Created in COMSOL Multiphysics 5.6

Modeling Rigid Bodies

Introduction

In many structural dynamics applications, some components are stiff compared to the supporting structure. Such a stiff part only contributes to the dynamic properties of the structure through its mass and moment of inertia. It is then possible to reduce the model size significantly by treating it as a rigid body.

In this example you compute the eigenfrequency of an assembly where one of the parts can be considered as a rigid domain.

Model Definition

An aluminum frame supports a heavy solid part made of steel. Due to the difference in stiffness between the two domains, the steel component can be regarded as rigid.

Figure 1 below shows the assembly geometry with the aluminum bracket shown in gray and the steel domain shown in blue.

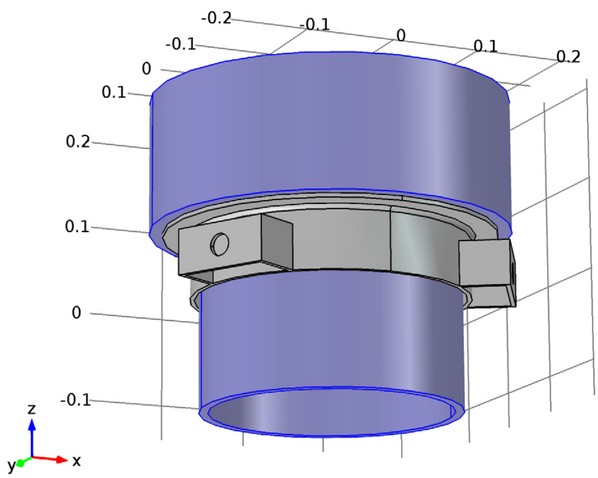


Figure 1: Model geometry with the rigid steel domain shown in blue.

The mounting frame is fixed at the three bolt holes and an eigenfrequency analysis is performed to find the first six natural frequencies of the assembly.

Results and Discussion

The plots in Figure 2 below show the eigenmodes for the first six natural frequencies.

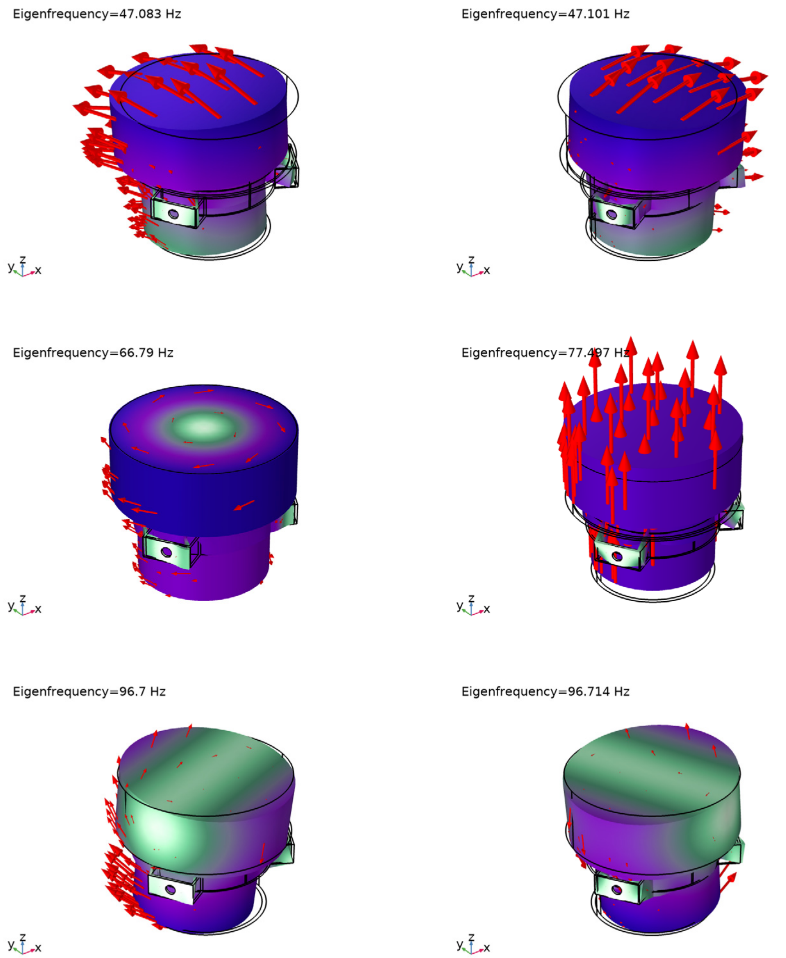


Figure 2: Displacement field for the six lowest eigenmodes.

Eigenfrequencies which are close to each other indicates that corresponding eigenmodes can be similar but due to the model symmetry they occur in different directions. Compare modes 1 with 2 and 5 with 6 in Figure 2.

Notes About the COMSOL Implementation

In COMSOL Multiphysics you can define a rigid domain as a special material model. The mass and moment of inertia properties are then computed, and the rigid domain is represented by only seven degrees of freedom: three for the translations and four for the rotations. The four rotational degrees of freedom are quaternion components which together represent a rotation vector and the rotation angle.

The rigid part still needs to have a mesh, because a numerical integration is performed over the domain in order to obtain the mass properties.The mesh can however be very coarse.

Application Library path: `Structural_Mechanics_Module/Connectors_and_Mechanisms/rigid_domain`

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>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

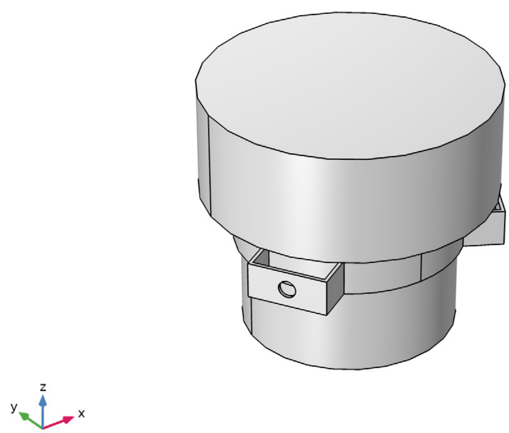
4 Browse to the model's Application Libraries folder and double-click the file `rigid_domain.mphbin`.

5 Click **Import**.

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**.



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>Aluminum**.

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

5 In the tree, select **Built-in>Structural steel**.

6 Click **Add to Component** in the window toolbar.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Structural steel (mat2)

Select Domain 2 only.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.
2. Select Boundaries 11, 12, 28, 29, 48, and 49 only.

Rigid Domain 1

1. In the **Physics** toolbar, click **Domains** and choose **Rigid Domain**.
2. Select Domain 2 only.

STUDY 1

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

RESULTS

Mode Shape (solid)

The default plot shows the mode shape for the first resonance frequency. To improve the visualization, add an arrow plot of the displacements.

Arrow Surface 1

Right-click **Mode Shape (solid)** and choose **Arrow Surface**.

Deformation 1

In the **Model Builder** window, right-click **Arrow Surface 1** and choose **Deformation**.

Arrow Surface 1

1. In the **Settings** window for **Arrow Surface**, click to expand the **Title** section.
2. From the **Title type** list, choose **None**.
3. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
4. Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `100`.
5. Locate the **Coloring and Style** section. Select the **Scale factor** check box.
6. In the associated text field, type `2e5`.

Surface 1

1. In the **Model Builder** window, click **Surface 1**.
2. In the **Settings** window for **Surface**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

To view the solution for the other eigenfrequencies shown in Figure 2, do the following:

In the **Mode shape (solid)** settings window, click the **forward pointing arrow** button to display the solution from the next eigenfrequency.