Model created in COMSOL Multiphysics 6.4

Assembly with a Hinge Joint

Introduction

In mechanical assemblies, parts are sometimes connected so that they are free to move relative to each other with one or multiple 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 possible to model the connection using joint nodes in the Multibody Dynamics Module.

This example illustrates how to model a barrel hinge connecting two solid objects in an assembly. Two different versions are studied. In one of them, both parts are flexible. In the other version, one of the parts is considered as rigid.

Model Definition

Figure 1 shows the model geometry.

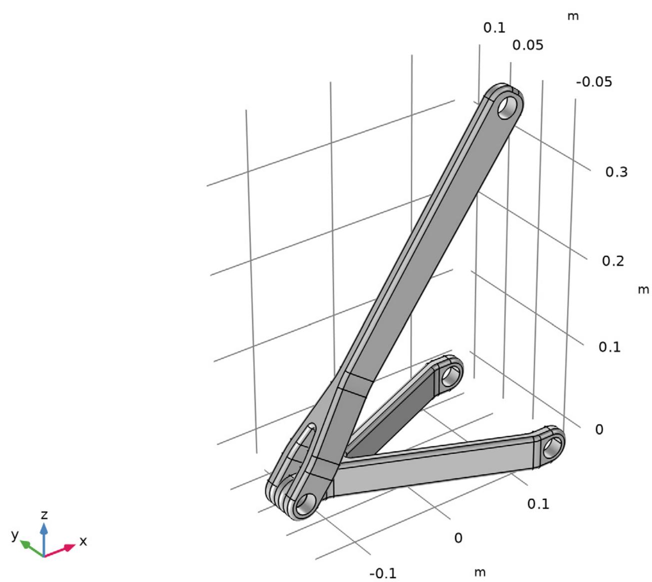

Figure 1: Model geometry.

Two parts of the assembly are connected through a barrel hinge that allows relative rotation as well as sliding along the axis of the pin hole. All the other degrees of freedom are shared between the two parts.

One hole of the forked bottom part is modeled as a hinge joint and other hole is modeled as a cylindrical joint allowing the axial motion.

The pin hole of the top part is constrained in the x direction so that it can slide in the y-z plane.

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

In a second study, the upper part is considered as rigid, and it only transmits the force through the hinge.

Results and Discussion

The default displacement plot for the flexible model is shown in Figure 2.

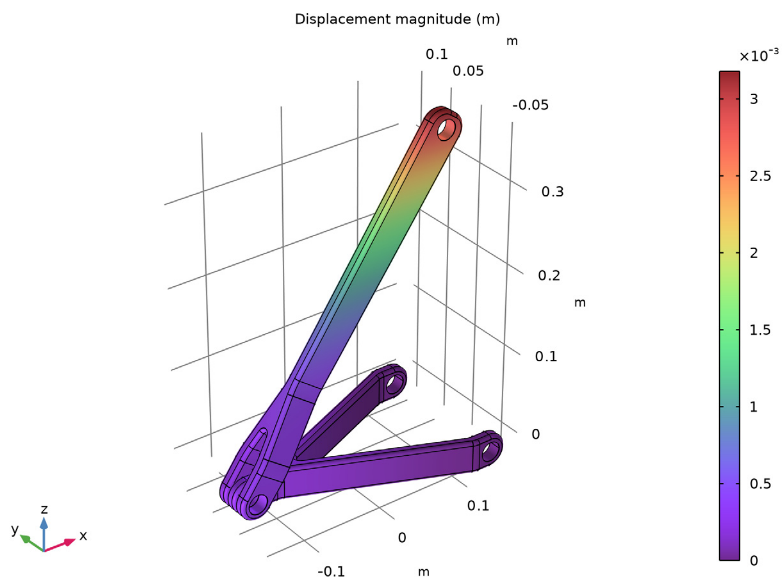

Figure 2: Displacement of the flexible structure.

In Figure 3 you can see the stress distribution.

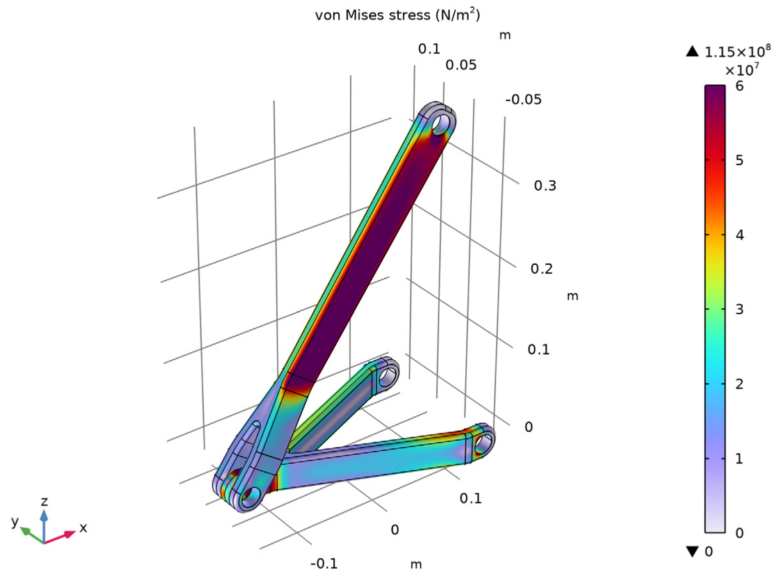

Figure 3: Equivalent stress in the flexible structure.

In a model that consists of a mix of rigid and flexible parts, stresses can only be displayed in the flexible parts. This is shown in Figure 4.

If you compare Figure 3 and Figure 4, you can see that the stress distribution in the lower part is essentially unaffected by the fact that the load is transmitted through a rigid body.

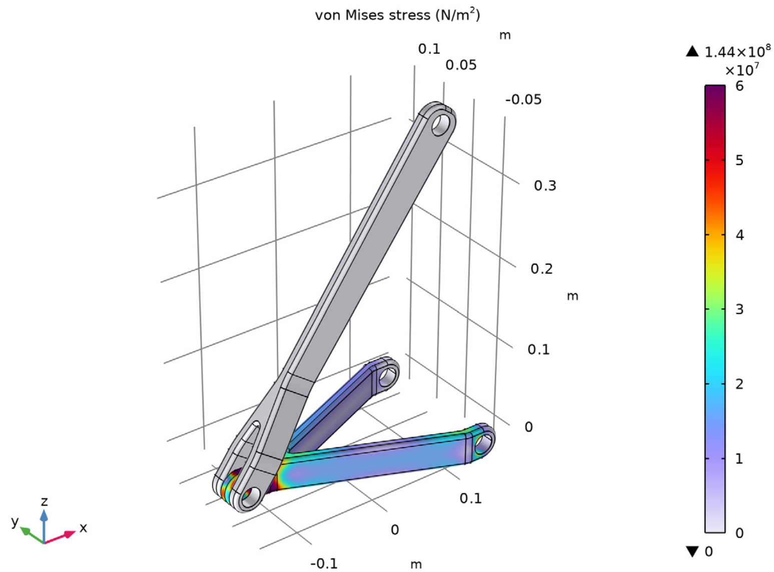

Figure 4: Equivalent stress in the flexible part when one component is rigid.

Notes About the COMSOL Implementation

When the flexibility of a component can be neglected and its stress distribution is not of interest, it is efficient to treat such a component as a rigid domain. The rigid domain is a domain that has a material model with only one material parameter, the density.

A **Joint** node can establish a connection directly between **Rigid Material** nodes however **Attachment** nodes are needed, defining the connection boundaries, for flexible elements.

Application Library path: `Multibody_Dynamics_Module/Tutorials/hinge_joint_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 > Multibody Dynamics (mbd)**.

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 **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 `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** checkbox.

5 Click **Build Selected**.

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** > **Structural steel**.

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

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

MULTIBODY DYNAMICS (MBD)

Fixed Constraint 1

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

2 Select Boundaries 133–136 only.

Use **Rigid Connector** node to constrain the motion and apply load.

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** checkbox.

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** checkbox.

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

0	x
-0.1	y
0	z

The center of rotation for a rigid connector is available in the variables xcx_*tag*, xcy_*tag*, and xcz_*tag*. The default position is the center of gravity of the attached boundaries, which in this case will be the center of the hole.

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

0	x
0	y
F	z

Attachment 1

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

2 Select Boundaries 16 and 17 only.

3 In the **Settings** window for **Attachment**, locate the **Connection Type** section.

4 From the list, choose **Flexible**.

Attachment 2

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

2 Select Boundaries 18 and 19 only.

3 In the **Settings** window for **Attachment**, locate the **Connection Type** section.

4 From the list, choose **Flexible**.

Attachment 3

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

2 Select Boundaries 75 and 76 only.

3 In the **Settings** window for **Attachment**, locate the **Connection Type** section.

4 From the list, choose **Flexible**.

Hinge Joint 1

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

2 In the **Settings** window for **Hinge Joint**, locate the **Attachment Selection** section.

3 From the **Source** list, choose **Attachment 1**.

4 From the **Destination** list, choose **Attachment 3**.

5 Locate the **Axis of Joint** section. Specify the $\mathbf{e}_0$ vector as

0	x
1	y
0	z

Cylindrical Joint 1

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

2 In the **Settings** window for **Cylindrical Joint**, locate the **Attachment Selection** section.

3 From the **Source** list, choose **Attachment 2**.

4 From the **Destination** list, choose **Attachment 3**.

5 Locate the **Axis of Joint** section. Specify the $\mathbf{e}_0$ vector as

0	x
1	y
0	z

If you want accurate stress results, then quadratic shape functions should be used for the displacements. The default value in the Multibody Dynamics interface is linear shape functions.

6 In the **Model Builder** window, click **Multibody Dynamics (mbd)**.

7 In the **Settings** window for **Multibody Dynamics**, click to expand the **Discretization** section.

8 From the **Displacement field** list, choose **Quadratic Lagrange**.

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.

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.002`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** 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 **Study** toolbar, click **Compute**.

RESULTS

Displacement (mbd)

1 In the **Displacement (mbd)** toolbar, click **Plot**.

The default plot shows the displacement of the assembly. Compare with Figure 2.

Reproduce the stress plot shown in Figure 3 with the following steps.

Stress

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Stress` in the **Label** text field.

Surface 1

1 Right-click **Stress** and choose **Surface**.

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)** > **Multibody Dynamics** > **Stress** > **mbd.misesGp - von Mises stress - N/m²**.

3 In the **Stress** toolbar, click **Plot**.

4 Click to expand the **Range** section. Select the **Manual color range** checkbox.

5 In the **Maximum** text field, type `6E7`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Stress

1 In the **Model Builder** window, click **Stress**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Select the **Show maximum and minimum values** checkbox.

4 In the **Stress** toolbar, click **Plot**.

Make the upper lever rigid and analyze that configuration in a new study.

MULTIBODY DYNAMICS (MBD)

Rigid Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.

2 Select Domain 1 only.

Use **Rigid Material** subnodes to constrain the motion and apply load.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Displacement/Rotation**.

2 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement at Center of Rotation** section.

3 Select the **Prescribed in x direction** checkbox.

4 Locate the **Center of Rotation** section. From the list, choose **Centroid of selected entities**.

Center of Rotation: Boundary 1

1 In the **Model Builder** window, expand the **Prescribed Displacement/Rotation 1** node, then click **Center of Rotation: Boundary 1**.

2 Select Boundaries 67 and 68 only.

Rigid Material 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Multibody Dynamics (mbd)** click **Rigid Material 1**.

Applied Force 1

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

Similar to the previous case, point of load application can be written like as follows:

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

3 From the list, choose **User defined**.

4 Specify the $\mathbf{X}_p$ vector as

mbd.rd1.pdr1.xcx	x
mbd.rd1.pdr1.xcy-0.1	y
mbd.rd1.pdr1.xcz	z

5 Locate the **Applied Force** section. Specify the **F** vector as

0	x
0	y
F	z

Add an extra **Hinge Joint** node, so that it is easy to rerun the original version of the model. Normally it would have been easier just to utilize the existing joint.

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 the **Add Study** button in the window toolbar.

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

STUDY 2

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

RESULTS

Stress

Duplicate the stress plot used for the fully flexible model to get Figure 4.

Stress 1

1 In the **Model Builder** window, right-click **Stress** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 In the **Stress 1** toolbar, click **Plot**.

Make sure that the first study still solves for the flexible structure.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Multibody Dynamics (mbd), Controls spatial frame** > **Rigid Material 1**.

5 Click **Disable**.