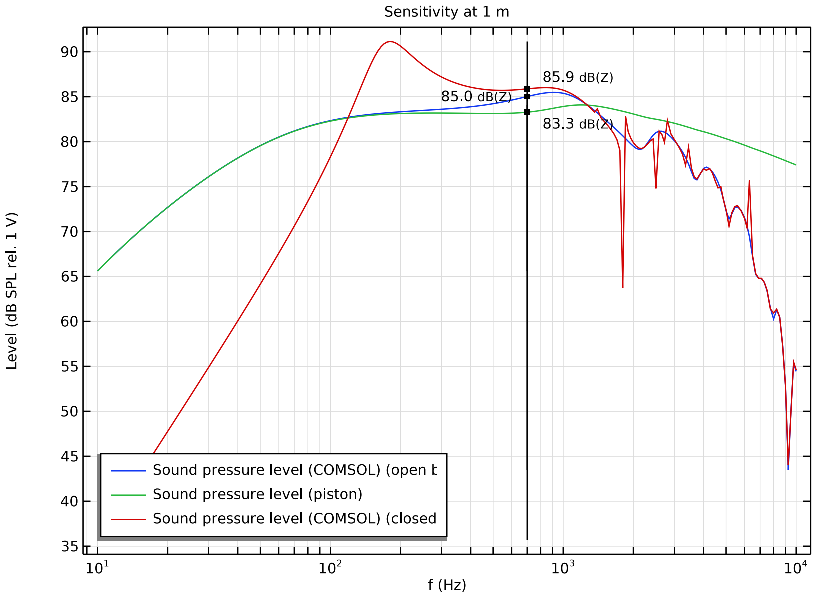


Figure 13: Sensitivity curves for the open back-volume configuration, the analytical results, and the closed back-volume configuration.

References

1. W. Marshall Leach, Jr., *Introduction to Electroacoustics and Audio Amplifier Design*, Kendall Hunt, 2010.

2. *Loudspeaker Driver Model Documentation*, from the COMSOL Application Library.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver`

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 **Acoustics** > **Pressure Acoustics** > **Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **AC/DC** > **Electrical Circuit (cir)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.
8. Click **Done**.

GLOBAL DEFINITIONS

Load all the model parameters from a file; they include all the small signal parameters (Table 2 and Table 3) as well as geometry parameters.

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 `lumped_loudspeaker_driver_parameters.txt`.

   Build the simple 2D axisymmetric geometry of the speaker driver by drawing some circles and lines.

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 `1[cm]`.
4. In the **Sector angle** text field, type `180`.
5. Locate the **Position** section. In the **r** text field, type `a`.

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 `Rair+Rpml`.
4. In the **Sector angle** text field, type `180`.
5. Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.
6. Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Rpml`

Polygon 1 (pol1)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Object Type** section.
3. From the **Type** list, choose **Open curve**.
4. Locate the **Coordinates** section. In the table, enter the following settings:

r (m)	z (m)
`3[cm]`	`-4[cm]`
`a-1[cm]`	`0`

Polygon 2 (pol2)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Object Type** section.
3. From the **Type** list, choose **Open curve**.
4. Locate the **Coordinates** section. In the table, enter the following settings:

r (m)	z (m)
`a+1[cm]`	`0`
`Rair`	`0`

5. Click **Build All Objects**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

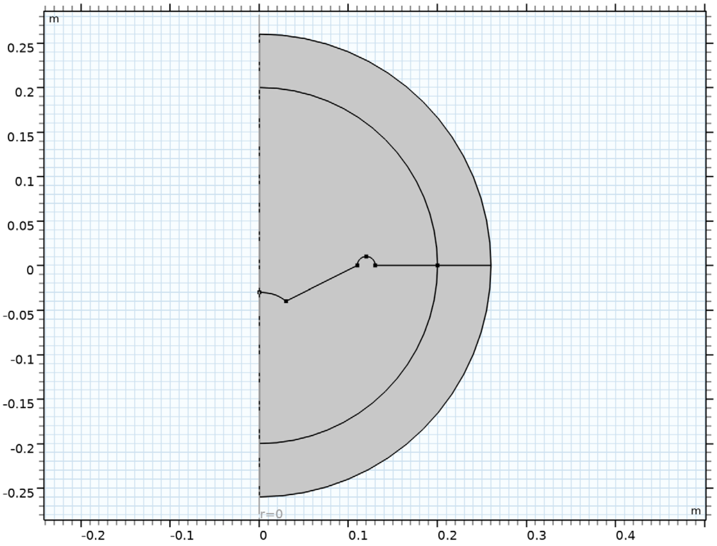
Quadratic Bézier 1 (qb1)

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 **2**, set **r** to `1.8[cm]`.
4. In row **3**, set **r** to `3[cm]`.
5. In row **1**, set **z** to `-3[cm]`.
6. In row **2**, set **z** to `-3.1[cm]`.
7. In row **3**, set **z** to `-4[cm]`.
8. Locate the **Weights** section. In the **2** text field, type `1.5`.
9. Click **Build All Objects**.

Delete Entities 1 (del1)

1. In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.
2. On the object **c1**, select Boundaries 3 and 4 only.
3. In the **Settings** window for **Delete Entities**, click **Build Selected**.



Polygon 3 (pol3)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Object Type** section.
3. From the **Type** list, choose **Open curve**.

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

r (m)	**z (m)**
`0.15`	`0`
`0.15`	`-0.1`
`0`	`-0.1`

5 Click **Build All Objects**.

Now, set up all the variables, selections, and component couplings under the **Definitions** node. Load the variables from the two variable files (one for model variables and one for the analytical piston expressions). The selections represent the speaker cone surface with an internal acceleration (red line in Figure 3) and the internal sound hard wall.

DEFINITIONS

Model variables

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

2 In the **Settings** window for **Variables**, type `Model variables` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `lumped_loudspeaker_driver_variables_1.txt`.

Analytical approximations

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

2 In the **Settings** window for **Variables**, type `Analytical approximations` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `lumped_loudspeaker_driver_variables_2.txt`.

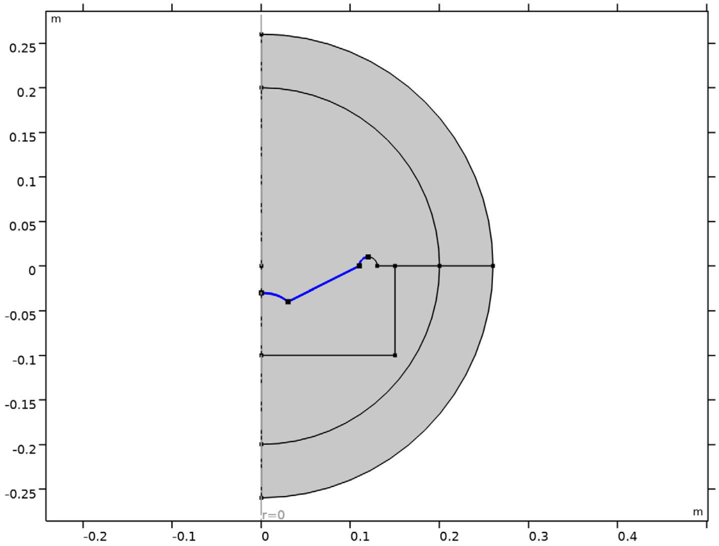
Speaker

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

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

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 8, 15, and 18 only.



Baffle (interior wall)

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

2 In the **Settings** window for **Explicit**, type `Baffle (interior wall)` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 9, 11, 12, and 19 only.

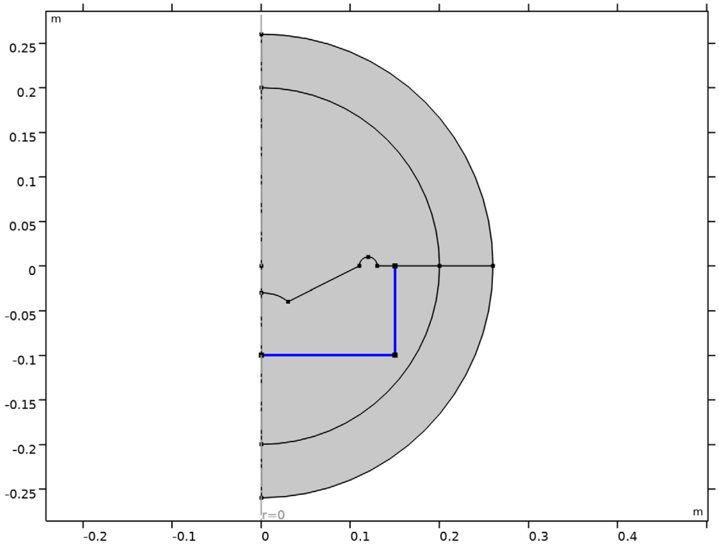
Back Volume (interior wall)

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

2 In the **Settings** window for **Explicit**, type `Back Volume (interior wall)` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 4 and 10 only.



Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 1 and 5 only.

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

4 From the **Coordinate stretching type** list, choose **Rational**.

5 In the **PML scaling factor** text field, type `0.5`.

6 In the **PML scaling curvature parameter** text field, type `5`.

You have now changed the default settings for the perfectly matched layer (PML). The new settings will improve the performance of the PML at very low frequencies. First of all, the acoustic radiated power should be positive. If the default settings had been used you would see a negative radiated power at low frequencies (see Figure 5 (top right) and the instructions on how to create the plot, further down). The issue is that at low frequencies the evanescent waves created by the moving speaker cone extend into the PML layer. The interaction between the scaled coordinate system in the PML and these waves may create an erroneous energy contribution to the model (can be either positive or negative). Note that the evanescent waves decay in only a fraction of a wavelength.

A good way to investigate the performance of the PML is to make a sensitivity analysis on some parameter (for example the total radiated acoustic power) with respect to changes in the PML parameters. This model does not include such a sensitivity analysis.

In general, increasing the curvature factor effectively shifts the resolving power of the PML toward the physical domain, which is necessary in this case since the evanescent components decay in only a fraction of a wavelength. However, if you increase it too much, you may lose resolution in the other end, that is, of the free space wavelength. Assuming that the PMLs work properly for high frequencies and curvature parameter 1, you can in principle do a convergence study increasing a constant curvature parameter until the low-frequency result converges (for 10 Hz) while making sure that the high-frequency result (for 10 kHz) is not affected. It turns out that a value of 5 yields good results in this model. A scaling factor of 0.5 further improves the results, but only by a small amount. Decreasing the scaling factor corresponds to compressing the PML layer (shortening it), which in turn effectively increases the mesh resolution.

ADD MATERIAL

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

Now, set up the physics and the boundary conditions for the model. Use the interior conditions at the diaphragm and on the infinite baffle. This condition will allow for a discontinuous pressure field.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Interior Sound Hard Boundary (Wall) 1

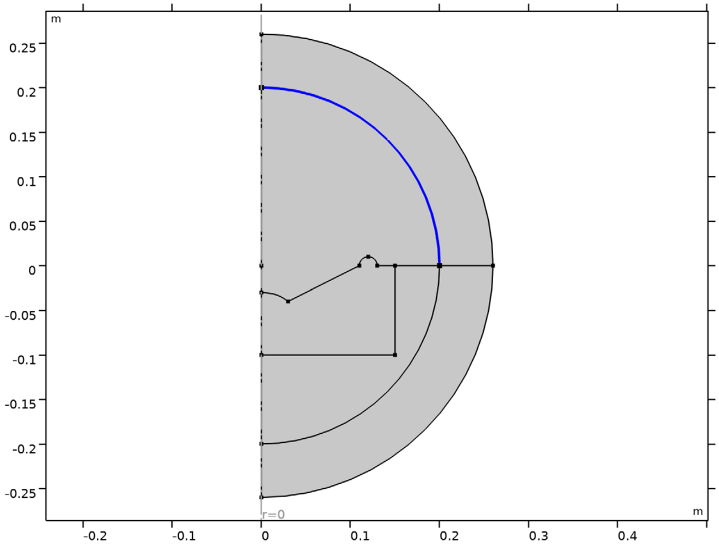
1. In the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.
2. In the **Settings** window for **Interior Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Baffle (interior wall)**.

Interior Lumped Speaker Boundary 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Interior Lumped Speaker Boundary**.
2. In the **Settings** window for **Interior Lumped Speaker Boundary**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Speaker**.

Exterior Field Calculation 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.
2. Select Boundary 16 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**.

   Note that you have applied the exterior-field calculation condition only to the front of the speaker. In reality, the exterior-field condition should be applied to boundaries surrounding all sources and scatterers. This is of course not possible with an infinite baffle. However, in this specific case, a trick can be used, namely employing the fact that symmetry (in $z = 0$) is equal to a sound-hard wall in pressure acoustics (as indicated in the UI). So, in the infinite baffle configuration, the exterior-field condition can still be used.

Interior Sound Hard Boundary (Wall) 2

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

2 In the **Settings** window for **Interior Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Back Volume (interior wall)**.

This interior wall condition will be deactivated in the first study when modeling the open configuration. In the second study the interior wall is active and the speaker operates with a closed back-volume.

Proceed to set up the electrical circuit system for the electric and mechanical model. When building this look at Figure 3 for the references to the node numbers used in the model.

ELECTRICAL CIRCUIT (CIR)

Voltage Source 1 (V1)

1 In the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. In the v_{src} text field, type `V0`.

Resistor 1 (R1)

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `R_g`.

Resistor 2 (R2)

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	3

4 Locate the **Device Parameters** section. In the R text field, type `R_E`.

Inductor 1 (L1)

1 In the **Electrical Circuit** toolbar, click **Inductor**.

2 In the **Settings** window for **Inductor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the L text field, type `L_E`.

Resistor 3 (R3)

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the R text field, type `Rp_E`.

Current-Controlled Voltage Source 1 (H1)

1 In the **Electrical Circuit** toolbar, click **Current-Controlled Voltage Source**.

2 In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Node Connections** section.

3. In the table, enter the following settings:

Label	Node names
p	4
n	0

4. Locate the **Current Measurement** section. In the **Gain** text field, type `BL[m/Wb*ohm]`.

   Remember to select the measured current when the mechanical circuit components have been set up.

Current-Controlled Voltage Source 2 (H2)

1. In the **Electrical Circuit** toolbar, click **Current-Controlled Voltage Source**.
2. In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	6
n	0

4. Locate the **Current Measurement** section. In the **Gain** text field, type `BL[m/Wb*ohm]`.
5. From the **Measure current for device** list, choose **Resistor 2 (R2)**.

Inductor 2 (L2)

1. In the **Electrical Circuit** toolbar, click **Inductor**.
2. In the **Settings** window for **Inductor**, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	6
n	7

4. Locate the **Device Parameters** section. In the *L* text field, type `M_MD[H/kg]`.

Current-Controlled Voltage Source 1 (H1)

1. In the **Model Builder** window, click **Current-Controlled Voltage Source 1 (H1)**.
2. In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Current Measurement** section.
3. From the **Measure current for device** list, choose **Inductor 2 (L2)**.

Resistor 4 (R4)

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	7
n	8

4 Locate the **Device Parameters** section. In the R text field, type `R_MS[ohm/kg*s]`.

Capacitor 1 (C1)

1 In the **Electrical Circuit** toolbar, click **Capacitor**.

2 In the **Settings** window for **Capacitor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	8
n	9

4 Locate the **Device Parameters** section. In the C text field, type `C_MS[F*N/m]`.

External I vs. U 1 (IvsU1)

1 In the **Electrical Circuit** toolbar, click **External I vs. U**.

2 In the **Settings** window for **External I vs. U**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	9
n	0

4 Locate the **External Device** section. From the V list, choose **Voltage from lumped speaker boundary (acpr/ilsb1)**.

The lumped electromechanical model defined by the electrical circuit is now fully coupled with the **Interior Lumped Speaker Boundary** feature in the finite element domain.

MESH

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

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3. In the table, clear the **Use** checkbox for **Electrical Circuit (cir)**.

   Now, proceed to the study and set the frequencies, before building the mesh and solving.

STUDY 1 - OPEN BACK-VOLUME

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study 1 - Open Back-Volume` in the **Label** text field.

Step 1: Frequency Domain

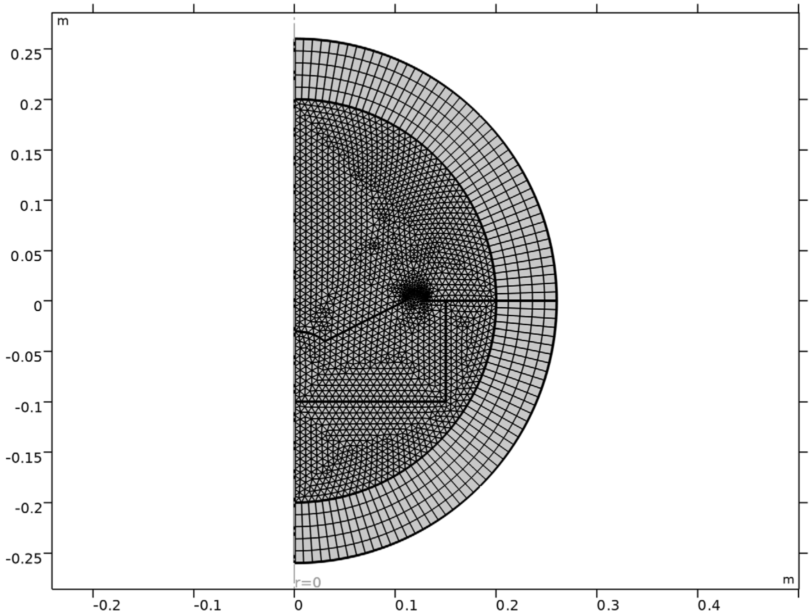
1. In the **Model Builder** window, under **Study 1 - Open Back-Volume** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. Click **Range**.
4. In the **Range** dialog, choose **ISO preferred frequencies** from the **Entry method** list.
5. In the **Start frequency** text field, type `10`.
6. In the **Stop frequency** text field, type `10000`.
7. From the **Interval** list, choose **1/24 octave**.
8. Click **Replace**.

   Disable the interior wall that defines the closed back-volume for this first study where you use the open configuration.

9. In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.
10. Select the **Modify model configuration for study step** checkbox.
11. In the tree, select **Component 1 (comp1)** > **Pressure Acoustics, Frequency Domain (acpr)** > **Interior Sound Hard Boundary (Wall) 2**.
12. Click **Disable**.

MESH 1

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



STUDY 1 - OPEN BACK-VOLUME

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

RESULTS

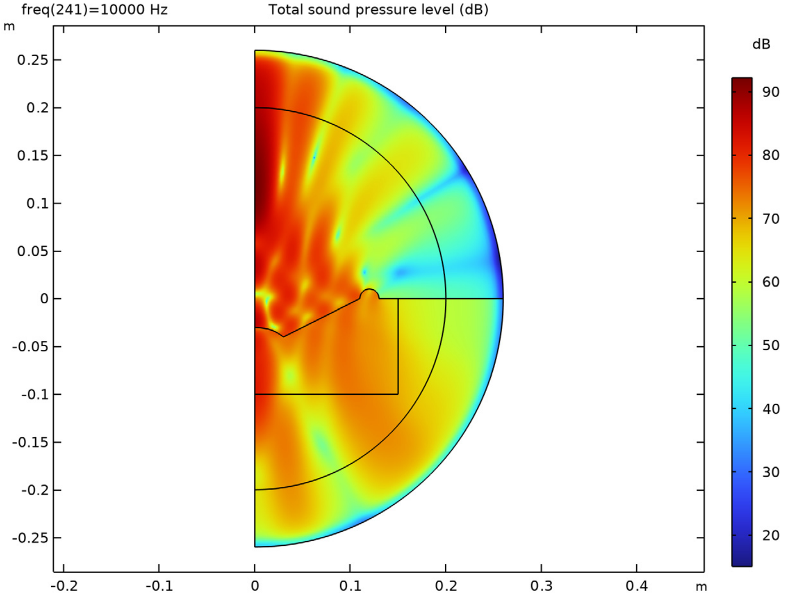
Acoustic Pressure (acpr)

First, look at the default plots. Investigate the 2D **Sound Pressure Level (acpr)** plot to verify the performance of the perfectly matched layer (PML). After doing this you can disable plotting in the PML region, which is unphysical. Secondly, look at the default exterior-field plots and make a few changes, before setting up a range of plots to investigate the loudspeaker driver performance.

Sound Pressure Level (acpr)

Look at the sound pressure level (SPL) plots at the frequencies of 10 kHz, 1 kHz, and 10 Hz. Note that the SPL decreases nearly 100 dB over the width of the thickness of the PML. This means that the outgoing waves are extremely damped.

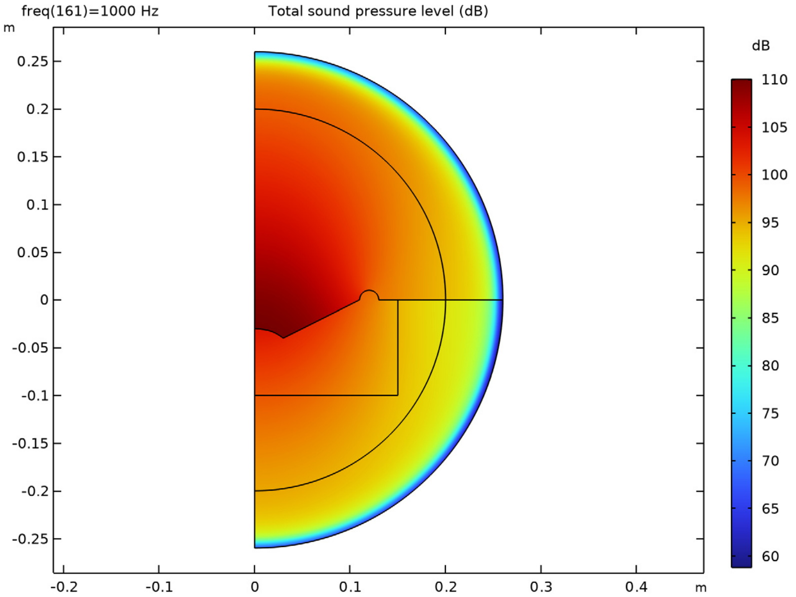
1 In the **Model Builder** window, click **Sound Pressure Level (acpr)**.



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

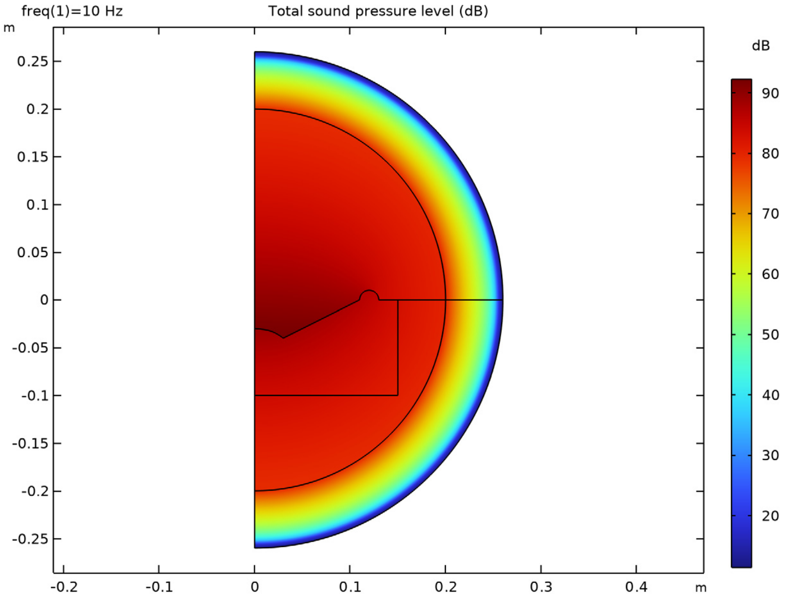
3 From the **Parameter value (freq (Hz))** list, choose **1000**.

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



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

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



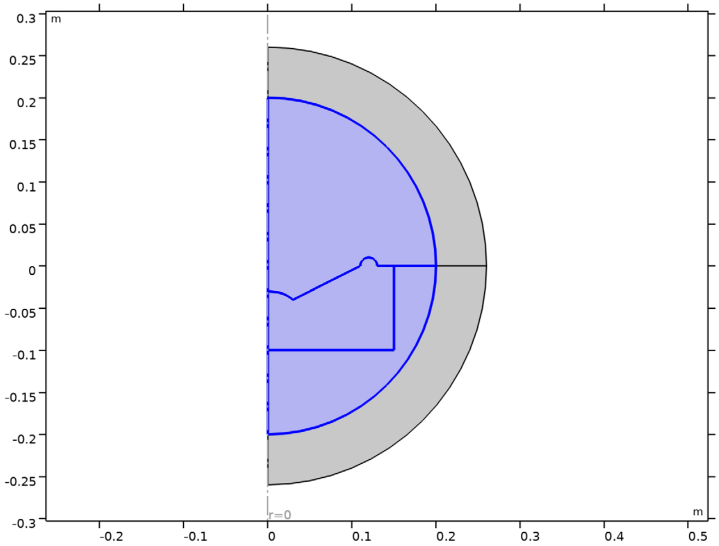
Study 1 - Open Back-Volume/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Study 1 - Open Back-Volume/Solution 1 (sol1)**.

Selection

1. In the **Results** toolbar, click **Attributes** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.

4. Select Domains 2–4 only.



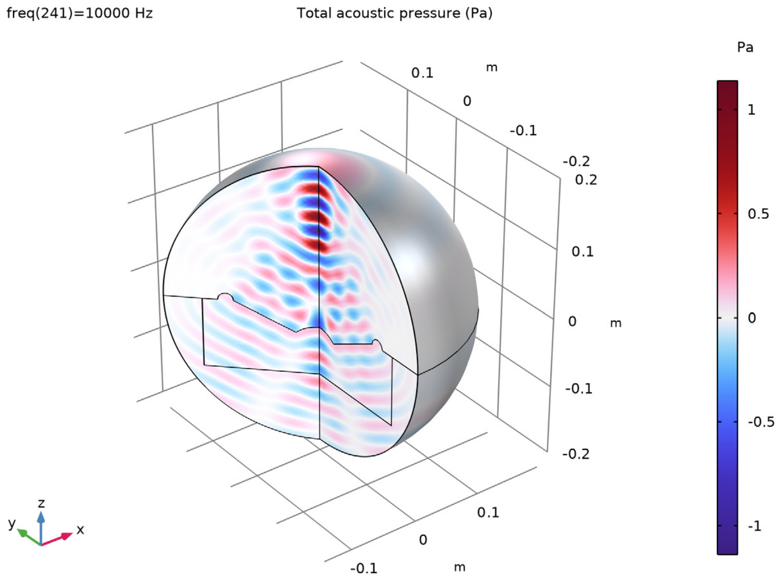
Acoustic Pressure (acpr)

1. In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Parameter value (freq (Hz))** list, choose **1000**.
4. In the **Acoustic Pressure (acpr)** toolbar, click **Plot**.
5. Click the **Zoom Extents** button in the **Graphics** toolbar.
6. From the **Parameter value (freq (Hz))** list, choose **5000**.
7. In the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

   These two plots should reproduce Figure 4.

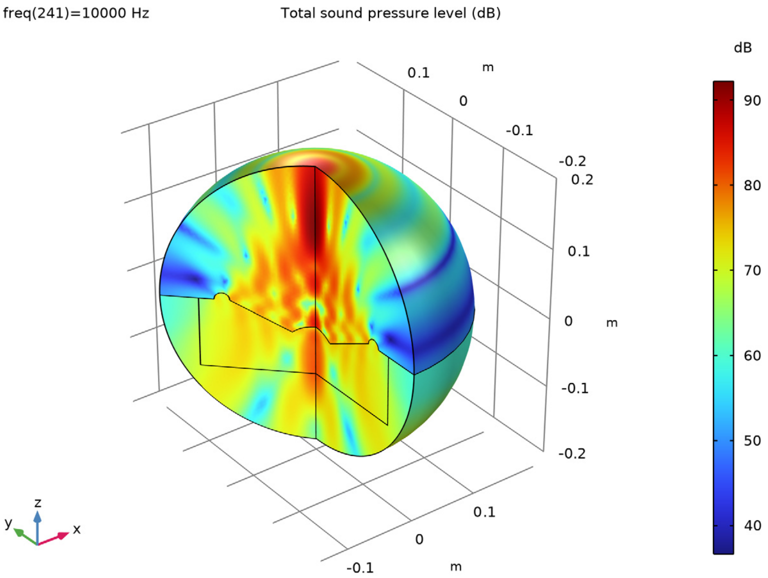
Acoustic Pressure, 3D (acpr)

In the **Model Builder** window, click **Acoustic Pressure, 3D (acpr)**.



Sound Pressure Level, 3D (acpr)

In the **Model Builder** window, click **Sound Pressure Level, 3D (acpr)**.



Radiation Pattern 1

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

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

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

Radiation Pattern 1

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

2 In the **Settings** window for **Radiation Pattern**, 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 **Exterior-Field Pressure (acpr)** toolbar, click **Plot**.

   These two polar plots should reproduce Figure 8.

Now, create nine 1D plots that depict various speaker characteristic plots. Each plot includes a number of steps to set up the title, the axis, the plot name and so on; these are optional and can be skipped. You can also just create the specific plot that is of interest to you. The next steps reproduce the graphs in Figure 5, Figure 6, and Figure 7.

The plots are the following:

1. Membrane Velocity
2. Acoustic Radiated Power
3. Electric Input Power
4. Efficiency
5. Force on Speaker Cone
6. Voice-Coil Impedance
7. Sensitivity
8. Phase
9. On-axis Exterior-Field Pressure

Diaphragm Velocity

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Diaphragm Velocity` in the **Label** text field.
3. Click to expand the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Plot Settings** section.
5. Select the **x-axis label** checkbox. In the associated text field, type `f (Hz)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Velocity amplitude (m/s)`.

Global 1

1. Right-click **Diaphragm Velocity** 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**
`abs(u_D)`	`m/s`	`COMSOL`
`abs(Qlf_D/S_D)`	`m/s`	`Low frequency approximation`
`abs(Qhf_D/S_D)`	`m/s`	`High frequency approximation`

4. In the **Diaphragm Velocity** toolbar, click **Plot**.
5. Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

   This plot should reproduce Figure 5 (top left).

Acoustic Radiated Power

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Acoustic Radiated Power` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Plot Settings** section.
5. Select the **x-axis label** checkbox. In the associated text field, type `f (Hz)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Power (W)`.
7. Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

Global 1

1. Right-click **Acoustic Radiated Power** and choose **Global**.
2. In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **P_AR - Radiated power (COMSOL) - W**.
3. Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **P_AR_ana - Radiated power (piston) - W**.
4. In the **Acoustic Radiated Power** toolbar, click **Plot**.
5. Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

   This plot should reproduce Figure 5 (top right).

Electric Input Power

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Electric Input Power` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

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

6 Select the **y-axis label** checkbox. In the associated text field, type `Power (W)`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **Electric Input 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)** > **Definitions** > **Variables** > **P_E - Electric input power (rms) - W**.

3 In the **Electric Input Power** toolbar, click **Plot**.

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

This plot should reproduce Figure 5 (bottom left).

Driver Efficiency

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

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

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

4 Locate the **Plot Settings** section.

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

6 Select the **y-axis label** checkbox. In the associated text field, type `Efficiency (%)`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

Global 1

1 Right-click **Driver Efficiency** 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**
`eta*100`	`1`	`Driver efficiency (COMSOL)`
`eta0*100`		`Reference efficiency (piston)`

4 In the **Driver Efficiency** toolbar, click **Plot**.

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

   This plot should reproduce Figure 5 (bottom right).

Force on Speaker Cone

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Force on Speaker Cone` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Plot Settings** section.
5. Select the **x-axis label** checkbox. In the associated text field, type `f (Hz)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Pressure force (N)`.
7. Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1. Right-click **Force on Speaker Cone** 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**
`sqrt(0.5*F_D*conj(F_D))`	`N`	`Pressure force (rms) (COMSOL)`

4. In the **Force on Speaker Cone** toolbar, click **Plot**.
5. Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

   This plot should reproduce Figure 6 (top left).

Voice-Coil Impedance

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Voice-Coil Impedance` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Plot Settings** section.
5. Select the **x-axis label** checkbox. In the associated text field, type `f (Hz)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Impedance (ohm)`.

Global 1

1. Right-click **Voice-Coil Impedance** 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
`abs(V0/cir.R1.i)`	`ohm`	`|Z|`
`real(V0/cir.R1.i)`	`ohm`	`Re(Z)`
`imag(V0/cir.R1.i)`	`ohm`	`Im(Z)`

4. In the **Voice-Coil Impedance** toolbar, click **Plot**.
5. Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

   This plot should reproduce Figure 6 (top right).

Sensitivity at 1 m

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Sensitivity at 1 m` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Plot Settings** section.
5. Select the **x-axis label** checkbox. In the associated text field, type `f (Hz)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Level (dB SPL rel. 1 V)`.
7. Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Octave Band 1

1. In the **Sensitivity at 1 m** toolbar, click **More Plots** and choose **Octave Band**.
2. In the **Settings** window for **Octave Band**, locate the **Selection** section.
3. From the **Geometric entity level** list, choose **Global**.
4. Locate the **y-Axis Data** section. In the **Expression** text field, type `pext(0,1)`.
5. Locate the **Plot** section. From the **Quantity** list, choose **Continuous power spectral density**.
6. Click to expand the **Legends** section. Select the **Show legends** checkbox.
7. From the **Legends** list, choose **Manual**.
8. In the table, enter the following settings:

Legends
`Sound pressure level (COMSOL)`

Graph Marker 1

1. Right-click **Octave Band 1** and choose **Graph Marker**.
2. In the **Settings** window for **Graph Marker**, locate the **Display** section.
3. From the **Display mode** list, choose **Line intersection**.
4. In the **x-coordinates** text field, type `700`.
5. Select the **Show lines** checkbox.
6. Locate the **Text Format** section. In the **Precision** text field, type `3`.
7. Select the **Include unit** checkbox.
8. Click to expand the **Coloring and Style** section. From the **Anchor point** list, choose **Middle right**.
9. In the **Sensitivity at 1 m** toolbar, click **Plot**.

Octave Band 2

1. In the **Model Builder** window, under **Results** > **Sensitivity at 1 m** right-click **Octave Band 1** and choose **Duplicate**.
2. In the **Settings** window for **Octave Band**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `sqrt(2)*prms`.
4. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Sound pressure level (piston)`

Graph Marker 1

1. In the **Model Builder** window, expand the **Octave Band 2** node, then click **Graph Marker 1**.
2. In the **Settings** window for **Graph Marker**, locate the **Coloring and Style** section.
3. From the **Anchor point** list, choose **Upper left**.
4. In the **Sensitivity at 1 m** toolbar, click **Plot**.

   This plot should reproduce Figure 6 (bottom left).

Phase Relative to a Plane Wave

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Phase Relative to a Plane Wave` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Plot Settings** section.

5. Select the **x-axis label** checkbox. In the associated text field, type `f (Hz)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Phase (deg)`.
7. Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1. Right-click **Phase Relative to a Plane Wave** 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
`arg(subst(acpr.efc1.pext,r,0,z,1[m])/(exp(-i*k0*1[m])*1[Pa]))`	`deg`	`Phase (COMSOL)`

4. Select the **Unwrap phase** checkbox.
5. In the **Phase Relative to a Plane Wave** toolbar, click **Plot**.
6. Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

   This plot should reproduce Figure 6 (bottom right).

   Set up a parametric curve used to evaluate the exterior-field outside of the computational mesh.

Parametric Curve 2D 1

1. In the **Results** toolbar, click **More Datasets** and choose **Parametric Curve 2D**.
2. In the **Settings** window for **Parametric Curve 2D**, locate the **Expressions** section.
3. In the **z** text field, type `s*5[m]+(1-s)*Rair`.
4. Select the **Only evaluate globally defined expressions** checkbox.

On-axis Pressure

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `On-axis Pressure` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Parametric Curve 2D 1**.
4. Locate the **Title** section. From the **Title type** list, choose **Label**.
5. Locate the **Plot Settings** section.
6. Select the **x-axis label** checkbox. In the associated text field, type `z (m)`.
7. Select the **y-axis label** checkbox. In the associated text field, type `Pressure (Pa)`.
8. Locate the **Data** section. From the **Parameter selection (freq)** list, choose **From list**.

9. In the **Parameter values (freq (Hz))** list box, select **1000**.

Line Graph 1

1. Right-click **On-axis Pressure** and choose **Line Graph**.
2. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `pext(r,z)`.
4. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
5. In the **Expression** text field, type `z`.
6. Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.
7. From the **Resolution** list, choose **Extra fine**.
8. Click to expand the **Legends** section. Select the **Show legends** checkbox.
9. From the **Legends** list, choose **Manual**.
10. In the table, enter the following settings:

Legends
`COMSOL`

Line Graph 2

1. Right-click **Line Graph 1** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `acpr.iomega*rho0*u_D*S_D*exp(-i*k0*z)/(2*pi*z)`.
4. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Piston`

5. In the **On-axis Pressure** toolbar, click **Plot**.

   This plot should reproduce Figure 7.

   Create a 2D intensity plot that includes the magnitude of the intensity vector `acpr.I_rms` as well as an arrow surface (vector field plot) of the intensity vector, with the components (`acpr.Ir`,`acpr.Iz`).

Intensity

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

Surface 1

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

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

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

Arrow Surface 1

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

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Pressure Acoustics, Frequency Domain** > **Intensity** > **acpr.Ir,acpr.Iz - Intensity**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Arrow Line 1

1 Right-click **Intensity** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Pressure Acoustics, Frequency Domain** > **Geometry** > **acpr.nr,acpr.nz - Normal vector**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Gray**.

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

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

This plot should reproduce the last frame in Figure 10, now change the evaluation frequency to 5000 Hz, 1000 Hz, and 100 Hz. This will reproduce the remaining frames in Figure 9 and Figure 10.

Intensity

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

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

3 From the **Parameter value (freq (Hz))** list, choose **5000**.

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

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

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

7 From the **Parameter value (freq (Hz))** list, choose **100**.

8 In the **Intensity** toolbar, click **Plot**.

Next create the directivity plot of the speaker.

Directivity

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

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

Directivity 1

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

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

3 Find the **Angles** subsection. In the **Number of angles** text field, type `180`.

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

5 In the ϕ **start** text field, type `-90`.

6 In the ϕ **range** text field, type `180`.

7 Click to expand the **Coloring and Style** section. From the **Layout** list, choose **Frequency on y-axis**.

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

9 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

This should reproduce the directivity plot depicted in Figure 11. You can tailor the plot to your needs using the normalization options or defining the specific levels to use in the contour plot.

Create an additional plot that shows the sensitivity and the phase in one plot using two y-axes.

Global 1

In the **Model Builder** window, under **Results** > **Phase Relative to a Plane Wave** right-click **Global 1** and choose **Copy**.

Sensitivity and Phase

1 In the **Model Builder** window, right-click **Sensitivity at 1 m** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Sensitivity and Phase` in the **Label** text field.

Octave Band 1

In the **Model Builder** window, expand the **Sensitivity and Phase** node.

Graph Marker 1

1 In the **Model Builder** window, expand the **Octave Band 1** node.

2 Right-click **Graph Marker 1** and choose **Delete**.

Octave Band 2

In the **Model Builder** window, right-click **Octave Band 2** and choose **Delete**.

Octave Band 1

1. In the **Settings** window for **Octave Band**, locate the **Plot** section.
2. From the **Quantity** list, choose **Band average power spectral density**.
3. From the **Band type** list, choose **1/3 octave**.

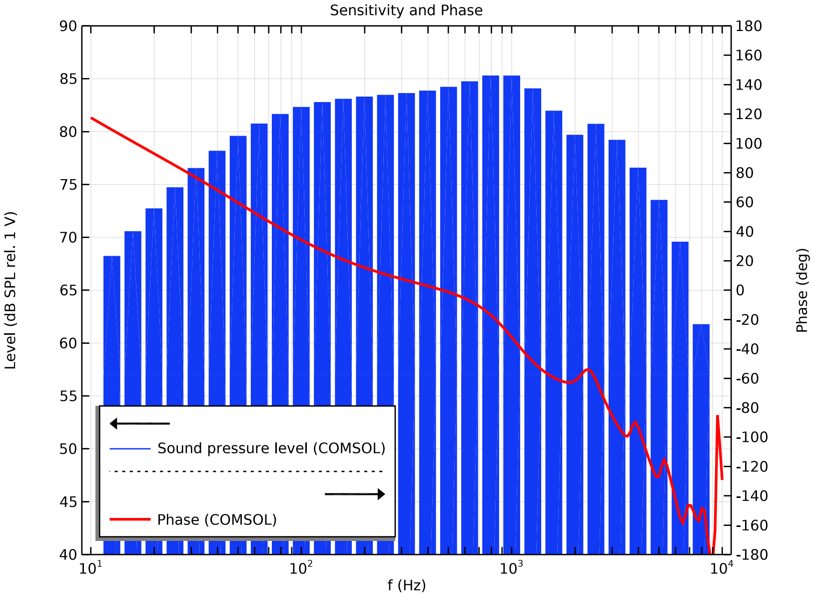
Global 1

1. In the **Model Builder** window, right-click **Sensitivity and Phase** and choose **Paste Global**.
2. In the **Settings** window for **Global**, click to expand the **Coloring and Style** section.
3. From the **Width** list, choose **2**.
4. From the **Color** list, choose **Red**.

Sensitivity and Phase

1. In the **Model Builder** window, click **Sensitivity and Phase**.
2. In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3. Select the **Two y-axes** checkbox.
4. In the table, select the **Plot on secondary y-axis** checkbox for **Global 1**.
5. Locate the **Axis** section. Select the **Manual axis limits** checkbox.
6. In the **y minimum** text field, type `40`.
7. In the **y maximum** text field, type `90`.
8. In the **Secondary y minimum** text field, type `-180`.
9. In the **Secondary y maximum** text field, type `180`.
10. Locate the **Plot Settings** section.
11. Select the **Secondary y-axis label** checkbox. In the associated text field, type `Phase (deg)`.

12 In the **Sensitivity and Phase** toolbar, click **Plot**.



It is time to solve the model with the closed back-volume configuration. After solving add a few extra graphs to the existing plots to see how the system response has changed. The closed back-volume acts as a spring and thus modifies the mechanical compliance of the system. The effect is, for example, seen in the electric impedance plot by moving the resonance up in frequency.

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** > **Frequency Domain**.
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: Frequency Domain

1. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
2. Click **Range**.

3. In the **Range** dialog, choose **ISO preferred frequencies** from the **Entry method** list.
4. In the **Start frequency** text field, type `10`.
5. In the **Stop frequency** text field, type `10000`.
6. From the **Interval** list, choose **1/24 octave**.
7. Click **Replace**.
8. In the **Model Builder** window, click **Study 2**.
9. In the **Settings** window for **Study**, type `Study 2 - Closed Back-Volume` in the **Label** text field.
10. Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.
11. In the **Study** toolbar, click = **Compute**.

Finally, modify two of the existing plots to understand the effect of a closed-back volume. To see the results of the closed configuration, in any of the plots, simply change the **Dataset** to the one from study 2.

RESULTS

Global 1

1. In the **Model Builder** window, under **Results** > **Voice-Coil Impedance** click **Global 1**.
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/cir.R1.i)`	`ohm`	`|Z| (open back-volume)`

Global 2

1. Right-click **Results** > **Voice-Coil Impedance** > **Global 1** and choose **Duplicate**.
2. In the **Settings** window for **Global**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 2 - Closed Back-Volume/Solution 2 (sol2)**.
4. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`abs(V0/cir.R1.i)`	`ohm`	`|Z| (closed back-volume)`

5. Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.
6. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 In the **Voice-Coil Impedance** toolbar, click **Plot**.

This plot should reproduce Figure 12.

Octave Band 1

1 In the **Model Builder** window, under **Results** > **Sensitivity at 1 m** click **Octave Band 1**.

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

3 In the table, enter the following settings:

Legends
`Sound pressure level (COMSOL) (open back-volume)`

Octave Band 3

1 Right-click **Results** > **Sensitivity at 1 m** > **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 - Closed Back-Volume/Solution 2 (sol2)**.

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

Legends
`Sound pressure level (COMSOL) (closed back-volume)`

Graph Marker 1

1 In the **Model Builder** window, expand the **Octave Band 3** node, then click **Graph Marker 1**.

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

3 From the **Anchor point** list, choose **Lower left**.

4 In the **Sensitivity at 1 m** toolbar, click **Plot**.

This plot should reproduce Figure 13.