Created in COMSOL Multiphysics 6.1

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.

Note: This application uses the Acoustic-Shell Interaction, Frequency Domain interface, which is available if you have both the Acoustics Module and the Structural Mechanics Module.

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. Such a coupling is set up automatically by the Acoustic-Structure Boundary multiphysics coupling.

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.

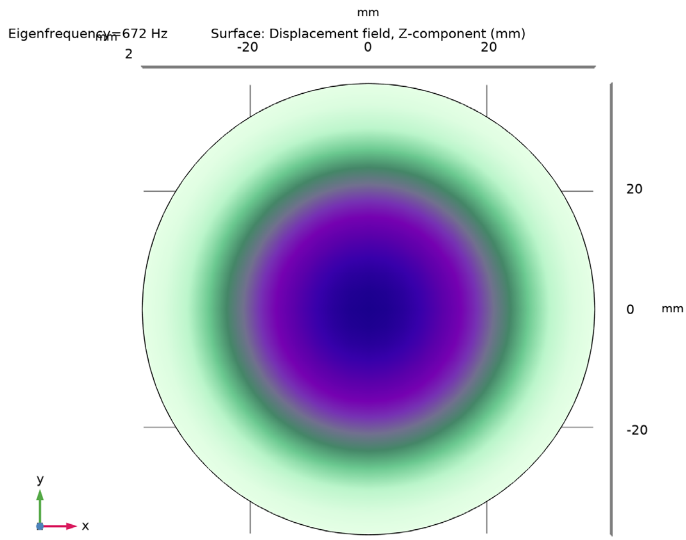

Figure 1: First structural mode displayed by vertical displacement.

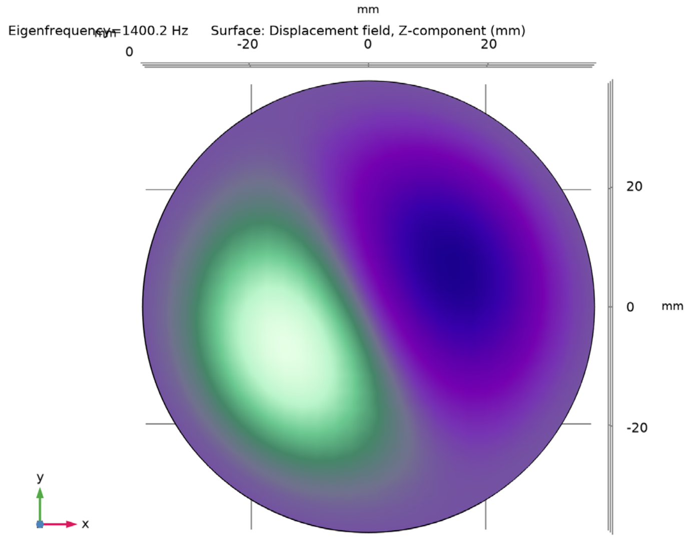

Figure 2: Second structural mode displayed by vertical displacement.

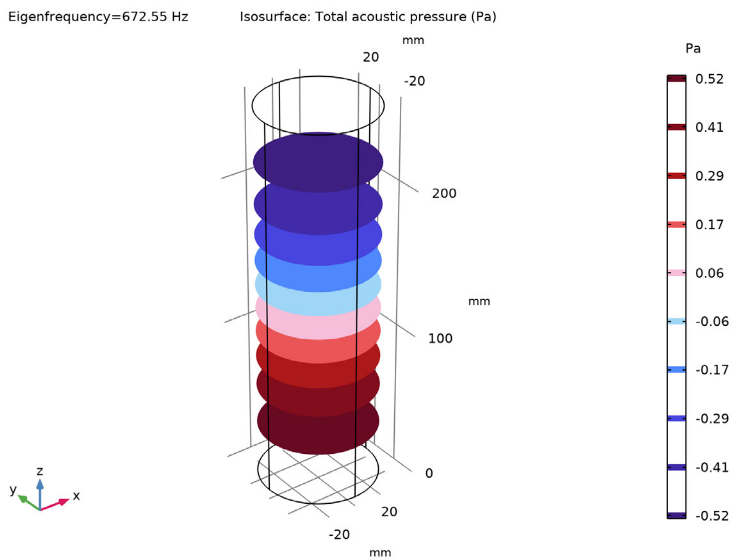

Figure 3: First acoustic mode displayed by pressure isosurfaces.

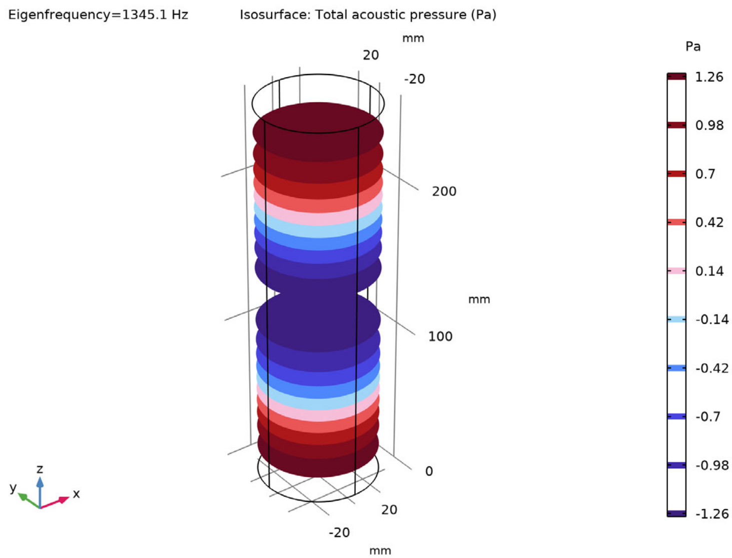

Figure 4: Second acoustic mode displayed by pressure isosurfaces.

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

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

Dominated by	**Semi-analytical (Hz)**	**Computed (Hz)**	**Experimental (Hz)**
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 modes are 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.

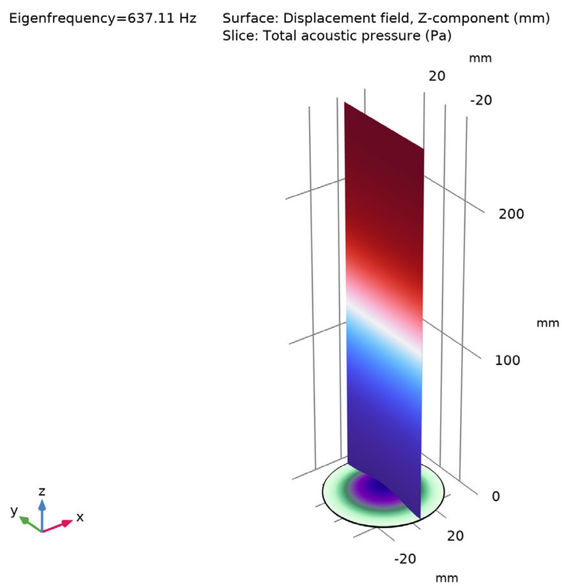

Figure 5: Disc deformation and pressure contours for the first coupled mode.

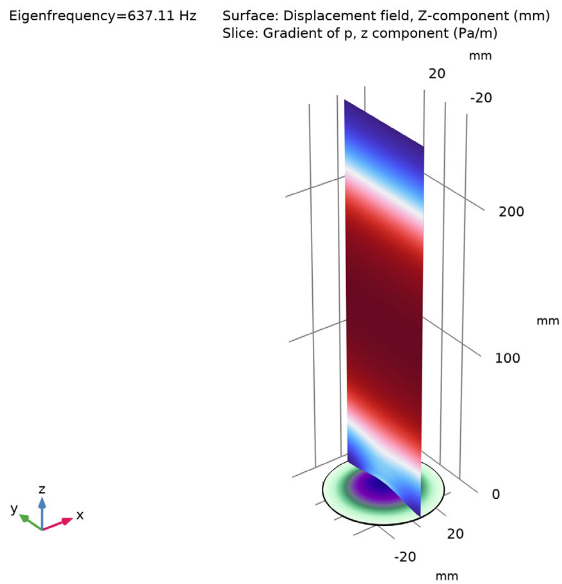

Figure 6: Disc deformation and pressure gradient contours for the first coupled mode.

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.

Notes About the COMSOL Implementation

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

This model uses the Acoustic-Shell Interaction, Frequency Domain interface. If you decide to couple Pressure Acoustics, Frequency Domain to Shell manually, 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 the Acoustic-Structure Boundary multiphysics coupling.

Application Library path: `Acoustics_Module/Verification_Examples/coupled_vibrations`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Shell Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

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

6 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**.

SHELL (SHELL)

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

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

3 Click **Clear Selection**.

4 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

Acoustic Material

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

2 In the **Settings** window for **Material**, type `Acoustic Material` in the **Label** text field.

3 Locate the **Material Contents** section. 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

Structural Material

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

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

3 Locate the **Geometric Entity Selection** section. 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	210[GPa]	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 **Boundary** 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 we use 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.

5 Click **Build Selected**.

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 only the structural problem.

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** text field, type `500`.

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell>Displacement>Displacement field - m>w - Displacement field, Z-component**.

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

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

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

Mode Shape, Structural Analysis

1 In the **Model Builder** window, click **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 a second study to solve the acoustics problem only.

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.

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** text field, type `500`.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Shell (shell)**.

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**.

ROOT

Finally, add a 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** text field, type `500`.

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

RESULTS

Mode Shape, Coupled Analysis

1 In the **Model Builder** window, under **Results** click **Mode Shape (shell)**.

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

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

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

Surface 1

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell> Displacement>Displacement field - m>w - Displacement field, Z-component**.

Slice 1

1 In the **Model Builder** window, right-click **Mode Shape, Coupled Analysis** and choose **Slice**.

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

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

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

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

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

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

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

9 Click **OK**.

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

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

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

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

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

Notice the Eigenfrequency evaluation groups located last 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 of the 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 simplifies debugging 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.