Created in COMSOL Multiphysics 6.1

Tweeter Dome and Waveguide Shape Optimization

Introduction

Loudspeaker drivers convert electric signals into sound. Due to the large frequency range of human hearing, loudspeaker systems usually have different drivers tuned to reproduce different frequency bands of the audible spectrum. At the low-frequency end, there are *subwoofer* and *woofer* drivers that work as almost omnidirectional sources and have large areas to create high sound-pressure levels. As the frequency increases, effects like cone breakup (the excitation of structural modes that make the membrane displace in and out of phase in some areas) limit the size of the speaker. The beaming effect reduces the area where high sound pressure levels are achieved, this occurs as the frequency increases and the wavelength becomes comparable to the size of the speaker. This is why *tweeters* — drivers tuned for reproducing the high-frequency content of the hearing range — are usually small and light.

An ideal tweeter driver would have a flat sensitivity curve, meaning that the on-axis sound-pressure level achieved, at a given distance from the driver, for a constant voltage input is constant as a function of frequency. The response should be similar, when the listening point is not exactly in front of the speaker. Both cone breakup and beaming are effects that are inherent to the loudspeaker driver design, but it is possible to tune a driver so that these two effects cancel out through the frequency range and create a speaker that is closer to an ideal speaker.

This model performs shape optimization of the dome (or speaker membrane) and the waveguide of the loudspeaker. By changing the shape of these two components, it is possible to create a tweeter that behaves closer to an ideal tweeter all through the range of frequencies of interest.

This tutorial model illustrates:

- How to obtain the sensitivity and directivity of a loudspeaker using a Thiele–Small equivalent for the electromagnetic circuit
- How to combine the frequency and spatial response to create a combined optimization objective
- How this objective can be used effectively to improve the design of the loudspeaker
- How the near acoustic field can be used to approximate the exterior acoustic field enabling the use of gradient-based optimization algorithms
- How changes in the shape of a boundary can indicate more radical changes in the design

Model Definition

The model uses a Thiele–Small analogous circuit to include the electromagnetic characteristics of the driver. This approach is discussed in the tutorial Lumped Loudspeaker Driver. The suspension, voice coil, and dome are captured through the Solid Mechanics and Shell physics interfaces.

If the electromagnetic circuit of the tweeter is known, the two main elements that influence the radiation characteristics are the dome and the waveguide. The dome is the radiation element and creates sound as it moves back and forth. The waveguide helps to direct the sound to the front of the speaker. Figure 1 shows the main components of the tweeter.

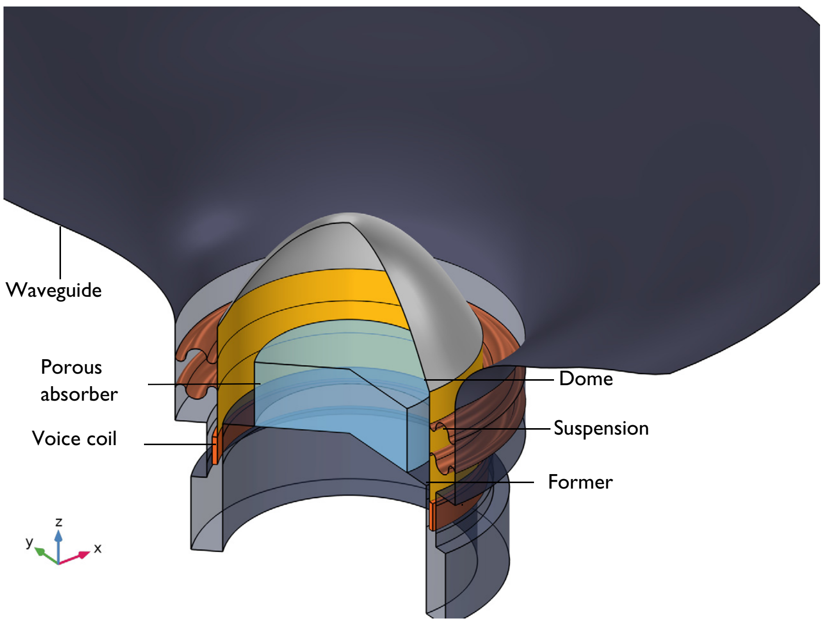

Figure 1: Geometry of the initial tweeter design.

Throughout the range of frequencies that a tweeter reproduces, it is almost unavoidable to excite different dynamic effects, like resonances or dome flexible modes. These effects, if undamped, will create a large variation of sensitivity just above and below the frequency where it is found. To avoid this variation and have a more constant sensitivity throughout the frequency range, foam inserts are sometimes added under the dome to add damping. This model includes one such foam insert and also includes some other sources of damping, like the losses in the narrow gaps around the voice coil and the structural damping of the dome and suspension.

The optimization objective is defined as the squared difference between the target and measured sound-pressure levels at four points in space. These points define the spatial extent where a uniform radiation is desired. The frequency at which the sum of these four squares is the largest, drives the optimization. This is a so-called min-max objective. A logarithmic function is applied to the objective function as the objective function is always positive. This improves convergence without introducing scaling of the objective.

In the model, the first study analyzes a traditional design to include as a reference. In the second study, a shape optimization problem is defined to obtain a tweeter that produces the same sound pressure level through the frequencies and angles of interest. The last study is an analysis of the optimized design through the complete frequency range and positions.

Results and Discussion

The on-axis sound pressure levels (SPL) or sensitivity for the initial and the optimized designs are shown in Figure 2. The target SPL is included as a dotted horizontal line. The plot also shows that there is a good agreement between the near acoustic field extrapolation (the values used in the optimization) and the actual exterior field evaluation.

Note how the initial design presents significant deviations from the target, while the optimized design produces a very flat response.

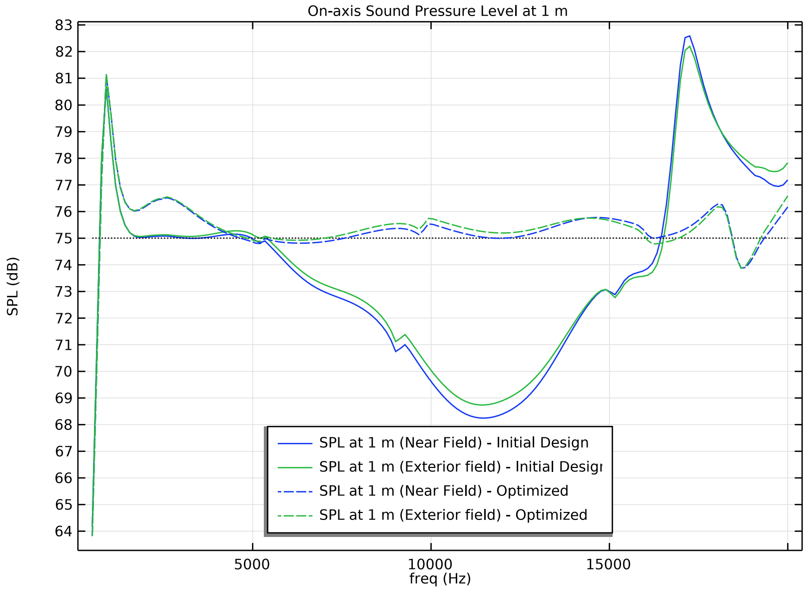

Figure 2: On-axis sound pressure level at 1 meter.

The shape optimization improves the objective by changing the shape of the dome and waveguide. Figure 3 shows the displacement produced in the dome and waveguide during the optimization. Note how the segment of the former that connects the dome and the suspension follows the same profile as the dome, indicating that this segment could be removed to make the dome directly connect to the suspension and the former. Note, as well, how the effective radius of the waveguide has been reduced, as the flat part of the waveguide is reached at a smaller radius. This has the advantage of limiting the space requirement of the tweeter while maintaining the sound-pressure level produced.

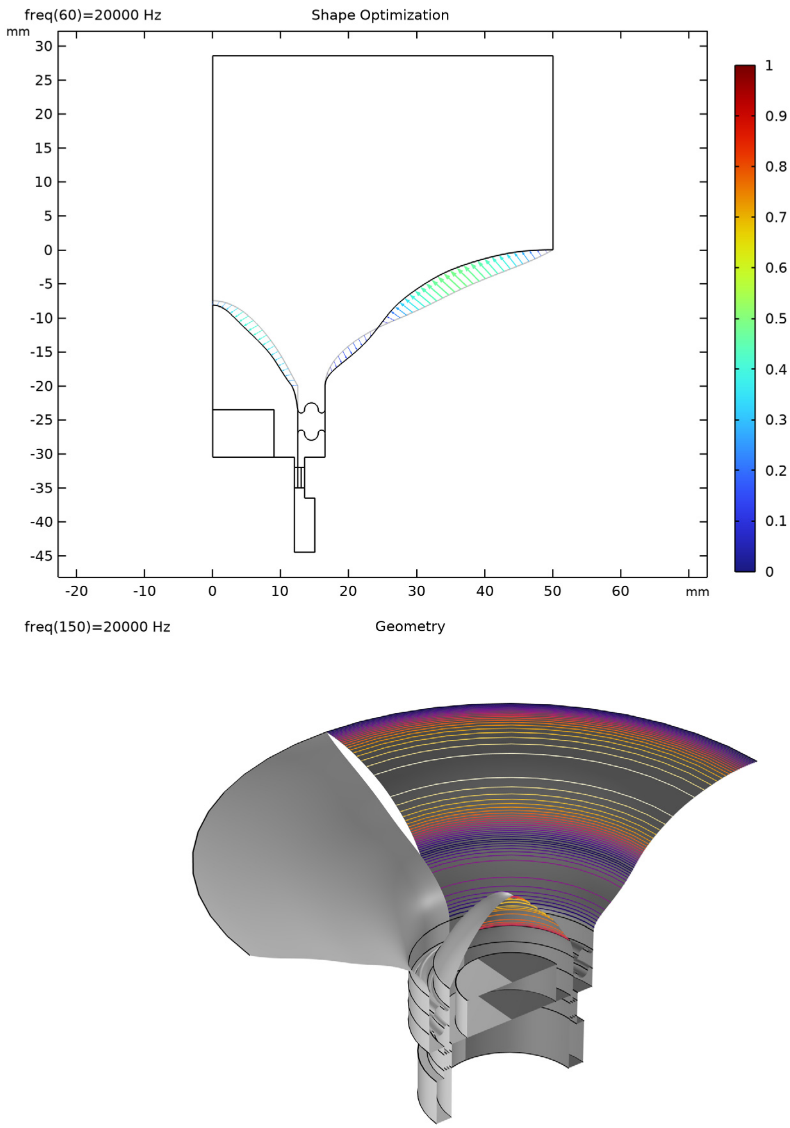

Figure 3: The optimization results (top) are shown together with the initial design (bottom).

Figure 4 shows the sound-pressure level at the maximum frequency for the initial and the optimized design. The geometry of the dome, former, and suspension is shown as a deformed black line. The change in direction between the dome and the former facilitates a more prominent cone breakup in the initial design (left part of the image).

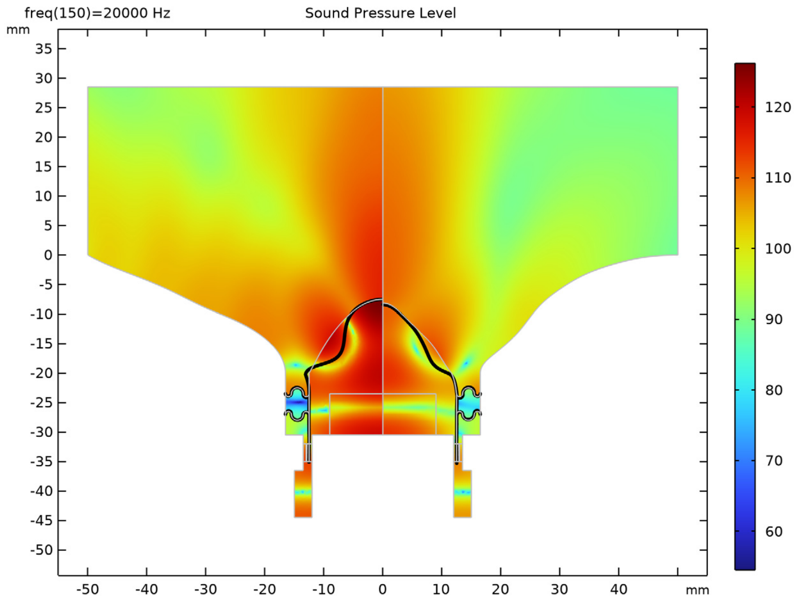

Figure 4: Sound-pressure level produced at the highest frequency and structural deformation on the dome, former, and suspension for the initial design (left) and optimized design (right).

The directivity of the initial and the optimized design is shown in Figure 5. The on-axis sound pressure level is within 3 dB through the frequency range in the optimized design and this on-axis behavior is maintained for the angles of interest.

This tutorial presents a possible method to optimize the performance of a tweeter speaker. Different design variables, constraints, frequencies of interest, or spatial distributions are also possible and can be modified. With the prominence of additive manufacturing the novel waveguides can, for example, be 3D printed. By using composite materials it is also possible to create the new shapes for the dome.

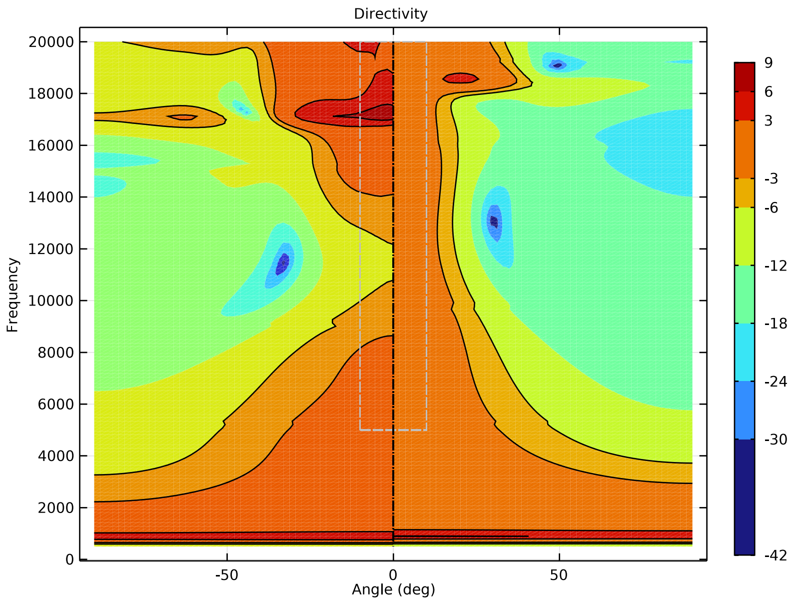

Figure 5: Directivity plot of the initial design (left) and optimized design (right). The colos show the deviation from the target sound pressure level. Black lines mark the ±3 dB and ±6 dB limits. The gray box marks the frequencies and locations used during the optimization.

Notes About the COMSOL Implementation

USING NEAR ACOUSTIC FIELD TO OBTAIN THE EXTERIOR FIELD

In order to use gradient-based optimization algorithms like MMA, the computation domain is artificially large to be able to use the near acoustic field to obtain the exterior field (the response at 1 m). This approach has to be compared to the results using the **Exterior Field Calculation** feature to check the validity of the assumption. In this case, the deviation is small so the approach is deemed as acceptable.

USING LOGARITHMIC FUNCTION TO TRANSFORM THE OBJECTIVE FUNCTION

In cases where the objective function is always positive, it is possible to transform it using a logarithmic function to overcome the need of an objective function scaling.

USING DOMAIN PROBES TO COMPUTE THE OPTIMIZATION OBJECTIVES

The objective of the optimization is based on a series of domain probes at the listening points (measurement points). The near acoustic field response is computed through these domain probes and extrapolated to the sensitivity distance of 1 m. These values are used in the optimization objective.

OPTIMIZING FOR A REDUCED NUMBER OF FREQUENCIES

The optimization study uses a smaller number of frequencies relative to the verification study. This is done to limit the computational time, but with COMSOL launched in cluster mode supports computation of the individual frequencies in parallel. With the appropriate hardware it is thus possible to include significantly more frequencies for the optimization without increasing the computational time.

Application Library path: `Acoustics_Module/Optimization/tweeter_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>Acoustic-Structure Interaction>Acoustic-Shell Interaction, Frequency Domain**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.
7 Click **Add**.
8 Click **Study**.
9 In the **Select Study** tree, select **General Studies>Frequency Domain**.
10 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

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

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

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

GLOBAL DEFINITIONS

Geometry Parameters

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

2 In the **Settings** window for **Parameters**, type `Geometry Parameters` in the **Label** text field.

Model Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

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

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

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

Thiele Small Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Thiele Small Parameters` in the **Label** text field.

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

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

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
L_E	(L_e/(sin(n_e*pi/2)))*(acpr.omega[s/rad])^(n_e-1)	H	Voice coil inductance (frequency dependent)
Rp_E	(L_e/(cos(n_e*pi/2)))*(acpr.omega[s/rad])^(n_e)[ohm/H]	Ω	Resistance (losses in magnetic system)
obj_1	(mic1-target_spl)^2+(mic2-target_spl)^2+(mic3-target_spl)^2+(mic4-target_spl)^2		Optimization objective

Domain Probe 1 (dom1)

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

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

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

4 In the **Variable name** text field, type `v0`.

5 Locate the **Expression** section. In the **Expression** text field, type `solid.u_tZ`.

Domain Point Probe 1

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

2 Select Domain 4 only.

3 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

4 In row **Coordinates**, set **z** to `r_eval`.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, type `mic1` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `acpr.Lp_t+10*log10((r_eval/1[m])^2)`.

4 Select the **Description** check box. In the associated text field, type `Microphone 1 SPL`.

Domain Point Probe 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **Domain Point Probe 1** and choose **Duplicate**.

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

3 In row **Coordinates**, set **r** to `r_eval*sin(angle_eval/2/4)`.

4 In row **Coordinates**, set **z** to `r_eval*cos(angle_eval/2/4)`.

Point Probe Expression 1 (ppb2)

1 In the **Model Builder** window, expand the **Domain Point Probe 2** node, then click **Point Probe Expression 1 (ppb2)**.

2 In the **Settings** window for **Point Probe Expression**, type `mic2` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Description** text field, type `Microphone 2 SPL`.

Domain Point Probe 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **Domain Point Probe 1** and choose **Duplicate**.

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

3 In row **Coordinates**, set **r** to `r_eval*sin(angle_eval/2/4*3)`.

4 In row **Coordinates**, set **z** to `r_eval*cos(angle_eval/2/4*3)`.

Point Probe Expression 1 (ppb3)

1 In the **Model Builder** window, expand the **Domain Point Probe 3** node, then click **Point Probe Expression 1 (ppb3)**.

2 In the **Settings** window for **Point Probe Expression**, type `mic3` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Description** text field, type `Microphone 3 SPL`.

Domain Point Probe 4

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **Domain Point Probe 1** and choose **Duplicate**.

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

3 In row **Coordinates**, set **r** to `r_eval*sin(angle_eval/2/4*4)`.

4 In row **Coordinates**, set **z** to `r_eval*cos(angle_eval/2/4*4)`.

Point Probe Expression 1 (ppb4)

1 In the **Model Builder** window, expand the **Domain Point Probe 4** node, then click **Point Probe Expression 1 (ppb4)**.

2 In the **Settings** window for **Point Probe Expression**, type `mic4` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Description** text field, type `Microphone 4 SPL`.

COMPONENT 1 (COMP1)

Free Shape Domain 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Shape Optimization> Free Shape Domain**.

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

3 Click **Clear Selection**.

4 Select Domains 2 and 3 only.

Free Shape Boundary 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Shape Optimization> Free Shape Boundary**.

2 In the **Settings** window for **Free Shape Boundary**, locate the **Boundary Selection** section.

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

4 Locate the **Filtering** section. From the R_{min} list, choose **Medium**.

5 Locate the **Control Variable Settings** section. In the d_{max} text field, type `disp_dome`.

Free Shape Boundary 2

1 In the **Definitions** toolbar, click **Optimization** and choose **Shape Optimization> Free Shape Boundary**.

2 In the **Settings** window for **Free Shape Boundary**, locate the **Boundary Selection** section.

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

4 Locate the **Filtering** section. From the R_{min} list, choose **Medium**.

5 Locate the **Control Variable Settings** section. In the d_{max} text field, type `disp_wave`.

Symmetry/Roller 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Shape Optimization> Symmetry/Roller**.

2 In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry/Roller Boundaries**.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **PML Domains**.

4 Locate the **Geometry** section. From the **Type** list, choose **Cylindrical**.

5 Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

6 Right-click **Perfectly Matched Layer 1 (pml1)** and choose **Materials>Browse Materials**.

MATERIAL BROWSER

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

2 From the Application Libraries root, browse to the folder `Acoustics_Module/Electroacoustic_Transducers` and double-click the file `loudspeaker_driver_materials.mph`.

3 Click **Done**.

ADD MATERIAL

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

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

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

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

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

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

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

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

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

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

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

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

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

MATERIALS

Glass Fiber (mat5)

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

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

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

Air (mat1)

1. In the **Model Builder** window, click **Air (mat1)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Air Domains**.

Composite (mat2)

1. In the **Model Builder** window, click **Composite (mat2)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Dome**.

Cloth (mat3)

1. In the **Model Builder** window, click **Cloth (mat3)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Suspension**.

Coil (mat4)

1. In the **Model Builder** window, click **Coil (mat4)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Voice Coil**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.
2. In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Air Domains**.

Narrow Region Acoustics 1

1. In the **Physics** toolbar, click **Domains** and choose **Narrow Region Acoustics**.
2. In the **Settings** window for **Narrow Region Acoustics**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Narrow Regions**.
4. Locate the **Duct Properties** section. From the **Duct type** list, choose **Slit**.
5. In the h text field, type `air_gap`.

Poroacoustics 1

1. In the **Physics** toolbar, click **Domains** and choose **Poroacoustics**.
2. In the **Settings** window for **Poroacoustics**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Porous Domain**.
4. Locate the **Porous Matrix Properties** section. From the R_f list, choose **User defined**. In the associated text field, type `rf_pa`.
5. From the **Constants** list, choose **Miki**.

Exterior Field Calculation 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.
2. In the **Settings** window for **Exterior Field Calculation**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Exterior Field**.
4. Locate the **Exterior Field Calculation** section. From the **Condition in the z =** z_0 **plane** list, choose **Symmetric/Infinite sound hard boundary**.

SHELL (SHELL)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.
2. In the **Settings** window for **Shell**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Shell Boundaries**.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Linear Elastic Material 1**.

Damping 1

1. In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2. In the **Settings** window for **Damping**, locate the **Damping Settings** section.
3. From the **Damping type** list, choose **Isotropic loss factor**.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 2

1. In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2. In the **Settings** window for **Damping**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Suspension**.

4 Locate the **Damping Settings** section. From the **Input parameters** list, choose **Damping ratios**.

5 In the f_1 text field, type `fmin`.

6 In the ζ_1 text field, type `damp_susp`.

7 In the f_2 text field, type `fmax`.

8 In the ζ_2 text field, type `damp_susp`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d_0 text field, type `th_dome`.

Thickness and Offset 2

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

2 In the **Settings** window for **Thickness and Offset**, locate the **Boundary Selection** section.

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

4 Locate the **Thickness and Offset** section. In the d_0 text field, type `th_former`.

Thickness and Offset 3

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

2 In the **Settings** window for **Thickness and Offset**, locate the **Boundary Selection** section.

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

4 Locate the **Thickness and Offset** section. In the d_0 text field, type `th_susp`.

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **Fixed Points**.

SOLID MECHANICS (SOLID)

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

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

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

Body Load 1

1 In the **Physics** toolbar, click **Domains** and choose **Body Load**.

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

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

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

0	r
BL*cir.R1_i	z

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **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`.

Voltage Source 2 (V2)

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	4
n	0

4 Locate the **Device Parameters** section. In the v_{src} text field, type `BL*v0`.

MULTIPHYSICS

Acoustic-Structure Boundary 2 (asb2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multiphysics** and choose **Acoustic-Structure Boundary**.

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

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

4 Locate the **Coupled Interfaces** section. From the **Structure** list, choose **Solid Mechanics (solid)**.

Solid-Thin Structure Connection 1 (sshc1)

1 In the **Model Builder** window, right-click **Multiphysics** and choose **Solid-Thin Structure Connection**.

2 In the **Settings** window for **Solid-Thin Structure Connection**, locate the **Connection Settings** section.

3 From the **Connection type** list, choose **Shared boundaries**.

MESH 1

Free Quad 1

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

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

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

4 Select Domains 1, 4, 7, 8, and 11 only.

Size 1

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

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

3 From the **Selection** list, choose **Narrow Regions**.

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

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

6 Select the **Maximum element size** check box. In the associated text field, type `air_gap`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `lam0/5`.

5 In the **Minimum element size** text field, type `r1_susp`.

6 Click **Build All**.

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 2, 3, 6, 9, and 10 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 From the **Selection** list, choose **Dome**.

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

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

7 Select the **Maximum element size** check box. In the associated text field, type `mesh_optim`.

Size 2

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

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

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

Mapped 1

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

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

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

4 From the **Selection** list, choose **PML Domains**.

Distribution 1

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

2 Select Boundaries 41 and 42 only.

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

4 In the **Number of elements** text field, type `6`.

5 Click **Build All**.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

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

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

4 Select Domains 3 and 4 only.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior Field**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `1`.

5 Click **Build All**.

STUDY 1 - INITIAL DESIGN

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

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

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

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - Initial Design** 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(fmin,(fmax-fmin)/(nf-1),fmax)`.

4 Click to expand the **Results While Solving** section. From the **Probes** list, choose **None**.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Deformed geometry (Component 1)** and **Shape Optimization (Component 1)**.

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

RESULTS

In the **Model Builder** window, expand the **Results** node.

Study 1 - Initial Design/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1 - Initial Design/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 From the **Selection** list, choose **Visualization Domains**.

Study 1 - Initial Design/Revolution

1 Right-click **Study 1 - Initial Design/Solution 1 (sol1)** and choose **Duplicate**.

2 In the **Model Builder** window, click **Study 1 - Initial Design/Solution 1 (2) (sol1)**.

3 In the **Settings** window for **Solution**, type `Study 1 - Initial Design/Revolution` in the **Label** text field.

Selection

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

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

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

4 From the **Selection** list, choose **Revolution Boundaries**.

Revolution 2D - Initial Design

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

2 In the **Settings** window for **Revolution 2D**, type `Revolution 2D - Initial Design` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Initial Design/Revolution (sol1)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `90`.

5 In the **Revolution angle** text field, type `90`.

6 Clear the **Add end caps if the revolution is not full** check box.

7 Click to expand the **Advanced** section. Select the **Define variables** check box.

Mirror 2D 1

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

2 In the **Settings** window for **Mirror 2D**, click to expand the **Advanced** section.

3 Select the **Define variables** check box.

On-axis Sound Pressure Level at 1 m

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

2 In the **Settings** window for **1D Plot Group**, type `On-axis Sound Pressure Level at 1 m` 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** check box. In the associated text field, type `freq (Hz)`.

6 Select the **y-axis label** check box. In the associated text field, type `SPL (dB)`.

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

Global 1

1 Right-click **On-axis Sound Pressure Level at 1 m** 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
mic1	dB	SPL at 1 m (Near Field) - Initial Design

4 In the **On-axis Sound Pressure Level at 1 m** toolbar, click **Plot**.

On-axis Sound Pressure Level at 1 m

In the **Model Builder** window, click **On-axis Sound Pressure Level at 1 m**.

Octave Band 1

1 In the **On-axis Sound Pressure Level 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[m])`.

5 Select the **Description** check box. In the associated text field, type `SPL at 1 m (Exterior field) - Initial Design`.

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

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 In the **On-axis Sound Pressure Level at 1 m** toolbar, click **Plot**.

Global 2

1 Right-click **On-axis Sound Pressure Level at 1 m** 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
target_spl	dB	

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

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

6 Click to expand the **Legends** section. Clear the **Show legends** check box.

7 In the **On-axis Sound Pressure Level at 1 m** toolbar, click **Plot**.

Directivity

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

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

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

Directivity 1

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

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

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

4 In the **Expression** text field, type `acpr.efc1.Lp_pext-target_spl`.

5 Locate the **Evaluation** section. Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.

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

7 In the ϕ **range** text field, type `90`.

8 Find the **Evaluation distance** subsection. In the **Radius** text field, type `1000`.

9 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

10 In the **Levels** text field, type `-42 -30 -24 -18 -12 -6 -3 3 6 9`.

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

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

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

Directivity 2

1 Right-click **Directivity 1** and choose **Duplicate**.
2 In the **Settings** window for **Directivity**, locate the **Levels** section.
3 In the **Levels** text field, type `-6 -3 3 6`.
4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
5 From the **Color** list, choose **Black**.
6 From the **Contour type** list, choose **Line**.
7 From the **Width** list, choose **1**.
8 In the **Directivity** toolbar, click **Plot**.

Sound Pressure Level

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Sound Pressure Level` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.
4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.
5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **Sound Pressure Level** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `acpr.Lp_t`.

Filter 1

1 Right-click **Surface 1** and choose **Filter**.
2 In the **Settings** window for **Filter**, locate the **Element Selection** section.
3 In the **Logical expression for inclusion** text field, type `mir1x<0`.
4 In the **Sound Pressure Level** toolbar, click **Plot**.

Line 1

1 In the **Model Builder** window, right-click **Sound Pressure Level** and choose **Line**.
2 In the **Settings** window for **Line**, locate the **Expression** section.
3 In the **Expression** text field, type `0`.
4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
5 From the **Color** list, choose **Gray**.

Filter 1

1 Right-click **Line 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `mir1x<0`.

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

Line 2

1 In the **Model Builder** window, under **Results>Sound Pressure Level** right-click **Line 1** and choose **Duplicate**.

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

3 From the **Line type** list, choose **Tube**.

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

Deformation 1

1 Right-click **Line 2** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type `10000`.

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

Geometry

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

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

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

Surface 1

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

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

3 In the **Expression** text field, type `0`.

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

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

Filter 1

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

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

3 In the **Logical expression for inclusion** text field, type `(rev1y>0)*(rev1x<0)`.

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

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 - OPTIMIZATION

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

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

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

Shape Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Shape Optimization**.

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

3 In the **Maximum number of iterations** text field, type `20`.

4 Locate the **Objective Function** section. From the **Solution** list, choose **Maximum of objectives**.

5 In the table, enter the following settings:

Expression	Description
`log10(comp1.obj_1)`	

6 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

Step 1: Frequency Domain

1 In the **Model Builder** window, 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(fmin_optim,(fmax_optim-fmin_optim)/(nf_optim-1),fmax_optim)`.

4 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Study 2 - Optimization /Solution 2 (sol2)

In the **Model Builder** window, under **Results>Datasets** click **Study 2 - Optimization / 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 From the **Selection** list, choose **Visualization Domains**.

Shape Optimization

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Optimization / Solution 2 (sol2)**.

4 In the **Shape Optimization** toolbar, click **Plot**.

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

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

7 Locate the **Plot Settings** section. From the **Color** list, choose **Gray**.

8 From the **Frame** list, choose **Geometry (Rg, PHIg, Zg)**.

Line 1

1 Right-click **Shape Optimization** and choose **Line**.

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

3 In the **Expression** text field, type `1`.

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

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

Arrow Line 1

1 In the **Model Builder** window, right-click **Shape Optimization** and choose **Arrow Line**.

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

3 In the **r-component** text field, type `fsd1.dRg`.

4 In the **z-component** text field, type `fsd1.dZg`.

5 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

6 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

7 From the **Arrow base** list, choose **Head**.

Color Expression 1

1 Right-click **Arrow Line 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `fsd1.rel_disp`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `1`.

6 In the **Shape Optimization** toolbar, click **Plot**.

STUDY 2 - OPTIMIZATION

Shape Optimization

1 In the **Model Builder** window, under **Study 2 - Optimization** click **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Shape Optimization**.

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

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(fmin,(fmax-fmin)/(nf-1),fmax)`.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Deformed geometry (Component 1)** and **Shape Optimization (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 - Optimization , Frequency Domain**.

7 Locate the **Results While Solving** section. From the **Probes** list, choose **None**.

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

9 In the **Settings** window for **Study**, locate the **Study Settings** section.

10 Clear the **Generate default plots** check box.

11 In the **Label** text field, type `Study 3 - Optimized Design`.

12 In the **Home** toolbar, click **= Compute**.

RESULTS

Probe Solution 4 (sol3)

In the **Model Builder** window, under **Results>Datasets** right-click **Probe Solution 4 (sol3)** and choose **Delete**.

Study 3 - Optimized Design/Solution 3 (sol3)

In the **Model Builder** window, under **Results>Datasets** click **Study 3 - Optimized Design/Solution 3 (sol3)**.

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 From the **Selection** list, choose **Visualization Domains**.

Study 3 - Optimized Design/Revolution

1 Right-click **Study 3 - Optimized Design/Solution 3 (sol3)** and choose **Duplicate**.

2 In the **Settings** window for **Solution**, type `Study 3 - Optimized Design/Revolution` in the **Label** text field.

Selection

1 In the **Model Builder** window, expand the **Results>Datasets>Study 3 - Optimized Design/Revolution (sol3)** node, then click **Selection**.

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

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

4 From the **Selection** list, choose **Revolution Boundaries**.

Revolution 2D - Optimized Design

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

2 In the **Settings** window for **Revolution 2D**, type `Revolution 2D - Optimized Design` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Optimized Design/ Revolution (sol3)**.

4 Locate the **Revolution Layers** section. In the **Revolution angle** text field, type `90`.

5 Clear the **Add end caps if the revolution is not full** check box.

6 Locate the **Advanced** section. Select the **Define variables** check box.

Global 3

1 In the **Model Builder** window, under **Results>On-axis Sound Pressure Level at 1 m** 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 - Optimized Design/Solution 3 (sol3)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
mic1	dB	SPL at 1 m (Near Field) - Optimized

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Cycle (reset)**.

Octave Band 2

1 In the **Model Builder** window, under **Results>On-axis Sound Pressure Level at 1 m** right-click **Octave Band 1** and choose **Duplicate**.

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

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

4 Locate the **y-Axis Data** section. In the **Description** text field, type `SPL at 1 m (Exterior field) - Optimized`.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

Directivity 3

1 In the **Model Builder** window, under **Results>Directivity** right-click **Directivity 1** and choose **Duplicate**.

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

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

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the ϕ **start** text field, type 0.

5 Locate the **Coloring and Style** section. Select the **Color legend** check box.

Directivity 4

1 In the **Model Builder** window, under **Results>Directivity** right-click **Directivity 2** and choose **Duplicate**.

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

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

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the ϕ **start** text field, type 0.

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

Line Segments 1

1 In the **Model Builder** window, right-click **Directivity** and choose **Line Segments**.

2 In the **Settings** window for **Line Segments**, locate the **x-Coordinates** section.

3 In the table, enter the following settings:

Expression	Unit	Description
-angle_eval/2	deg	
angle_eval/2	deg	
angle_eval/2	deg	
-angle_eval/2	deg	
-angle_eval/2	deg	

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

Expression	Unit	Description
fmin_optim	Hz	Minimum frequency optimized for
fmin_optim	Hz	Minimum frequency optimized for
fmax_optim	Hz	Maximum frequency optimized for
fmax_optim	Hz	Maximum frequency optimized for
fmin_optim	Hz	Minimum frequency optimized for

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

7 From the **Width** list, choose **1**.

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

Line Segments 2

1 Right-click **Directivity** and choose **Line Segments**.

2 In the **Settings** window for **Line Segments**, locate the **x-Coordinates** section.

3 In the table, enter the following settings:

Expression	Unit	Description
0	1	
0	1	

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

Expression	Unit	Description
fmin	Hz	Minimum frequency analyzed
fmax	Hz	Maximum frequency analyzed

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

6 From the **Color** list, choose **Black**.

7 From the **Width** list, choose **1**.

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

Surface 2

1 In the **Model Builder** window, right-click **Sound Pressure Level** and choose **Surface**.

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `acpr.Lp_t`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

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

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

Line 3

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

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `0`.

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

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

Line 4

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

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `0`.

5 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Line 2**.

Deformation 1

Right-click **Line 4** and choose **Deformation**.

Surface 2

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

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

3 From the **Dataset** list, choose **Revolution 2D - Optimized Design**.

4 Locate the **Expression** section. In the **Expression** text field, type `0`.

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

6 From the **Color** list, choose **Custom**.

7 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

8 Click **Define custom colors**.

9 Set the RGB values to 105, 105, and 105, respectively.

10 Click **Add to custom colors**.

11 Click **Show color palette only** or **OK** on the cross-platform desktop.

Filter 1

1 Right-click **Surface 2** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(rev2y>0)*(rev2x>0)`.

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

Line 1

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

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

3 From the **Dataset** list, choose **Revolution 2D - Optimized Design**.

4 Locate the **Expression** section. In the **Expression** text field, type `0`.

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

6 From the **Color** list, choose **Black**.

7 In the **Geometry** toolbar, click **Plot**.

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

Contour 1

1 Right-click **Geometry** and choose **Contour**.

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

3 From the **Dataset** list, choose **Revolution 2D - Optimized Design**.

4 Locate the **Expression** section. In the **Expression** text field, type `fsd1.rel_disp`.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Thermal>HeatCamera** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Contour**, locate the **Coloring and Style** section.

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

10 In the **Geometry** toolbar, click **Plot**.

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

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 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

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

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

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 r_model+th_pml.

4 In the **Height** text field, type h_model+th_pml.

5 Click to expand the **Layers** section. Locate the **Position** section. In the **z** text field, type -h_former+h_voice_coil/2-h_top_plate/2-h_back-h_waveguide.

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

Layer name	Thickness (mm)
Layer 1	th_pml

7 Select the **Layers to the right** check box.

8 Clear the **Layers on bottom** check box.

9 Select the **Layers on top** check box.

10 Click **Build Selected**.

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 th_voice_coil+2*air_gap.

4 In the **Height** text field, type h_voice_coil.

5 Locate the **Position** section. In the **r** text field, type d_voice_coil/2-air_gap.

6 In the **z** text field, type -h_former-h_waveguide.

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

Layer name	Thickness (mm)
Layer 1	air_gap

8 Select the **Layers to the left** check box.

9 Select the **Layers to the right** check box.

10 Clear the **Layers on bottom** check box.

11 Click **Build Selected**.

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 (mm)	z (mm)
0	-h_former+h_voice_coil/2+h_top_plate/2-h_waveguide
d_voice_coil/2-air_gap	-h_former+h_voice_coil/2+h_top_plate/2-h_waveguide
d_voice_coil/2-air_gap	-h_former+h_voice_coil/2-h_top_plate/2-h_back-h_waveguide
d_magnet/2	-h_former+h_voice_coil/2-h_top_plate/2-h_back-h_waveguide
d_magnet/2	-h_former+h_voice_coil/2-h_top_plate/2-h_waveguide
d_voice_coil/2+th_voice_coil+air_gap	-h_former+h_voice_coil/2-h_top_plate/2-h_waveguide
d_voice_coil/2+th_voice_coil+air_gap	-h_former+h_voice_coil/2+h_top_plate/2-h_waveguide
d_voice_coil/2+4*r1_susp+2*r2_susp	-h_former+h_voice_coil/2+h_top_plate/2-h_waveguide
d_voice_coil/2+4*r1_susp+2*r2_susp	-h_waveguide

5 Click **Build Selected**.

Interpolation Curve 1 (ic1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Interpolation Curve**.

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

3 In the table, enter the following settings:

r (mm)	**z (mm)**
0	d1-h_waveguide
(d_voice_coil/2)*1/5	d2-h_waveguide
(d_voice_coil/2)*2/5	d3-h_waveguide
(d_voice_coil/2)*3/5	d4-h_waveguide
(d_voice_coil/2)*4/5	d5-h_waveguide
(d_voice_coil/2)*5/5	-h_waveguide

4 Locate the **End Conditions** section. From the **Condition at starting point** list, choose **Tangent direction**.

5 In the **r** text field, type `cos(alpha1)`.

6 In the **z** text field, type `-sin(alpha1)`.

7 From the **Condition at endpoint** list, choose **Tangent direction**.

8 In the **r** text field, type `cos(alpha2)`.

9 In the **z** text field, type `-sin(alpha2)`.

10 Click **Build Selected**.

Interpolation Curve 2 (ic2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Interpolation Curve**.

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

3 In the table, enter the following settings:

r (mm)	**z (mm)**
d_voice_coil/2+4*r1_susp+2*r2_susp	-h_waveguide
d_voice_coil/2+4*r1_susp+2*r2_susp+(d_waveguide/2-(d_voice_coil/2+4*r1_susp+2*r2_susp))*1/5	w1-h_waveguide
d_voice_coil/2+4*r1_susp+2*r2_susp+(d_waveguide/2-(d_voice_coil/2+4*r1_susp+2*r2_susp))*2/5	w2-h_waveguide
d_voice_coil/2+4*r1_susp+2*r2_susp+(d_waveguide/2-(d_voice_coil/2+4*r1_susp+2*r2_susp))*3/5	w3-h_waveguide

r (mm)	z (mm)
d_voice_coil/2+4*r1_susp+2*r2_susp+(d_waveguide/2-(d_voice_coil/2+4*r1_susp+2*r2_susp))*4/5	w4-h_waveguide
d_voice_coil/2+4*r1_susp+2*r2_susp+(d_waveguide/2-(d_voice_coil/2+4*r1_susp+2*r2_susp))*5/5	0

4 Locate the **End Conditions** section. From the **Condition at starting point** list, choose **Tangent direction**.

5 In the **r** text field, type `cos(beta1)`.

6 In the **z** text field, type `sin(beta1)`.

7 From the **Condition at endpoint** list, choose **Tangent direction**.

8 In the **r** text field, type `cos(beta2)`.

9 In the **z** text field, type `sin(beta2)`.

10 Click **Build Selected**.

Former

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

2 In the **Settings** window for **Line Segment**, type `Former` in the **Label** text field.

3 On the object **ic1**, select Point 2 only.

4 Locate the **Endpoint** section. Find the **End vertex** subsection. Click to select the **Activate Selection** toggle button.

5 On the object **r2**, select Point 3 only.

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

7 Click **Build Selected**.

Line Segment 2 (ls2)

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

2 On the object **ic2**, select Point 2 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

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

5 In the **r** text field, type `r_model+th_pml`.

6 Click **Build Selected**.

Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **r** text field, type `d_voice_coil/2+r1_susp`.

4 In the **z** text field, type `(-h_former+h_voice_coil/2+h_top_plate/2)*2/3-h_waveguide`.

5 Locate the **Radius** section. In the **Radius** text field, type `r1_susp`.

6 Locate the **Angles** section. In the **End angle** text field, type `180`.

7 Click **Build Selected**.

Circular Arc 2 (ca2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **r** text field, type `d_voice_coil/2+2*r1_susp+r2_susp`.

4 In the **z** text field, type `(-h_former+h_voice_coil/2+h_top_plate/2)*2/3-h_waveguide`.

5 Locate the **Radius** section. In the **Radius** text field, type `r2_susp`.

6 Locate the **Angles** section. In the **Start angle** text field, type `-180`.

7 In the **End angle** text field, type `0`.

8 Click **Build Selected**.

Circular Arc 3 (ca3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **r** text field, type `d_voice_coil/2+3*r1_susp+2*r2_susp`.

4 In the **z** text field, type `(-h_former+h_voice_coil/2+h_top_plate/2)*2/3-h_waveguide`.

5 Locate the **Radius** section. In the **Radius** text field, type `r1_susp`.

6 Locate the **Angles** section. In the **End angle** text field, type `180`.

7 Click **Build Selected**.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **ca1**, **ca2**, and **ca3** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **z** text field, type (-h_former+h_voice_coil/2+h_top_plate/2)/2 -h_waveguide.

6 Locate the **Normal Vector to Line of Reflection** section. In the **r** text field, type 0.

7 In the **z** text field, type 4.

8 Click **Build Selected**.

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 (mm)	z (mm)
0	r_model/4
d_voice_coil/2+4*r1_susp+2*r2_susp+(d_waveguide/2-(d_voice_coil/2+4*r1_susp+2*r2_susp))	r_model/4
d_voice_coil/2+4*r1_susp+2*r2_susp+(d_waveguide/2-(d_voice_coil/2+4*r1_susp+2*r2_susp))	0

5 Click **Build Selected**.

Porous Domain

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

2 In the **Settings** window for **Rectangle**, type Porous Domain in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type d_porous/2.

4 In the **Height** text field, type h_porous.

5 Locate the **Position** section. In the **z** text field, type -h_former+h_voice_coil/2+h_top_plate/2-h_waveguide.

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

7 Click **Build Selected**.

Union 1 (uni1)

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

2 Select the object **r1** only.

3 Click the **Select All** button in the **Graphics** toolbar.

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

Delete Entities 1 (del1)

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

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

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

4 On the object **uni1**, select Domains 1, 13, and 14 only.

5 Click **Build Selected**.

PML Domains

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **del1**, select Domains 5, 12, and 13 only.

Voice Coil

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **del1**, select Domain 8 only.

Air Domains

1 In the **Geometry** toolbar, click **Selections** and choose **Complement Selection**.

2 In the **Settings** window for **Complement Selection**, type `Air Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Voice Coil** in the **Selections to invert** list.

5 Click **OK**.

Narrow Regions

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **del1**, select Domains 7 and 11 only.

Visualization Domains

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **del1**, select Domains 1–3, 6, 7, and 9–11 only.

Exterior Field

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

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

4 On the object **del1**, select Boundaries 9 and 39 only.

Dome

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

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

4 On the object **del1**, select Boundary 46 only.

Symmetry/Roller Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Symmetry/Roller Boundaries` in the **Label** text field.

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

4 On the object **del1**, select Boundaries 3 and 5 only.

Suspension

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

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

4 On the object **del1**, select Boundaries 47–52 only.

Revolution Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

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

4 On the object **del1**, select Boundaries 2, 4, 11–15, 17, 19–25, 28–36, and 46–53 only.

Shell Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Shell Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Former**, **Dome**, and **Suspension**.

6 Click **OK**.

Waveguide

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

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

4 On the object **del1**, select Boundary 53 only.

Fixed Points

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Point**.

4 On the object **del1**, select Points 31 and 32 only.