Created in COMSOL Multiphysics 6.2

Modeling a Radial Cam Based Valve Opening Mechanism

Introduction

In this example, a spring-loaded valve opening mechanism with a rocker arm and a radial cam is studied. All the components of the system are modeled as rigid, and are connected through prismatic, hinge, and slot joints. The cam-follower connection as well as other joint connections are modeled using built-in nodes of the Multibody Dynamics interface.

A transient analysis is performed for various values of the valve spring stiffness. The output from the model includes, among many things, the follower velocity, follower acceleration, cam-follower connection force, and the torque required to rotate the cam shaft.

Model Definition

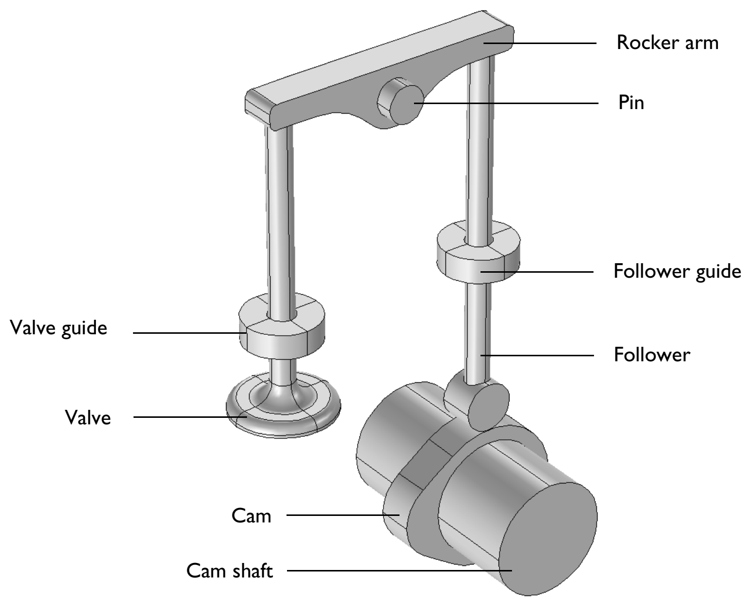

Figure 1: Geometry of a valve opening mechanism with a rocker arm and a radial cam.

The geometry of a valve opening mechanism is shown in Figure 1. All the components of the mechanism are assumed rigid and are divided into two categories based on whether they are movable or fixed.

The moving components of the mechanism are:

- Cam and cam shaft
- Follower
- Rocker arm
- Valve

The fixed components of the mechanisms are:

- Follower guide
- Pin
- Valve guide

In this mechanism, the cam rotation is prescribed and the spring is attached to the valve to restrict its motion. The objective of this analysis is twofold:

- To compute the displacement, velocity, and acceleration of the follower for a given cam shaft RPM.
- To compute the cam-follower connection force, and the torque required to rotate the cam shaft at a given RPM for different values of valve spring stiffness.

One of the main objectives behind mounting the spring on the valve is to force the follower to follow the cam profile and to avoid intermittent contacts between the cam and the follower. Hence the optimal value of the valve spring stiffness should be the one which enforces the contact between the cam and follower all the time while at the same time requiring the least torque to rotate the cam shaft.

CAM PROFILE

The profile of the cam used in this mechanism together with the maximum follower lift is shown in Figure 2. It can be seen that the cam profile is divided into four regions based on the follower or valve motion:

- Heal — Valve remains closed in this region.
- Opening flank — Valve starts opening in this region.
- Nose — Valve is in nearly fully open state in this region.
- Closing flank — Valve starts closing in this region.

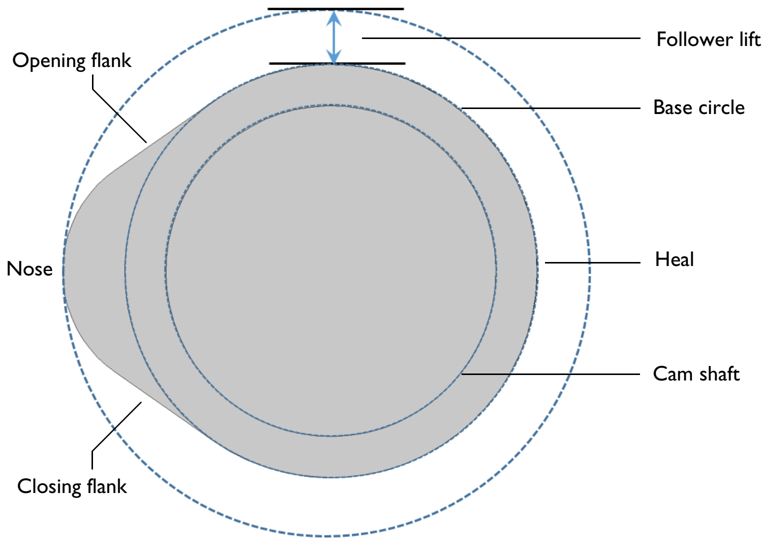

Figure 2: The profile of a cam used for the opening and closing of a valve.

MESHING OF A CAM SURFACE

A point on the follower body follows the cam profile, which is discretized through a finite element mesh. Hence in order to get accurate results and to avoid spurious oscillations, it is necessary to have a mesh which reproduces the cam profile accurately as shown in Figure 3.

JOINTS

Apart from a cam-follower connection, the following types of joints are used to connect various components of the system:

- Two hinge joints
- Two prismatic joints
- Two slot joints

MATERIAL

Structural steel is used as the material for all the components.

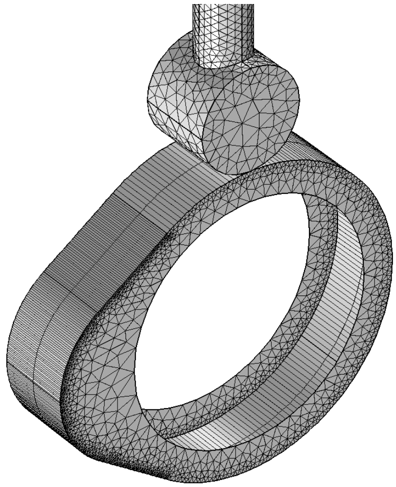

Figure 3: A fine mapped mesh created on the cam surface in order to get an accurate surface representation.

MODEL PARAMETERS

The parameters used in the model are given in the table below:

TABLE 1: MODEL PARAMETERS

DESCRIPTION	NAME	EXPRESSION
Follower radius	rf	(0.025/3) m
Spring stiffness	k	5 kN/m
Cam shaft RPM	N	1200

The model is solved with 4 different values of valve spring stiffness: 5 kN/m, 10 kN/m, 20 kN/m, and 30 kN/m.

Results and Discussion

Figure 4 shows the follower velocity together with the follower displacement as a function of cam rotation. It can be seen that the follower velocity is nonzero only for a small range of cam rotations (35°–145°).

The follower velocity first increases when the follower comes in contact with the opening flank region of the cam profile, later it decreases and becomes zero at the tip of the nose region. Similarly it increases in the reverse direction and becomes zero when the follower comes in contact with the heal region of the cam profile.

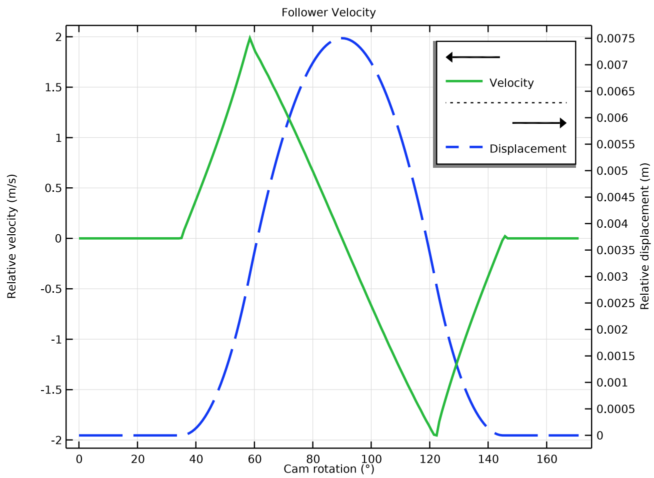

Figure 4: Variation of the follower velocity with cam rotation.

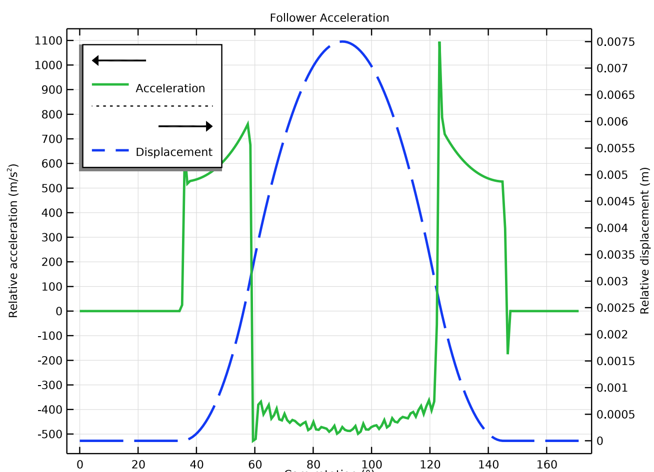

Figure 5: Variation of the follower acceleration with cam rotation.

The corresponding follower acceleration is shown in Figure 5. It can be seen that the acceleration values are negative in the range of 60° to 120° of cam rotation. This is the region where the follower has a tendency to leave the contact with the cam profile which depends on the valve spring stiffness value for a given camshaft RPM.

Figure 6 and Figure 7 show the cam-follower connection force and the torque required to rotate the cam shaft respectively. In order to have a continuous contact between the cam and follower, the connection force should be negative all the time.

It can be seen that out of the four valve stiffness values, only 20 kN/m and 30 kN/m can enforce a continuous contact between the cam and follower. In order to choose the optimal value for the valve spring stiffness, we can look at the required torque plot in Figure 7. It can be noticed in this plot that the required torque is lesser for a value of 20 kN/m and hence it is the optimal value of the valve stiffness among the values considered in this analysis.

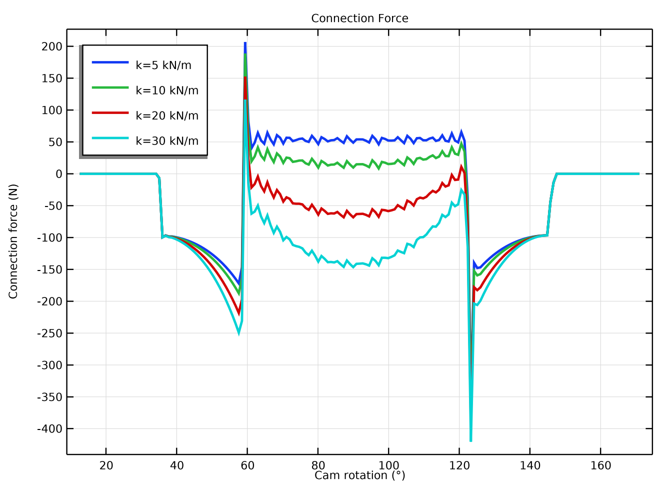

Figure 6: Variation of the cam-follower connection force with cam rotation for different values of the valve spring stiffness.

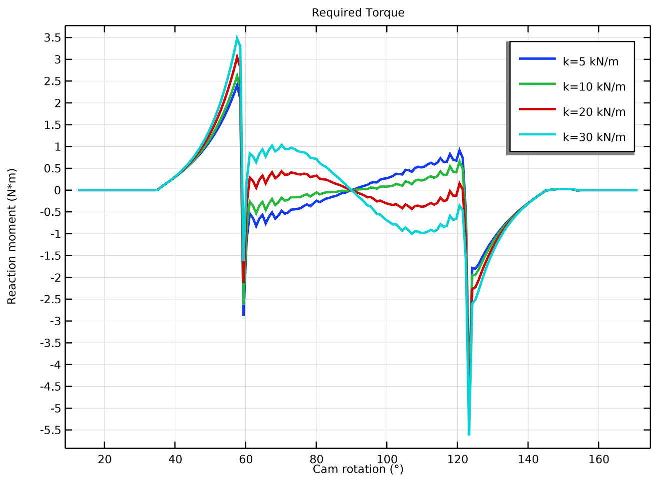

Figure 7: Variation of the torque required to rotate the cam shaft for different values of the valve spring stiffness.

Notes About the COMSOL Implementation

- Change the discretization order of the dependent variable to **Quadratic** in order to increase the geometry shape order and improve the accuracy of the mesh normal used in the **Cam-Follower** connection node.
- Use a fine mapped mesh, if possible, on a cam surface in order to improve the accuracy of the mesh normal used in the **Cam-Follower** connection node.
- In case of evaluating reaction forces using weak constraints on joints and cam-follower connection nodes, try increasing the default scaling of reaction force and moment variables in order to improve the convergence of the solution.

Application Library path: `Multibody_Dynamics_Module/Tutorials/radial_cam_follower`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Multibody Dynamics (mbd)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `radial_cam_follower_parameters.txt`.

GEOMETRY 1

Import 1 (imp1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

2 Right-click **Geometry 1** and choose **Import**.

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

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `radial_cam_follower.mphbin`.

6 Click **Import**.

Form Union (fin)

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

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

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

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

5 In the **Home** toolbar, click **Build All**.

GLOBAL DEFINITIONS

Step 1 (step1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Step**.

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

3 In the **Location** text field, type `T/40`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `T/20`.

DEFINITIONS

Cam Surface

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 89 only.

5 Select the **Group by continuous tangent** check box.

6 In the **Label** text field, type `Cam Surface`.

Cam Surface: Adjacent Edges

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Under **Input selections**, click + **Add**.

5 In the **Add** dialog box, select **Cam Surface** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent**, locate the **Output Entities** section.

8 From the **Geometric entity level** list, choose **Adjacent edges**.

9 In the **Label** text field, type `Cam Surface: Adjacent Edges`.

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)

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

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

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

Start by defining different components of the mechanism as rigid bodies.

Rigid Material: Cam

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

2 In the **Settings** window for **Rigid Material**, type `Rigid Material: Cam` in the **Label** text field.

3 Select Domain 5 only.

Rigid Materials

Create more components with the information given in the table below:

Label	**Selection**
Rigid Material: Follower	7
Rigid Material: Rocker Arm	3
Rigid Material: Valve	1
Rigid Material: Follower Guide	6
Rigid Material: Pin	4
Rigid Material: Valve Guide	2

Rigid Material: Follower Guide

In the **Model Builder** window, click **Rigid Material: Follower Guide**.

Fixed Constraint 1

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

Rigid Material: Pin

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

Fixed Constraint 1

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

Rigid Material: Valve Guide

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

Fixed Constraint 1

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

Now define the cam-follower connection and joints to connect different components of the mechanism.

Cam-Follower 1

1 In the **Physics** toolbar, click **Global** and choose **Cam-Follower**.
2 In the **Settings** window for **Cam-Follower**, locate the **Boundary Selection, Cam** section.
3 From the **Selection** list, choose **Cam Surface**.
4 Locate the **Point Selection, Follower** section. Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `148 164` in the **Selection** text field.
6 Click **OK**.
7 In the **Settings** window for **Cam-Follower**, locate the **Cam** section.
8 In the X_{offset} text field, type `rf`.
9 Locate the **Connection Force** section. From the list, choose **Computed using weak constraints**.

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 **Rigid Material: Cam**.
5 Locate the **Center of Joint** section. From the **Entity level** list, choose **Point**.
6 Locate the **Axis of Joint** section. Specify the $\mathbf{e}_0$ vector as

0	x
1	y
0	z

Center of Joint: Point 1

1. In the **Model Builder** window, click **Center of Joint: Point 1**.
2. In the **Settings** window for **Center of Joint: Point**, locate the **Point Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog box, type `108 128` in the **Selection** text field.
5. Click **OK**.

Joints

Create the remaining connections using the information given in the table below:

Label	Source	Destination	Center of Joint (points)	Axis of Joint
Hinge Joint 2	Rigid Material: Pin	Rigid Material: Rocker Arm	`94, 100`	`{0,1,0}`
Prismatic Joint 1	Rigid Material: Follower Guide	Rigid Material: Follower	`133, 147`	`{0,0,1}`
Prismatic Joint 2	Rigid Material: Valve Guide	Rigid Material: Valve	`35, 49`	`{0,0,1}`
Slot joint 1	Rigid Material: Rocker Arm	Rigid Material: Follower	`151, 162`	`Default`
Slot joint 2	Rigid Material: Rocker Arm	Rigid Material: Valve	`58, 62`	`Default`

Next, prescribe the cam rotation and add a spring to the valve motion.

Hinge Joint 1

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

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 `omega*step1(t)`.
5. Click to expand the **Reaction Force Settings** section. Select the **Evaluate reaction forces** check box.

Prismatic Joint 2

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

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: Translational** section.

3 In the k_u text field, type `k`.

To create node groups for the physics features, do the following:

Rigid Material: Cam, Rigid Material: Follower, Rigid Material: Follower Guide, Rigid Material: Pin, Rigid Material: Rocker Arm, Rigid Material: Valve, Rigid Material: Valve Guide

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)**, Ctrl-click to select **Rigid Material: Cam**, **Rigid Material: Follower**, **Rigid Material: Rocker Arm**, **Rigid Material: Valve**, **Rigid Material: Follower Guide**, **Rigid Material: Pin**, and **Rigid Material: Valve Guide**.

2 Right-click and choose **Group**.

Rigid Materials

In the **Settings** window for **Group**, type `Rigid Materials` in the **Label** text field.

Hinge Joint 1, Hinge Joint 2

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

2 Right-click and choose **Group**.

Hinge Joints

In the **Settings** window for **Group**, type `Hinge Joints` in the **Label** text field.

Prismatic Joint 1, Prismatic Joint 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)**, Ctrl-click to select **Prismatic Joint 1** and **Prismatic Joint 2**.

2 Right-click and choose **Group**.

Prismatic Joints

In the **Settings** window for **Group**, type `Prismatic Joints` in the **Label** text field.

Slot Joint 1, Slot Joint 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)**, Ctrl-click to select **Slot Joint 1** and **Slot Joint 2**.

2 Right-click and choose **Group**.

Slot Joints

In the **Settings** window for **Group**, type `Slot Joints` in the **Label** text field.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 In the **Settings** window for **Edge**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Cam Surface: Adjacent Edges**.

Size 1

1 Right-click **Edge 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Curvature factor** check box. In the associated text field, type `0.015`.

6 Select the **Maximum element growth rate** check box. In the associated text field, type `1.1`.

7 Click **Build Selected**.

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

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

3 From the **Selection** list, choose **Cam Surface**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `199` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `3`.

Free Tetrahedral 1

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

Size

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

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,T/400,T/2)`.
4. Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
5. Click **Add**.
6. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
k (Spring stiffness)	5 10 20 30	kN/m

Solution 1 (sol1)

1. In the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3. In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Connection force (comp1.mbd.cfc1.F)**.
4. In the **Settings** window for **State**, locate the **Scaling** section.
5. In the **Scale** text field, type `1e9`.
6. In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Reaction moment (comp1.mbd.hgj1.pm1.RM)**.
7. In the **Settings** window for **State**, locate the **Scaling** section.
8. In the **Scale** text field, type `1e7`.
9. In the **Study** toolbar, click **Compute**.

Follow the instructions below to plot the velocity and acceleration of the follower as shown in Figure 4 and Figure 5 respectively.

RESULTS

Follower Velocity

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Data** section. From the **Parameter selection (k)** list, choose **First**.

5 Locate the **Plot Settings** section. Select the **Two y-axes** check box.

Global 1

1 Right-click **Follower Velocity** 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>Prismatic joints>Prismatic Joint 1>mbd.prj1.u - Relative displacement - m**.

3 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** check box.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 1>mbd.hgj1.th - Relative rotation - rad**.

6 Locate the **x-Axis Data** section. From the **Unit** list, choose **°**.

7 Select the **Description** check box. In the associated text field, type `Cam rotation`.

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

9 From the **Width** list, choose **2**.

10 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`Displacement`

12 Right-click **Global 1** and choose **Duplicate**.

Global 2

1 In the **Model Builder** window, click **Global 2**.

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>Prismatic joints>Prismatic Joint 1>mbd.prj1.u_t - Relative velocity - m/s**.

3 Locate the **y-Axis** section. Clear the **Plot on secondary y-axis** check box.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Velocity`

6 In the **Follower Velocity** toolbar, click **Plot**.

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

Follower Velocity

In the **Model Builder** window, right-click **Follower Velocity** and choose **Duplicate**.

Follower Acceleration

1 In the **Model Builder** window, under **Results** click **Follower Velocity 1**.

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

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

Global 2

1 In the **Model Builder** window, expand the **Follower Acceleration** node, then click **Global 2**.

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>Prismatic joints>Prismatic Joint 1>mbd.prj1.u_tt - Relative acceleration - m/s²**.

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

Legends
`Acceleration`

4 In the **Follower Acceleration** toolbar, click **Plot**.

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

Follow the instructions below to plot the connection force and required torque as shown in Figure 6 and Figure 7 respectively.

Connection Force

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

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

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Data** section. From the **Time selection** list, choose **Manual**.

5 In the **Parameter indices (1-201)** text field, type `range(25,1,201)`.

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

Global 1

1 Right-click **Connection Force** 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>Cam-followers>Cam-Follower 1>mbd.cfc1.F - Connection force - N**.

3 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint 1>mbd.hgj1.th - Relative rotation - rad**.

5 Locate the **x-Axis Data** section. From the **Unit** list, choose **°**.

6 Select the **Description** check box. In the associated text field, type `Cam rotation`.

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

8 Locate the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

9 In the **Connection Force** toolbar, click **Plot**.

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

Connection Force

In the **Model Builder** window, right-click **Connection Force** and choose **Duplicate**.

Required Torque

1 In the **Model Builder** window, under **Results** click **Connection Force 1**.

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

3 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Global 1

1 In the **Model Builder** window, expand the **Required Torque** 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 1>mbd.hgj1.pm1.RM - Reaction moment - N·m**.
3 In the **Required Torque** toolbar, click **Plot**.
4 Click the **Zoom Extents** button in the **Graphics** toolbar.

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.