Created in COMSOL Multiphysics 6.1

Slider Crank Mechanism

Introduction

In this model you simulate the dynamic behavior of a slider crank mechanism, when the initial velocity of the slider is prescribed and the system is subjected to a gravity load. This is a benchmark problem to test the numerical algorithms in the area of multibody dynamics. This mechanism has only one degree of freedom, but it goes through singular positions during the operation. At the singular position, the mechanism has two instantaneous degrees of freedom, which in general is difficult to handle. The details about the complexity of the problem can be found in Ref. 1.

This model is simulated using the Multibody Dynamics interface and the results of the analysis are compared with those obtained in Ref. 1.

Model Definition

The geometry, which is a simplified version of a slider crank mechanism, is shown in Figure 1. It consists of only two links and the slider is not modeled physically.

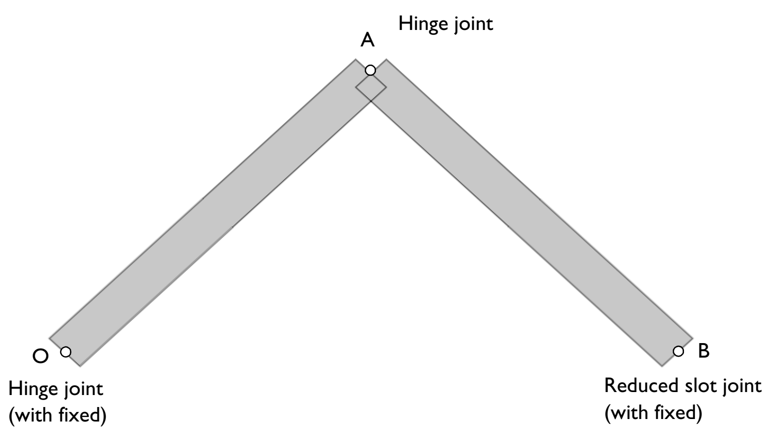

Figure 1: Model geometry.

The two links are connected by a hinge joint at point A. One of the links is connected to the ground at point O using a hinge joint. The other one is connected to the ground at point B using a reduced slot joint.

Both links are 1 m long and have a uniformly distributed mass of 1 kg. Initially the crank forms an angle of 45° with the horizontal axis. The slider (point B) is given an initial velocity of 4 m/s in the negative x direction. The whole assembly is subjected to a gravity load, which acts in the negative y direction.

Results and Discussion

The computed results are compared with the solution obtained in Ref. 1. The comparison shows that the computed results are in very good agreement with the results given in the reference.

Figure 2 shows the displacement of the links at t = 10 s and the trajectory of the various points on the arm.

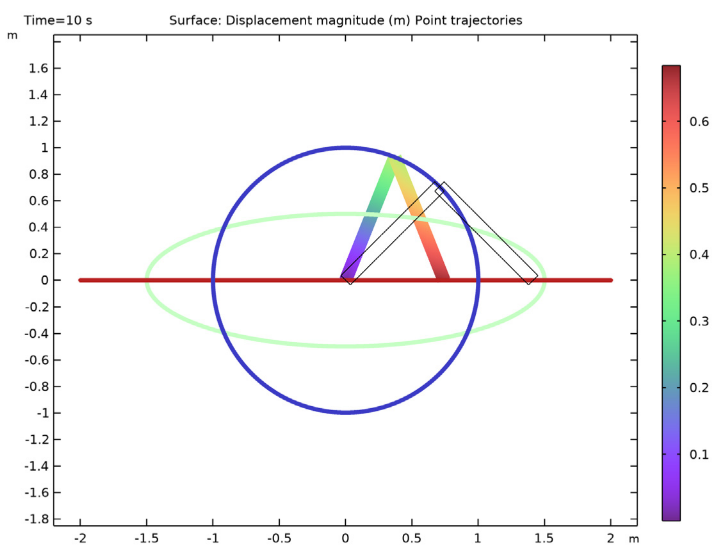

Figure 2: Displacement of the mechanism at t=10 s.

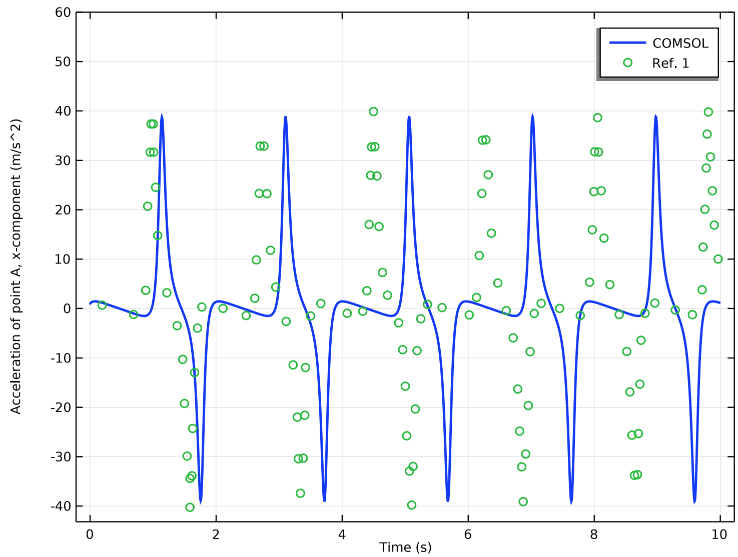

Figure 3: Comparison of the x-component of the acceleration at point A with Ref. 1.

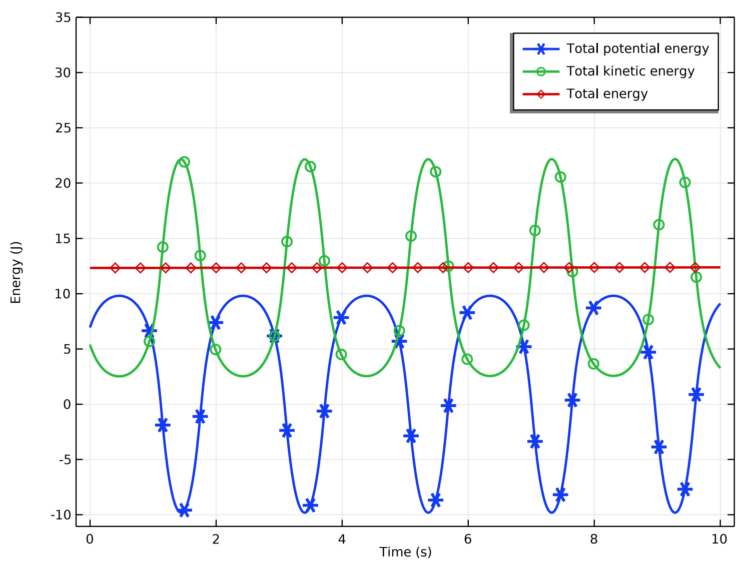

Figure 4: Time variation of kinetic, potential and total energy.

Figure 3 displays the time variation of the acceleration of point A. The computed acceleration is in very good agreement with the values obtained in Ref. 1.

Figure 4 shows the variation of kinetic, potential, and total energy with time. It can be seen that the potential energy is converted into the kinetic energy during the motion and vice-versa, conserving the total energy of the system.

Notes About the COMSOL Implementation

- In this model, linkages are modeled as rigid elements using the **Rigid Material** node as we are only interested in the kinematics of the mechanism.
- A **Joint** node can establish a connection between a **Rigid Material** or an **Attachment** node and the ground (**Fixed**). This helps in avoiding extra geometry components.
- The given initial velocity of the slider is enforced by choosing the **Force initial values** option in the **Consistent initialization** list found in the **Rigid Material** node.
- A single mesh element is enough to model a regular rigid domain if the body load is constant over the domain.

Reference

1. E. Bayo and A. Avello, "Singularity-Free Augmented Lagrangian Algorithms for Constrained Multibody Dynamics," *Nonlinear Dynamics*, vol. 5, pp. 209–231, 1994.

Application Library path: `Multibody_Dynamics_Module/Verification_Examples/slider_crank_mechanism`

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 **2D**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
d	0.1[m]	0.1 m	Thickness
l	1[m]	1 m	Length
m	1[kg]	1 kg	Mass

GEOMETRY 1

Rectangle 1 (r1)

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

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

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

4 Locate the **Position** section. In the **x** text field, type `-d/2`.

Rotate 1 (rot1)

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

2 Select the object **r1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `-45`.

Mirror 1 (mir1)

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

2 Select the object **rot1** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **x** text field, type `l/sqrt(2)`.

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 In the **Geometry** toolbar, click **Build All**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Define the density of the material using the mass and geometric-property parameters.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.
3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	m/(l*d^2)	kg/m³	Basic

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**, locate the **Thickness** section.
3 In the d text field, type d.

Rigid Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.
2 Select Domain 1 only.

Rigid Material 2

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.
2 Select Domain 2 only.

Define the initial velocity of the slider and force the same to consistently initialize the remaining degrees of freedom.

3 In the **Settings** window for **Rigid Material**, locate the **Initial Values** section.
4 From the list, choose **Locally defined**.

5 From the **Consistent initialization** list, choose **Force initial values**.

6 Select the **Translation along first axis** check box.

Initial Values 1

1 In the **Model Builder** window, expand the **Rigid Material 2** node, then click **Initial Values**.

2 In the **Settings** window for Initial Values, locate the **Initial Values: Translational** section.

3 Specify the d**u**/dt vector as

4 In the table, enter the following settings:

-4	x
0	y

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

Center of Rotation: Boundary 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)>Rigid Material 2>Initial Values 1** click **Center of Rotation: Boundary 1**.

2 Select Boundary 8 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 **Rigid Material 1**.

Center of Joint: Boundary 1

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

2 Select Boundary 1 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 **Rigid Material 1**.

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

Center of Joint: Boundary 1

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

2 Select Boundary 5 only.

Reduced Slot Joint 1

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

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

3 From the **Source** list, choose **Fixed**.

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

5 Locate the **Axes of Joint** section. From the **Joint translational axis** list, choose **Attached on source**.

The default value of the joint axis can be used here.

Center of Joint: Boundary 1

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

2 Select Boundary 8 only.

Use the **Gravity** node to apply the gravity load.

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

MESH 1

Since both components are rigid, a single element is sufficient for computing the inertial properties.

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

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

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

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type 1.

5 Click **Build All**.

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*y*d)	J	Total potential energy
W	Wp+mbd.Wk_tot	J	Total energy

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(0,0.005,10)`.

Increase the BDF order for the accurate evaluation of acceleration.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** 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 **Maximum BDF order** list, choose **3**.

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

The two default plots show the displacement and velocity profile of the slider crank mechanism. Add the trajectories of the points on the arm to the first default plot shown in Figure 2.

Create the cut points on the arm at various locations to plot the trajectory.

RESULTS

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **X** text field, type `1/sqrt(2) 1.5/sqrt(2) sqrt(2)`.

4 In the **Y** text field, type `1/sqrt(2) 0.5/sqrt(2) 0`.

Cut Point 2D 2

1 Right-click **Cut Point 2D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **X** text field, type `1/sqrt(2)`.

4 In the **Y** text field, type `1/sqrt(2)`.

Displacement (mbd)

In the **Model Builder** window, under **Results** click **Displacement (mbd)**.

Point Trajectories 1

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

2 In the **Settings** window for **Point Trajectories**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Point 2D 1**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Coloring and Style** section. 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 `X`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

8 Clear the **Color legend** check box.

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

Displacement (mbd)

1 In the **Model Builder** window, under **Results** click **Displacement (mbd)**.

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

3 From the **Frame** list, choose **Material (X, Y, Z)**.

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

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

Table 1

1. In the **Results** toolbar, click **Table**.

   Import the acceleration data obtained from Ref. 1 for comparison.

2. In the **Settings** window for **Table**, locate the **Data** section.
3. Click **Import**.
4. Browse to the model's Application Libraries folder and double-click the file `slider_crank_mechanism_aA.txt`.

Use the following instructions to plot the acceleration of point A as shown in Figure 3.

Acceleration: point A

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Cut Point 2D 2**.
4. In the **Label** text field, type `Acceleration: point A`.

Point Graph 1

1. Right-click **Acceleration: point A** and choose **Point Graph**.
2. In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Acceleration and velocity>Acceleration - m/s²>mbd.u_ttX - Acceleration, X component**.
3. Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.
4. Click to expand the **Legends** section. Select the **Show legends** check box.
5. From the **Legends** list, choose **Manual**.
6. In the table, enter the following settings:

Legends
COMSOL

Table Graph 1

1. In the **Model Builder** window, right-click **Acceleration: point A** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
3. Find the **Line style** subsection. From the **Line** list, choose **None**.
4. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
5. Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Ref. 1`

8 In the **Acceleration: point A** toolbar, click **Plot**.

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

Acceleration: point A

1 In the **Model Builder** window, click **Acceleration: point A**.

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

3 Select the **Manual axis limits** check box.

4 In the **y maximum** text field, type `60`.

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

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `Time (s)`.

8 Select the **y-axis label** check box. In the associated text field, type `Acceleration of point A, x-component (m/s^2)`.

9 In the **Acceleration: point A** toolbar, click **Plot**.

Use the following instructions to plot the energy variation in the slider crank mechanism as shown in Figure 4.

Energy

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` in the **Label** text field.

Global 1

1 Right-click **Energy** 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 - Total 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)>Definitions>Variables>W - Total energy - J**.

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

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

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

8 In the **Number** text field, type 24.

Energy

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

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

3 Select the **Manual axis limits** check box.

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

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

6 In the **y maximum** text field, type 35.

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

8 Locate the **Plot Settings** section. Select the **x-axis label** check box.

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

10 In the **Energy** toolbar, click **Plot**.

Finally, to generate an animation of the slider crank mechanism, follow these instructions:

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 100.

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