Created in COMSOL Multiphysics 6.2

Bracket — Rigid Connector Analysis

Introduction

Rigid connectors provide an alternative way for modeling geometrical parts that are of little interest in the analysis and have a negligible deformation. Such parts can be replaced with virtual rigid bodies with appropriate boundary conditions. This saves computational time and memory.

In this example, you study the stress in a bracket connected to a pin where a load is applied. The pin is modeled using rigid connector.

It is recommended that you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This model is an extension of the example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

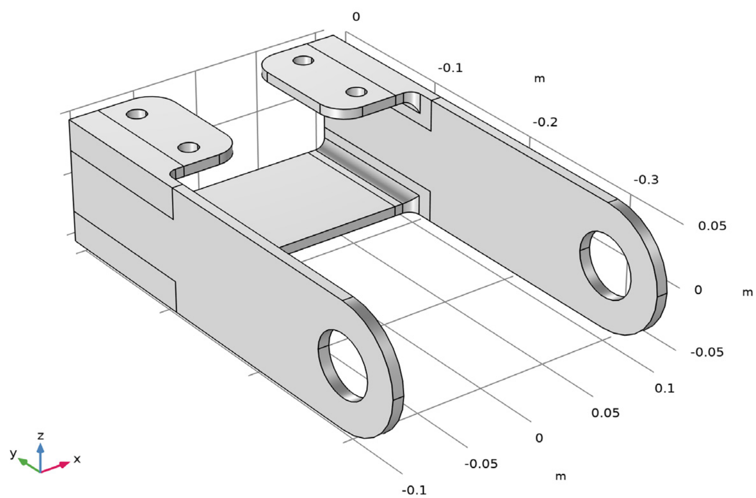

Figure 1: Bracket geometry.

A pin is connected between the bracket hole where a load is applied. The pin is assumed to be perfectly rigid and is modeled with a rigid connector. Thus, the pin is not represented in the model geometry.

The pin is forced to rotate 0.05° around the y-axis. The center of rotation is a fixed point located at (0, -0.4, -0.1) m.

Figure 2 below shows the boundaries connected to the rigid connector and the position of its center of rotation.

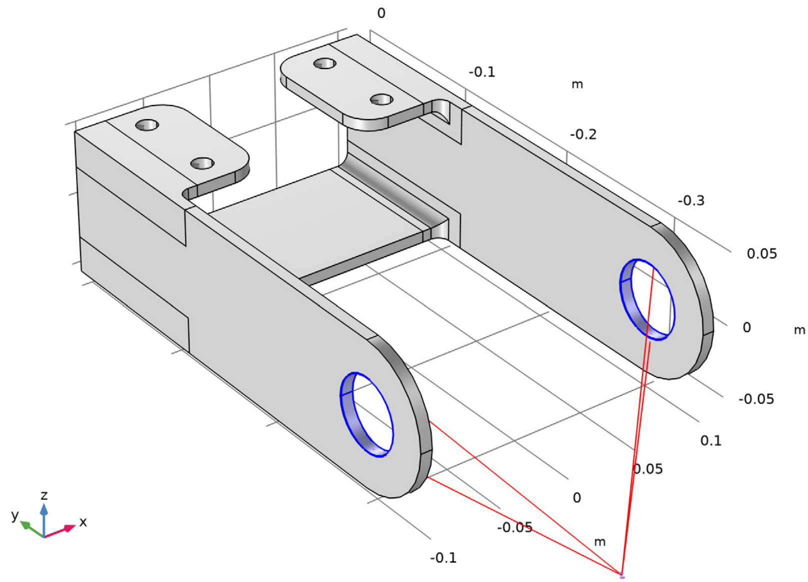

Figure 2: Center of rotation of the rigid connector applied to the bracket geometry.

Results and Discussion

Figure 3 shows the von Mises stress in a deformed geometry. You can see the effect that the pin's rotation and applied force has on the bracket arms.

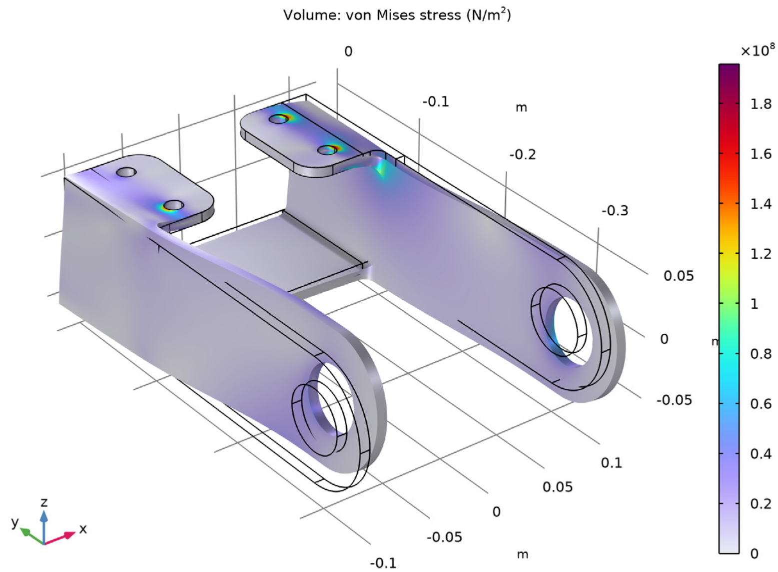

Figure 3: Von Mises stress distribution.

Notes About the COMSOL Implementation

The rigid connector has the translational and rotational degrees of freedom of a rigid body. Options for this feature include applying an external load or moment to the rigid body, or defining mass and moments of inertia in dynamic analyses. The rigid connector can also be used to apply loads and constraints at points so that these are distributed and thereby preventing singularities.

To visualize the position of the center of rotation you need to enable the physics symbols. This is done in the settings for the physics interface.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_rigid_connector`

Modeling Instructions

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Structural Mechanics Module>Tutorials>bracket_basic** in the tree.

3 Click **Open**.

SOLID MECHANICS (SOLID)

Rigid Connector 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Connections>Rigid Connector**.

Add a rigid connector that connects the holes in the bracket arms to simulate the presence of the pin.

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

3 From the **Selection** list, choose **Pin Holes**.

By default, the location of the center of rotation is computed automatically. You can also manually specify its location.

4 Locate the **Center of Rotation** section. From the list, choose **User defined**.

5 Specify the $\mathbf{X}_c$ vector as

0	x
-0.40	y
-0.10	z

To visualize its position you need to enable the physics symbols.

6 In the **Model Builder** window, click **Solid Mechanics (solid)**.

7 In the **Settings** window for **Solid Mechanics**, locate the **Physics Symbols** section.

8 Select the **Enable physics symbols** check box.

The displacements of the rigid connector are constrained in the $x-$ and $z-$ directions at its center of rotation.

9 In the **Model Builder** window, click **Rigid Connector 1**.

10 In the **Settings** window for **Rigid Connector**, locate the **Prescribed Displacement at Center of Rotation** section.

11 Select the **Prescribed in x direction** check box.

12 Select the **Prescribed in z direction** check box.

13 Locate the **Prescribed Rotation** section. From the **By** list, choose **Prescribed rotation**.

Apply a prescribed rotation of the rigid body of 0.05 degrees around the *y*-axis.

14 Specify the Ω vector as

0	x
1	y
0	z

15 In the ϕ_0 text field, type `0.05[deg]`.

Applied Force 1

1 In the **Physics** toolbar, click **Attributes** and choose **Applied Force**.

Apply an external load of 10 kN at the center of rotation of the rigid body.

2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.

3 Specify the **F** vector as

0	x
-10[kN]	y
0	z

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>Stationary**.

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

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

STUDY 1

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

RESULTS

Stress (solid)

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

The default plot shows the von Mises stress on a deformed geometry.