Created in COMSOL Multiphysics 6.2

Vibrations of a Disk Backed by an Air-Filled Cylinder

Introduction

The vibration modes of a thin or thick circular disc are well known, and it is possible to compute the corresponding eigenfrequencies with an arbitrary precision from a series solution. The same is true for the acoustic modes of an air-filled cylinder with perfectly rigid walls. A more interesting question to ask is: What happens if the cylinder is sealed in one end not by a rigid wall but by a thin disc? This is the question addressed in this tutorial.

The application uses the Structural Mechanics Module's Shell interface and the Pressure Acoustics interface from COMSOL Multiphysics. If you have a license for the Acoustics Module, see Vibrations of a Disk Backed by an Air-Filled Cylinder in the Acoustics Module Application Library for a model version that uses the Acoustic-Shell Interaction, Frequency Domain multiphysics interface.

Model Definition

The geometry is a rigid steel cylinder with a height of 255 mm and a radius of 38 mm. One end is welded to a heavy slab, while the other is sealed with a steel disc only 0.38 mm thick. The disc is modeled using shell elements with the outer edge of the disc fixed. The acoustics in the cylinder is described in terms of the acoustic (differential) pressure. The eigenvalue equation for the pressure is

$$-\Delta p = \frac{\omega^2}{c^2} p$$

where c is the speed of sound and $\omega = 2\pi f$ defines the eigenfrequency, f.

A first step is to calculate the eigenfrequencies for the disc and the cylinder separately and compare them with theoretical values. This way you can verify the basic components of the model and assess the accuracy of the finite-element solution before modeling the coupled system. When computing the decoupled problem, the acoustic domain is completely surrounded by sound hard boundaries. In the coupled analysis, the boundary at the disc instead has the accelerations of the disc as boundary conditions. At the same time, the acoustic pressure supplies a load on the disc.

Results and Discussion

To be able to study the effects of the coupling, we first look at the solution of the uncoupled problem. Figure 1 through Figure 4 show the two first uncoupled structural and acoustic modes.

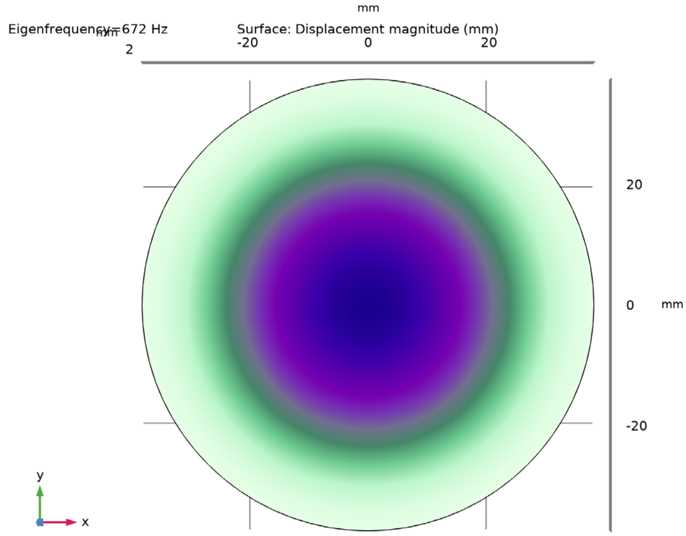

Figure 1: First structural mode displayed by displacement magnitude.

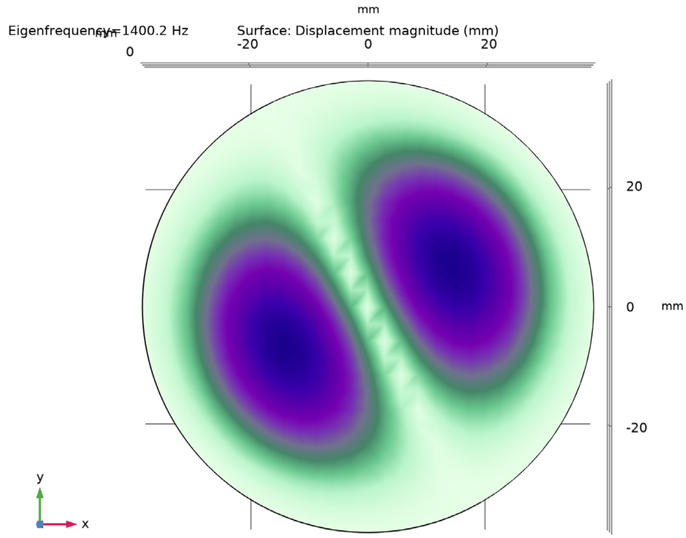

Figure 2: Second structural mode displayed by displacement magnitude.

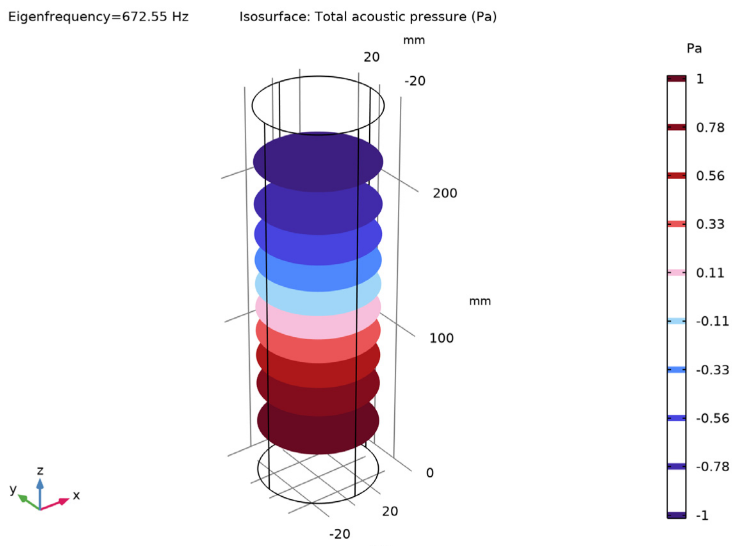

Figure 3: First acoustic mode displayed by pressure isosurfaces.

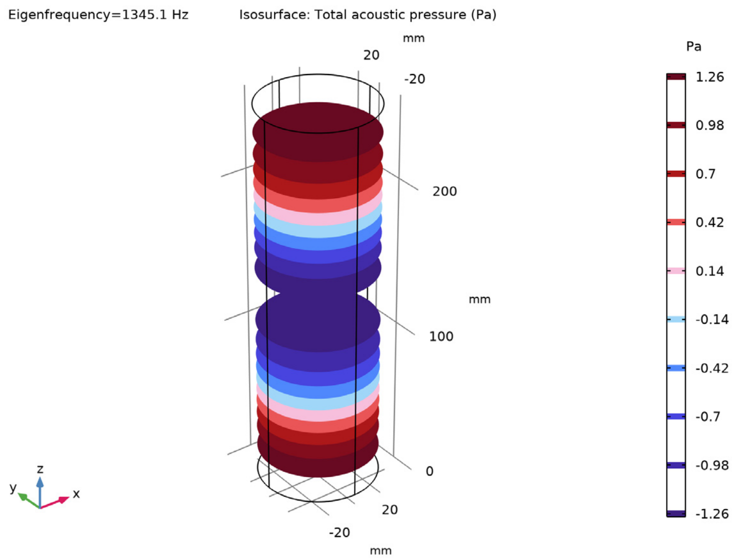

Figure 4: Second acoustic mode displayed by pressure isosurfaces.

In Ref. 1, D.G. Gorman and others have thoroughly investigated the coupled model at hand, and they have developed a semi-analytical solution verified by experiments. Their results for the coupled problem are presented in Table 1, together with the computed results from the COMSOL Multiphysics analysis.

TABLE 1: RESULTS FROM SEMI-ANALYTICAL AND COMSOL MULTIPHYSICS ANALYSIS AND EXPERIMENTAL DATA.

Dominated by	**Semi-analytical (Hz)**	**Computed (Hz)**	**Experimental (Hz)**
str/ac	636.9	637.1	630
str/ac	707.7	707.6	685
ac	1347	1347	1348
str	1394	1396	1376
ac	2018	2019	2040
str	2289	2292/2304	2170
str/ac	2607	2623	2596
ac	2645	2646	–
str/ac	2697	2697	2689
ac	2730	2730	2756
ac	2968	2968	2971

As the table shows, the computed eigenfrequencies are in good agreement with both the theoretical predictions and the experimentally measured values. The table also states whether the mode is structurally dominated (str), acoustically dominated (ac), or tightly coupled (str/ac).The eigenfrequency precision is generally better for the acoustically dominated modes.

Most of the modes show rather weak coupling between the structural bending of the disc and the pressure field in the cylinder. It is, however, interesting to note that some of the uncoupled modes have been split into one covibrating and one contravibrating mode with distinct eigenfrequencies. This is, for example, the case for modes 1 and 2 in the FEM solution.

In Figure 5, the first coupled mode is shown in terms of disc displacements and air pressure. The coupling effect can be clearly displayed using a plot of pressure gradients, as in Figure 6.

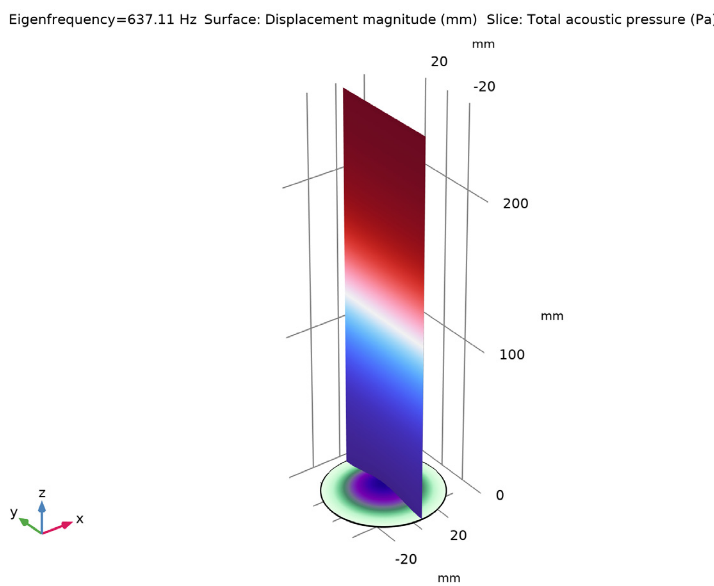

Figure 5: Disc deformation and pressure slice plot for the first coupled mode.

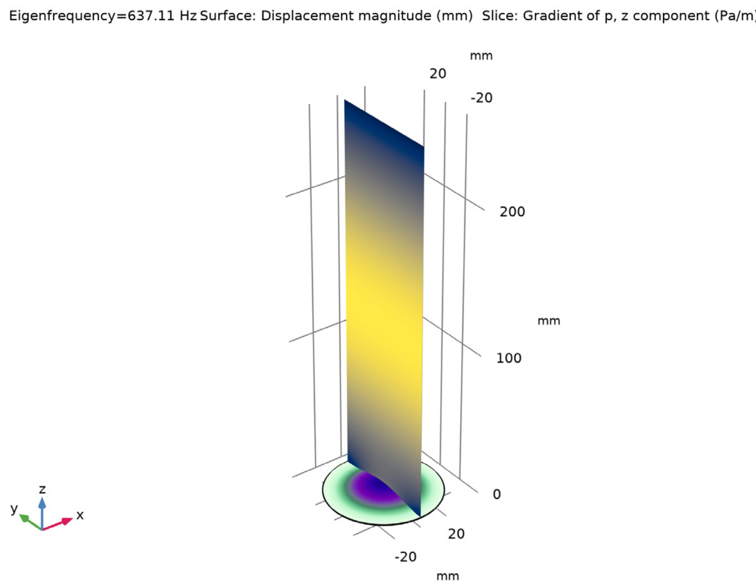

Figure 6: Disc deformation and pressure gradient slice plot for the first coupled mode.

Notes About the COMSOL Implementation

You specify the part of the physics for which to compute the uncoupled eigenvalues by selecting the physics interface in the eigenfrequency study.

When coupling the two types of physics, be careful when selecting the sign of the coupling terms, so that they act in the intended direction. You should specify the acceleration in the inward normal direction for the pressure acoustics domain, which in this case is the positive z-acceleration of the disc. The acceleration is denoted `wtt` as it is the second time derivative of the variable `w`. The pressure on the shell can be given using global directions, so that a positive pressure acts as a face load in the negative z direction. This is handled automatically by COMSOL in the setup in this example.

Reference

1. D.G. Gorman, J.M. Reese, J. Horacek, and D. Dedouch, “Vibration Analysis of a Circular Disk Backed by a Cylindrical Cavity,” *Proc. Instn. Mech. Engrs.*, Part C, vol. 215, no. 11, pp. 1303–1311, 2001.

Application Library path: `Structural_Mechanics_Module/Acoustic-Structure_Interaction/coupled_vibrations_manual`

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 **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Eigenfrequency**.

8 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**.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `38`.

4 In the **Height** text field, type `255`.

5 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

SHELL (SHELL)

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

2 Select Boundary 3 only.

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 `0.38[mm]`.

Fixed Constraint 1

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

2 Select Edges 2, 3, 7, and 10 only.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	1.2	kg/m³	Basic
Speed of sound	c	343	m/s	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

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

4 Select Boundary 3 only.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	2.1e11	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7800	kg/m³	Basic

MESH 1

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

Free Quad 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

2 Select Boundary 3 only.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 10.

In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details, see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide*. The mesh size used in this model corresponds to resolving the wavelength with about 9 elements at 4000 Hz, the maximum eigenfrequency studied in the model. The wavelength in air at 4000 Hz is 8.6 cm.

Free Quad 1

In the **Model Builder** window, right-click **Free Quad 1** and choose **Build Selected**.

Swept 1

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

2 In the **Settings** window for **Swept**, click **Build Selected**.

STRUCTURAL ANALYSIS

In the first study, you solve the structural problem only.

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

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

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Structural Analysis** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `20`.

4 In the **Search for eigenfrequencies around shift** text field, type `500`.

Exclude the Pressure Acoustics interface.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain (acpr)**.

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

RESULTS

Mode Shape, Structural Analysis

In the **Settings** window for **3D Plot Group**, type `Mode Shape, Structural Analysis` in the **Label** text field.

Surface 1

1 Click the **Go to XY View** button in the **Graphics** toolbar.

2 Click the **Scene Light** button in the **Graphics** toolbar.

Mode Shape, Structural Analysis

1 In the **Model Builder** window, expand the **Mode Shape, Structural Analysis** node, then click **Results>Mode Shape, Structural Analysis**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **1400.2**.

4 In the **Mode Shape, Structural Analysis** toolbar, click **Plot**.

ROOT

Add the second study to solve the pure acoustics problem.

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> Eigenfrequency**.

For this study, exclude the Shell interface.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Shell (shell)**.

5 Click **Add Study** in the window toolbar.

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

ACOUSTICS ANALYSIS

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

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

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Acoustics Analysis** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `20`.

4 In the **Search for eigenfrequencies around shift** text field, type `500`.

5 From the **Search method around shift** list, choose **Larger real part**.

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

RESULTS

Acoustic Pressure, Acoustics Analysis, Isosurfaces

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

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Acoustics Analysis, Isosurfaces` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **672.55**.

4 In the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

6 In the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.

7 From the **Eigenfrequency (Hz)** list, choose **1345.1**.

8 In the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.

Add the boundary conditions that couple the Pressure Acoustics and the Shell interfaces.

SHELL (SHELL)

Face Load 1

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

2 Select Boundary 3 only.

The acoustic pressure exerts a normal load on the plate.

3 In the **Settings** window for **Face Load**, locate the **Force** section.

4 From the $\mathbf{F}_A$ list, choose **Acoustic load per unit area (acpr/fpam1)**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

Normal Acceleration 1

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

2 Select Boundary 3 only.

The inward normal acceleration at the plate equals the second time derivative of the vertical displacement.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 From the **Type** list, choose **Acceleration**.

5 From the $\mathbf{a}_0$ list, choose **Acceleration (shell/emm1)**.

Add the third study for the coupled problem.

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> Eigenfrequency**.

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

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

COUPLED ANALYSIS

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

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

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Coupled Analysis** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `20`.

4 In the **Search for eigenfrequencies around shift** text field, type `500`.

5 From the **Search method around shift** list, choose **Larger real part**.

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

RESULTS

Mode Shape, Coupled Analysis

1 In the **Settings** window for **3D Plot Group**, type `Mode Shape, Coupled Analysis` in the **Label** text field.

2 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **637.11**.

Slice 1

1 In the **Model Builder** window, expand the **Mode Shape, Coupled Analysis** node.

2 Right-click **Mode Shape, Coupled Analysis** and choose **Slice**.

3 In the **Settings** window for **Slice**, locate the **Data** section.

4 From the **Dataset** list, choose **Coupled Analysis/Solution 3 (sol3)**.

5 From the **Solution parameters** list, choose **From parent**.

6 Locate the **Expression** section. In the **Expression** text field, type `acpr.p_t`.

7 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

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

9 In the **Color Table** dialog box, select **Wave>Wave** in the tree.

10 Click **OK**.

11 In the **Settings** window for **Slice**, locate the **Coloring and Style** section.

12 From the **Scale** list, choose **Linear symmetric**.

13 In the **Mode Shape, Coupled Analysis** toolbar, click **Plot**.

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

Plot the pressure gradient to display the connection to the disk shape.

Mode Shape, Coupled Analysis

Right-click **Mode Shape, Coupled Analysis** and choose **Duplicate**.

Coupling Effect

1 In the **Model Builder** window, under **Results** click **Mode Shape, Coupled Analysis 1**.

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

Slice 1

1 In the **Model Builder** window, expand the **Coupling Effect** node, then click **Slice 1**.

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

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

4 In the **Unit** field, type `Pa/m`.

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

6 Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

8 Click **OK**.

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

2 In the **Coupling Effect** toolbar, click **Plot**.

Notice the Eigenfrequency evaluation groups located under the Results node. These are automatically created when an eigenfrequency analysis is carried out. The generated table shows the eigenfrequency, angular frequency, damping ratio, and Q factor.

Eigenfrequencies (Coupled Analysis)

1 In the **Model Builder** window, under **Results** click **Eigenfrequencies (Coupled Analysis)**.

2 In the **Eigenfrequencies (Coupled Analysis)** toolbar, click **Evaluate**.

The final instruction steps are optional. Here you rename the remaining plot groups to reflect which study and physics they refer to. It is always good modeling practice to give plots and plot groups proper names. This makes it easier to debug the models and gives a better overview.

Acoustic Pressure, Acoustics Analysis

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

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Acoustics Analysis` in the **Label** text field.

Sound Pressure Level, Acoustics Analysis

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

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level, Acoustics Analysis` in the **Label** text field.

Acoustic Pressure, Coupled Analysis

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

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Coupled Analysis` in the **Label** text field.

Sound Pressure Level, Coupled Analysis

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

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level, Coupled Analysis` in the **Label** text field.

Acoustic Pressure, Coupled Analysis, Isosurfaces

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

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Coupled Analysis, Isosurfaces` in the **Label** text field.