Created in COMSOL Multiphysics 5.6

Assembly with a Hinge

Introduction

In mechanical assemblies, parts are sometimes connected so that they are free to move relative to each other in one or more degrees of freedom. Examples of such connections are ball joints, hinges, and different types of bearings. If the details of the connection are not the subjects of the analysis, it is often possible to model the connection using the Rigid Connector feature in COMSOL Multiphysics combined with some extra equations.

The current example illustrates how to model a barrel hinge connecting two solid objects in an assembly.

You can create a model like this much more conveniently by using the Multibody Dynamics Module. There you will for example find a predefined hinge joint.

Model Definition

Figure 1 shows the model geometry.

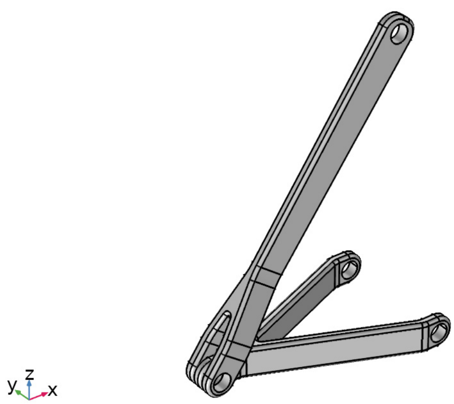

Figure 1: Model geometry.

The two parts of the assembly are connected through a barrel hinge that allows relative rotation only along the axis of the pin hole. All other degrees of freedom are common between the two parts.

The two holes of the forked bottom part are bolted, and can be considered as fully constrained.

The pin hole of the top part is constrained in the X direction so that it can slide in the YZ-plane.

A force of 1 kN is applied in the Z direction at a distance 10 cm in the negative Y direction from the center of the upper pin hole. The offset of the load thus introduces both tension and bending of the member.

Results and Discussion

The default plot shown in Figure 2 shows the von Mises stress in the model. You can see the bending of the top part due to the offset of the load. You can also see the stress that is transmitted around the hinge.

In Figure 3 the color gradient in the X direction displacement indicates that the lower, forked, part of the assembly is subjected to bending around the Y axis. Without the hinge, bending from one part would be transmitted to the next part. The upper part, however, shows a fairly constant displacement along the height, which indicates that it has a free rotation around the Y direction in the connection point.

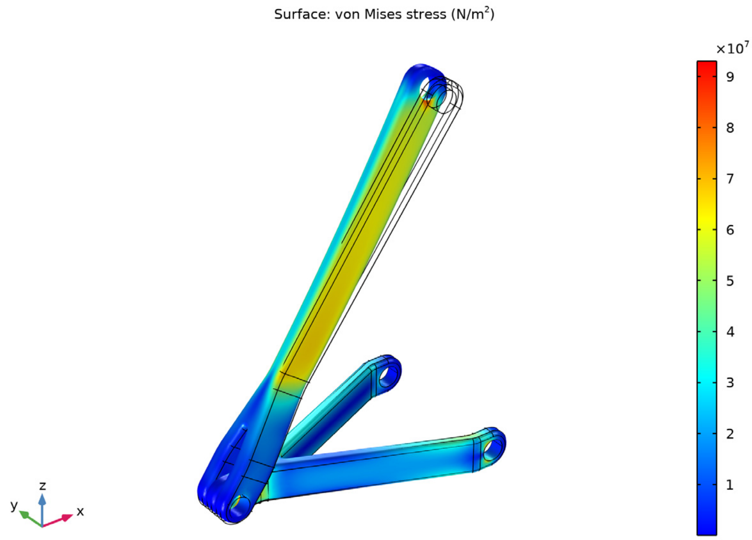

Figure 2: von Mises stress distribution in the hinge assembly.

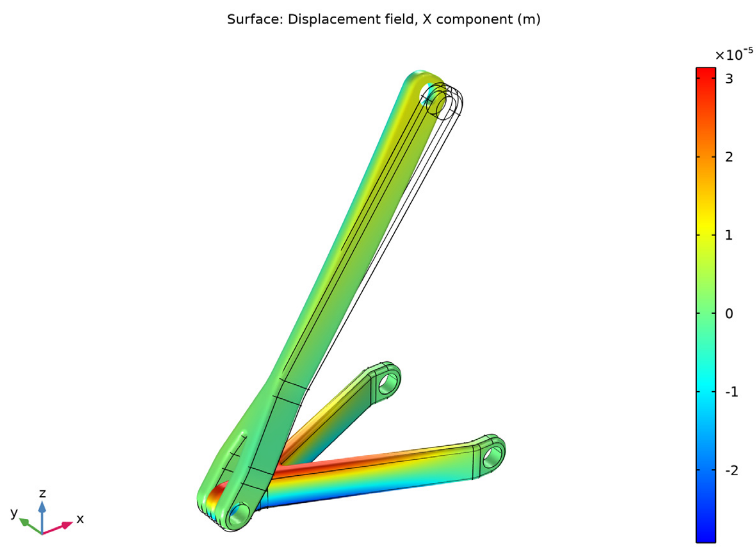

Figure 3: X direction displacement.

Notes About the COMSOL Implementation

The approach when modeling the mechanism is to attach rigid connectors to both parts, make sure that they have a common center of rotation, and then couple relevant degrees of freedom between them in order to obtain the desired function.

When you model a hinge, all translations and two rotations should be equal in the two parts. As in this case, the displacement and the rotation in the hinge remain small, the procedure simply consists of linking the displacement in all directions as well as the rotation around the X and Z directions. You connect the displacements of the rigid connectors directly using the prescribed displacement setting available in the rigid connector node. In order to constrain the rotation directions independently, you need however to add two global constraints, one for each rotational degree of freedom. The rigid connector uses a quaternion representation of the rotation. For details, see the description in the *Structural Mechanics User's Guide*. In this specific example both the b and the d rotation variables are coupled between the two rigid connectors, because they directly correspond to the rotation around X and Z axes for small rotations.

Another rigid connector is used for applying the force to the upper pin hole.

Application Library path: `Structural_Mechanics_Module/Connectors_and_Mechanisms/hinge_assembly`

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>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
F	1e3[N]	1000 N	Applied load

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 `hinge_assembly.mphbin`.

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 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>Structural steel**.

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

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

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 133–136 only.

Rigid Connector 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundaries 67 and 68 only.

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

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

Applied Force 1

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

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

3 Select the **Offset** check box.

4 Specify the $\mathbf{X}_{\mathrm{offset}}$ vector as

0	x
-0.1	y
0	z

5 Locate the **Applied Force** section. Specify the $\mathbf{F}$ vector as

0	x
0	y
F	z

Rigid Connector 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundaries 75 and 76 only.

Rigid Connector 3

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundaries 16–19 only.

Set the translations of this rigid connector equal to the translations of the second rigid connector.

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

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

5 In the u_{0x} text field, type `solid.rig2.u`.

6 Select the **Prescribed in y direction** check box.

7 In the u_{0y} text field, type `solid.rig2.v`.

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

9 In the u_{0z} text field, type `solid.rig2.w`.

10 Click the **Show More Options** button in the **Model Builder** toolbar.

11 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

12 Click **OK**.

Connect the rotations around the x and z directions, using the quaternion degrees of freedom 'b' and 'd'.

Global Constraint 1

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

2 In the **Settings** window for **Global Constraint**, locate the **Global Constraint** section.

3 In the **Constraint expression** text field, type `comp1.solid.rig2.b-comp1.solid.rig3.b`.

Global Constraint 2

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

2 In the **Settings** window for **Global Constraint**, locate the **Global Constraint** section.

3 In the **Constraint expression** text field, type `comp1.solid.rig2.d-comp1.solid.rig3.d`.

MESH 1

Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size 1

1. Right-click **Free Tetrahedral 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 41, 42, and 53 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7. In the associated text field, type `0.002`.

Size

1. In the **Model Builder** window, click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Fine**.
4. Click **Build All**.

STUDY 1

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

RESULTS

Stress (solid)

The default plot shows the von Mises stress distribution on the surface of the assembly. Compare with Figure 2.

Finally, reproduce the *x*-displacement plot shown in Figure 3 with the following steps:

Stress (solid) 1

Right-click **Stress (solid)** and choose **Duplicate**.

Surface 1

1. In the **Model Builder** window, expand the **Stress (solid) 1** 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)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X component**.
3. In the **Stress (solid) 1** toolbar, click **Plot**.

x-Displacement

1. In the **Model Builder** window, under **Results** click **Stress (solid) 1**.
2. In the **Settings** window for **3D Plot Group**, type `x-Displacement` in the **Label** text field.