Created in COMSOL Multiphysics 5.6

Optimizing the Shape of a Horn

Introduction

This example shows how to apply boundary shape optimization to a simple axisymmetric horn. For the sake of simplicity, the far-field sound pressure level is maximized for a single frequency and in a single direction. The focus is on the optimization procedure, which involves choice of objective function, and optimization solver settings. The deformation of the geometry is introduced using the *Shape Optimization* functionality of COMSOL Multiphysics.

The model was inspired by the work of Erik Bängtsson, Daniel Noreland, and Martin Berggren (Ref. 1).

Note: This application requires the Acoustics Module and the Optimization Module.

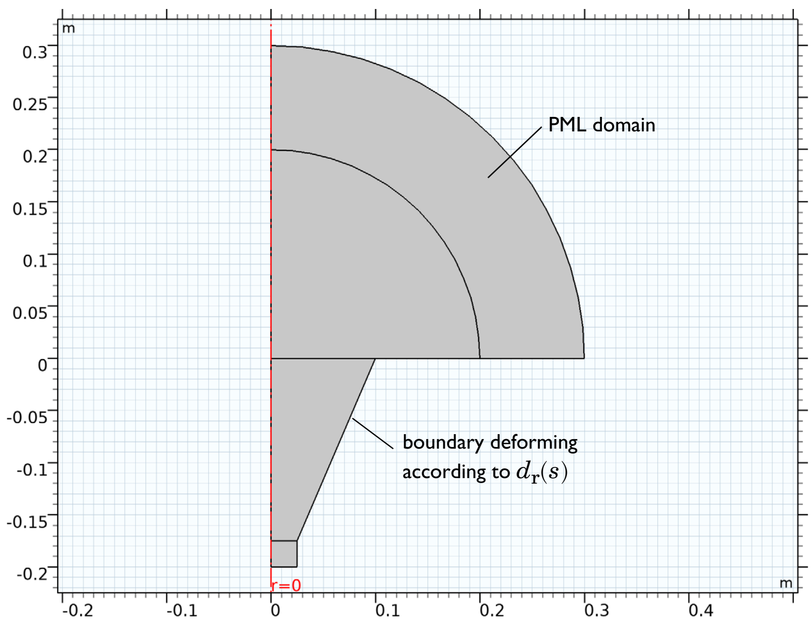


Figure 1: The initial configuration is a simple cone (the z<0 part of the gray area) in an infinite baffle.

Model Definition

A plane-wave mode feeds an axisymmetric horn radiating from an infinite baffle toward an open half space; see Figure 1. The radius of the feeding waveguide is assumed to be fixed, as well as the depth of the horn and the size of the hole where the horn is attached to the baffle. By varying the curvature of the initially conical surface of the horn, its directivity and impedance can be changed.

The surface is parameterized assuming that the radius and the z position of the horn deviates from the simple cone by a set of 8th-order Bernstein polynomials. The maximum displacement (in the two directions) is given by the d_{max} parameter. The number of optimization variables is determined by the order of the polynomial. Using a higher order gives more freedom and potentially a better final value of the objective function, but it also makes the optimization process more sensitive and can generate a shape that is less suitable for production.

Optimization can only be applied to real-valued functions because the minimum of a complex-valued function is not well-defined. But the raw result from a frequency-domain acoustics simulation is a complex-valued pressure field. From this you generate a scalar, real-valued quantity to be used as the *objective function* in the optimization process. However, any operation which converts a complex number to a real value is necessarily non-analytical, which means that its derivative is not uniquely defined.

The gradient-based optimization solver in the Optimization Module by default evaluates derivatives of the objective function via the solution of an *adjoint equation*. This procedure requires that the symbolic derivative of any non-analytic function is selected in a special way. The default behavior of the composite functions abs(z) and conj(z), which are most commonly used to obtain a real-valued objective function, is to return a derivative parallel to the real axis. However, this behavior is not appropriate for the adjoint method, where you instead need the definitions

$$\begin{aligned}\frac{d}{dz}|z| &= \frac{\bar{z}}{|z|}\\ \frac{d}{dz_1}(z_1\overline{z_2}) &= \overline{z_2}\\ \frac{d}{dz_2}(z_1\overline{z_2}) &= \overline{z_1}\end{aligned} \qquad (1)$$

It is indeed possible to redefine the symbolic derivatives of built-in functions in COMSOL Multiphysics, but in this case it is more convenient to use the special function realdot(z_1, z_2), which evaluates as real(z_1·conj(z_2)) but differentiates according to Equation 1. In particular, as a measure of the transmission properties of the horn, use an expression of the form realdot(p_m, p_m)/p_0^2, where p_m is the pressure measured at a specific point in front of the horn and p_0 is the (real-valued and constant) amplitude of the incoming wave.

If you choose to evaluate p_m in the near-field, or can afford to include a sufficiently large domain in front of the horn to effectively measure a far-field value at a point in the model, you can simply measure p_m as the local pressure in a geometry vertex. However, in order to optimize the directivity pattern in an efficient way, p_m should be defined using an integral representation of the exterior-field pressure as a function of the angle from the axis. Typically, for speaker it will be the pressure or SPL evaluated at a 1 m distance.

COMSOL Multiphysics contains optimized code for evaluating such exterior field integrals. This is, however, a pure postprocessing feature that does not support the automatic differentiation required by the adjoint method. Therefore, return to the definition of the *Helmholtz-Kirchhoff integral* as given in its axisymmetric form. See the *Theory Background for the Pressure Acoustics Module* section in the *Acoustics Module User's Guide* for further details. To simplify things we will use the expression of the exterior field in the far-field limit.

$$
\begin{aligned}
p_{\mathrm{far}}(\mathbf{R}) \cong -\frac{1}{2}\int_S r e^{ik\frac{zZ}{|\mathbf{R}|}}\Big[J_0\Big(\frac{krR}{|\mathbf{R}|}\Big)\nabla p(\mathbf{r})\cdot\mathbf{n} - \\
\frac{ikp(\mathbf{r})}{|\mathbf{R}|}\Big(in_r R J_1\Big(\frac{krR}{|\mathbf{R}|}\Big) + n_z Z J_0\Big(\frac{krR}{|\mathbf{R}|}\Big)\Big)\Big]dS \\
\mathbf{R} = (R, Z) \qquad |\mathbf{R}| \to \infty
\end{aligned}
\tag{2}
$$

If the infinite baffle is placed at $z = 0$, its effect is the same as that of adding a mirror image of the horn and at the same time removing the baffle. If, in addition, the integration surface is taken to be the wide end of the horn, in the plane of the baffle, most of the terms in Equation 2 cancel out, and all that is left is

$$
p_m(\theta) = \frac{1}{L_{\mathrm{far}}}\left|\int_S J_0(kr\sin(\theta))\frac{\partial p}{\partial z}\frac{1}{2\pi}dr\right| \tag{3}
$$

where J_0 is the Bessel function of the first kind of order 0, L_{far} is the far-field evaluation distance, and the angle θ from the axis has been introduced as a parameter. This integral is easily implemented in COMSOL Multiphysics using an integration operator.

In this way the metric for the optimization is based on the far-field limit of the exterior field, which is valid for distances larger than the Rayleigh radius given by

$$R_0 = \frac{1}{2}ka^2 = 0.45 \text{ m at } f = 5000 \text{ Hz}$$

Optimization as a rule implies many evaluations of the model for different designs, which can be very time consuming. In addition, the solver can be asked to evaluate each design at a number of frequencies and optimize with respect to the sum of the objective function values for each frequency. In this tutorial, a single frequency of 5000 Hz has been selected in order to make it possible to experiment with other aspects of the model. For example, changing the parameter θ, you can easily study the effect on the horn shape of optimizing the output at a specified angle from the axis.

Results and Discussion

By changing the shape of the horn within the limits of the selected parameterization, the on-axis sound pressure level can be raised by about 1 dB compared to the simple cone in Figure 1.

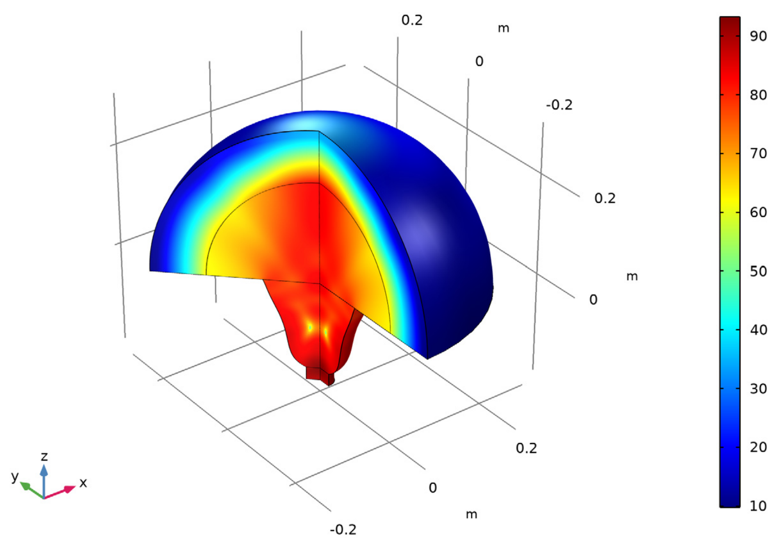


Figure 2: The final shape of the horn, optimized for on-axis SPL at 5000 Hz.

The improvement is rather small, because the initial design also shows a marked directivity, as can be seen from Figure 3. Obviously, the optimal shape with respect to on-axis SPL leads to deep undesirable minima in other directions.

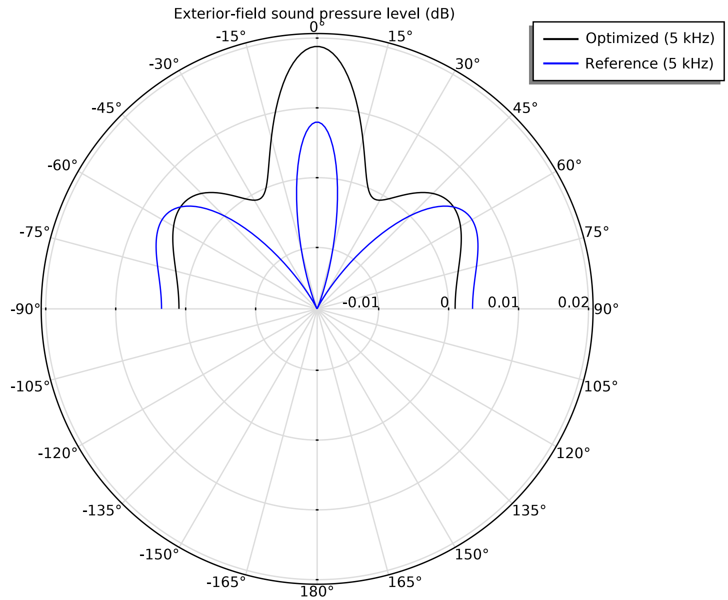


Figure 3: Radiation plot of the original (dashed blue) and final (solid black) designs.

Optimizing with respect to a slight off-axis direction can give you a more uniform far-field pattern, but may also result in a deep minimum on the axis. Try for example to set the off-axis angle θ to 22° (in the parameters list set `theta` equal to `22[deg]`).

Varying the frequency reveals that the on-axis improvement is robust over a wide frequency band; see Figure 4.

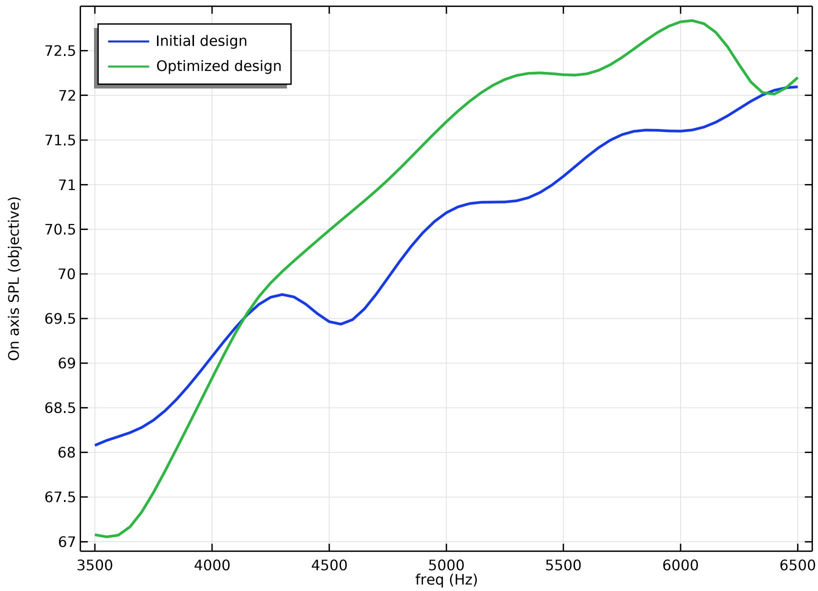


Figure 4: The objective function is plotted as a function of the frequency for the initial as well as the optimized design.

To search for a stable and practically useful horn design, you might instead create a composite objective function as a weighted sum of transmission values evaluated for a number of discrete directions, or choose to minimize the deviation from the mean SPL over a range of angles. In addition, you would also want to optimize with respect to more than one frequency, and experiment with different parameterizations.

Notes About the COMSOL Implementation

COMSOL Multiphysics implements the parameterization as a prescribed boundary displacement in the Moving Mesh feature. The mesh is allowed to move freely in the conical part of the horn, but otherwise kept fix. Some measures must be taken to avoid inverted elements when the shape of the cone is changed. Firstly, a quad mesh is used, since quads are less likely to become inverted, compared to triangles.

Secondly, the amplitude of the boundary displacement is restricted by the d_{max} parameter and the polynomial order. These constraints are intended to keep the mesh element volumes positive at all times and must not be active at the optimum point. You perform this sanity check as a final postprocessing step.

A time-harmonic Pressure Acoustics, Frequency Domain interface solves for the pressure field inside the horn and in a small spherical domain surrounding its opening. The air domain is terminated by a spherical PML layer which absorbs outgoing waves in such a way that the artificial termination of the domain has no influence on the near field. An accurate near field is sufficient, since the far-field result is based on an integral representation evaluated in the plane of the baffle. A plane wave radiation boundary condition on the waveguide attached to the narrow end of the horn feeds the horn with a plane wave of amplitude 1 Pa.

Set up the optimization problem in the **Optimization** study node. The pressure measured by an integration coupling variable, according to Equation 3, is inserted into a scalar objective function equal to the SPL value, or $10 \cdot \log_{10}(0.5 \cdot \mathrm{realdot}(p_\mathrm{m}, p_\mathrm{m})/(20 \cdot 10^{-6})^2)$.

Reference

1. E. Bängtsson, D. Noreland, and M. Berggren, "Shape Optimization of an Acoustic Horn," *Technical Report 2002-019*, Department of Information Technology, Uppsala University, May 2002.

Application Library path: `Acoustics_Module/Optimization/horn_shape_optimization`

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

Square 1 (sq1)

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

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

3 In the **Side length** text field, type `0.025`.

4 Locate the **Position** section. In the **z** text field, type `-0.2`.

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 `0.3`.

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	0.1

6 Click **Build Selected**.

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

Polygon 1 (pol1)

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

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

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

4 In the **r** text field, type `0 0.1 0.1 0.025 0.025 0`.

5 In the **z** text field, type `0 0 0 -0.175 -0.175 -0.175`.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
theta	0[deg]	0 rad	Polar angle
f0	5[kHz]	5000 Hz	Optimization frequency
df	50[Hz]	50 Hz	Optimization frequency bandwidth
Lfar	1[m]	1 m	Far-field evaluation distance

DEFINITIONS

Integration 1 (intop1)

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

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

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

4 Select Boundary 6 only.

Variables 1

1 In the **Definitions** toolbar, click a= **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
pm	intop1(besselj(0,acpr.k*r*sin(theta))*pz/2/pi/Lfar)	N/m²	Measured pressure

In the definition of pm, intop1() is the name of your integration operator, r is the radial coordinate, acpr.k is the local wave number, theta is the observation angle, and pz is the derivative of the pressure with respect to the z-coordinate.

Global Variable Probe 1 (var1)

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type obj in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type 10*log10(0.5*realdot(pm,pm)/acpr.pref_SPL^2).

Perfectly Matched Layer 1 (pml1)

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

2 Select Domain 4 only.

Now set up the shape optimization using a **Free Shape Domain** and a **Polynomial Boundary** feature.

Free Shape Domain 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Free Shape Domain**.
2 Select Domain 2 only.

Polynomial Boundary 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Polynomial Boundary**.
2 Select Boundary 9 only.
3 In the **Settings** window for **Polynomial Boundary**, locate the **Polynomial** section.
4 In the n text field, type 8.

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)

Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Pressure Acoustics, Frequency Domain (acpr)** and choose **Port**.
2 Select Boundary 2 only.
3 In the **Settings** window for **Port**, locate the **Port Properties** section.
4 From the **Type of port** list, choose **Circular**.
5 Locate the **Incident Mode Settings** section. In the A^{in} text field, type 1.

   This gives you a plane wave with the amplitude 1 Pa propagating in the positive z direction

The default boundary condition is sound hard, which is appropriate for the horn surface and the baffle, and does no harm at the PML domain’s outer boundary. The PML and the incident plane-wave condition therefore fully specify the physics of the model. In order to prepare for the results processing, add an Exterior-Field Calculation node.

Exterior Field Calculation 1

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

2 Select Boundary 6 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**.

5 From the **Type of integral** list, choose **Far-field integral approximation for r-> infinity**.

MESH 1

Run the model at a frequency of 5 kHz, corresponding to a wavelength of just under 7 cm. Using the standard at-least-six-elements-per-wavelength rule, a maximum element size of 1 cm seems like a good choice. However, the shape wavelength along the edge of the cone should also be resolved, and the maximum wavelength is therefore set to 0.5 cm. A quad mesh is in general more resistant to element warping when the mesh is deformed. Therefore, use an unstructured quad mesh everywhere except in the PMLs, which perform better with a mapped mesh aligned with the radial and tangential directions.

Free Triangular 1

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

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

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

4 Select Domains 1–3 only.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.005`.

Mapped 1

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

Distribution 1

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

2 Select Boundary 7 only.

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

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

5 Click **Build All**.

STUDY 1 - REFERENCE SOLUTION

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

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

Before starting the actual optimization, it is good practice to check the model setup by solving once with the default parameters. This way, you can also study the reference state on which you intend to improve.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - Reference Solution** 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(3500,50,6500)`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

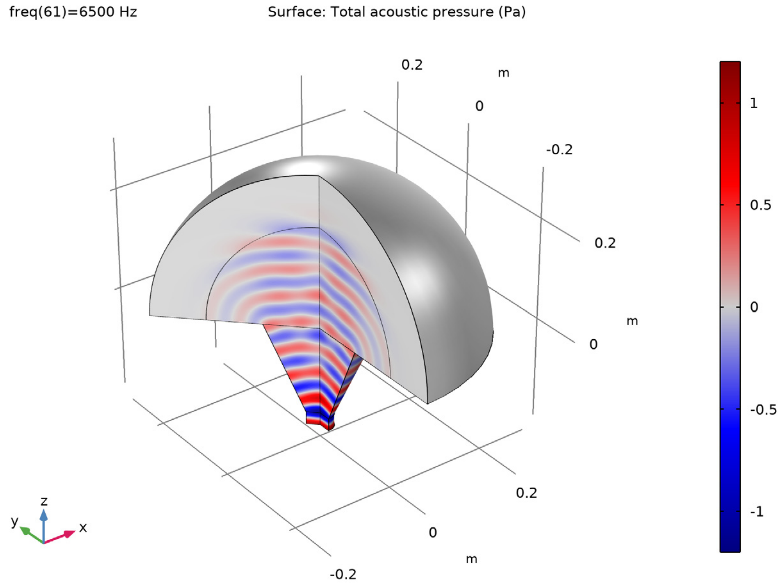
5 In the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure, 3D (acpr)

The default plot in the main window shows the distribution of the instantaneous pressure in the physical domain and the PML. Note that the pressure near the outer boundary of the PML is practically zero. This has no physical relevance, but indicates that the PML is doing a good job absorbing the sound.

1 Click the ![icon] **Zoom Extents** button in the **Graphics** toolbar.



Probe Plot Group 8, Shape Optimization

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Shape Optimization** and **Probe Plot Group 8**.

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

Objective Validation

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

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

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

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Objective Validation** 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
subst(acpr.efc1.Lp_pext,r,0,z,Lfar)	dB	Exterior field

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>obj - Global Variable Probe 1**.

5 In the **Objective Validation** toolbar, click **Plot**.

Acoustic Pressure (acpr), Acoustic Pressure, 3D (acpr), Exterior-Field Pressure (acpr), Exterior-Field Sound Pressure Level (acpr), Objective Validation, Sound Pressure Level (acpr), Sound Pressure Level, 3D (acpr)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr)**, **Sound Pressure Level (acpr)**, **Acoustic Pressure, 3D (acpr)**, **Sound Pressure Level, 3D (acpr)**, **Exterior-Field Sound Pressure Level (acpr)**, **Exterior-Field Pressure (acpr)**, and **Objective Validation**.

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

Reference solution

In the **Settings** window for **Group**, type `Reference solution` in the **Label** text field.

ROOT

Next, add a new study for the optimization.

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 **Add Study** 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 In the **Frequencies** text field, type `f0-df/2 f0 f0+df/2`.

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

4 In the **Settings** window for **Study**, type `Study 2 - Optimized Solution` in the **Label** text field.

Shape Optimization

1 Right-click **Study 2 - Optimized Solution** and choose **Optimization>Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 Clear the **Move limits** check box.

4 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>comp1.obj - Global Variable Probe 1**.

Set up a MaxMin problems such that frequency associated with the worst objective function is prioritized.

5 Locate the **Objective Function** section. From the **Type** list, choose **Maximization**.

6 From the **Solution** list, choose **Minimum of objectives**.

Solution 2 (sol2)

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

By making the nonlinear tolerance stricter than that of the optimization solver, you ensure that the optimization does not fail because each solution is not sufficiently converged. An optimality tolerance of 1e-4 is still stricter than the accuracy of this low-resolution finite element model.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2 - Optimized Solution>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1** node, then click **Stationary 1**.

4 In the **Settings** window for **Stationary**, locate the **General** section.

5 In the **Relative tolerance** text field, type `1e-6`.

6 From the **Linearity** list, choose **Nonlinear**.

7 In the **Model Builder** window, expand the **Study 2 - Optimized Solution>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1>Stationary 1** node, then click **Fully Coupled 1**.

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

9 In the **Minimum damping factor** text field, type `1e-4`.

10 Right-click **Study 2 - Optimized Solution>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1>Stationary 1>Fully Coupled 1** and choose **Compute**.

RESULTS

Since the PML does not add any physical information it can be excluded for clarity.

Probe Solution 2 (sol2)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Probe Solution 2 (sol2)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

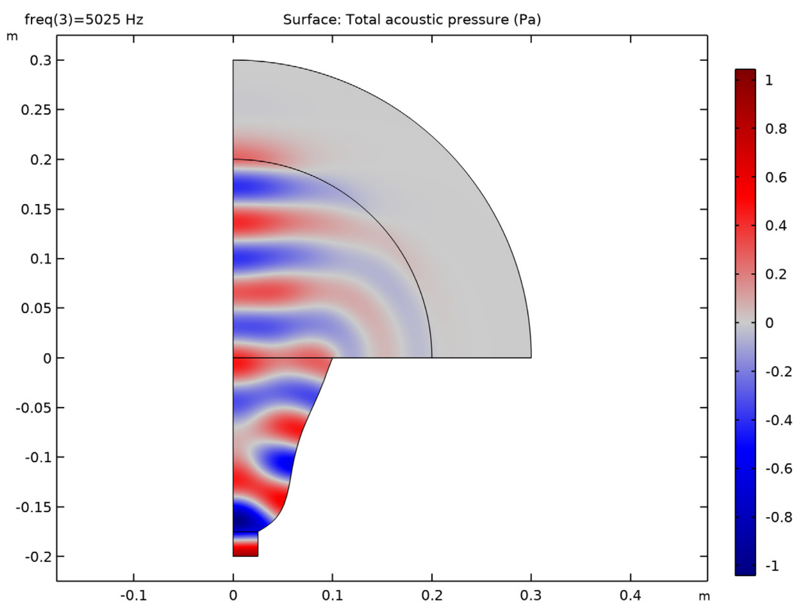
2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

4 Select Domains 1–3 only.

Acoustic Pressure (acpr) 1

The first default plot is the acoustic pressure. It should look like the image below.



Sound Pressure Level, 3D (acpr) 1

1 In the **Model Builder** window, click **Sound Pressure Level, 3D (acpr) 1**.

2 In the **Sound Pressure Level, 3D (acpr) 1** toolbar, click **Plot**.

Your plot of the sound pressure level should now look like Figure 2.

To see a direct comparison of the exterior-field polar pattern before and after optimization, modify the second to last default plot. This is an exterior-field plot of the sound pressure level in the *rz*-plane. Modify it to only plot the results in the positive half plane ($z > 0$), increase the resolution, and change some Coloring and Style options. The resulting plot of the exterior field should look like Figure 3. Note that 0 deg on the polar graph corresponds to the vertical *z*-axis.

Exterior-Field Pressure (acpr) 1

1 In the **Model Builder** window, click **Exterior-Field Pressure (acpr) 1**.

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

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

4 In the **Parameter indices (1-3)** text field, type `2`.

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

6 In the **Title** text area, type `Exterior-field sound pressure level (dB)`.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field Pressure (acpr) 1** 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 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Optimized (5 kHz)`

8 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Black**.

Radiation Pattern 2

1 Right-click **Results>Exterior-Field Pressure (acpr) 1>Radiation Pattern 1** and choose **Duplicate**.

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

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

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

5 In the **Parameter values (freq (Hz))** list, select **5000**.

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

Legends
`Reference (5 kHz)`

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

8 In the **Exterior-Field Pressure (acpr) 1** toolbar, click **Plot**.

Acoustic Pressure (acpr) 1, Acoustic Pressure, 3D (acpr) 1, Exterior-Field Pressure (acpr) 1, Exterior-Field Sound Pressure Level (acpr) 1, Shape Optimization, Sound Pressure Level (acpr) 1, Sound Pressure Level, 3D (acpr) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr) 1**, **Sound Pressure Level (acpr) 1**, **Acoustic Pressure, 3D (acpr) 1**, **Sound Pressure Level, 3D (acpr) 1**, **Exterior-Field Sound Pressure Level (acpr) 1**, **Exterior-Field Pressure (acpr) 1**, and **Shape Optimization**.

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

Optimized solution

In the **Settings** window for **Group**, type `Optimized solution` in the **Label** text field.

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 **Add Study** in the window toolbar.

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

STUDY 3

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `range(3500,50,6500)`.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

4 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 2 - Optimized Solution, Frequency Domain**.

7 In the **Model Builder** window, click **Study 3**.

8 In the **Settings** window for **Study**, type `Study 3 - Frequency Sweep (Optimized)` in the **Label** text field.

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

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

RESULTS

Optimized solution

Add **1D Plot Group** showing the objective function as a function of the frequency for the initial as well as the optimized design.

Response

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

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

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

4 Locate the **Plot Settings** section. Select the **y-axis label** check box.

5 In the associated text field, type `On axis SPL (objective)`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Response** 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>obj - Global Variable Probe 1**.

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

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

5 In the table, enter the following settings:

Legends
`Initial design`

Global 2

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

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

3 From the **Dataset** list, choose **Study 3 - Frequency Sweep (Optimized)/Solution 3 (sol3)**.

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

Legends
Optimized design

Probe Plot Group 15

In the **Model Builder** window, right-click **Probe Plot Group 15** and choose **Delete**.