Created in COMSOL Multiphysics 6.1

Double-Pendulum Dynamics

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This tutorial model shows the modeling of a double pendulum using the Multibody Dynamics interface in COMSOL Multiphysics. The example begins with the creation of a simple hinge joint and then extends the model with several nodes available for joints such as constraints, locking, spring, damper, prescribed motion, and friction.

Model Definition

The model describes the motion of a double pendulum under gravity. The double pendulum system shown in Figure 1 behaves linearly for small angles of rotation, but it becomes highly nonlinear as the angle of rotation increases, eventually leading to a chaotic system.

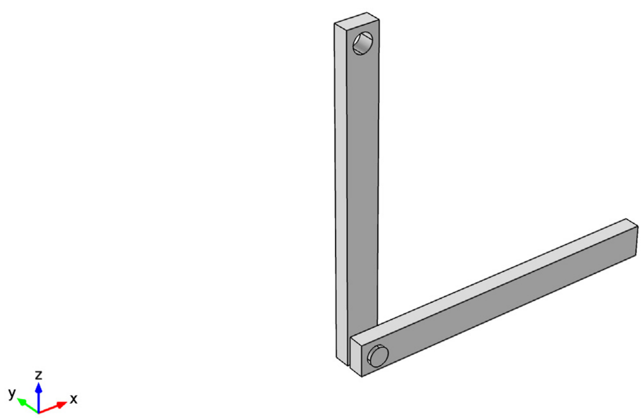

Figure 1: Geometry of the double pendulum.

Here, the arms of the double pendulum are connected through a hinge joint. A hinge joint has one rotational degree of freedom about the joint axis. The remaining degrees of freedom of both components defining the joint are constrained to have the same values at the center of the joint.

The complete example is divided into six parts in order to illustrate the available functionality on the joint node and related subnodes.

A new study is added for each of the cases to provide more clarity. This also helps in storing all the model settings, which is needed when recomputing the solution in the solved model.

The different cases are as follows.

- Case-1: Basic hinge joint
- Case-2: Constraints on joint
- Case-3: Locking on joint
- Case-4: Spring and damper on joint
- Case-5: Prescribed motion on joint
- Case-6: Friction on joint

The details of each case along with the results and modeling instructions are given at a later point in this document.

Notes About the COMSOL Implementation

- For the first three cases, both arms are modeled as flexible elements using the **Linear Elastic Material** model, and in the last three cases they are modeled as rigid elements using the **Rigid Material** model. Rigid elements can be used if the stresses and the deformation in the components are of no interest.
- A **Joint** node can directly establish a connection between **Rigid Material** nodes. However, in flexible elements, **Attachment** nodes are needed to define the connection boundaries.
- Constraint boundary conditions, like **Rigid Connector** and **Fixed Constraint**, cannot be used with the **Rigid Material** node. Hence, the **Prescribed Displacement/Rotation** node (subnode of **Rigid Material**) is used to prescribe the degrees of freedom.

Application Library path: `Multibody_Dynamics_Module/Tutorials/double_pendulum`

Model Definition: Case-1 (Basic Hinge Joint)

The top surface of the upper arm is constrained so that it cannot translate but it is free to rotate about the y-axis. The whole system is placed in the gravitational field to analyze the dynamics of the system.

In this case, the arms of the pendulum are modeled as flexible elements. Hence, the stresses generated in the components can be evaluated while computing the system's dynamics.

Results and Discussion

Figure 2 and Figure 3 show the relative rotation between the arms and the reaction forces at the hinge joint during the oscillation of the double pendulum. The x and z components of the reaction forces balance the gravity load in the lower arm, as the pendulum oscillates in the xz-plane.

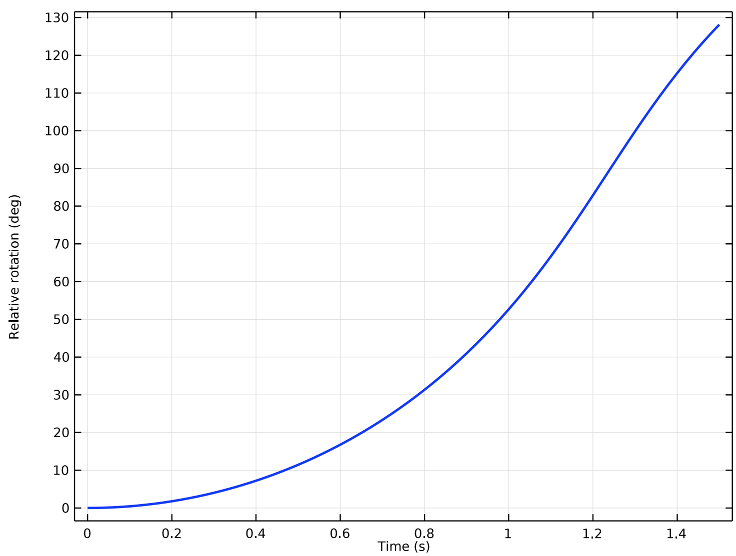

Figure 2: Relative rotation of the arms at the hinge joint (Case-1).

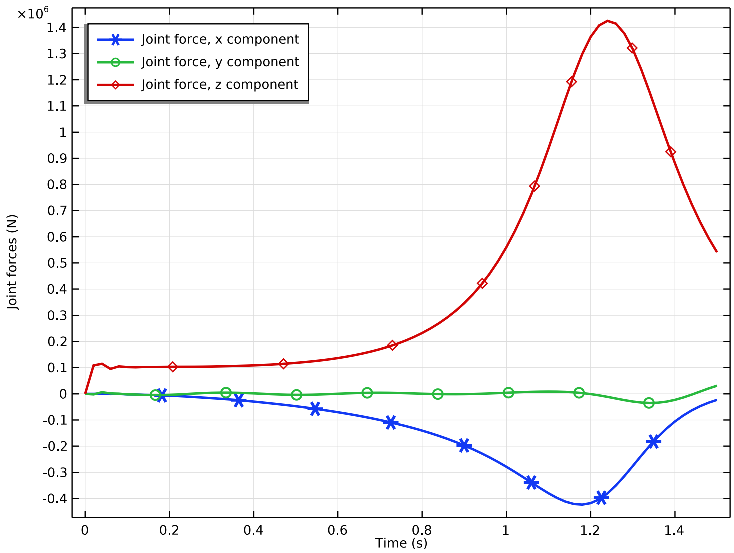

Figure 3: Reaction forces at the hinge joint (Case-1).

Model Definition: Case-2 (Constraints on Joint)

The top surface of the upper arm is fixed. A constraint is added at 45° relative rotation between both the arms. The whole system is placed in the gravitational field and the dynamics is analyzed.

Both arms are modeled as flexible parts. The deformation, as well as the stresses generated in the components, is significant during and after the activation of the constraint condition.

Results and Discussion

The relative rotation between the arms, when a constraint is added to the joint, is shown in Figure 4. In this case, the relative rotation first increases due to the gravitational force; then it decreases after reaching the constrained maximum limit as the lower arm bounces back. The relative rotation stays at its maximum limit for certain duration. This happens because of the bending of the flexible arms due to the high moment of inertia, and the sudden application of the constraint condition.

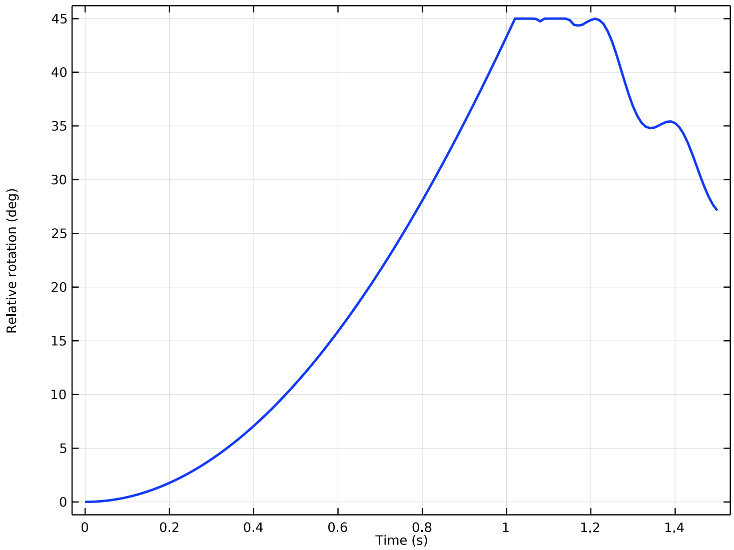

Figure 4: Relative rotation of the arms at the hinge joint (Case-2).

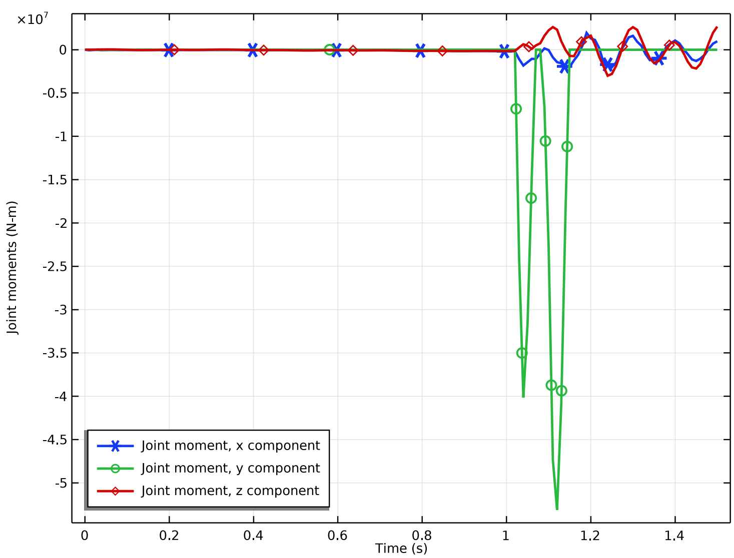

Figure 5: Reaction moments at the hinge joint (Case-2).

Figure 5 shows the reaction moments at the joint when using a constraint condition. The joint allows the arms to rotate about the y-axis, hence the reaction moment should be zero in this direction. However, when the constraint condition is active, the same joint restricts this motion and hence, a high reaction moment in the y direction can be seen. After the constraint condition has stopped the relative motion, the pendulum tries to rotate about other two axes due to its unsymmetrical geometry. Therefore, nonzero moments in these directions are also visible.

Figure 6 shows the variation of different forms of energy in the system when using a constraint condition. Before the constraint condition becomes active, potential energy is converted into kinetic energy and the strain energy is negligible. During the period when the constraint condition is active, the relative velocity goes to zero before changing sign. In this period, all kinetic energy is transformed into strain energy. When the constraint condition is no longer in action, the strain energy is converted back into kinetic energy. Structural waves persist in the components due to their flexible nature, therefore a nonzero strain energy can be seen.

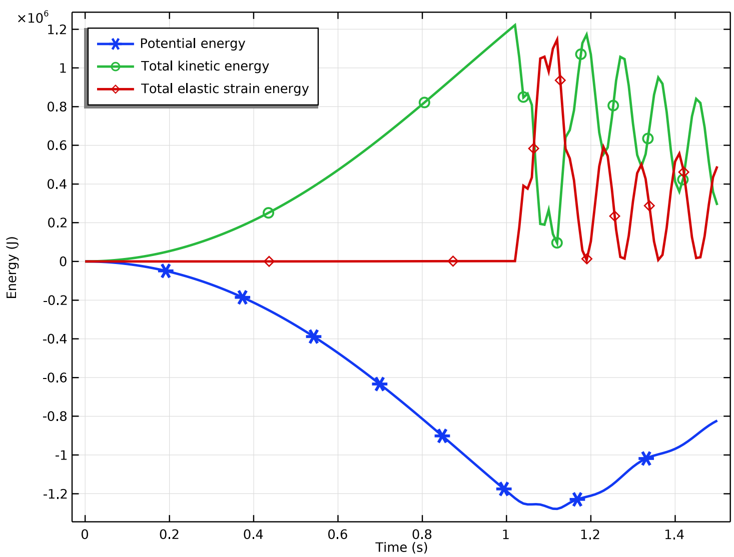

Figure 6: Time variation of different forms of energy in Case-2.

Model Definition: Case-3 (Locking on Joint)

The top surface of the upper arm is constrained such that it cannot translate but is free to rotate about the y-axis. A lock condition is added at 45° relative rotation between both the arms of the pendulum. The whole system is placed in the gravitational field and the dynamics of the system is analyzed. Both the arms of the pendulum are modeled as flexible elements.

Results and Discussion

Figure 7 shows the relative rotation between the arms when a lock is present in the joint. In this case, the relative rotation first increases due to the gravitational force and then it is locked at the constrained maximum limit. The relative velocity is forced to zero for the rest of the simulation once the relative rotation reaches the lock position.

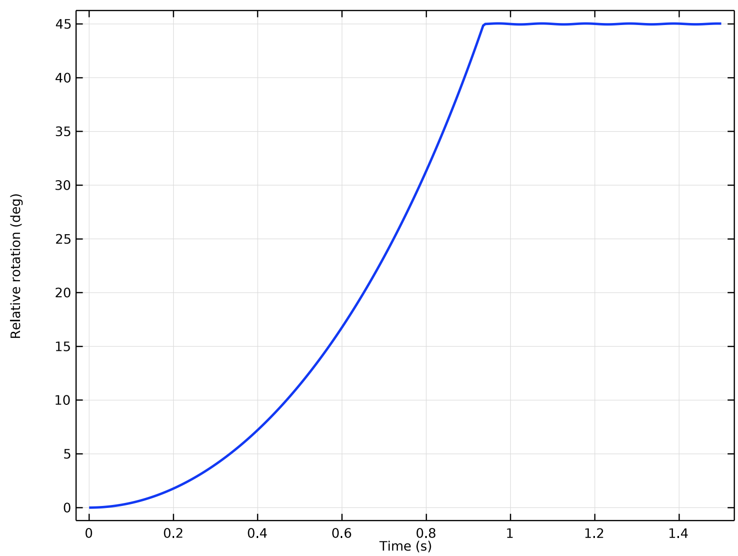

Figure 7: Relative rotation of the arms at the hinge joint (Case-3).

Figure 8 shows the reaction moments at the joint when a lock condition is used. The joint initially allows the arms to rotate relative to each other about the y-axis so that the reaction moment should be zero in this direction. However, after applying the lock condition, the same joint restricts this motion so that a high reaction moment in the y direction can be seen. After the lock condition is active, the pendulum also tries to rotate about the other two axes because of its unsymmetrical geometry. Therefore, nonzero moments in these directions are also visible.

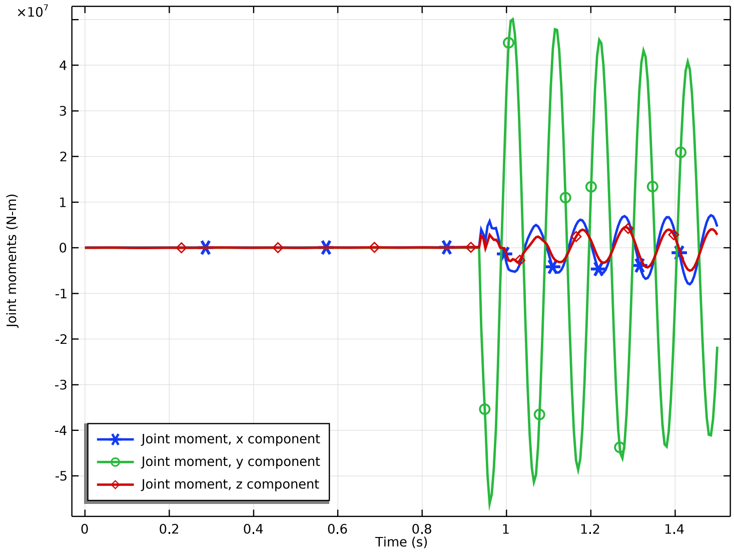

Figure 8: Reaction moments at the hinge joint (Case-3).

Model Definition: Case-4 (Spring and Damper on Joint)

In this case, both arms of the pendulum are modeled as rigid elements. The upper arm is fixed. A torsional spring and damper are added on the relative rotation between the arms. The spring constant and the damping coefficient are **5e6 N-m** and **1e6** N-m-s, respectively. The dynamics of the system is analyzed under gravity load.

Results and Discussion

Figure 9 shows the relative rotation between the arms when a spring and a damper are added to the joint. In this case, the torsional spring tries to restrict the relative motion at the joint and balances the moment caused by the gravity load. Without the damper the arms would oscillate about the equilibrium position. Here, a damper is added to the joint, so that the fluctuation in the relative rotation decreases and the arms tend to the equilibrium position.

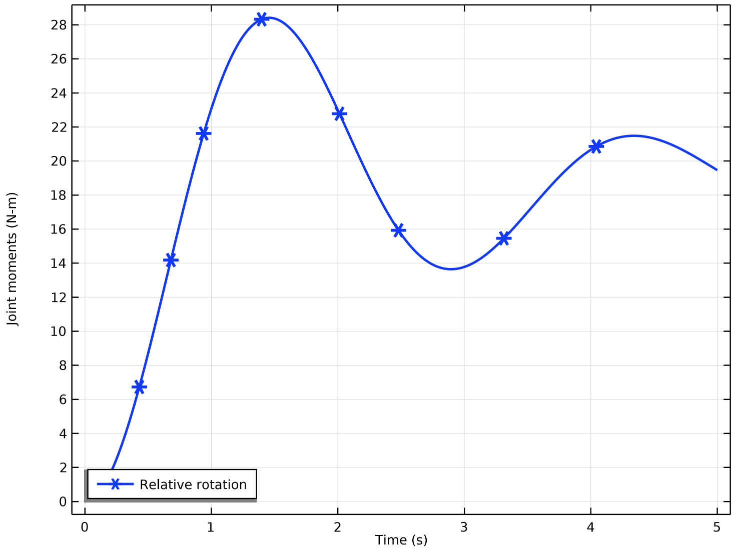

Figure 9: Relative rotation of the arms at the hinge joint (Case-4).

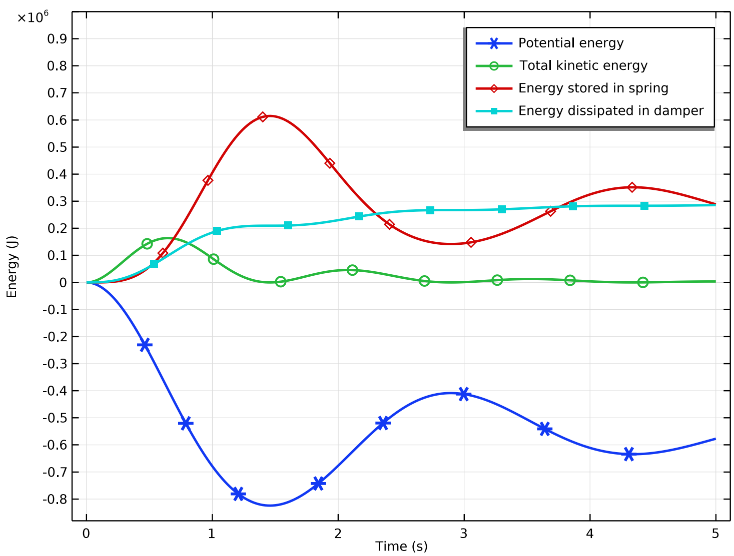

Figure 10: Time variation of different forms of energy (Case-4).

Figure 10 shows the variation of different forms of energy in the system when a spring and damper are added to the joint. Initially, the potential energy is converted into kinetic energy. After some time, the spring and damper effects will however dominate. The damper dissipates energy and thus the kinetic energy is reduced to almost zero. As the energy dissipated in the damper is proportional to the velocity, it tends to a constant value. The energy stored in the spring is slowly converted into potential energy and vice-versa.

Model Definition: Case-5 (Prescribed Motion on Joint)

As in the previous case, arms of the pendulum are modeled as rigid elements. The upper arm is fixed. The relative angular velocity between both arms is prescribed for certain duration, after which each joint degree of freedom is made free. The prescribed angular velocity is a linear function of time (t rad/s) and reaches it's maximum value 1 rad/s at 1 s.

Results and Discussion

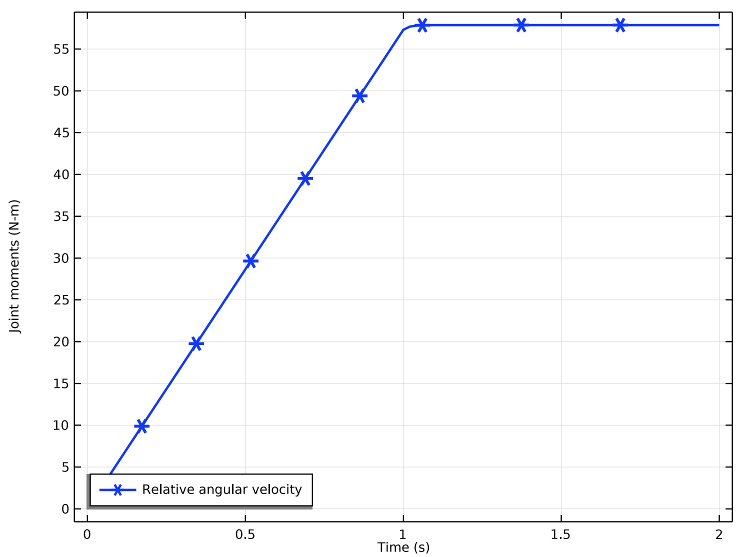

Figure 11: Relative angular velocity of the arms at the hinge joint (Case-5).

Figure 11 shows the relative angular velocity between the arms when prescribed for the given duration. The velocity is prescribed until 1 s, and hence, it increases to it's maximum value in this time interval. After that, due to the inertia of the components and without any losses, the velocity is maintained to the maximum value for the rest of the simulation. The relative rotation between the arms is shown in Figure 12. It is clear from this plot that before 1 s, the rotation increases quadratically with time, after which time it increases linearly.

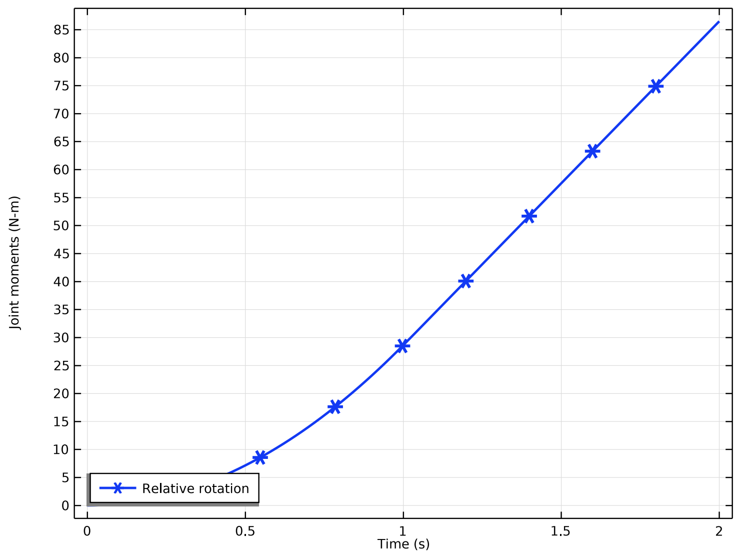

Figure 12: Relative rotation of the arms at the hinge joint (Case-5).

Model Definition: Case-6 (Friction on Joint)

As in the previous case, the arms of the pendulum are modeled as rigid bodies. The upper arm is fixed. The dynamics of the system is analyzed under gravity load and with frictional losses in the joint. The effective friction moment is computed using the forces acting on a joint and it is applied on the relative rotation between the arms.

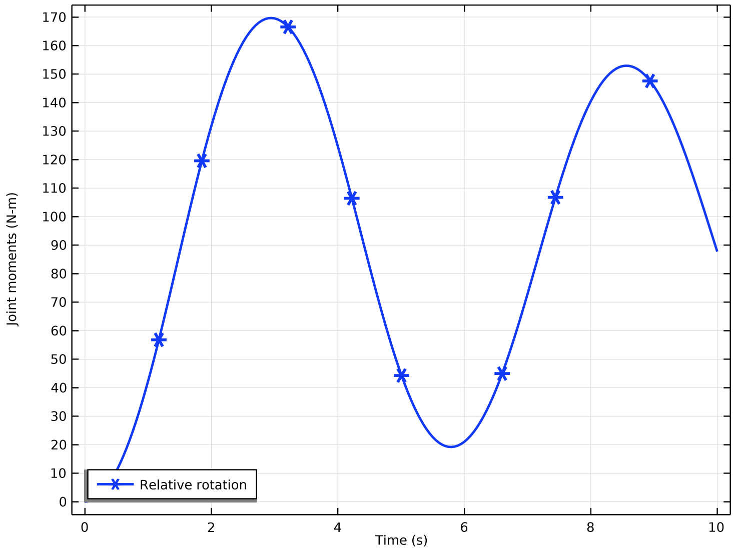

Figure 13: Relative rotation of the arms at the hinge joint (Case-6).

Figure 13 shows the relative rotation between the arms. The decay in the magnitude of relative rotation due to the frictional losses at the hinge joint, can be seen.

The time variation of the friction moment is shown in Figure 14. The average value of friction moment is governed by the gravity load acting on both the arms and the fluctuations in friction moment are governed by the inertial forces in both the arms. A fast variation in the friction moment can be seen when the direction of motion is getting reversed. This mimics the behavior of Coulomb's friction law, but the friction moment is not fully discontinuous as it would be in Coulomb's friction law.

Figure 15 shows the variation of different forms of energy in the system when frictional losses are added to the joint. The increasing frictional energy loss can be seen in the plot.

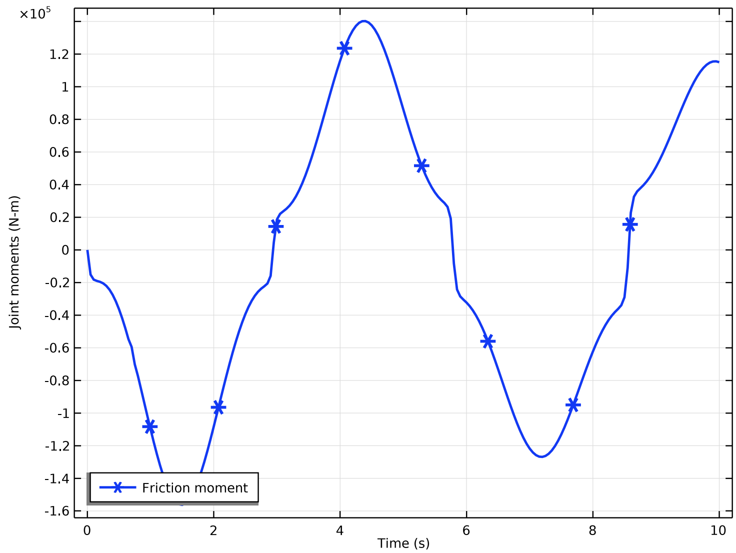

Figure 14: Friction moment profile at the hinge joint (Case-6).

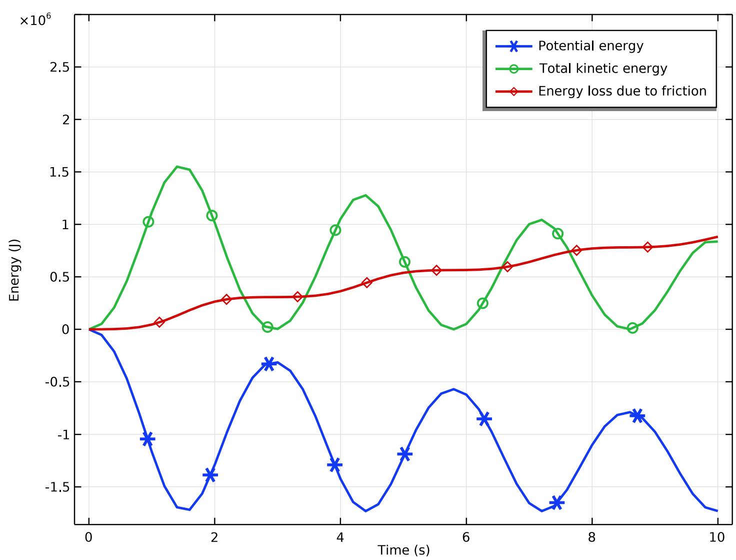

Figure 15: Time variation of different forms of energy (Case-6).

Modeling Instructions (Basic Hinge Joint)

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>Time Dependent**.

6 Click **Done**.

GEOMETRY 1

If you do not want to build the geometry step by step, you can load it from the stored model: In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `double_pendulum.mph`. You can then continue to the **Definitions** section below.

To build the geometry from scratch, continue here.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Depth** text field, type `0.5`.

4 In the **Height** text field, type `10`.

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.3`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **x** text field, type `0.5`.

6 In the **z** text field, type `9.5`.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **blk1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **cyl1** only.

Cylinder 2 (cyl2)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.3`.

4 In the **Height** text field, type `1.25`.

5 Locate the **Position** section. In the **x** text field, type `0.5`.

6 In the **y** text field, type `-0.75`.

7 In the **z** text field, type `0.5`.

8 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **cyl2** only.

Block 2 (blk2)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `10`.

4 In the **Depth** text field, type `0.5`.

5 Locate the **Position** section. In the **y** text field, type `-0.625`.

Difference 2 (dif2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **blk2** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **copy1** only.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **cyl2** and **dif1** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

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 Click **Build Selected**.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 Click in the **Graphics** window and then press Ctrl+A to select both domains.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Wp	intop1(mbd.rho*g_const*w)	J	Potential energy

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.

MULTIBODY DYNAMICS (MBD)

Gravity 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multibody Dynamics (mbd)** and choose **Gravity**.

Attachment 1

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

2 Select Boundaries 19, 20, 24, and 25 only.

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 **Fixed**.

4 From the **Destination** list, choose **Attachment 1**.

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

0	x
1	y
0	z

6 Locate the **Joint Forces and Moments** section. From the list, choose **Do not compute**.

As first attachment, select the boundaries of the protruding cylinder.

Attachment 2

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

2 Select Boundaries 16, 17, 22, and 23 only.

As second attachment, select the interior boundaries of the hole in the outer bar.

Attachment 3

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

2 Select Boundaries 6–9 only.

Hinge Joint 2

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 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

STUDY 1

Step 1: Time Dependent

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

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.02,1.5)`.

4 In the **Model Builder** window, click **Study 1**.

5 In the **Settings** window for **Study**, type `Study: Basic` in the **Label** text field.

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

RESULTS

Displacement (mbd)

The two default plots show the displacement and velocity profile in the arms of the double pendulum. The stresses can also be visualized by changing the expression in the first plot.

Add the point trajectories in the Displacement plot to visualize the direction of the motion of each arm.

Point Trajectories 1

1 In the **Displacement (mbd)** toolbar, click **More Plots** and choose **Point Trajectories**.

2 Select Points 14 and 21 only.

3 In the **Settings** window for **Point Trajectories**, locate the **Coloring and Style** section.

4 Find the **Line style** subsection. From the **Type** list, choose **Tube**.

Color Expression 1

1 Right-click **Point Trajectories 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `t`.

4 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

Use the following instructions to reproduce Figure 2:

Relative rotation

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Relative rotation` in the **Label** text field.

Global 1

1 Right-click **Relative rotation** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 2>mbd.hgj2.th - Relative rotation - rad**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mbd.hgj2.th`	`deg`	`Relative rotation`

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Click to expand the **Legends** section. Clear the **Show legends** check box.

6 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

7 In the **Relative rotation** toolbar, click **Plot**.

To generate the joint forces plot given in Figure 3, follow the instructions below:

Joint forces

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Joint forces` in the **Label** text field.

Global 1

1 Right-click **Joint forces** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 2>Joint force - N>mbd.hgj2.Fx - Joint force, x component**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 2>Joint force - N>mbd.hgj2.Fy - Joint force, y component**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 2>Joint force - N>mbd.hgj2.Fz - Joint force, z component**.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **Interpolated**.

Joint forces

1 In the **Model Builder** window, click **Joint forces**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type `Joint forces (N)`.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 In the **Joint forces** toolbar, click **Plot**.

Joint forces in the joint's local coordinate system can also be plotted by doing similar steps.

Modeling Instructions (Constraints on Joint)

The same hinge joint is used for the first three cases by controlling its subnodes at the study node. The joint's center and axis are computed with different techniques. This is to demonstrate various types of techniques available, where each one can be useful in certain cases.

MULTIBODY DYNAMICS (MBD)

Hinge Joint 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Joint 2**.

2 In the **Settings** window for **Hinge Joint**, locate the **Center of Joint** section.

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

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

0.5	x
0	y
0.5	z

5 Locate the **Axis of Joint** section. From the list, choose **From selected coordinate system**.

6 From the **Axis to use** list, choose **2**.

Constraints 1

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

2 In the **Settings** window for **Constraints**, locate the **Rotational Constraints** section.

3 In the θ_{max} text field, type `pi/4`.

The relative motion between the source and destination in the first hinge joint should be set to zero.

Hinge Joint 1

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

Prescribed Motion 1

In the **Physics** toolbar, click **Attributes** and choose **Prescribed Motion**.

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> Time Dependent**.

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

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,0.01,1.5)`.

3 In the **Model Builder** window, click **Study 2**.

4 In the **Settings** window for **Study**, type `Study: Constraints` in the **Label** text field.

5 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

Duplicate the plots generated in the first case to reproduce the results shown in Figure 4 and Figure 5:

RESULTS

Relative rotation: Constraints

1. In the **Model Builder** window, right-click **Relative rotation** and choose **Duplicate**.
2. In the **Settings** window for **1D Plot Group**, type `Relative rotation: Constraints` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study: Constraints/ Solution 2 (sol2)**.
4. In the **Relative rotation: Constraints** toolbar, click **Plot**.

Joint moments: Constraints

1. In the **Model Builder** window, right-click **Joint forces** and choose **Duplicate**.
2. In the **Settings** window for **1D Plot Group**, type `Joint moments: Constraints` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study: Constraints/ Solution 2 (sol2)**.

Global 1

1. In the **Model Builder** window, expand the **Joint moments: Constraints** node, then click **Global 1**.
2. In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Multibody Dynamics>Hinge joints>Hinge Joint 2>Joint moment - N·m>mbd.hgj2.Mx - Joint moment, x component**.
3. Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 2> Joint moment - N·m>mbd.hgj2.My - Joint moment, y component**.
4. Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 2> Joint moment - N·m>mbd.hgj2.Mz - Joint moment, z component**.

Joint moments: Constraints

1. In the **Model Builder** window, click **Joint moments: Constraints**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. From the **Position** list, choose **Lower left**.
4. Locate the **Plot Settings** section. In the **y-axis label** text field, type `Joint moments (N-m)`.

5 In the **Joint moments: Constraints** toolbar, click **Plot**.

Follow these instructions to reproduce energy plot shown in Figure 6:

Energy: Constraints

1 Right-click **Joint moments: Constraints** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Energy: Constraints` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Energy: Constraints** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> Variables>Wp - Potential energy - J**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Global>mbd.Wk_tot - Total kinetic energy - J**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Global>mbd.Ws_tot - Total elastic strain energy - J**.

Energy: Constraints

1 In the **Model Builder** window, click **Energy: Constraints**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Energy (J)`.

5 In the **Energy: Constraints** toolbar, click **Plot**.

Modeling Instructions (Locking on Joint)

MULTIBODY DYNAMICS (MBD)

Hinge Joint 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Joint 2**.

2 In the **Settings** window for **Hinge Joint**, locate the **Axis of Joint** section.

3 From the list, choose **Select a parallel edge**.

Joint Axis 1

1 In the **Model Builder** window, click **Joint Axis 1**.

2 Select Edge 33 only.

Hinge Joint 2

1 In the **Model Builder** window, click **Hinge Joint 2**.

2 In the **Settings** window for **Hinge Joint**, locate the **Center of Joint** section.

3 From the list, choose **Centroid of selected entities**.

4 From the **Entity level** list, choose **Edge**.

Center of Joint: Edge 1

1 In the **Model Builder** window, click **Center of Joint: Edge 1**.

2 Select Edges 34, 35, 46, and 48 only.

Hinge Joint 2

In the **Model Builder** window, click **Hinge Joint 2**.

Locking 1

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

2 In the **Settings** window for **Locking**, locate the **Rotational Locking** section.

3 In the θ_{max} text field, type `pi/4`.

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> Time Dependent**.

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

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

STUDY 3

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,0.005,1.5)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 1>Prescribed Motion 1** and **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 2>Constraints 1**.

5 Right-click and choose **Disable**.

6 In the **Model Builder** window, click **Study 3**.

7 In the **Settings** window for **Study**, type `Study: Locking` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solution 3 (sol3)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Intermediate**.

5 In the **Model Builder** window, expand the **Study: Locking>Solver Configurations> Solution 3 (sol3)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

6 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

7 From the **Nonlinear method** list, choose **Automatic (Newton)**.

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

To generate the relative rotation plot shown in Figure 7, follow the instruction below:

RESULTS

Relative rotation: Locking

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

2 In the **Settings** window for **1D Plot Group**, type `Relative rotation: Locking` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Locking/Solution 3 (sol3)**.

4 In the **Relative rotation: Locking** toolbar, click **Plot**.

To generate a plot for joint moments shown in Figure 8, follow the instructions below:

Joint moments: Locking

1 In the **Model Builder** window, right-click **Joint moments: Constraints** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Joint moments: Locking` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Locking/Solution 3 (sol3)**.

4 In the **Joint moments: Locking** toolbar, click **Plot**.

Modeling Instructions (Spring and Damper on Joint)

The hinge joint used in the previous cases could be used here to define the connection between the rigid components. A new hinge joint is however created so that it is easy to rerun each study independently without changing the attachment selection in the hinge joint.

MULTIBODY DYNAMICS (MBD)

Rigid Material 1

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

2 Select Domain 2 only.

Use the **Rigid Material's** subnode to prescribe or constrain its motion.

Fixed Constraint 1

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

Rigid Material 2

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

2 Select Domain 1 only.

Hinge Joint 3

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 **Rigid Material 1**.

4 From the **Destination** list, choose **Rigid Material 2**.

5 Locate the **Center of Joint** section. From the list, choose **User defined**.

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

0.5	x
0	y
0.5	z

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

0	x
1	y
0	z

Spring and Damper 1

1 In the **Physics** toolbar, click **Attributes** and choose **Spring and Damper**.

2 In the **Settings** window for **Spring and Damper**, locate the **Spring and Damper: Rotational** section.

3 In the k_θ text field, type `5e6`.

4 In the c_θ text field, type `1e6`.

Add an ODE to calculate the energy dissipated in damper.

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

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

7 Click **OK**.

Energy: Damper

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

2 In the **Settings** window for **Global Equations**, type `Energy: Damper` in the **Label** text field.

3 Locate the **Global Equations** section. In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
Wd	d(Wd,t)-mbd.hgj3.Qdamper	0	0	Energy dissipated in damper

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `energy` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Energy (J)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `power` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Power (W)**.

14 Click **OK**.

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>Time Dependent**.

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

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

STUDY 4

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,0.02,5)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

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

5 Right-click and choose **Disable**.

6 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 2**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 3**.

7 Right-click and choose **Disable**.

8 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 2**.

9 Right-click and choose **Disable**.

10 In the **Model Builder** window, click **Study 4**.

11 In the **Settings** window for **Study**, type `Study: Spring-Damper` in the **Label** text field.

12 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

Follow the instructions to generate the relative rotation plot shown in Figure 9:

RESULTS

Relative rotation 1

In the **Model Builder** window, right-click **Relative rotation** and choose **Duplicate**.

Relative rotation: Spring-Damper

1 In the **Settings** window for **1D Plot Group**, type `Relative rotation: Spring-Damper` in the **Label** text field.

  Set the **Dataset** to **None** for now to avoid the error in automatic plotting of a variable, which is not available in the selected dataset.

2 Locate the **Data** section. From the **Dataset** list, choose **None**.

Global 1

1 In the **Model Builder** window, expand the **Relative rotation: Spring-Damper** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Multibody Dynamics>Hinge joints>Hinge Joint 3>comp1.mbd.hgj3.th - Relative rotation - rad**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`comp1.mbd.hgj3.th`	`deg`	`Relative rotation`

Relative rotation: Spring-Damper

1 In the **Model Builder** window, click **Relative rotation: Spring-Damper**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Spring-Damper/Solution 4 (sol4)**.

4 In the **Relative rotation: Spring-Damper** toolbar, click **Plot**.

The energy plot in Figure 10 can be generated by following the steps below:

Energy: Spring-Damper

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Energy: Spring-Damper` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Spring-Damper/ Solution 4 (sol4)**.

Global 1

1 Right-click **Energy: Spring-Damper** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> Variables>Wp - Potential energy - J**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Global>mbd.Wk_tot - Total kinetic energy - J**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mbd.hgj3.Wspring`	`J`	`Energy stored in spring`
`Wd`	`J`	`Energy dissipated in damper`

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **Interpolated**.

Energy: Spring-Damper

1 In the **Model Builder** window, click **Energy: Spring-Damper**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type `Energy (J)`.

4 In the **Energy: Spring-Damper** toolbar, click **Plot**.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **y maximum** text field, type `1e6`.

7 In the **Energy: Spring-Damper** toolbar, click **Plot**.

MULTIBODY DYNAMICS (MBD)

Hinge Joint 3

In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Joint 3**.

Prescribed Motion 1

1. In the **Physics** toolbar, click **Attributes** and choose **Prescribed Motion**.
2. In the **Settings** window for **Prescribed Motion**, locate the **Prescribed Rotational Motion** section.
3. From the **Prescribed motion through** list, choose **Angular velocity**.
4. In the ω_p text field, type `t`.
5. From the **Activation condition** list, choose **Conditionally active**.
6. In the $i_{\theta p}$ text field, type `t>1`.

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> Time Dependent**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 5

Step 1: Time Dependent

1. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
2. In the **Output times** text field, type `range(0,0.02,2)`.
3. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
4. In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Gravity 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 2**, **Component 1 (comp1)>Multibody Dynamics (mbd),**

Controls spatial frame>Attachment 3, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 2**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3>Spring and Damper 1**, and **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Energy: Damper**.

5 Right-click and choose **Disable**.

6 In the **Model Builder** window, click **Study 5**.

7 In the **Settings** window for **Study**, type `Study: Prescribed motion` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

Follow these instructions to generate the relative velocity (Figure 11) and the relative rotation (Figure 12) plots:

RESULTS

Relative rotation: Prescribed motion

1 In the **Model Builder** window, right-click **Relative rotation: Spring-Damper** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Relative rotation: Prescribed motion` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Prescribed motion/ Solution 5 (sol5)**.

4 In the **Relative rotation: Prescribed motion** toolbar, click **Plot**.

Relative velocity: Prescribed motion

1 Right-click **Relative rotation: Spring-Damper** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Relative velocity: Prescribed motion` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Prescribed motion/ Solution 5 (sol5)**.

Global 1

1 In the **Model Builder** window, expand the **Relative velocity: Prescribed motion** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>**

Multibody Dynamics>Hinge joints>Hinge Joint 3>mbd.hgj3.th_t - Relative angular velocity - rad/s.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mbd.hgj3.th_t`	`deg/s`	`Relative angular velocity`

Relative velocity: Prescribed motion

1 In the **Model Builder** window, click **Relative velocity: Prescribed motion**.

2 In the **Relative velocity: Prescribed motion** toolbar, click **Plot**.

Modeling Instructions (Friction on Joint)

MULTIBODY DYNAMICS (MBD)

Hinge Joint 3

Compute joint forces to evaluate normal force in the friction node.

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Joint 3**.

2 In the **Settings** window for **Hinge Joint**, locate the **Joint Forces and Moments** section.

3 From the list, choose **Computed using weak constraints**.

Friction 1

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

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

3 In the μ text field, type `0.6`.

4 In the *r* text field, type `0.3`.

Add an ODE to calculate the energy loss due to friction.

Energy: Friction

1 In the **Model Builder** window, right-click **Energy: Damper** and choose **Duplicate**.

2 In the **Settings** window for **Global Equations**, type `Energy: Friction` in the **Label** text field.

3 Locate the **Global Equations** section. In the table, enter the following settings:

Name	f(u,ut,utt,t) (W)	Initial value (u_0) (J)	Initial value (u_t0) (W)	Description
`Wf`	`d(Wf,t)-mbd.hgj3.Qfriction`	`0`	`0`	`Energy loss due to friction`

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> Time Dependent**.

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

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

STUDY 6

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,0.05,10)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 2**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Attachment 3**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 2**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3>Spring and Damper 1**, **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3>Prescribed Motion 1**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame> Energy: Damper**.

5 Right-click and choose **Disable**.

6 In the **Model Builder** window, click **Study 6**.

7 In the **Settings** window for **Study**, type `Study: Friction` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solution 6 (sol6)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node.

3 In the **Model Builder** window, expand the **Study: Friction>Solver Configurations>Solution 6 (sol6)>Dependent Variables 1** node, then click **Energy loss due to friction (comp1.ODE1)**.

4 In the **Settings** window for **State**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `1e6`.

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

Follow the instructions below to generate the relative rotation and friction moment plots shown in Figure 13 and Figure 14 respectively:

RESULTS

Relative rotation: Friction

1 In the **Model Builder** window, right-click **Relative rotation: Spring-Damper** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Relative rotation: Friction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Friction/Solution 6 (sol6)**.

4 In the **Relative rotation: Friction** toolbar, click **Plot**.

Friction moment

1 Right-click **Relative rotation: Friction** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Friction moment` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Friction moment** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`mbd.hgj3.fric1.Mf`	`N*m`	`Friction moment`

Friction moment

1 In the **Model Builder** window, click **Friction moment**.

2 In the **Friction moment** toolbar, click **Plot**.

The energy plot in Figure 15 can be generated by following the steps below:

Energy: Friction

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Energy: Friction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Friction/Solution 6 (sol6)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(0,0.2,10)`.

Global 1

1 Right-click **Energy: Friction** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Wp - Potential energy - J**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Global>mbd.Wk_tot - Total kinetic energy - J**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
Wf	J	Energy loss due to friction

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **Interpolated**.

Energy: Friction

1 In the **Model Builder** window, click **Energy: Friction**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type `Energy (J)`.

4 In the **Energy: Friction** toolbar, click **Plot**.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **y maximum** text field, type `3e6`.

7 In the **Energy: Friction** toolbar, click **Plot**.

You can also generate an animation of the double pendulum. Follow the steps given below to generate the animation for Case-1:

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 In the **Number of frames** text field, type 50.

4 Click the **Play** button in the **Graphics** toolbar.

The analyses are now finished. If you want to store this model and reuse it later, you need to disable the nodes in all the studies that are added at a later stage.

STUDY: PRESCRIBED MOTION

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study: Prescribed motion** click **Step 1: Time Dependent**.

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

3 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3>Friction 1** and **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Energy: Friction**.

4 Right-click and choose **Disable**.

STUDY: SPRING-DAMPER

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study: Spring-Damper** click **Step 1: Time Dependent**.

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

3 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3>Prescribed Motion 1**, **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3>Friction 1**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame> Energy: Friction**.

4 Right-click and choose **Disable**.

STUDY: LOCKING

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study: Locking** click **Step 1: Time Dependent**.

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

3 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Rigid Material 1**, **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Rigid Material 2**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame> Energy: Damper**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Energy: Friction**.

4 Right-click and choose **Disable**.

STUDY: CONSTRAINTS

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study: Constraints** click **Step 1: Time Dependent**.

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 2>Locking 1**, **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Rigid Material 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame> Rigid Material 2**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Energy: Damper**, and **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Energy: Friction**.

5 Right-click and choose **Disable**.

STUDY: BASIC

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study: Basic** click **Step 1: Time Dependent**.

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 1>Prescribed Motion 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 2>Constraints 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 2>Locking 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Rigid Material 1**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Rigid Material 2**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint 3**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Energy: Damper**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Energy: Friction**.

5 Right-click and choose **Disable**.