Created in COMSOL Multiphysics 5.5

Cylindrical Subwoofer

Introduction

This is an introduction to modeling with the Acoustics Module. The step-by-step instructions take you through the process of setting up a model of the sound field created by a cylindrical down-firing subwoofer placed on a floor. The important result of the simulation is the contribution to the system's mechanical impedance induced by the coupling between the speaker membrane and the air inside and outside the speaker enclosure. A polar plot of the exterior radiation pattern as well as a directivity plot are also presented to demonstrate the uniform intensity distribution of the subwoofer.

Model Definition

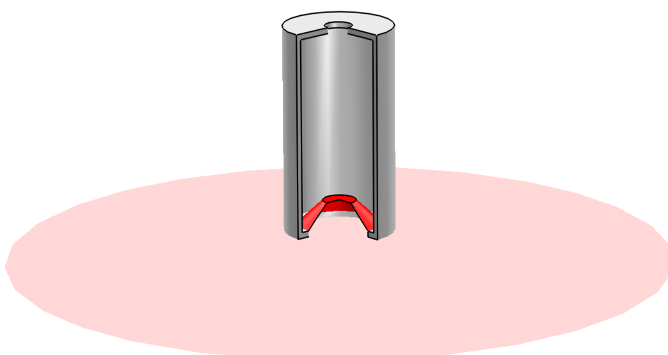


Figure 1: The geometry of the subwoofer with a slice removed to show its interior. In the illustration the floor is cut off but in the model it is assumed to extend to infinity.

Figure 1 shows the geometry simulated in this model. A down-firing cylindrical subwoofer is placed on a floor bounding an infinite half-space. The subwoofer enclosure has a height of 0.7 m and a diameter of 0.4 m. The bottom of the subwoofer is dominated by a 12-inch (0.3 m diameter) membrane. The top has a vent with a diameter of 0.1 m.

The membrane vibrates harmonically with the acceleration $a = a_0 e^{i\omega t}$ where $a_0 = 10\ \mathrm{m/s^2}$ and $\omega = 2\pi f$ is the angular frequency (rad/s). The frequencies considered are $f = 50$–70 Hz, which are in the high end of the frequency range that subwoofers are typically used for. It is assumed that the walls of the enclosure are perfectly rigid. The acoustic medium is air.

The feet that the speaker would need to stand on are judged to have a negligible effect on the sound field. With this assumption, all geometric features and physics have a rotational symmetry with respect to the axis of the speaker. This makes it natural to set up the model in a 2D axisymmetric physics interface.

Results and Discussion

By assuming the membrane is inflexible, you can write Newton's second law for the voice coil and membrane as

$$ma = F_{\text{ext}} + F_{\text{emf}} + F_m + F_a$$

where the forces acting on the system have been split into four components: the applied external force F_{ext}, an electromagnetic force opposing the movement of the coil F_{emf}, a mechanical part F_m independent of the acoustic environment (elastic behavior of the suspensions, spider and surround in a speaker), and a term F_a containing only the effects of the fluid loading on the membrane. The external force is typically proportional to the applied voltage, while the latter three contributions are proportional to the velocity and directed to oppose the movement. Introducing corresponding mechanical impedances (with proper normal directions assumed), and using the time-harmonic assumption, it holds that

$$F_{\text{ext}} = v(i\omega m + Z_{\text{emf}} + Z_m + Z_a)$$

If you know the three impedances as functions of frequency, you can predict the system's electrical and mechanical behavior. The electrical force can be measured or simulated with a blocked voice coil, while Z_m is directly related to the mass of the voice coil and membrane as well as to the stiffness of the baffle keeping the voice coil centered. These contributions are therefore independent of the environment in which the speaker operates.

The acoustic contribution to the mechanical impedance Z_a, on the other hand, depends on the enclosure geometry and on reflecting surfaces in the immediate surroundings. In Figure 2 you can study the real and imaginary parts of Z_a as functions of frequency in the simulated range. The real part represents radiation and acts as a resistance in the electromechanical system while the imaginary part represents a reactance.

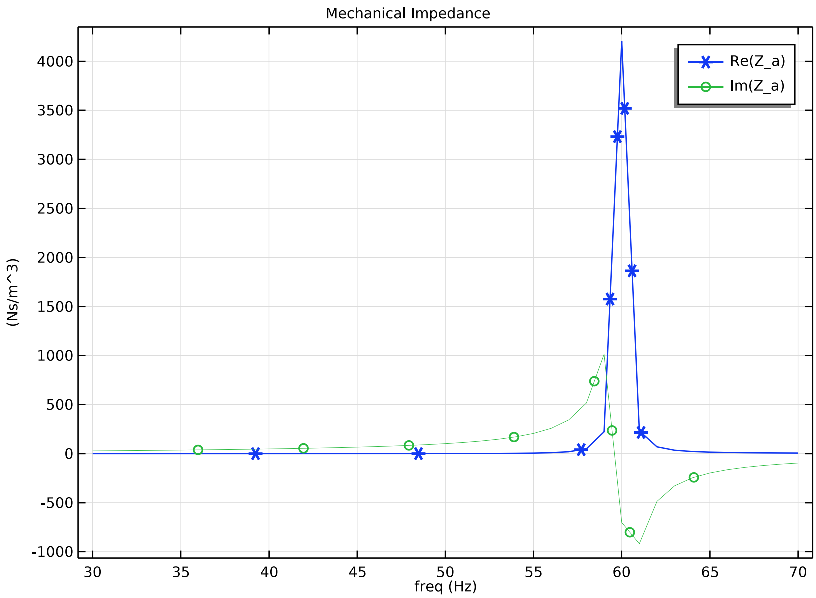


Figure 2: The resistive (real) and reactive (imaginary) parts of the contribution of the air to the mechanical impedance of the speaker.

Note that the reactance switches sign from positive to negative at approximately 60 Hz. This means that at frequencies below this visible resonance, the surrounding air acts as an added mass on the membrane, while at higher frequencies its action is spring-like.

Figure 3 displays the exterior-field sound pressure level in the *rz*-plane as a polar plot evaluated at 2 m from the speaker.

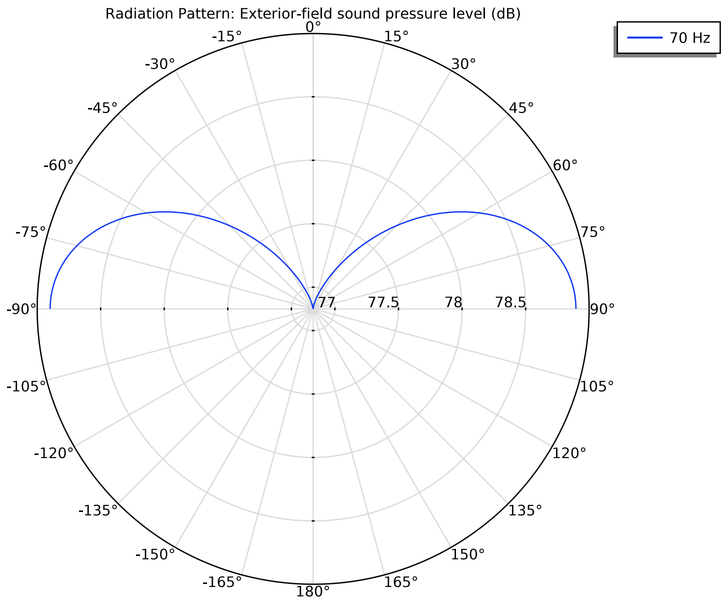


Figure 3: Exterior-field sound level in dB at a distance of 2 m from the floor below the subwoofer, 0^0 corresponds to the vertical z-direction. Note the 77 dB offset. The large wavelength compared to the size of the speaker system makes the response rather uniform.

Notes About the COMSOL Implementation

This model is set up in 2D axisymmetry using the Pressure Acoustics, Frequency Domain interface, which is described in the *Acoustics Module User's Guide*. The modeled physical domain is a hemisphere with a radius of 1 m. To minimize the effect of nonphysical reflections at the exterior boundary of this domain, an absorbing perfectly matched layer (PML) is added outside of it. For more information about PMLs see the *COMSOL Multiphysics Reference Manual*.

The membrane is modeled with zero thickness. The *Interior Normal Acceleration* boundary condition allows the pressure field to be discontinuous across the membrane.

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

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 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Frequency Domain**.
6 Click **Done**.

GEOMETRY 1

Create the geometry.

To simplify this step, you can insert a prepared geometry sequence. In the **Geometry** toolbar, click **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `cylindrical_subwoofer.mph`. Click **Build All** in the **Geometry** toolbar. Then, continue with the instruction after the geometry plot below.

Otherwise, proceed with the following instructions to create the geometry from scratch:

Begin by drawing the air domain and the surrounding PML.

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 `1.5`.
4 In the **Sector angle** text field, type `90`.
5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.5

6 Click **Build Selected**.

Next, add the walls of the subwoofer.

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 `0.02`.

4 In the **Height** text field, type `0.66`.

5 Locate the **Position** section. In the **r** text field, type `0.18`.

6 In the **z** text field, type `0.12`.

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 `0.05`.

4 In the **Height** text field, type `0.02`.

5 Locate the **Position** section. In the **r** text field, type `0.15`.

6 In the **z** text field, type `0.1`.

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 `0.15`.

4 In the **Height** text field, type `0.02`.

5 Locate the **Position** section. In the **r** text field, type `0.05`.

6 In the **z** text field, type `0.78`.

Difference 1 (dif1)

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

2 Select the object **c1** only to add it to the **Objects to add** list.

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

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

5 Select the objects **r1**, **r2**, and **r3** only.

6 Click **Build All Objects**.

Finally, create the membrane.

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. From the **Data source** list, choose **Vectors**.
5. In the **r** text field, type `0 0.06 0.06 0.15`.
6. In the **z** text field, type `0.2 0.2 0.2 0.12`.

GLOBAL DEFINITIONS

This model uses a parameter to represent the membrane's peak acceleration and variables for the total force on the membrane and the mechanical impedance of the membrane-air system. An integral operator takes part in the force computation.

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
a0	10[m/s^2]	10 m/s²	Peak acceleration

DEFINITIONS

Integration 1 (intop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, type `mem_int` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 4 and 9 only.
5. Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Variables 1

1. In the **Definitions** toolbar, click **Local Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
F_a	mem_int(2*pi*r*nz*(down(p)-up(p)))	N	Net force on membrane
Z_a	-F_a/(a0/acpr.iomega)	N·s/m	Mechanical impedance

Perfectly Matched Layer 1 (pml1)

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

2 Select Domain 2 only.

Note that the default PML center coordinate (0,0) applies because the spherical mantle that it constitutes is centered at the origin. The evanescent part of the wave leaving from the vent is expected to decay with a characteristic length much shorter than the wavelength.

ADD MATERIAL

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

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

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

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

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Interior Normal Acceleration 1

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

2 Select Boundaries 4 and 9 only.

3 In the **Settings** window for **Interior Normal Acceleration**, locate the **Interior Normal Acceleration** section.

4 Specify the $\mathbf{a}_0$ vector as

0	r
a0	z

The membrane boundaries are considered interior because they have a pressure distribution on both sides. The Sound Hard Boundary (Wall) condition applies per default to all the remaining exterior boundaries. This includes the surface of the enclosure.

Exterior Field Calculation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.
2 Select Boundary 18 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**.

The hard floor is excluded from the integral by noting that the radiation pattern is exactly the same as if replacing the floor with a mirror image of the subwoofer that is instead included in the integral. The exterior-field integral is set to use the full integral version; this means that the exterior field can be evaluated in any point outside the computational selection. This option will give the full information about both amplitude and phase, not only in the mathematical far-field.

MESH 1

In acoustic models, the wavelength must be resolved by the mesh. In addition, small features in the geometry can induce high local pressure gradients that must be resolved properly to obtain a consistent global solution. In this particular model, the wavelength is greater than 5 m, so the focus is on resolving the geometry. Use a mapped mesh in the PML domain.

Free Triangular 1

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

Size

1 In the **Settings** window for **Size**, locate the **Element Size** section.
2 From the **Predefined** list, choose **Finer**.

Free Triangular 1

1 In the **Model Builder** window, 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 Domain 1 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. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 4, 6, and 9 only.
5. Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.
6. Click the **Custom** button.
7. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
8. In the associated text field, type `1e-3`.
9. Select the **Maximum element growth rate** check box.
10. Click **Build All**.

Mapped 1

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

STUDY 1

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. In the **Frequencies** text field, type `range(30,1,70)`.

   This gives you frequencies from 30 to 70 Hz with a pitch of 1 Hz.
4. In the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

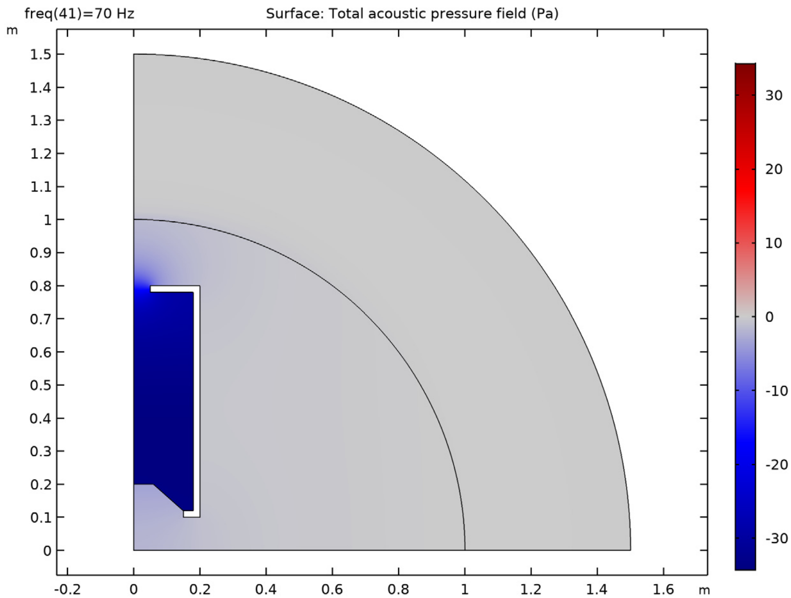
The first default plot shows the pressure field for the last parameter value, 70 Hz. The postprocessing algorithms automatically try to smooth discontinuous fields. You can see this effect if you zoom in on the membrane, where the field really should be discontinuous. To turn off the smoothing, follow these steps:

Surface 1

1. In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, click to expand the **Quality** section.

3 From the **Smoothing** list, choose **None**.

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



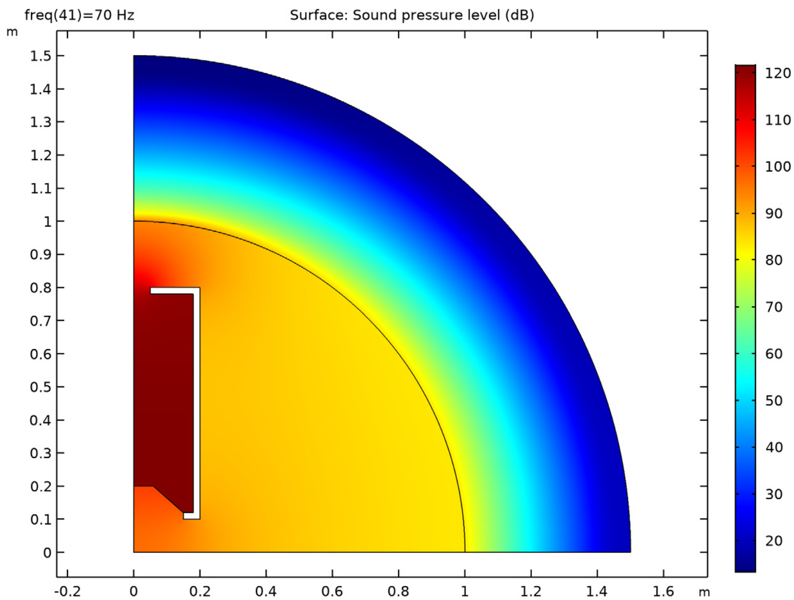
Note that the pressure drops rapidly toward zero in the PML domain.

For a better view of the damping, look at the second default plot, which shows the sound pressure level in dB.

Sound Pressure Level (acpr)

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

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



With a visual inspection you can see that the pressure drop in the PML is roughly 80 dB. This means that the part of the wave that is reflected at the exterior boundary experiences a total of 160 dB damping before it returns to the physical domain. In other words, the reflected wave has virtually no effect on the solution.

Study 1/Solution 1 (sol1)

For a 3D representation of the sound pressure level, look at the fourth default plot.

First create a selection to hide the PML domain:

Selection

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

2 Right-click **Study 1/Solution 1 (sol1)** and choose **Selection**.

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

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

5 Select Domain 1 only.

Contour 1

1 In the **Model Builder** window, right-click **Sound Pressure Level, 3D (acpr)** and choose **Contour**.

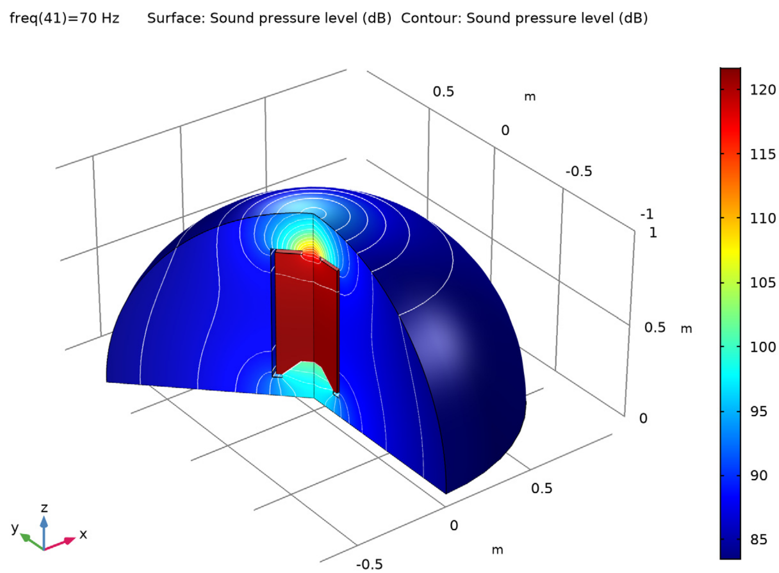
2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level - dB**.

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

4 From the **Color** list, choose **White**.

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

6 In the **Sound Pressure Level, 3D (acpr)** toolbar, click **Plot**.



Exterior-Field Sound Pressure Level (acpr)

The last two default plots represent the exterior-field sound pressure level and the exterior-field pressure evaluated at 1 m in the rz-plane (change the evaluation to 2 m, to be outside the computational domain). Look at the first of those to check that the exterior-field pressure distribution is indeed as uniform as you would expect from a subwoofer.

Initially the spatial response is depicted for all azimuthal angles (0 deg to 360 deg). The radiation-pattern plot has the angle defined such that 0 deg is along the z-axis; the reference direction is per default in the z-axis direction.

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 Find the **Evaluation distance** subsection. In the **Radius** text field, type `2`.

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

The plot indicates a maximum in the vicinity of 79 dB along the floor and a minimum of 77 dB straight above the speaker, and is depicted in Figure 3.

Proceed to display the previously defined impedance variable `Z_a` as function of frequency.

1D Plot Group 7

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

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

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

4 In the **Title** text area, type `Mechanical Impedance`.

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

6 In the associated text field, type `freq (Hz)`.

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

8 In the associated text field, type `(Ns/m^3)`.

Global 1

1 Right-click **Mechanical 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
real(Z_a)	N*s/m	Re(Z_a)
imag(Z_a)	N*s/m	Im(Z_a)

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 In the **Mechanical Impedance** toolbar, click **Plot**.

Your plot should be similar to Figure 2.