Created in COMSOL Multiphysics 6.3

Shape Optimization of an Acoustic Demultiplexer

Introduction

This model shows how shape optimization can be used to design an acoustic demultiplexer. A demultiplexer is a data distributing device, in this case it will distribute acoustic energy. The geometry consists of a circular domain with one input port and two output ports. The domain has the structure of a sonic crystal, it has 19 circular cavities which are deformed such that the energy goes to one output port for one frequency band and to another output port for another frequency band.

This model requires both the Acoustics Module and the Optimization Module.

Model Definition

In this model all boundaries are modeled as sound hard except for the three ports as shown for the initial geometry in Figure 1. A number of circular cavities are introduced and it is the shape of these that will be optimized.

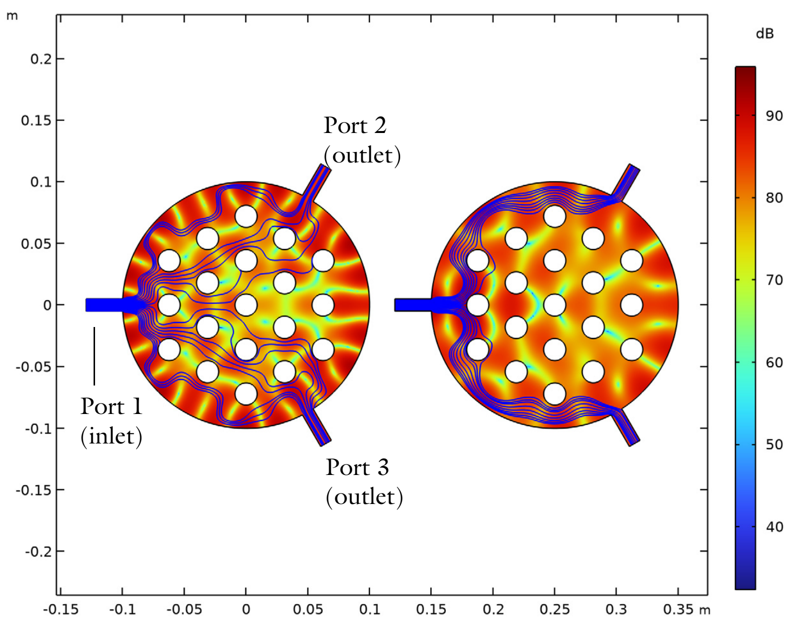


Figure 1: The sound pressure level distribution for the initial geometry is plotted together with blue lines, indicating the acoustic power flow from the input port to the left to the output ports to the right. The left figure is for 550 Hz and the right figure is for 750 Hz.

To set up the model, use the **Pressure Acoustics, Frequency Domain** interface together with the **Free Shape Domain** and **Free Shape Boundary** features.

DOMAIN EQUATIONS

The default Pressure Acoustics feature models harmonic sound waves in the domain by means of the Helmholtz equation for sound pressure:

$$\nabla \cdot \left(-\frac{1}{\rho_c}\nabla p\right) - \frac{\omega^2 p}{\rho_c c^2} = 0$$

Here the acoustic pressure is a harmonic quantity, $p = p_0 e^{i\omega t}$, and p is the pressure (SI unit: N/m^2), ρ_c is the density (kg/m^3), ω is the angular frequency (SI unit: rad/s), and c is the speed of sound (SI unit: m/s). Two frequency bands will be investigated as characterized by the three parameters shown in the table below.

TABLE 1: ACOUSTICS DOMAIN DATA.

QUANTITY	VALUE	DESCRIPTION
f_1	550 Hz	Frequency 1
f_2	750 Hz	Frequency 2
d_f	5 Hz	Frequency bandwidth

OPTIMIZATION SETUP

The cavities will be deformed using the **Free Shape Boundary** and **Free Shape Domain** features.

The objective function will be expressed as a MaxMin problem. This type of problem can be solved using the MMA optimization method.

$$\text{MinMax}(P_{\text{ratio}}) \quad , \text{ where } P_{\text{ratio}} = \begin{cases} P_{\text{port 2}}/P_{\text{port 3}}, & 2f < f_1 + f_2 \\ P_{\text{port 3}}/P_{\text{port 2}}, & \text{otherwise} \end{cases}$$

Where P_{port2} and P_{port3} are the output port powers, which can be computed using the built in port variables `acpr.port2.P_out` and `acpr.port3.P_out`. Similarly `freq` can be used to get the frequency, f. The governing equations will be solved for 5 frequencies around f_1 and f_2 for a total of 10 inner solutions. For some problems like this one, MMA can converge slowly. The number of maximum optimization iterations has been limited to 25 to limit the run time. A larger number of iterations will produce a better design but will increase the run time.

Results and Discussion

Figure 2 displays the sound pressure level (SPL) distribution for the optimized geometry at the center of both frequency bands (left 550 Hz and right 750 Hz). The figure also shows the power flow by plotting streamlines (blue lines) of the acoustic intensity field. This plot is made using the raw optimization results, but these can be verified by generating a mesh in the deformed configuration and running the simulation again as shown by means of a spectrum in Figure 3. This shows that we can achieve 35 dB power difference between the two ports across both frequency bands.

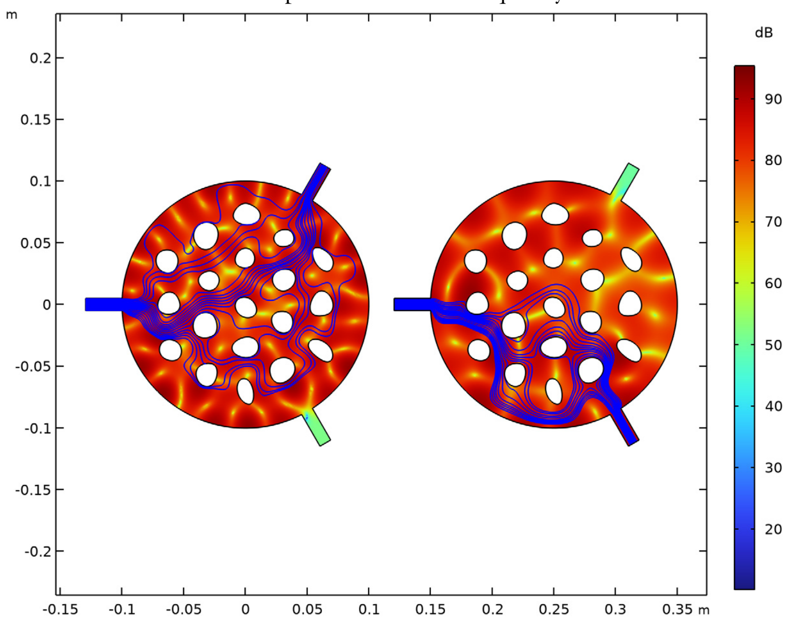


Figure 2: The intensity is plotted at the center of both frequency bands. The blue lines indicate the acoustic power flow.

Looking at the intensity streamlines in Figure 2 and the spectrum in Figure 3, it is clear that a node in the wave field is moved to the port for which no power is desired. This makes the performance of the design specific to the considered frequency bands and one can thus expect the minimum power ratio to decrease, if the bandwidth is increased.

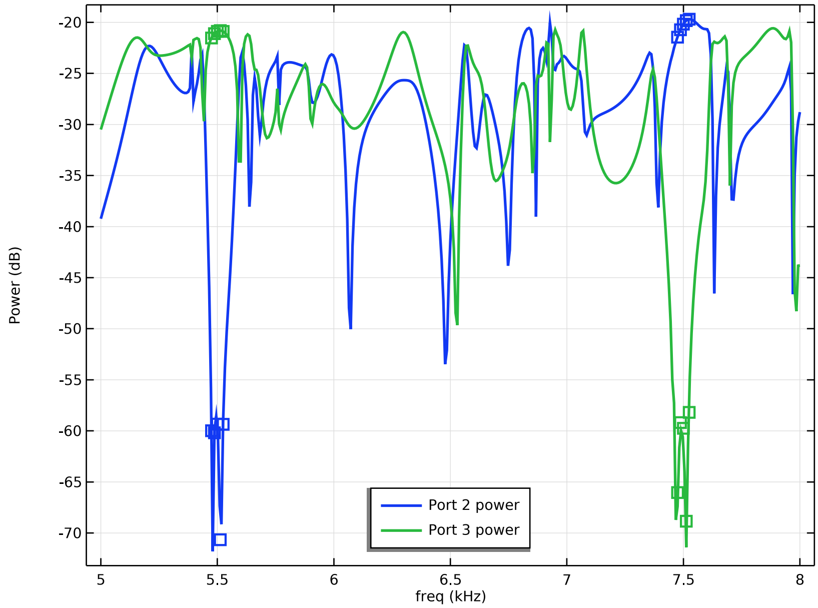


Figure 3: Spectrum for the two ports using the optimization mesh (points) and a finer mesh generated in the deformed configuration (lines).

Application Library path: `Acoustics_Module/Optimization/demultiplexer_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**.

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**.

GLOBAL DEFINITIONS

Parameters 1

You can import the table contents from demultiplexer_shape_optimization_parameters.txt.

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
f1	5.5[kHz]	5500 Hz	Frequency 1
f2	7.5[kHz]	7500 Hz	Frequency 2
df	50[Hz]	50 Hz	Frequency bandwidth
dfN	5	5	Frequencies per band
meshsz	340[m/s]/f2/6	0.0075556 m	Mesh size
Pc	1[mW/m]	0.001 W/m	Characteristic power

GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

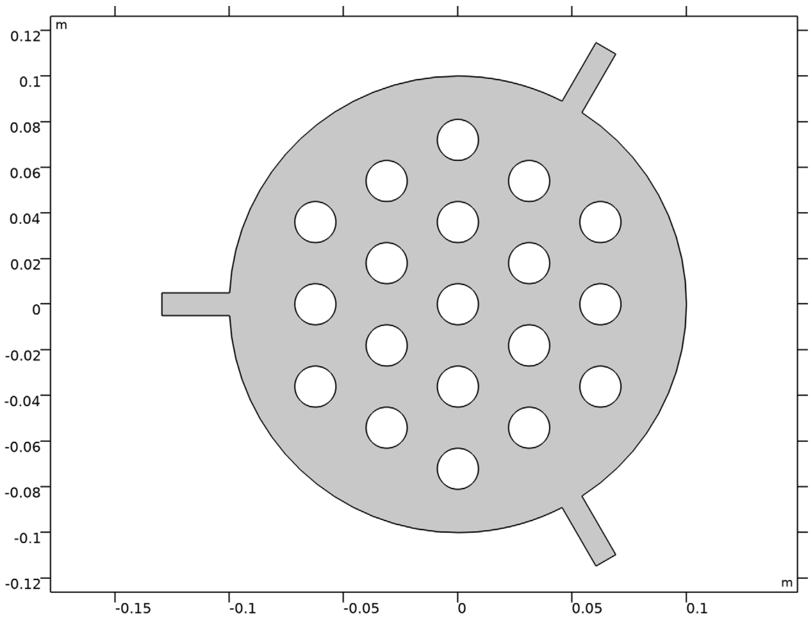
1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

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

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

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

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



The geometry should now look like that in Figure 1.

6 In the **Model Builder** window, collapse the **Geometry 1** node.

ADD MATERIAL

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

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

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

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

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

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

3 From the **Selection** list, choose **Port 1**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Slit**.

5 Locate the **Incident Mode Settings** section. In the A_p^in text field, type `1`.

Port 2

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

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

3 From the **Selection** list, choose **Port 2**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Slit**.

Port 3

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

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

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

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Slit**.

MESH 1

Size 1

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

Size

1 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

2 In the **Maximum element size** text field, type `meshsz`.

3 In the **Minimum element size** text field, type `meshsz/2`.

Size 1

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

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 **All boundaries**.

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

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `meshsz/2`.

Free Triangular 1

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

2 In the **Settings** window for **Free Triangular**, 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.

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 `f1 f2`.
4. In the **Study** toolbar, click **Compute**.

DEFINITIONS

Next define an objective function based on the power ratio between the two ports to the right.

Port powers

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, type `Port powers` in the **Label** text field.
3. Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
power2	10*log10(acpr.port2.P_out/Pc)		Port 2 power
power3	10*log10(acpr.port3.P_out/Pc)		Port 3 power
power_rat	log10(acpr.port2.P_out/Pc)-log10(acpr.port3.P_out/Pc)		Power ratio
obj	if(2*freq<f1+f2,power_rat,-power_rat)/10		Objective

Next define a shape optimization problem using the **Free Shape Domain** and **Free Shape Boundary** functionality.

COMPONENT 1 (COMP1)

Free Shape Domain 1

In the **Physics** toolbar, click **Optimization** and choose **Shape Optimization**.

Free Shape Boundary 1

1. In the **Shape Optimization** toolbar, click **Free Shape Boundary**.

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

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

4 Locate the **Control Variable Settings** section. In the text field, type `0.5*Rhole`.

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

ROOT

Add a 2nd 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 the **Add Study** button in the window toolbar.

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

STUDY 2: OPTIMIZATION

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

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 Clear the **Move limits** checkbox.

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

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

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(f1-0.5*df,df/(dfN-1),f1+0.5*df) range(f2-0.5*df,df/(dfN-1),f2+0.5*df)`.

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

RESULTS

Arrow Line 1

1 In the **Model Builder** window, expand the **Shape Optimization** node, then click **Arrow Line 1**.
2 In the **Settings** window for **Arrow Line**, locate the **Arrow Positioning** section.
3 From the **Placement** list, choose **Mesh nodes**.

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** checkbox.
4 From the **Plot group** list, choose **Shape Optimization**.
5 Locate the **Optimization Solver** section. In the **Maximum number of iterations** text field, type `25`.
6 In the **Study** toolbar, click = **Compute**.

RESULTS

Study 2: Optimization/Solution 2 (sol2)

Add a 2nd mesh generated in the deformed configuration and use it in two verification studies. The first verification study will only solve at `f1` and `f2`, while the 2nd will solve for many frequencies, but only save the data on the ports.

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.
2 Right-click **Results** > **Datasets** > **Study 2: Optimization/Solution 2 (sol2)** and choose **Remesh Deformed Configuration**.

MESH 2

Size 1

1 In the **Model Builder** window, expand the **Deformed Configuration 1 (frommesh1)** node.
2 Right-click **Component 1 (comp1)** > **Meshes** > **Deformed Configuration 1 (frommesh1)** > **Mesh 2** and choose **Size**.

Size

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

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `meshsz/2`.

4 In the **Minimum element size** text field, type `meshsz/4`.

Size 1

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

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 **Circles**.

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

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `meshsz/4`.

Free Triangular 1

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

Reference 1

In the **Model Builder** window, right-click **Reference 1** and choose **Disable**.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

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

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 `f1 f2`.

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

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 In the **Model Builder** window, click **Study 3**.
8 In the **Settings** window for **Study**, locate the **Study Settings** section.
9 Clear the **Generate default plots** checkbox.
10 In the **Label** text field, type `Study 3: Verification (f1,f2)`.
11 In the **Study** toolbar, click **Compute**.

VERIFICATION (MESH2)

1 In the **Model Builder** window, under **Study 4** click **Step 1: Frequency Domain**.
2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3 Click **Range**.
4 In the **Range** dialog, choose **Number of values** from the **Entry method** list.
5 In the **Start** text field, type `f1-10*df`.
6 In the **Stop** text field, type `f2+10*df`.
7 In the **Number of values** text field, type `401`.
8 Click **Replace**.
9 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.
10 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Deformed Geometry**.
11 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
12 From the **Method** list, choose **Solution**.
13 From the **Study** list, choose **Study 2: Optimization, Frequency Domain**.
14 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output
Pressure Acoustics, Frequency Domain (acpr)	Selection

15 Click to select row number 1 in the table.
16 Under **Selections**, click **Add**.
17 In the **Add** dialog, select **Ports** in the **Selections** list.

18 Click **OK**.

19 In the **Settings** window for **Frequency Domain**, locate the **Store in Output** section.

20 In the table, enter the following settings:

Interface	Output
Deformed geometry (Component 1)	Selection

21 Click to select row number 2 in the table.

22 Under **Selections**, click **Add**.

23 In the **Add** dialog, select **Ports** in the **Selections** list.

24 Click **OK**.

25 In the **Model Builder** window, click **Study 4**.

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

27 Clear the **Generate default plots** checkbox.

28 In the **Label** text field, type `Verification (mesh2)`.

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

RESULTS

Add a 1D plot group showing the optimized spectrum on both meshes.

Spectrum

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Verification (mesh2)/ Solution 4 (sol4)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

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

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

Global 1

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

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **power2 - Port 2 power**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **power3 - Port 3 power**.

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **kHz**.

5 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

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 2: Optimization/Solution 2 (sol2)**.

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

Expression	Unit	Description
power2		Port 2 (mesh1)
power3		Port 3 (mesh1)

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

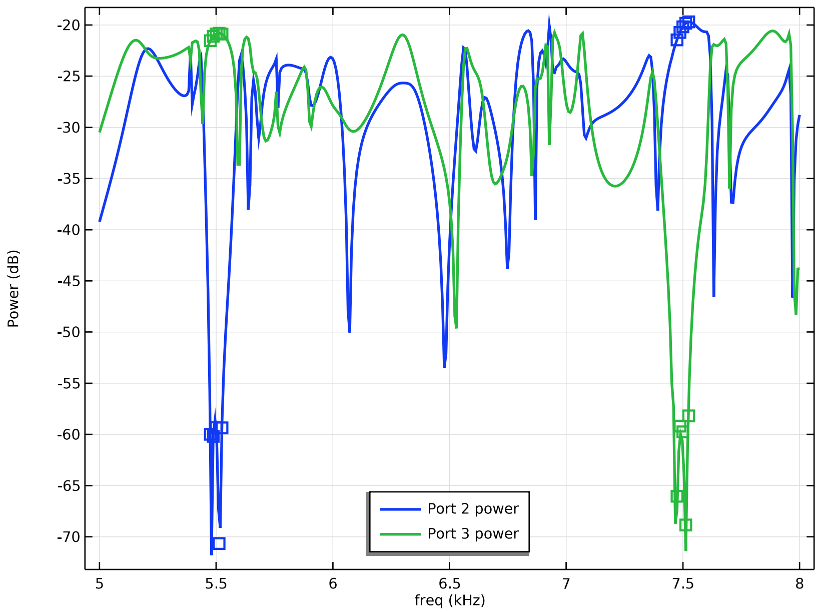
6 From the **Color** list, choose **Cycle (reset)**.

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

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

9 In the **Spectrum** toolbar, click **Plot**.

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



Modify the 4th plot group to animate the pressure for both frequencies.

Study 3: Verification (f1,f2)/Solution 3 (sol3)

1 In the **Model Builder** window, under **Results** > **Datasets** click **Study 3: Verification (f1,f2)/ Solution 3 (sol3)**.

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

3 From the **Frame** list, choose **Spatial (x, y, z)**.

Animation

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Surface 2

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

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

3 In the **Expression** text field, type `with(1,acpr.p_t)`.

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

Transformation 1

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

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

3 In the **X** text field, type `1.25*L`.

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

Surface 1

1 In the **Model Builder** window, under **Results** > **Animation** click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** checkbox.

4 In the **Minimum** text field, type `-1`.

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

Line 1

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

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

3 From the **Coloring** list, choose **Uniform**.

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

Transformation 1

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

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

3 In the **X** text field, type `1.25*L`.

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

5 Click **Plot**.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Animation**.

4 Locate the **Animation Editing** section. From the **Sequence type** list, choose **Dynamic data extension**.

5 Click the **Play** button in the **Graphics** toolbar.

6 Locate the **Playing** section. From the **Repeat** list, choose **Forever**.

Modify the 5th plot group to show the intensity for both frequencies.

Surface 2

1. In the **Model Builder** window, expand the **Sound Pressure Level (acpr) 1** node.
2. Right-click **Results** > **Sound Pressure Level (acpr) 1** > **Surface 1** and choose **Duplicate**.
3. In the **Settings** window for **Surface**, locate the **Expression** section.
4. In the **Expression** text field, type `with(1,acpr.Lp_t)`.
5. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Transformation 1

1. Right-click **Surface 2** and choose **Transformation**.
2. In the **Settings** window for **Transformation**, locate the **Transformation** section.
3. In the **X** text field, type `1.25*L`.
4. In the **Sound Pressure Level (acpr) 1** toolbar, click **Plot**.

Sound Pressure Level (acpr) 1

1. In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr) 1**.
2. In the **Settings** window for **2D Plot Group**, locate the **Title** section.
3. From the **Title type** list, choose **None**.
4. Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Streamline 1

1. Right-click **Sound Pressure Level (acpr) 1** and choose **Streamline**.
2. In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Pressure Acoustics, Frequency Domain** > **Intensity** > **acpr.Ix,acpr.Iy - Intensity (spatial and material frames)**.
3. Locate the **Selection** section. From the **Selection** list, choose **Port 1**.
4. Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Blue**.

Streamline 2

1. Right-click **Streamline 1** and choose **Duplicate**.
2. In the **Settings** window for **Streamline**, locate the **Expression** section.
3. In the **X-component** text field, type `with(1,acpr.Ix)`.
4. In the **Y-component** text field, type `with(1,acpr.Iy)`.

Transformation 1

1 Right-click **Streamline 2** and choose **Transformation**.

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

3 In the **X** text field, type `1.25*L`.

4 In the **Sound Pressure Level (acpr) 1** toolbar, click **Plot**.

Line 1

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

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

3 From the **Coloring** list, choose **Uniform**.

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

Transformation 1

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

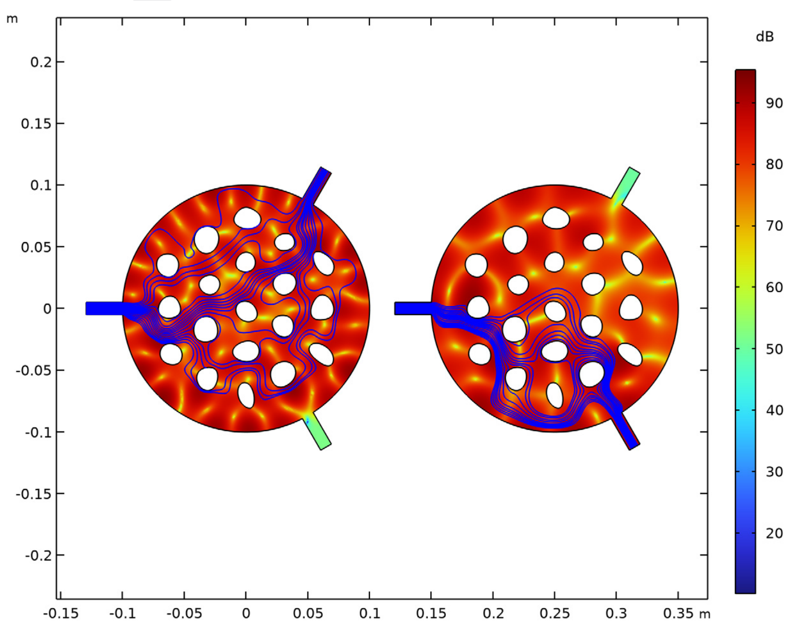
2 In the **Settings** window for **Transformation**, locate the **Transformation** section.

3 In the **X** text field, type `1.25*L`.

4 In the **Sound Pressure Level (acpr) 1** toolbar, click **Plot**.

5 Click **Plot**.

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



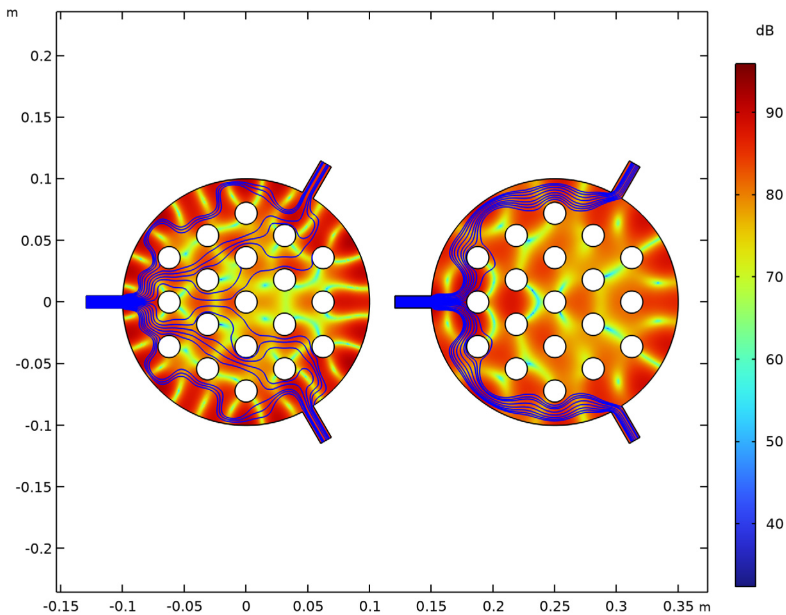
STUDY 1: INITIAL DESIGN

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 **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Deformed Geometry**.



Geometry Modeling Instructions

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

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
L	20[cm]	0.2 m	Domain diameter
L1	0.05*L	0.01 m	Port width
Lperiod	0.09*L	0.018 m	Hole period
Rhole	0.5*Lperiod	0.009 m	Hole radius

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 1

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type L/2.

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 3*L1.

4 In the **Height** text field, type L1.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **x** text field, type -L/2-L1*1.45.

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

Rotate 1 (rot1)

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

2 In the **Settings** window for **Rotate**, locate the **Input** section.

3 From the **Input objects** list, choose **Rectangle 1**.

4 Select the **Keep input objects** checkbox.

5 Locate the **Rotation** section. In the **Angle** text field, type 120 -120.

6 Click **Build Selected**.

Union 1 (uni1)

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

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

3 Clear the **Keep interior boundaries** checkbox.

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

5 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Circle 2 (c2)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Rhole`.

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

Move 1 (mov1)

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

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Circle 2**.

4 Select the **Keep input objects** checkbox.

5 Locate the **Displacement** section. In the **x** text field, type `2*Lperiod*sin(pi/3)`.

6 In the **y** text field, type `2*Lperiod*cos(pi/3)`.

Array 1 (arr1)

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

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Circle 2**.

4 Locate the **Size** section. In the **x size** text field, type `round(0.5*L/Lperiod)`.

5 In the **y size** text field, type `round(0.5*L/Lperiod)`.

6 Locate the **Displacement** section. In the **x** text field, type `4*Lperiod*sin(pi/3)`.

7 In the **y** text field, type `2*Lperiod`.

Mirror 1 (mir1)

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

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

3 From the **Input objects** list, choose **Circle 2**.

4 Select the **Keep input objects** checkbox.

Mirror 2 (mir2)

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

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

3 From the **Input objects** list, choose **Circle 2**.

4 Select the **Keep input objects** checkbox.

5 Locate the **Normal Vector to Line of Reflection** section. In the **x** text field, type `0`.

6 In the **y** text field, type `1`.

Disk Selection 1 (disksel1)

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

2 In the **Settings** window for **Disk Selection**, locate the **Input Entities** section.

3 From the **Entities** list, choose **From selections**.

4 Click **Add**.

5 In the **Add** dialog, select **Circle 2** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Disk Selection**, locate the **Size and Shape** section.

8 In the **Outer radius** text field, type `Inf`.

9 In the **Inner radius** text field, type `L/2-Rhole`.

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 From the **Selection** list, choose **Disk Selection 1**.

Difference 1 (dif1)

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

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

3 From the **Objects to add** list, choose **Union 1**.

4 From the **Objects to subtract** list, choose **Circle 2**.

Ports

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

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

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

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Inf`.

5 In the **Inner radius** text field, type `L*0.51`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside disk**.

Circles

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

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

3 Locate the **Size and Shape** section. In the **Inner radius** text field, type `0`.

4 In the **Outer radius** text field, type `L*0.49`.

Port 1

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

2 In the **Settings** window for **Box Selection**, type `Port 1` in the **Label** text field.

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

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Ports** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.

9 In the **x maximum** text field, type `0`.

Port 2

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

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

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

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Ports** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.

9 In the **y minimum** text field, type `0`.

10 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Port 3

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

2 In the **Settings** window for **Box Selection**, type `Port 3` in the **Label** text field.

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

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Ports** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.

9 In the **y maximum** text field, type `0`.

10 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

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

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

The model geometry is now complete.