



Model created in COMSOL Multiphysics 6.4

# 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 eigenfrequencies 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.

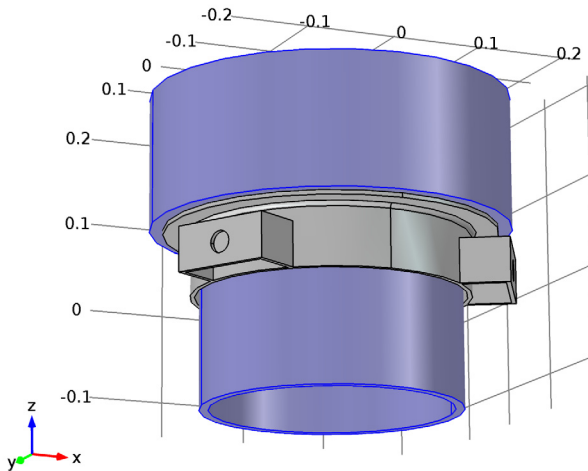
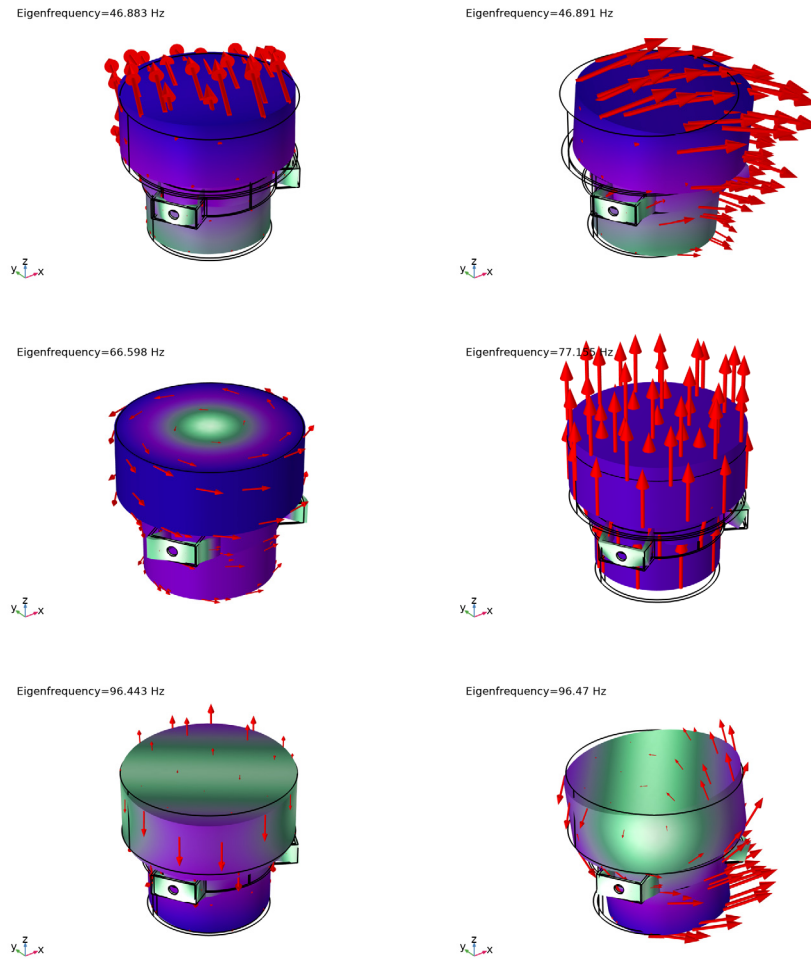


Figure 1: Model geometry with the rigid steel domain shown in blue.

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 indicate that corresponding eigenmodes can be similar, but due to the model symmetry the deformation occurs in different directions. Compare mode 1 with 2, and mode 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 fairly 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 (impl)*

- 1 In the **Geometry** toolbar, click  **Import**.
- 2 In the **Settings** window for **Import**, locate the **Source** section.
- 3 Click  **Browse**.

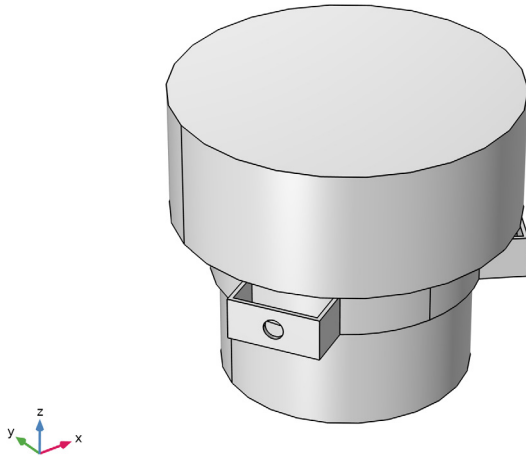
4 Browse to the model's Application Libraries folder and double-click the file rigid\_domain.mphbin.

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**.



## **DEFINITIONS**

### *Bolt Holes*

1 In the **Definitions** toolbar, click  **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select the **Group by continuous tangent** checkbox.


5 Select Boundaries 11, 12, 28, 29, 48, and 49 only.

6 In the **Label** text field, type Bolt Holes.

## **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 > Aluminum**.
- 4 Click the **Add to Component** button in the window toolbar.
- 5 In the tree, select **Built-in > Structural steel**.
- 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

*Structural steel (mat2)*

Select Domain 2 only.

## SOLID MECHANICS (SOLID)


*Fixed Constraint 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.
- 2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Bolt Holes**.

*Rigid Material 1*

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

## STUDY 1

In the **Study** 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 I**.
- 4** Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type 120.
- 5** Locate the **Coloring and Style** section.
- 6** Select the **Scale factor** checkbox. In the associated text field, type 2e5.

#### *Surface I*

- 1** In the **Model Builder** window, click **Surface I**.
- 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 button with the forward pointing arrow to display the solution for the next eigenfrequency.