Dynamics of Double Pendulum

Introduction

This tutorial example shows how you can model a double pendulum using the Multibody Dynamics Module. The example begins with the creation of a simple joint and then extends the model with features such as constraint, locking, spring, and damper as well as applied forces.

Model Definition

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

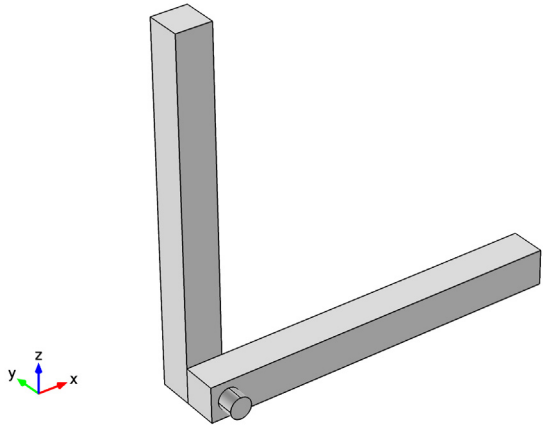

Figure 1: Geometry of the double pendulum.

Both arms of the pendulum are modeled as flexible elements, and the stress generated in both components are evaluated concurrently while computing the dynamics of the system. The two arms are connected using a hinge joint. In a hinge joint, there is one rotational degree of freedom about the joint axis. Both the components defining the

hinge joint are constrained to have the same translations at the joint. The two rotational directions orthogonal to the joint axis are also constrained to each other.

During the modeling, new subfeatures are added in order to illustrate various features available within a joint. A new study is added with each new addition of the subfeature to provide more clarity. Below is a list of different studies performed to demonstrate joint and its subfeatures:

- Case 1—Hinge joint: The top surface of the upper arm is constrained so that it cannot translate but is free to rotate about the y-axis. The whole system undergoes motion in the presence of gravitational field.
- Case 2—Constraint: The top surface of the upper arm is fixed. A constraint is added at 45° relative rotation between both arms of the joint. The whole system undergoes motion in the presence of gravitational field.
- Case 3—Locking: The top surface of the upper arm is constrained so that it cannot translate but is free to rotate about y-axis. A lock condition is added at 45° relative rotation between both the arms of the joint. The whole system undergoes motion in the presence of gravitational field.
- Case 4—Spring and damper: The top surface of the upper arm is constrained so that it cannot translate but is free to rotate about y-axis. A spring is added on the relative rotation between both the arms of the joint. The whole system undergoes motion in the presence of gravitational field.
- Case 5—Prescribed motion: The top surface of the upper arm is fixed and the relative rotation between both arms of the joint is prescribed.
- Case 6—Applied force and moment: The top surface of the upper arm is constrained so that it cannot translate but is free to rotate about y-axis. A moment is applied on the lower arm about the y-axis.

Results and Discussion

Figure 2 and Figure 3 show the relative rotation between the arms and forces on hinge joint respectively when double pendulum undergoes motion in a gravitational field (Case 1).

 ©2012 COMSOL

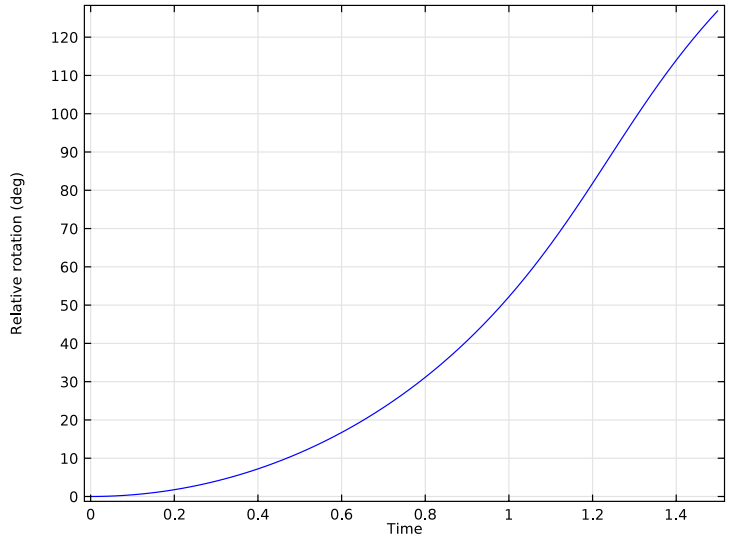

Figure 2: Relative rotation of arms in Case 1—Hinge joint.

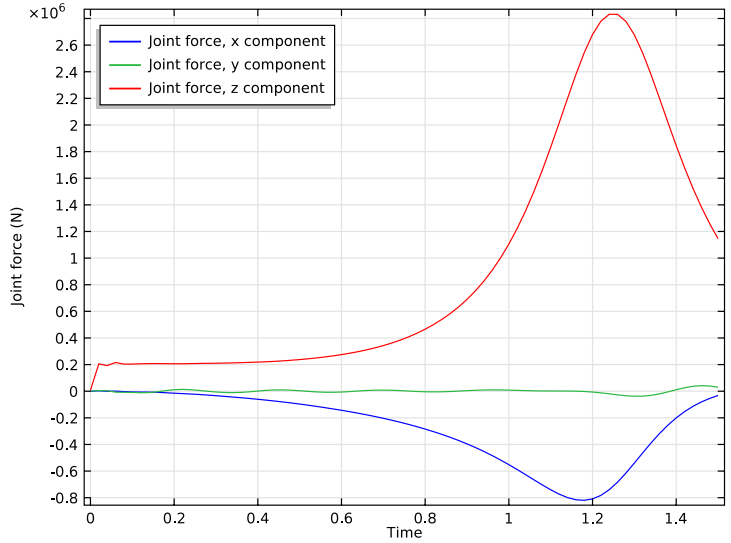

Figure 3: Forces in hinge joint in Case 1—Hinge joint.

Figure 4 shows the relative rotation between the arms when a constraint is added to the joint (Case 2). In this case the relative rotation first increases due to gravitational force, but as soon as it reaches its constrained maximum limit it becomes constant for some time and then decreases as the lower arm bounces back. The relative rotation stays at its maximum limit for some time due to bending of the flexible arms.

Figure 5 shows the relative rotation between the arms when a lock condition is applied on the joint (Case 3). Once the relative rotation reaches its prescribed lock value, it does not change further. After this, both the arms move together as a single pendulum.

The energy variation in the double pendulum when the lock condition is applied on the joint is shown in Figure 6 (Case 3). Variations in strain energy, kinetic energy, relative potential energy (considering the initial configuration as reference), and total energy are plotted as function of time. As expected, initially, kinetic energy increases and potential energy decreases and there is no change in strain energy. During the locking there is sudden drop in kinetic energy, as both arms try to acquire a common angular velocity. Due to a sudden shock, the arms also deforms, and the strain energy increases. This is reflected in the strain energy curve. Reduction in the potential energy after the locking follows a different behavior since both arms now move together. The total energy plot reflects the loss of energy during the locking process.

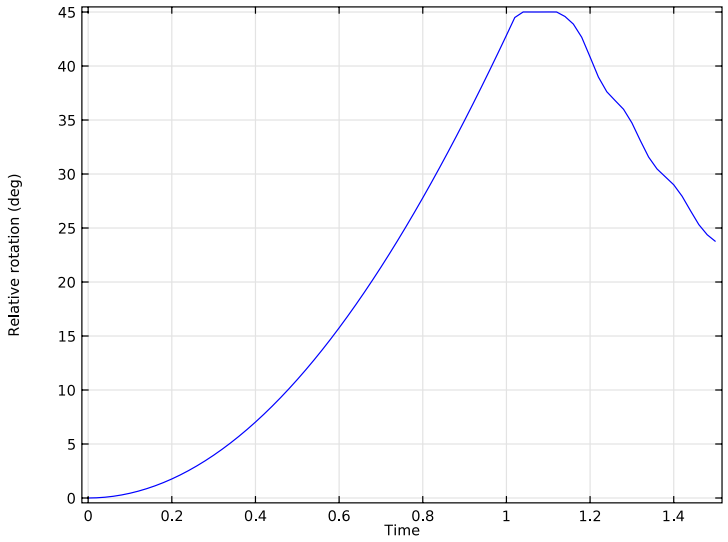

Figure 4: Relative rotation of arms in Case 2—Constraint.

 ©2012 COMSOL

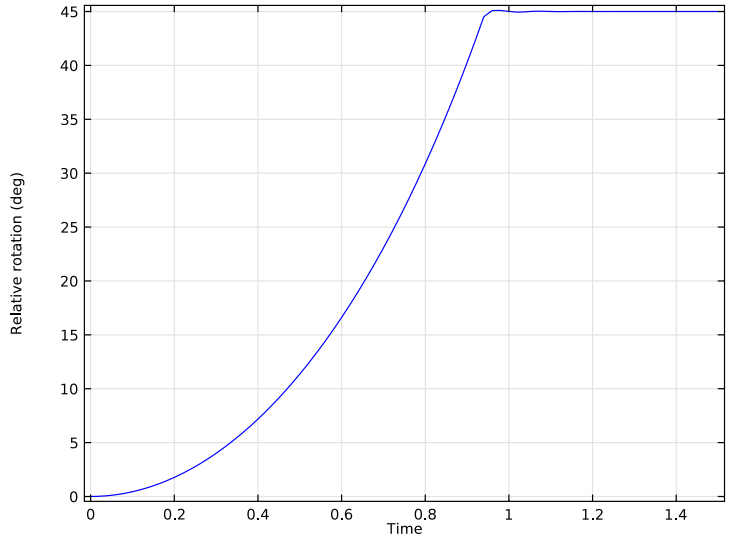

Figure 5: Relative rotation of arms in Case 3—Locking.

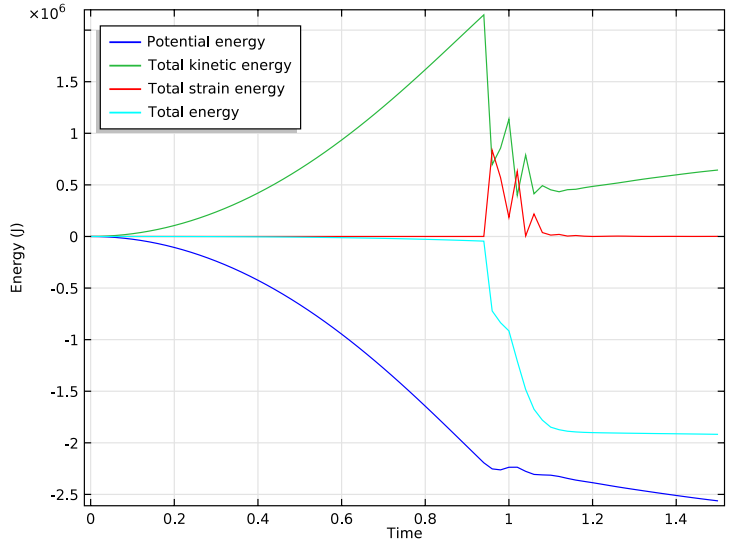

Figure 6: Energy variation in Case 3—Locking.

Figure 7 shows the relative rotation between the arms when a spring and damper is attached in the joint (Case 4). Unlike Figure 2, in this case we see that there is only a small relative rotation due to the presence of the spring. The relative rotation also decays because of the damper.

Figure 8 shows the variation in the relative rotation when it is prescribed for the certain duration and then left free (Case 5). The change occurs a $t = 0.5$ s. During the period when a relative rotation is prescribed, upper arm which is fixed at the top bends a little whereas lower arm moves freely in accordance with the prescribed value. Once the joint is free, upper arm releases the strain energy due to bending which results into the fluctuations in the relative rotation. However, lower arm keeps rotating because of its inertia.

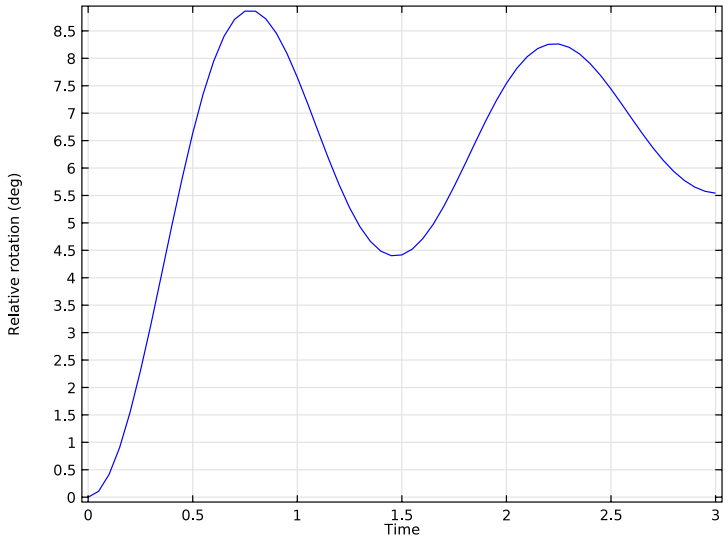

Figure 7: Relative rotation of arms in Case 4—Spring and damper.

 ©2012 COMSOL

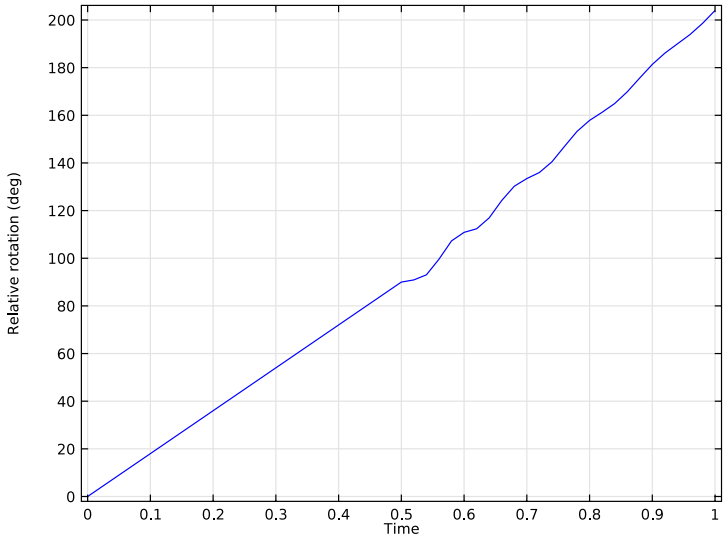

Figure 8: Relative rotation of arms in Case 5—Prescribed motion.

Model Library path: `Multibody_Dynamics_Module/Tutorial_Models/double_pendulum`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Multibody Dynamics (mbd)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

6 Click **Finish**.

GEOMETRY 1

Block 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Block**.

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

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

4 Locate the **Position** section. In the **y** edit field, type `-1`.

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

6 Locate the **Rotation Angle** section. In the **Rotation** edit field, type `-90`.

Cylinder 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.

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

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

4 In the **Height** edit field, type `2`.

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

6 In the **y** edit field, type `-1.5`.

7 In the **z** edit field, type `-0.5`.

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

Cylinder 2

Right-click **Model 1>Geometry 1>Cylinder 1** and choose **Duplicate**.

Difference 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Boolean Operations>Difference**.

2 Select the object **blk1** only.

3 In the **Difference** settings window, locate the **Difference** section.

4 Under **Objects to subtract**, click **Activate Selection**.

5 Select the object **cyl1** only.

Block 2

1 Right-click **Geometry 1** and choose **Block**.

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

3 In the **Height** edit field, type `10`.

4 Locate the **Position** section. In the **z** edit field, type `-1`.

 ©2012 COMSOL

Union 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.

2 Select the objects **cyl2** and **blk2** only.

3 In the **Union** settings window, locate the **Union** section.

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

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** click **Form Union**.

2 In the **Finalize** settings window, locate the **Finalize** section.

3 From the **Finalization method** list, choose **Form an assembly**.

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

5 Click the **Build Selected** button.

DEFINITIONS

Integration 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Model Couplings>Integration**.

2 Select Domains 1 and 2 only.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
Wp	intop1(mbd.rho*g_const*w)	Potential energy
Wt	mbd.Ws_tot+mbd.Wk_tot+Wp	Total energy

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Structural steel**.

4 Right-click and choose **Add Material to Model** from the menu.

Case 1—Hinge Joint

MULTIBODY DYNAMICS

Attachment 1

1 In the **Model Builder** window, under **Model 1** right-click **Multibody Dynamics** and choose **Attachment**.

2 Select Boundaries 16, 17, 19, and 20 only.

Attachment 2

1 In the **Model Builder** window, right-click **Multibody Dynamics** and choose **Attachment**.

2 Select Boundaries 6–9 only.

Hinge Joint 1

1 Right-click **Multibody Dynamics** and choose **Joints>Hinge Joint**.

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

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

4 From the **Destination** list, choose **Attachment 2**.

5 Locate the **Axis of Joint** section. From the list, choose **Select a parallel edge**.

Joint Axis 1

1 In the **Model Builder** window, expand the **Hinge Joint 1** node, then click **Joint Axis 1**.

2 Select Edge 40 only.

Body Load 1

1 In the **Model Builder** window, right-click **Multibody Dynamics** and choose **Body Load**.

2 Select Domains 1 and 2 only.

3 In the **Body Load** settings window, locate the **Force** section.

4 In the $\mathbf{F}_V$ table, enter the following settings:

0	x
0	y
-mbd.rho*g_const	z

Rigid Connector 1

1 Right-click **Multibody Dynamics** and choose **Rigid Connector**.

2 Select Boundary 14 only.

 ©2012 COMSOL

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

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

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

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

7 Locate the **Prescribed Rotation at Center of Rotation** section. From the **By** list, choose **Constrained rotation**.

8 Select the **Constrain rotation around x-axis** check box.

9 Select the **Constrain rotation around z-axis** check box.

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Coarse**.

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Times** edit field, type `range(0,0.02,1.5)`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Displacement

1 In the **Model Builder** window, under **Results** right-click **Displacement** and choose **Rename**.

2 Go to the **Rename 3D Plot Group** dialog box and type `Stress` in the **New name** edit field. Click **OK**.

Stress

1 In the **Model Builder** window, expand the **Results>Stress** node, then click **Surface**.

2 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Multibody Dynamics>Stress>von Mises stress (mbd.mises)**.

3 Click the **Go to Default 3D View** button on the Graphics toolbar.

The following instructions will generate Figure 2:

1D Plot Group 2

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

1 Right-click **Results>1D Plot Group 2** and choose **Global**.

2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Relative rotation (mbd.hgj1.th)**.

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

Unit	Description
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 the **Plot** button.

7 Right-click **1D Plot Group 2** and choose **Rename**.

8 Go to the **Rename 1D Plot Group** dialog box and type `theta` in the **New name** edit field. Click **OK**.

The following instructions will generate Figure 3:

1D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

1 Right-click **Results>1D Plot Group 3** and choose **Global**.

2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint force>Joint force, x component (mbd.hgj1.Fx)**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint force>Joint force, y component (mbd.hgj1.Fy)**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint force>Joint force, z component (mbd.hgj1.Fz)**.

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

6 In the **Model Builder** window, click **1D Plot Group 3**.

7 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

 ©2012 COMSOL

8 Select the **y-axis label** check box.

9 In the associated edit field, type `Joint force (N)`.

10 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

11 Right-click **1D Plot Group 3** and choose **Rename**.

12 Go to the **Rename 1D Plot Group** dialog box and type `Joint force, spatial` in the **New name** edit field. Click **OK**.

Plots for joint forces in local coordinate system, as well as joint moments (spatial and local coordinate system) can be created in a similar way.

Case 2—Constraint on Joint

MULTIBODY DYNAMICS

Linear Elastic Material 1

Add damping to damp out the stress waves when a constraint or lock condition is applied.

Damping 1

1 In the **Model Builder** window, under **Model 1>Multibody Dynamics** right-click **Linear Elastic Material 1** and choose **Damping**.

2 In the **Damping** settings window, locate the **Damping Settings** section.

3 In the α_{dM} edit field, type `0.03`.

4 In the β_{dK} edit field, type `5e-4`.

The following changes to the hinge joint settings are made just to demonstrate the different options available.

Hinge Joint 1

1 In the **Model Builder** window, under **Model 1>Multibody Dynamics** click **Hinge Joint 1**.

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

3 From the list, choose **Centroid of destination**.

4 Locate the **Axis of Joint** section. From the list, choose **Specify direction**.

5 In the *e* table, enter the following settings:

0	x
1	y
0	z

Constraints 1

1 Right-click **Model 1>Multibody Dynamics>Hinge Joint 1** and choose **Constraints**.

2 In the **Constraints** settings window, locate the **Rotational Constraints** section.

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

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Multibody Dynamics** and choose **Fixed Constraint**.

2 Select Boundary 14 only.

The fixed constraint will now override the Rigid connector on the same boundary.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

3 Click **Finish**.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.02,1.5)`.

4 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

The following instructions will generate Figure 4:

theta (constraints)

1 In the **Model Builder** window, under **Results** right-click **theta** and choose **Duplicate**.

2 Right-click **theta 1** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `theta (constraints)` in the **New name** edit field. Click **OK**.

4 In the **1D Plot Group** settings window, locate the **Data** section.

5 From the **Data set** list, choose **Solution 2**.

 ©2012 COMSOL

6 Click the **Plot** button.

1D Plot Group 6

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

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

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>1D Plot Group 6** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint moment>Joint moment, x component (mbd.hgj1.Mx)**.

6 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint moment>Joint moment, y component (mbd.hgj1.My)**.

7 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint moment>Joint moment, z component (mbd.hgj1.Mz)**.

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

9 In the **Model Builder** window, click **1D Plot Group 6**.

10 In the **1D Plot Group** settings window, locate the **Legend** section.

11 From the **Position** list, choose **Lower left**.

12 Locate the **Plot Settings** section. Select the **y-axis label** check box.

13 In the associated edit field, type `Joint moments (N-m)`.

14 Right-click **1D Plot Group 6** and choose **Rename**.

15 Go to the **Rename 1D Plot Group** dialog box and type `Joint moments, spatial` in the **New name** edit field. Click **OK**.

Case 3—Locking in Joint

DEFINITIONS

Rotated System 2

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Coordinate Systems>Rotated System**.

2 In the **Rotated System** settings window, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the α edit field, type `pi/2`.

MULTIBODY DYNAMICS

The following changes to the hinge joint settings are made just to demonstrate the different options available.

Hinge Joint 1

1 In the **Model Builder** window, under **Model 1>Multibody Dynamics** click **Hinge Joint 1**.

2 In the **Hinge Joint** settings window, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 2**.

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

5 In the X_c table, enter the following settings:

0.5	x
0	y
-0.5	z

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

Locking 1

1 Right-click **Model 1>Multibody Dynamics>Hinge Joint 1** and choose **Locking**.

2 In the **Locking** settings window, locate the **Rotational Locking** section.

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

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

3 Click **Finish**.

STUDY 3

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.02,1.5)`.

 ©2012 COMSOL

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Constraints 1**. Click **Disable**.

6 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Fixed Constraint 1**. Click **Disable**.

7 In the **Physics and variables selection** tree, select **Model 1**.

8 In the **Model Builder** window, right-click **Study 3** and choose **Compute**.

RESULTS

The following instructions will generate Figure 5:

theta (locking)

1 In the **Model Builder** window, under **Results** right-click **theta** and choose **Duplicate**.

2 Right-click **theta 1** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `theta (locking)` in the **New name** edit field. Click **OK**.

4 In the **1D Plot Group** settings window, locate the **Data** section.

5 From the **Data set** list, choose **Solution 3**.

6 Click the **Plot** button.

1D Plot Group 9

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

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

3 From the **Data set** list, choose **Solution 3**.

4 Right-click **Results>1D Plot Group 9** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Potential energy (Wp)**.

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

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

8 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Total energy (Wt)**.

9 In the **Model Builder** window, click **1D Plot Group 9**.

10 In the **1D Plot Group** settings window, click to expand the **Legend** section.

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

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

13 Locate the **Plot Settings** section. Select the **y-axis label** check box.

14 In the associated edit field, type `Energy (J)`.

15 Right-click **1D Plot Group 9** and choose **Rename**.

16 Go to the **Rename 1D Plot Group** dialog box and type `Energy` in the **New name** edit field. Click **OK**.

1D Plot Group 10

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

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

3 From the **Data set** list, choose **Solution 3**.

4 Right-click **Results>1D Plot Group 10** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint force, local coordinate system>Joint force, local coordinate system, 1 component (mbd.hgj1.FI1)**.

6 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint force, local coordinate system>Joint force, local coordinate system, 2 component (mbd.hgj1.FI2)**.

7 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint force, local coordinate system>Joint force, local coordinate system, 3 component (mbd.hgj1.FI3)**.

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

9 In the **Model Builder** window, click **1D Plot Group 10**.

10 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

11 Select the **y-axis label** check box.

12 In the associated edit field, type `Joint force, local (N)`.

13 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

14 Right-click **1D Plot Group 10** and choose **Rename**.

15 Go to the **Rename 1D Plot Group** dialog box and type `Joint force, local` in the **New name** edit field. Click **OK**.

 ©2012 COMSOL

Case 4—Spring and Damper in Joint

MULTIBODY DYNAMICS

Spring and Damper 1

1 In the **Model Builder** window, under **Model 1>Multibody Dynamics** right-click **Hinge Joint 1** and choose **Spring and Damper**.

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

3 In the k_θ edit field, type `3e7`.

4 In the c_θ edit field, type `2e6`.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

3 Click **Finish**.

STUDY 4

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 4** click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.05,3.0)`.

4 In the **Time Dependent** settings window, locate the **Physics and Variables Selection** section.

5 Select the **Modify physics tree and variables for study step** check box.

6 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Linear Elastic Material 1>Damping 1**. Click **Disable**.

7 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Constraints 1**. Click **Disable**.

8 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Locking 1**. Click **Disable**.

9 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Fixed Constraint 1**. Click **Disable**.

©2012 COMSOL

10 In the **Physics and variables selection** tree, select **Model 1**.

11 In the **Model Builder** window, right-click **Study 4** and choose **Compute**.

RESULTS

The following instructions will generate Figure 7:

theta (spring)

1 In the **Model Builder** window, under **Results** right-click **theta** and choose **Duplicate**.

2 Right-click **theta 1** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `theta (spring)` in the **New name** edit field. Click **OK**.

4 In the **1D Plot Group** settings window, locate the **Data** section.

5 From the **Data set** list, choose **Solution 4**.

6 Click the **Plot** button.

1D Plot Group 13

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

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

3 From the **Data set** list, choose **Solution 4**.

4 Right-click **Results>1D Plot Group 13** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint moment, local coordinate system>Joint moment, local coordinate system, 1 component (mbd.hgj1.Ml1)**.

6 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint moment, local coordinate system>Joint moment, local coordinate system, 2 component (mbd.hgj1.Ml2)**.

7 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Multibody Dynamics>Hinge joints>Hinge Joint 1>Joint moment, local coordinate system>Joint moment, local coordinate system, 3 component (mbd.hgj1.Ml3)**.

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

9 In the **Model Builder** window, click **1D Plot Group 13**.

10 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

11 Select the **y-axis label** check box.

 ©2012 COMSOL

12 In the associated edit field, type `Joint moment, local (N)`.

13 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

14 Click to expand the **Axis** section. Select the **Manual axis limits** check box.

15 In the **y minimum** edit field, type `-7e6`.

16 Right-click **1D Plot Group 13** and choose **Rename**.

17 Go to the **Rename 1D Plot Group** dialog box and type `Joint moment, local` in the **New name** edit field. Click **OK**.

Case 5—Prescribed Motion in Joint

MULTIBODY DYNAMICS

Prescribed Motion 1

1 In the **Model Builder** window, under **Model 1>Multibody Dynamics** right-click **Hinge Joint 1** and choose **Prescribed Motion**.

2 In the **Prescribed Motion** settings window, locate the **Prescribed Rotational Motion** section.

3 In the θ_p edit field, type `pi*t[1/s]`.

The prescribed motion constraint is removed when the deactivation condition is satisfied.

4 Select the **De-activation condition** check box.

5 In the $i_{\theta p}$ edit field, type `(t>0.5)`.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

3 Click **Finish**.

STUDY 5

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 5** click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.02,1)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Linear Elastic Material 1>Damping 1**. Click **Disable**.

6 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Constraints 1**. Click **Disable**.

7 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Locking 1**. Click **Disable**.

8 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Spring and Damper 1**. Click **Disable**.

9 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Body Load 1**. Click **Disable**.

10 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Rigid Connector 1**. Click **Disable**.

11 In the **Physics and variables selection** tree, select **Model 1**.

12 In the **Model Builder** window, right-click **Study 5** and choose **Compute**.

RESULTS

The following instructions will generate Figure 8:

theta (prescribed)

1 In the **Model Builder** window, under **Results** right-click **theta** and choose **Duplicate**.

2 Right-click **theta 1** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `theta (prescribed)` in the **New name** edit field. Click **OK**.

4 In the **1D Plot Group** settings window, locate the **Data** section.

5 From the **Data set** list, choose **Solution 5**.

6 Click the **Plot** button.

 ©2012 COMSOL

Case 6—Applied Force and Moment on Joint

MULTIBODY DYNAMICS

Applied Force and Moment 1

1 In the **Model Builder** window, under **Model 1>Multibody Dynamics** right-click **Hinge Joint 1** and choose **Applied Force and Moment**.

2 In the **Applied Force and Moment** settings window, locate the **Applied on Selected Attachment** section.

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

4 Locate the **Applied Moment** section. In the **M** table, enter the following settings:

0	x
1e7*t	y
0	z

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

3 Click **Finish**.

STUDY 6

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 6** click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.02,1.0)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Linear Elastic Material 1>Damping 1**. Click **Disable**.

6 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Constraints 1**. Click **Disable**.

©2012 COMSOL

7 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Locking 1**. Click **Disable**.

8 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Prescribed Motion 1**. Click **Disable**.

9 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Spring and Damper 1**. Click **Disable**.

10 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Body Load 1**. Click **Disable**.

11 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Fixed Constraint 1**. Click **Disable**.

12 In the **Physics and variables selection** tree, select **Model 1**.

13 In the **Model Builder** window, right-click **Study 6** and choose **Compute**.

RESULTS

theta (applied moment)

1 In the **Model Builder** window, under **Results** right-click **theta** and choose **Duplicate**.

2 Right-click **theta 1** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `theta (applied moment)` in the **New name** edit field. Click **OK**.

4 In the **1D Plot Group** settings window, locate the **Data** section.

5 From the **Data set** list, choose **Solution 6**.

6 Click the **Plot** button.

Export

You can generate an animation of the double pendulum with locking using the steps below.

1 In the **Model Builder** window, under **Results** right-click **Export** and choose **Player**.

2 In the **Player** settings window, locate the **Scene** section.

3 From the **Subject** list, choose **Displacement 1**.

4 Locate the **Frames** section. In the **Number of frames** edit field, type `100`.

5 Right-click **Results>Export>Player 1** and choose **Play**.

The analysis is now finished. If you want to store this model and re-use it later, you will need to disable the subfeatures in all the studies which are added at a later stage.

 ©2012 COMSOL

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Linear Elastic Material 1>Damping 1**. Click **Disable**.

5 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Constraints 1**. Click **Disable**.

6 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Locking 1**. Click **Disable**.

7 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Spring and Damper 1**. Click **Disable**.

8 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Prescribed Motion 1**. Click **Disable**.

9 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Applied Force and Moment 1**. Click **Disable**.

10 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Fixed Constraint 1**. Click **Disable**.

STUDY 2

1 In the **Physics and variables selection** tree, select **Model 1**.

2 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Time Dependent**.

3 In the **Time Dependent** settings window, locate the **Physics and Variables Selection** section.

4 Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Locking 1**. Click **Disable**.

6 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Spring and Damper 1**. Click **Disable**.

©2012 COMSOL

7 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Prescribed Motion 1**. Click **Disable**.

8 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Applied Force and Moment 1**. Click **Disable**.

STUDY 3

1 In the **Physics and variables selection** tree, select **Model 1**.

2 In the **Model Builder** window, expand the **Study 3** node, then click **Step 1: Time Dependent**.

3 In the **Time Dependent** settings window, locate the **Physics and Variables Selection** section.

4 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Spring and Damper 1**. Click **Disable**.

5 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Prescribed Motion 1**. Click **Disable**.

6 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Applied Force and Moment 1**. Click **Disable**.

STUDY 4

1 In the **Physics and variables selection** tree, select **Model 1**.

2 In the **Model Builder** window, expand the **Study 4** node, then click **Step 1: Time Dependent**.

3 In the **Time Dependent** settings window, locate the **Physics and Variables Selection** section.

4 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Prescribed Motion 1**. Click **Disable**.

5 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Applied Force and Moment 1**. Click **Disable**.

STUDY 5

1 In the **Physics and variables selection** tree, select **Model 1**.

2 In the **Model Builder** window, expand the **Study 5** node, then click **Step 1: Time Dependent**.

3 In the **Time Dependent** settings window, locate the **Physics and Variables Selection** section.

 ©2012 COMSOL

4 In the **Physics and variables selection** tree, select **Model 1>Multibody Dynamics>Hinge Joint 1>Applied Force and Moment 1**. Click **Disable**.

 ©2012 COMSOL