



Model created in COMSOL Multiphysics 6.4

Eigenmodes in a Muffler with Elastic Walls

Introduction

This model is an extension of the model [Eigenmodes in a Muffler](#). The latter studies the propagation of acoustic waves through the cross section of the [Absorptive Muffler](#) chamber, provided that the muffler walls were sound hard. Here the muffler walls are considered to be made of a linear elastic material to account for their influence on the modes propagating through the cross section of the chamber. This is done using the **Acoustic–Solid Interaction, Frequency Domain** multiphysics interface. The **Mode following** functionality is used to compute the dispersion diagram for the two configurations.

Model Definition

The muffler chamber has the same shape and dimensions as in example ([Eigenmodes in a Muffler](#)). The only difference is that an elastic wall of the thickness W surrounds the inner (acoustic) domain as shown in [Figure 1](#).

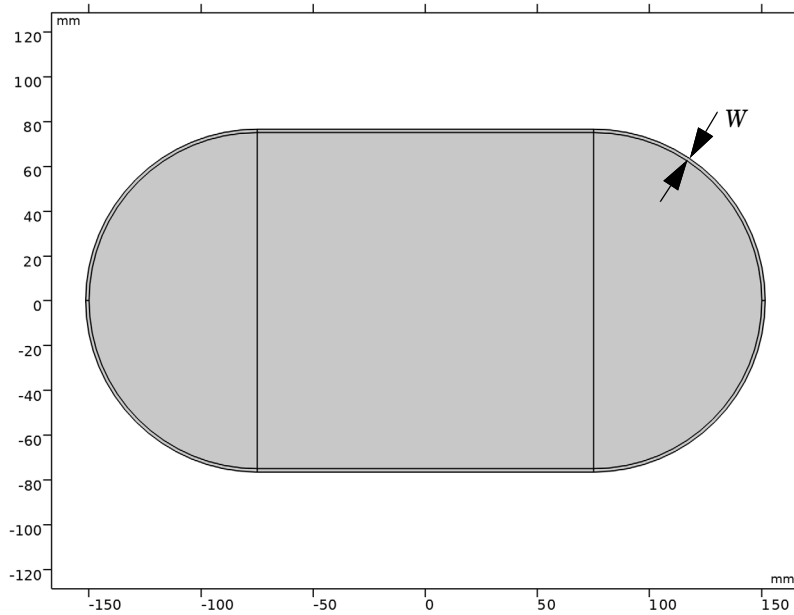


Figure 1: Geometry of model: the chamber of the same cross section as in [Eigenmodes in a Muffler](#) with the elastic wall to the thickness W .

The chamber is filled with air and the elastic wall is made of steel. Assuming the time-harmonic process with the angular frequency $\omega = 2\pi f$, the Mode Analysis study solves the

equations for the out-of-plane wave number κ_z as the function of given parameter f . The cutoff frequency for the j -th mode is then calculated as

$$f_j = \frac{\sqrt{\omega^2 - c^2 \kappa_z^2}}{2\pi} \quad (1)$$

Results and Discussion

The results of the simulations show that the presence of a thin elastic wall significantly affects the number of modes propagating through the cross section of the chamber. It becomes greater than that for the chamber with the sound hard wall boundary studied in the model [Eigenmodes in a Muffler](#). For example, the first propagating mode with the wave number lower than that of the plane wave mode has its cutoff frequency around 218 Hz. The former mode is shown in [Figure 2](#); the latter, in [Figure 3](#). Both figures shows the acoustic pressure in the chamber and the deformation of the muffler wall.

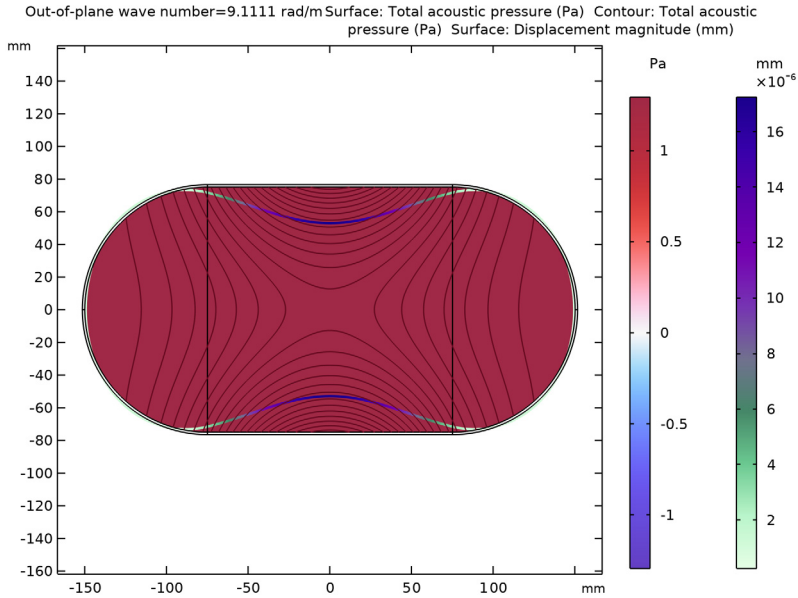


Figure 2: The plane wave propagating through the cross section of the chamber.

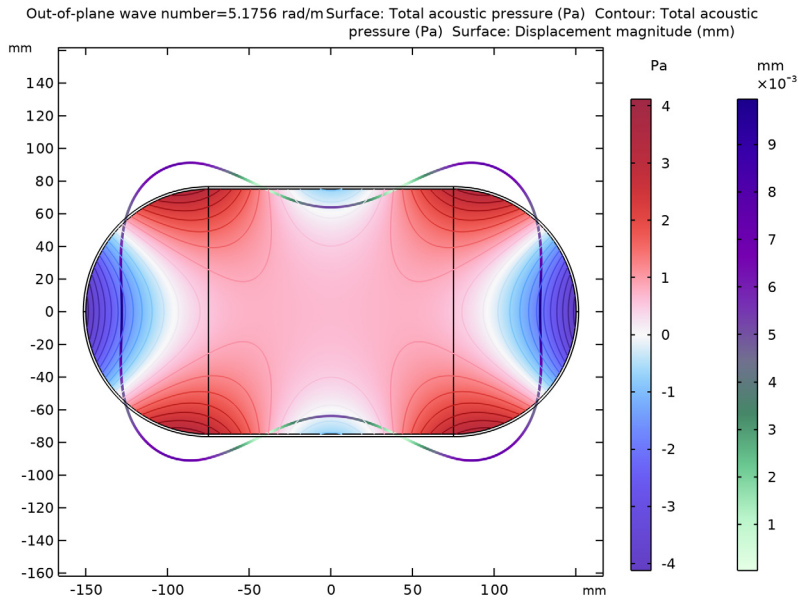


Figure 3: The first propagating mode different from the plane wave.

The model [Eigenmodes in a Muffler](#) gives the first least nonzero cutoff frequency equal to 635 Hz. A good understanding of the difference between these two cases is provided by the dispersion diagrams shown in [Figure 4](#). The top figure shows the diagram for the system with elastic walls while the bottom figure is the case for the hard wall configuration.

Another feature that distinguishes the chamber with the wall is that there can be modes having the wave number greater than that for the plane-wave mode, which is due to acoustics-structure interaction. In [Figure 5](#) at a frequency of 400 Hz the pure acoustic plane wave-number is 7.33 1/m (the actual plane wave mode is at 7.27 1/m). While the wave number selected has a value of 7.87 1/m. This corresponds to the point above the red dotted line at 400 Hz in [Figure 4](#). This means that their cutoff frequencies f_j calculated from [Equation 1](#) become pure imaginary. Such modes do not exist in the model [Eigenmodes in a Muffler](#).

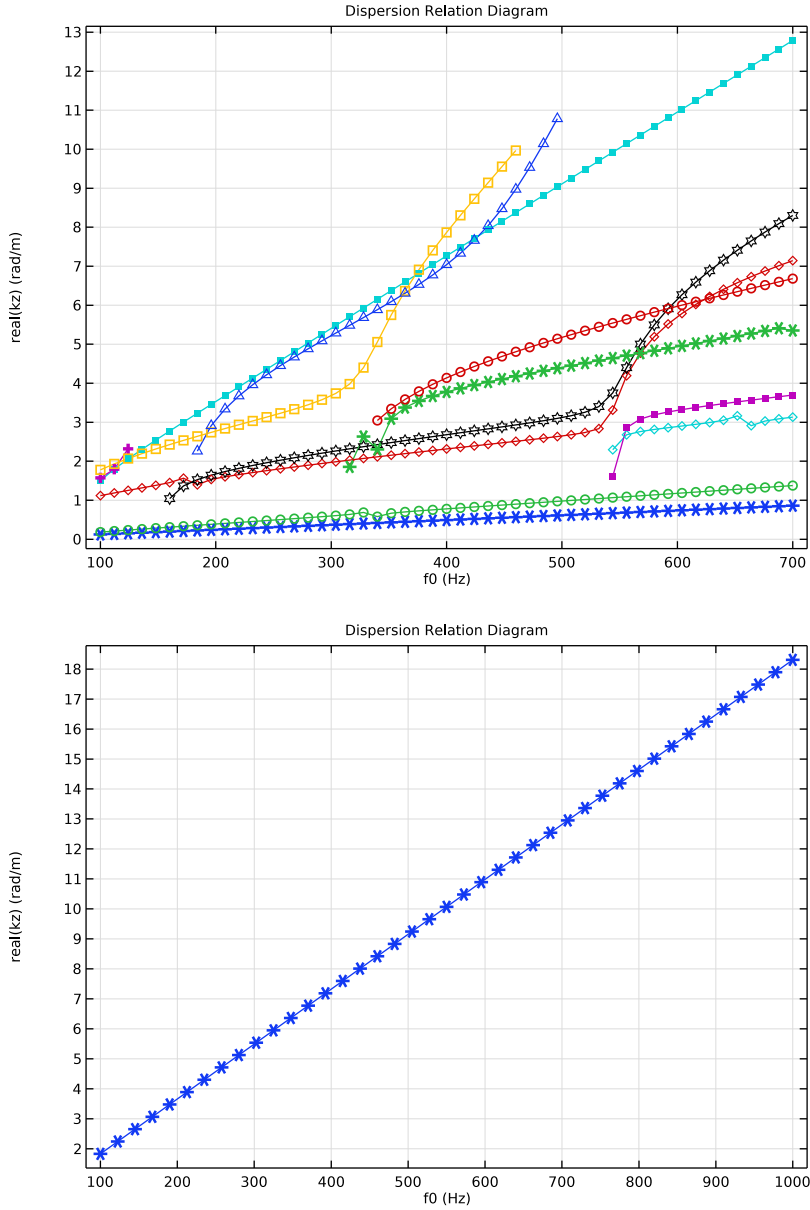


Figure 4: Dispersion diagram for the elastic wall case (top) and the hard wall case (bottom).

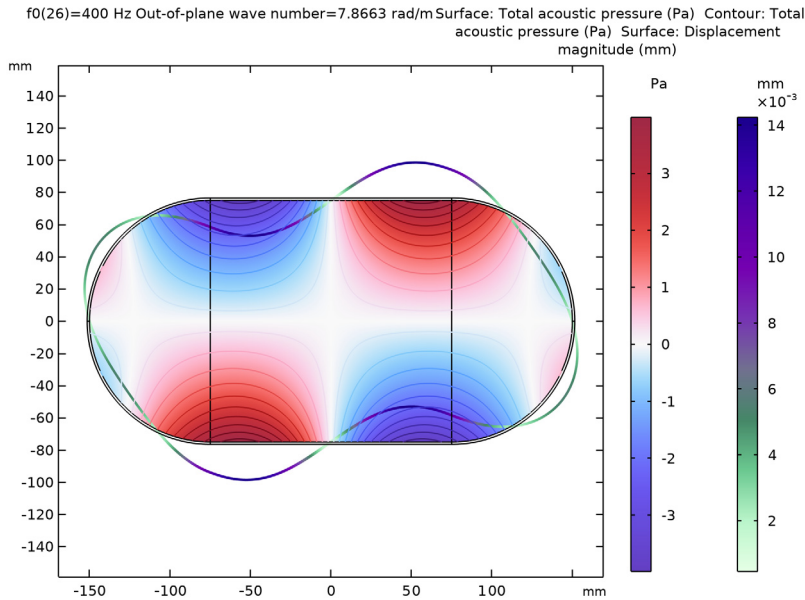


Figure 5: A propagating mode with the wave number greater than that of the plane wave.

Notes About the COMSOL Implementation

In this model **Mode following** is used when computing the dispersion diagram. It will ensure that a diagram can be plotted and modes are sorted and located in the same position in the list if **Out-of-plane wave number** when varying the outer parameter for the frequency f_0 .

Note that when plotting and referring to a dataset generated with the study that uses mode following, entries of the type $\text{NaN} + i\text{inf} * i$ can exist. These are “empty” solutions or placeholders that are filled as the frequency parameter increases and new modes appear.

Filtering is also used in this model for two cases:

- Removing evanescent (non-propagating) modes using the logic expression:
 $\text{abs}(\text{imag}(\text{comp1}.\text{acpr}.\text{kz})) / \text{abs}(\text{comp1}.\text{acpr}.\text{kz}) < 1e-3$

This removes modes that have imaginary parts that is larger than a factor 1000 of the absolute value of the wave number.


- Removing modes that propagate in one direction only to avoid duplicates using the logic expression $\text{real}(\text{comp1}.\text{acpr}.\text{kz}) > 0$.

Application Library path: Acoustics_Module/Automotive/
eigenmodes_in_muffler_elastic




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** > **Acoustic–Structure Interaction** > **Acoustic–Solid Interaction, Frequency Domain**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Mode Analysis**.
- 6 Click  **Done**.

GLOBAL DEFINITIONS

Parameters I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
W	1.5[mm]	0.0015 m	Muffler wall thickness
f0	500[Hz]	500 Hz	Cutoff frequency
c0	343[m/s]	343 m/s	Speed of sound in the air
lam0	c0/f0	0.686 m	Plane wave wavelength
k0	2*pi/lam0	9.1592 1/m	Free space plane-wave wave number

The parameter k_0 stands for the wave number of the plane wave generated at the cutoff frequency f_0 : $k_0 = 2\pi/\lambda_0 = 2\pi f_0/c_0$.


GEOMETRY I

1 In the **Model Builder** window, expand the **Component 1 (comp1) > Geometry 1** node, then 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 150.

4 In the **Height** text field, type $150 + 2*W$.

5 Locate the **Position** section. In the **x** text field, type -75.

6 In the **y** text field, type $-75 - W$.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	W

8 Select the **Layers on top** checkbox.

9 Click  **Build Selected**.

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 $75 + W$.

- 4 In the **Sector angle** text field, type 180.
- 5 Locate the **Position** section. In the **x** text field, type -75.
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type 90.
- 7 Click to expand the **Layers** section. In the table, enter the following settings:




Layer name	Thickness (mm)
Layer 1	W

- 8 Click  **Build Selected**.

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 $75 + W$.
- 4 In the **Sector angle** text field, type 180.
- 5 Locate the **Position** section. In the **x** text field, type 75.
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.
- 7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	W

- 8 Click  **Build Selected**.
- 9 In the **Geometry** toolbar, click  **Build All**.
- 10 Click the  **Zoom Extents** button in the **Graphics** toolbar.

The geometry should look like the one in [Figure 1](#).

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 tree, select **Built-in > Steel AISI 4340**.
- 6 Click the **Add to Component** button in the window toolbar.
- 7 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

Steel AISI 4340 (mat2)

Select Domains 1–3, 6, 7, and 9 only.

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 **Manual**.
- 4 Select Domains 4, 5, and 8 only.


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 **Manual**.
- 4 Select Domains 1–3, 6, 7, and 9 only.
Enable the **Out-of-plane mode extension (time-harmonic)** to perform the **Mode Analysis** for elastic waves in the muffler wall domain.
- 5 Locate the **2D Approximation** section. Select the **Out-of-plane mode extension (time-harmonic)** checkbox.

MESH 1

In this model, the mesh is set up manually. Proceed by directly adding the desired mesh component.

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 4, 5, and 8 only.


Size 1

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

Create a **Mapped** mesh for the muffler wall domain and choose the **Distribution** depth of 3 elements.

Mapped 1

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

Use the **Zoom Box** functionality to investigate the boundary layer mesh.

Distribution 1

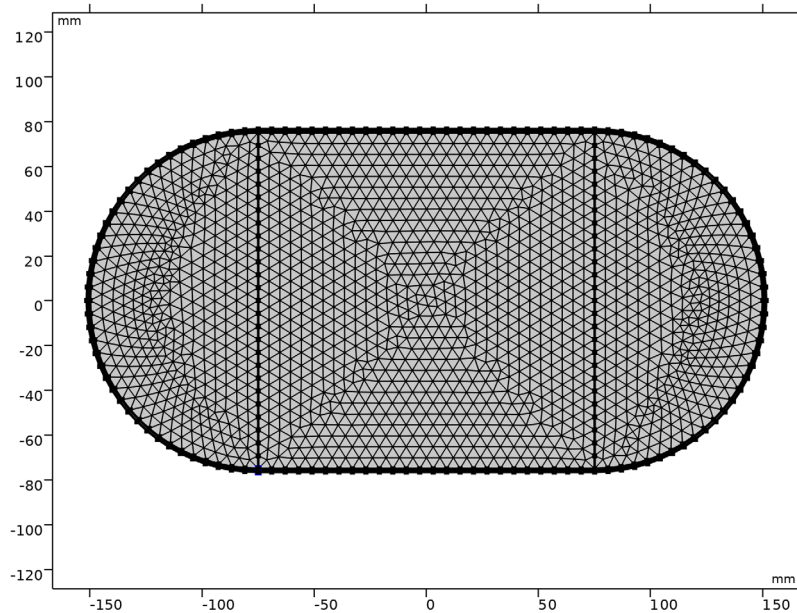
1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type 3.

5 Click  **Build All**.




STUDY 1

Step 1: Mode Analysis

1 In the **Model Builder** window, under **Study 1** click **Step 1: Mode Analysis**.

2 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.

- 3 In the **Mode analysis frequency** text field, type f_0 .
Set the solver to search for the first 6 propagating modes with the wave numbers lower than k_0 .
- 4 Select the **Desired number of modes** checkbox.
Search for the modes below the plane wave mode.
- 5 Select the **Search for modes around shift** checkbox. In the associated text field, type $1.1 * k_0$.
- 6 From the **Search method around shift** list, choose **Smaller real part**.
- 7 In the **Study** toolbar, click  **Compute**.

RESULTS

Acoustic Pressure (acpr)

Now, select the out-of-plane wave number with a value close to 5.18. This is the first nonplane mode.

- 1 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 2 From the **Out-of-plane wave number (rad/m)** list, choose **5.1756**.

Surface 2

- 1 In the **Model Builder** window, right-click **Acoustic Pressure (acpr)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `solid.disp`.
- 4 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraBorealis**.
- 5 From the **Scale** list, choose **Linear**.



Deformation 1

- 1 Right-click **Surface 2** and choose **Deformation**.
The results should look like the one depicted in [Figure 3](#).

Dispersion Relation Calculation

The next steps are optional and are added to show you how to get the dispersion relation curves that relate the frequency and the wave number. The **Mode following** option is used for this analysis, it is found in the **Filtering and Sorting** section of the study step. The dispersion relations for the chamber with elastic and hard walls are calculated and compared below.

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 **Preset Studies for Selected Physics Interfaces > Mode Analysis**.
- 4 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Mode Analysis


- 1 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.
- 2 In the **Mode analysis frequency** text field, type f_0 .
- 3 Select the **Desired number of modes** checkbox.
- 4 Select the **Search for modes around shift** checkbox.
- 5 In the **Desired number of modes** text field, type 12.
- 6 In the **Search for modes around shift** text field, type $1.2 \cdot k_0$.
Search, starting a bit higher than the plane wave mode and look for 12 modes.
- 7 From the **Search method around shift** list, choose **Smaller real part**.
- 8 Click to expand the **Filtering and Sorting** section. Find the **Filtering** subsection. In the table, enter the following settings:


Filter expression (store if positive)	Description
$\text{abs}(\text{imag}(\text{comp1.acpr.kz})) / \text{abs}(\text{comp1.acpr.kz}) < 1e-3$	Remove non-propagating modes
$\text{real}(\text{comp1.acpr.kz}) > 0$	Only propagation in one direction

Use the filtering functionality to remove evanescent modes and also only retain modes that propagate in one direction (to not get duplicates).



- 9 Find the **Sorting** subsection. Select the **Mode following** checkbox.

Parametric Sweep



- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 Click **+ Add**.
- 4 From the list in the **Parameter name** column, choose **f0 (Cutoff frequency)**.

- 5 Click  **Range**.
- 6 In the **Range** dialog, choose **Number of values** from the **Entry method** list.
- 7 In the **Start** text field, type 100.
- 8 In the **Stop** text field, type 700.
- 9 In the **Number of values** text field, type 51.
- 10 Click **Replace**.
- 11 In the **Model Builder** window, click **Study 2**.
- 12 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 13 Clear the **Generate default plots** checkbox.

Solution 2 (sol2)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
- 3 In the **Model Builder** window, expand the **Study 2 > Solver Configurations > Solution 2 (sol2) > Eigenvalue Solver 1** node, then click **Mode Following 1**.
- 4 In the **Settings** window for **Mode Following**, locate the **Mode Following** section.
- 5 Select the **Follow extra modes** checkbox.
- 6 In the **Extra modes allocation factor** text field, type 2.
Because of the problem type more modes will appear as the frequency increases. This option allows new modes to be followed also.
- 7 In the **Study** 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 **Preset Studies for Selected Physics Interfaces > Mode Analysis**.
- 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 3

Step 1: Mode Analysis





- 1 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.
- 2 In the **Mode analysis frequency** text field, type f_0 .

- 3 Select the **Desired number of modes** checkbox.
- 4 Select the **Search for modes around shift** checkbox. In the associated text field, type $1.2 * k_0$.
- 5 From the **Search method around shift** list, choose **Smaller real part**.
- 6 Click to expand the **Filtering and Sorting** section. Find the **Filtering** subsection. In the table, enter the following settings:

Filter expression (store if positive)	Description
$\frac{\text{abs}(\text{imag}(\text{comp1.acpr.kz}))}{\text{abs}(\text{comp1.acpr.kz})} < 1e-3$	Remove non-propagating modes
$\text{real}(\text{comp1.acpr.kz}) > 0$	Only propagation in one direction


- 7 Find the **Sorting** subsection. Select the **Mode following** checkbox.
Now, disable the **Solid Mechanics** physics which results in the outer wall of the acoustic domain to be sound hard.
- 8 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Solid Mechanics (solid)**.

Parametric Sweep

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 Click  **Add**.
- 4 From the list in the **Parameter name** column, choose **f0 (Cutoff frequency)**.
- 5 Click  **Range**.
- 6 In the **Range** dialog, choose **Number of values** from the **Entry method** list.
- 7 In the **Start** text field, type 100.
- 8 In the **Stop** text field, type 1000.
- 9 In the **Number of values** text field, type 41.
- 10 Click **Replace**.
- 11 In the **Model Builder** window, click **Study 3**.
- 12 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 13 Clear the **Generate default plots** checkbox.
- 14 In the **Study** toolbar, click  **Compute**.

RESULTS

Dispersion Relation for Elastic Walls

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Dispersion Relation for Elastic Walls in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Dispersion Relation Diagram.
- 5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 1 (sol3)**.
- 6 Locate the **Plot Settings** section.
- 7 Select the **x-axis label** checkbox. In the associated text field, type f_0 (Hz).
- 8 Select the **y-axis label** checkbox. In the associated text field, type $\text{real}(kz)$ (rad/m).
- 9 Locate the **Legend** section. Clear the **Show legends** checkbox.

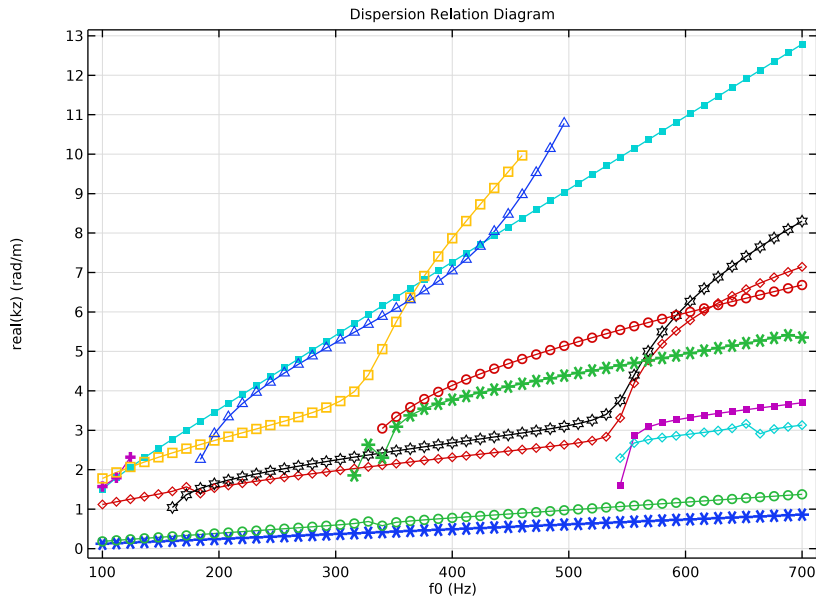
Global 1

- 1 Right-click **Dispersion Relation for Elastic Walls** 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
$\text{real}(acpr.kz)$	rad/m	

- 4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.
- 5 From the **Parameter** list, choose **Parameter value**.
- 6 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 In the **Dispersion Relation for Elastic Walls** toolbar, click  **Plot**.



Dispersion Relation for Hard Walls

- 1 In the **Model Builder** window, right-click **Dispersion Relation for Elastic Walls** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Dispersion Relation for Hard Walls in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 3/ Parametric Solutions 2 (sol56)**.

4 In the **Dispersion Relation for Hard Walls** toolbar, click  **Plot**.

