Created in COMSOL Multiphysics 6.1

Eigenmodes in a Muffler with Elastic Walls

Introduction

The model below is an extension of the model Eigenmodes in a Muffler. The latter studied 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-Structure Interaction multiphysics interface.

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.

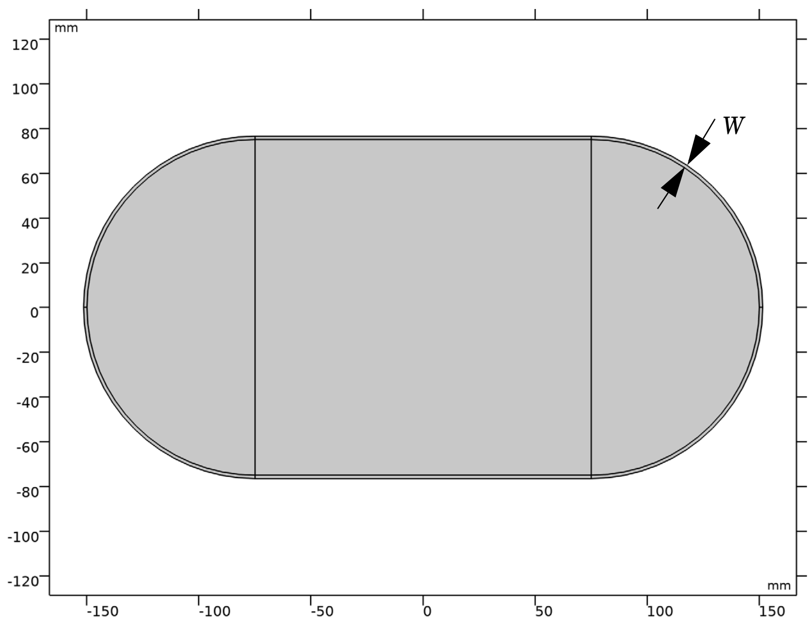


Figure 1: Geometry of model: the chamber of the same cross section as in Eigenmodes in a Muffler *with the elastic wall to the thickness* W.

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

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

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.

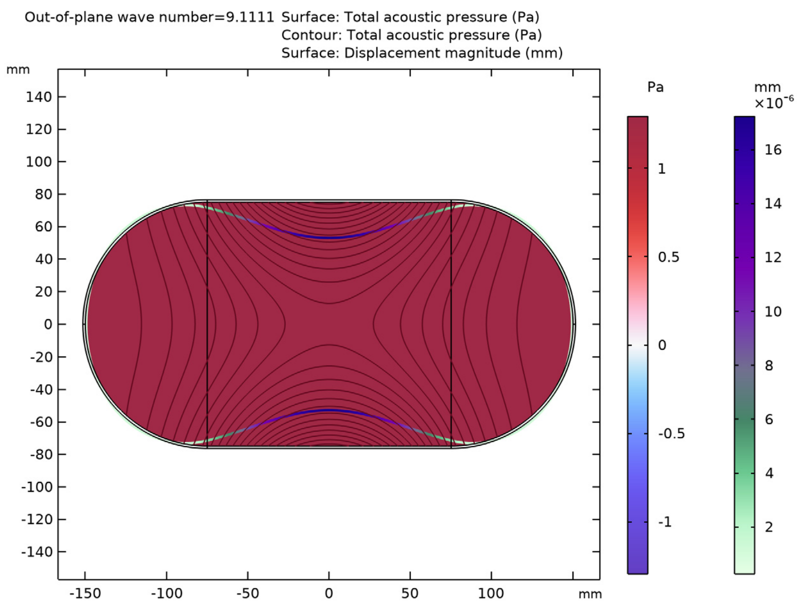


Figure 2: The plane wave propagating through the cross section of the chamber.

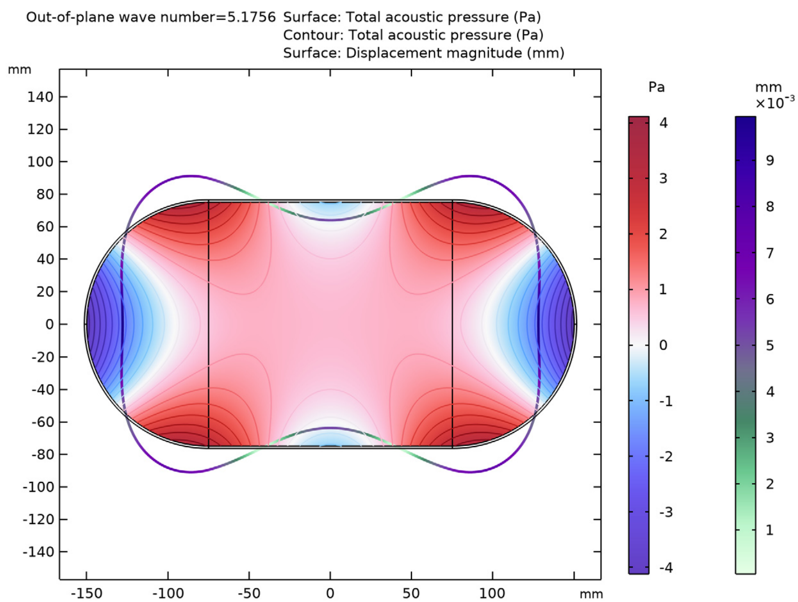


Figure 3: The first propagating mode different from the plane wave.

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 diagram shown in Figure 4, where the results corresponding to the presence of the wall are in black, and those obtained with the sound hard wall boundary condition are in red.

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.

.

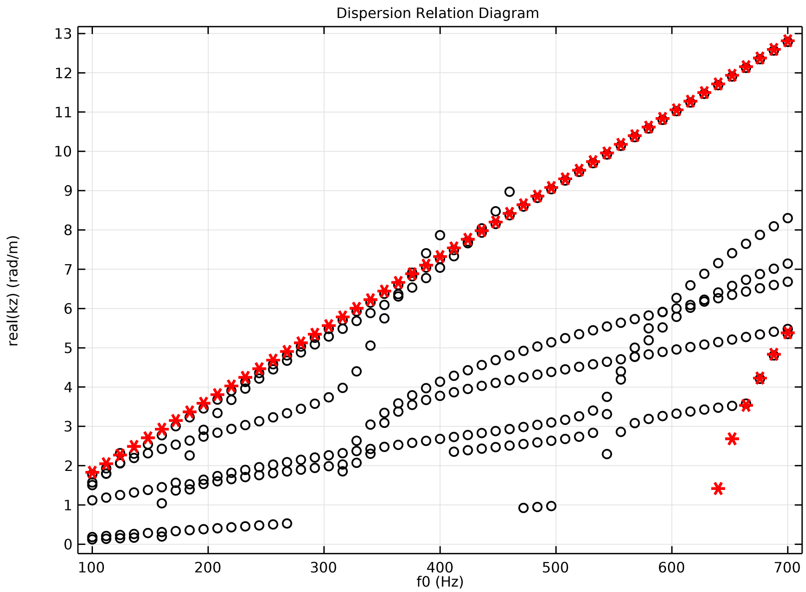


Figure 4: Dispersion diagram: the red asterisks correspond to the sound hard wall boundary; the black circles, to the elastic wall.

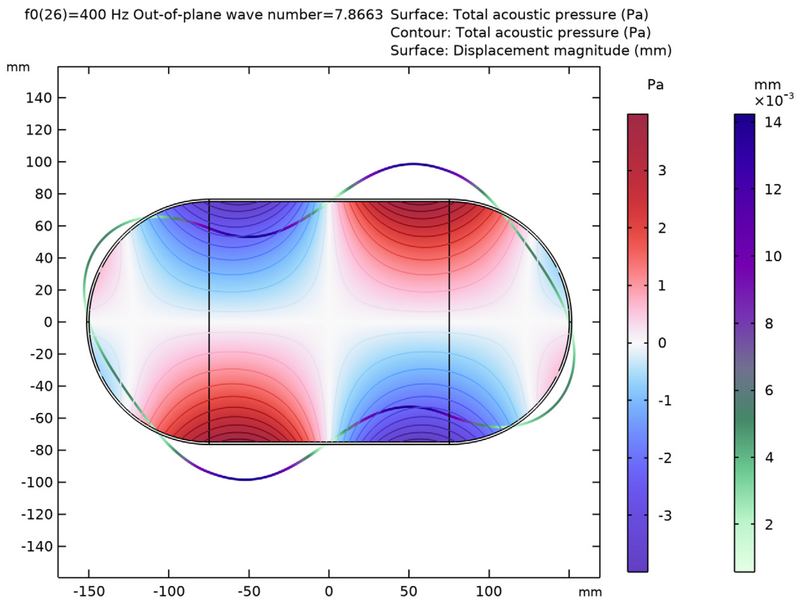


Figure 5: A propagating mode with the wave number greater that of the plane wave.

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 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
`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 1

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** check box.

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 **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 **Built-in>Steel AISI 4340**.
6. Click **Add to Component** in the window toolbar.
7. In the **Home** 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)** check box.

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

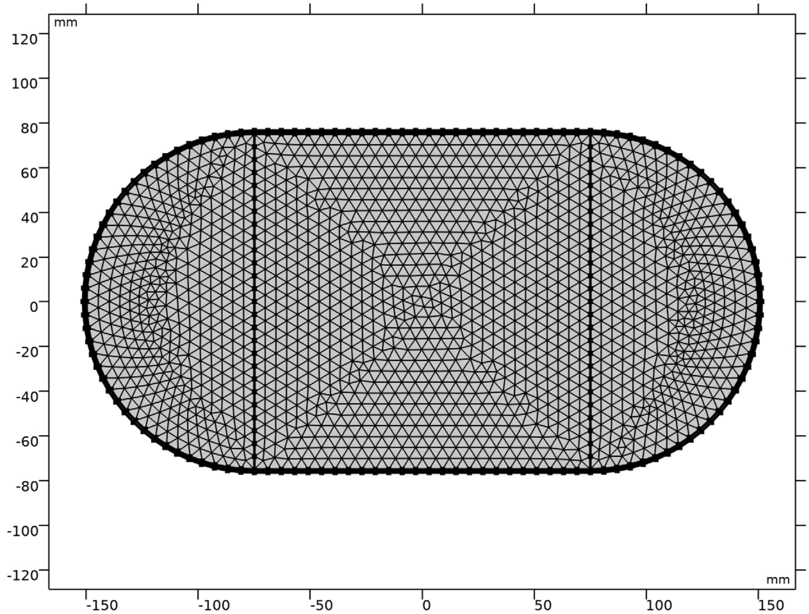
1 Right-click **Mapped 1** and choose **Distribution**.

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 `f0`.

Set the solver to search for the first 6 propagating modes with the wave numbers lower that k_0.

4 Select the **Desired number of modes** check box.

Search for the modes below the plane wave mode.

5 Select the **Search for modes around** check box. In the associated text field, type `1.1*k0`.

6 From the **Mode search method around shift** list, choose **Smaller real part**.

7 In the **Home** 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 non-plane mode.

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

2 From the **Out-of-plane wave number** 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. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Aurora>AuroraBorealis** in the tree.

6 Click **OK**.

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 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 **Add Study** 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 `f0`.

3 Select the **Desired number of modes** check box.

4 Select the **Search for modes around** check box. In the associated text field, type `1.1*k0`.

5 From the **Mode search method around shift** list, choose **Smaller real part**.

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 box, 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** check box.

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

ADD STUDY

1 In the **Study** 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 **Add Study** in the window toolbar.

5 In the **Study** 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 `f0`.

3 Select the **Desired number of modes** check box.

4 Select the **Search for modes around** check box. In the associated text field, type `1.1*k0`.

5 From the **Mode search method around shift** list, choose **Smaller real part**.

Now, disable the **Solid Mechanics** physics which results in the outer wall of the acoustic domain to be sound hard.

6 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box 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 box, 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 3**.

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

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

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

RESULTS

Dispersion Relation

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

2 In the **Settings** window for **1D Plot Group**, type `Dispersion Relation` 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** check box. In the associated text field, type `f0 (Hz)`.

8 Select the **y-axis label** check box. In the associated text field, type `real(kz) (rad/m)`.

9 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Dispersion Relation** 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
`if(abs(imag(acpr.kz))/abs(acpr.kz) < 1e-3, abs(real(acpr.kz)), NaN)`	`rad/m`	

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

5 From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `f0`.

7 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

9 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

Global 2

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

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

3 From the **Color** list, choose **Red**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

5 Locate the **Data** section. From the **Dataset** list, choose **Study 3/ Parametric Solutions 2 (sol56)**.

6 In the **Dispersion Relation** toolbar, click **Plot**.

The dispersion relation diagrams should look like the ones in Figure 4.