Model created in COMSOL Multiphysics 6.4

Four-Bar Mechanism with Assembly Defect

Introduction

This model simulates the dynamic behavior of a planar four-bar mechanism when one of the joints has a defect. There is an out-of-plane motion in the mechanism due to this defect. Flexible parts are used to model the links in the mechanism as the mechanism locks if the links are rigid.

The mechanism is modeled using the Multibody Dynamics interface and the results of the analysis are compared with those available in Ref. 1.

Model Definition

The geometry of the four-bar mechanism is shown in Figure 1. The geometry consists of three links. The connections between the links are modeled using hinge joints.

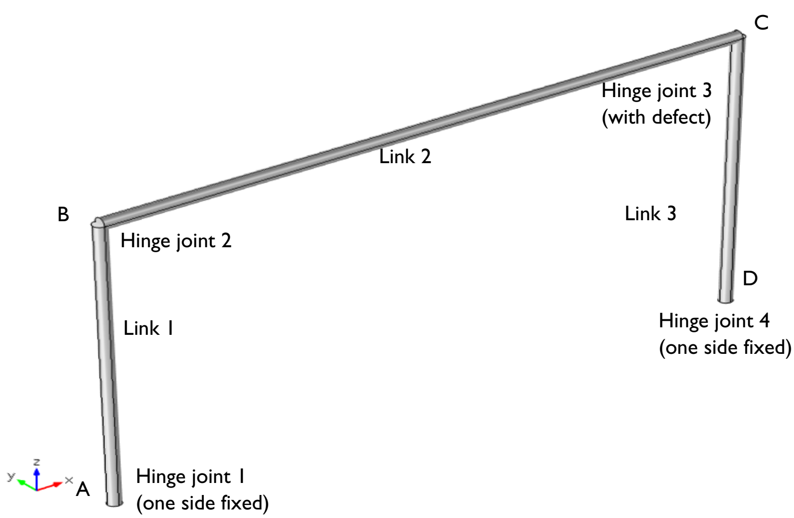

Figure 1: The geometry.

One end of each of the links 1 and 3 is connected to the ground using a hinge joint. The other end of these links are connected to link 2 using hinge joints. In case of no defect, the axis of rotation of all four hinge joints is perpendicular to the plane of the mechanism, so that the mechanism moves only in the plane. In this case, however, there is a defect in the joint between link 2 and link 3. The axis of rotation of this joint is at an angle of 5° from the normal to the plane. This simulates an assembly defect in the mechanism.

The length of links 1 and 3 is 0.12 m, and the length of link 2 is 0.24 m. The cross section of all the links is circular with a diameter of 5 mm. The links have the following material data:

- Modulus of elasticity: 70 GPa
- Poisson's ratio: 0.33.
- Density: 3000 kg/m^3

The angular velocity of the left crank (link 1) is prescribed as 1 rad/s. The effect of gravity is neglected.

Results and Discussion

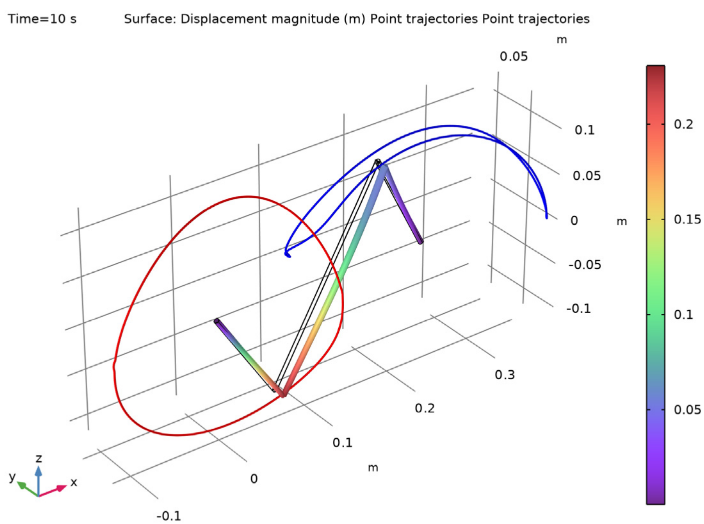

Figure 2: Configuration of the four-bar mechanism at t=10 sec. Trajectory of point B and point C is also shown.

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

Figure 2 shows the configuration of the four-bar mechanism at *t*=10 sec. The out-of-plane displacement is scaled by a factor of 20 for better visualization. The trajectory of point B and point C can also be seen.

[Figure 3](#) shows the y-component of the displacement at the joint between link 1 and link 2. If there is no defect in the joint, the out-of-plane displacement vanish.

Figure 4 displays the y-component of the displacement at the joint between link 2 and link 3.

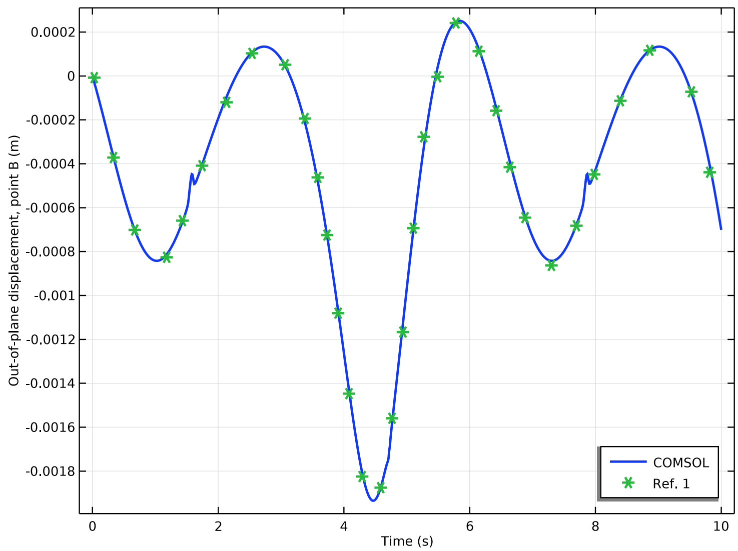

Figure 3: Comparison of out-of-plane displacement of point B with Ref. 1.

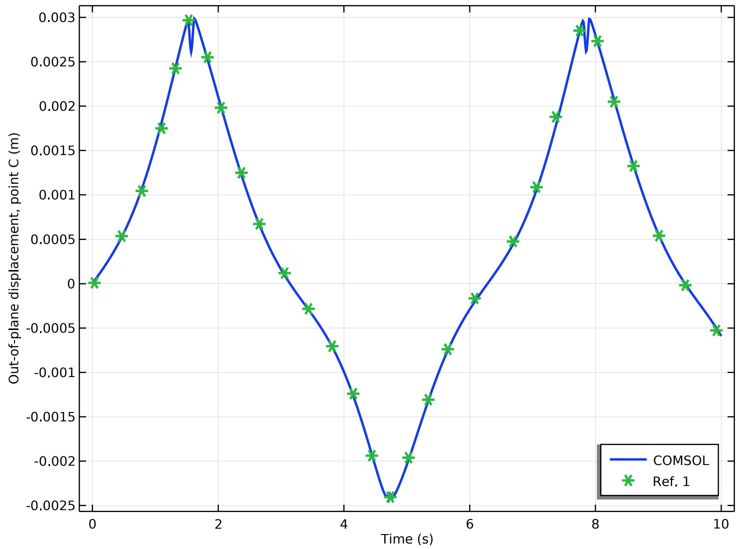

Figure 4: Comparison of out-of-plane displacement of point C with Ref. 1.

Notes About the COMSOL Implementation

- In this model, the links are modeled as flexible parts using the **Linear Elastic Material** node. Modeling the links as rigid locks the assembly.
- A **Joint** can have one side fixed by selecting the **Source** as **Fixed**. In this way you can avoid creating an extra geometry components for the "ground".
- The shape function order has been increased to quadratic. The default in the Multibody Dynamics interface is to use linear shape functions for the displacements. Such a simulation, using linear shape functions, can give an overly stiff structure, unless a fine mesh is used.

Reference

1. J. Cuadrado and others, "A Comparison in Terms of Accuracy and Efficiency between a MBS Dynamic Formulation with Stress Analysis and a Non-linear FEA Code," *Int. J. for Numerical Methods in Engineering*, vol. 51, pp. 1033–1052, 2001.

Application Library path: `Multibody_Dynamics_Module/Verification_Examples/crooked_four_bar_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 **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 In the table, enter the following settings:

Name	Expression	Value	Description
d	5[mm]	0.005 m	Diameter of links
l1	0.12[m]	0.12 m	Length of vertical link
l2	0.24[m]	0.24 m	Length of horizontal link
theta	5[deg]	0.087266 rad	Offset angle

GEOMETRY 1

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 `d/2`.

4 In the **Height** text field, type `l1`.

Cylinder 2 (cyl2)

1 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder**, locate the **Position** section.

3 In the **x** text field, type `l2`.

Cylinder 3 (cyl3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

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

3 In the **Height** text field, type `l2`.

4 Locate the **Position** section. In the **z** text field, type `l1`.

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

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**.

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
Young's modulus	E	7e10	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.33	1	Young's modulus and Poisson's ratio
Density	rho	3000	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**, click to expand the **Discretization** section.

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

Attachment 1

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

2 Select Boundary 3 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. From the list, choose **From selected coordinate system**.

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

Attachment 2

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

2 Select Boundary 4 only.

Attachment 3

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

2 Select Boundary 7 only.

Hinge Joint 2

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

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**.

Attachment 4

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

2 Select Boundary 12 only.

Attachment 5

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

2 Select Boundary 16 only.

Hinge Joint 3

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

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

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

4 From the **Destination** list, choose **Attachment 5**.

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

6 Specify the $\mathbf{e}_0$ vector as

sin(theta)	x
cos(theta)	y
0	z

Attachment 6

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

2 Select Boundary 15 only.

Hinge Joint 4

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

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

3 From the **Destination** list, choose **Attachment 6**.

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 `-1`.

Attachment 1, Attachment 2, Attachment 3, Attachment 4, Attachment 5, Attachment 6

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multibody Dynamics (mbd)**, Ctrl-click to select **Attachment 1**, **Attachment 2**, **Attachment 3**, **Attachment 4**, **Attachment 5**, and **Attachment 6**.

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

Attachments

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

Hinge Joint 1, Hinge Joint 2, Hinge Joint 3, Hinge Joint 4

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multibody Dynamics (mbd)**, Ctrl-click to select **Hinge Joint 1**, **Hinge Joint 2**, **Hinge Joint 3**, and **Hinge Joint 4**.

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

Hinge Joints

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

MESH 1

Use a swept mesh since the geometry consists of slender components.

Free Triangular 1

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

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

3 Select Boundaries 3, 7, and 16 only.

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 **Fine**.

Swept 1

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

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.
2 Select Domains 1 and 3 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `10`.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.
2 Select Domain 2 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `20`.
5 In the **Model Builder** window, right-click **Mesh 1** and choose **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(0,0.01,10)`.
4 In the **Study** toolbar, click **Compute**.

RESULTS

Displacement (mbd)

The two default plots show the displacement and velocity profile of the four bar mechanism. Follow the instructions to add the trajectory of points B and C in the first plot shown in Figure 2.

1 In the **Model Builder** window, expand the **Displacement (mbd)** node.

Surface

Scale the out-of-plane displacement by a factor of 20 for better visualization.

Deformation

1 In the **Model Builder** window, expand the **Results** > **Displacement (mbd)** > **Surface** node, then click **Deformation**.

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

3 In the **Y-component** text field, type `20*v`.

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 **Trajectory Data** section.

3 In the **Y-expression** text field, type `Y+20*v`.

4 Select Point 11 only.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

Point Trajectories 2

1 Right-click **Point Trajectories 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Point 15 only.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Blue**.

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

Displacement (mbd)

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

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

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

Import the data obtained from Ref. 1 for comparison.

vB

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

2 In the **Settings** window for **Table**, type `vB` in the **Label** text field.

3 Locate the **Data** section. Click **Import**.

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

vC

1. In the **Results** toolbar, click **Table**.
2. In the **Settings** window for **Table**, type `vC` in the **Label** text field.
3. Locate the **Data** section. Click **Import**.
4. Browse to the model's Application Libraries folder and double-click the file `crooked_four_bar_mechanism_vC.txt`.

Use the following instructions to plot the out of plane displacement of point B shown in Figure 3.

y-Displacement: Point B

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `y-Displacement: Point B` in the **Label** text field.

Point Graph 1

1. Right-click **y-Displacement: Point B** and choose **Point Graph**.
2. Select Point 4 only.
3. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `v`.
5. Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.
6. Click to expand the **Legends** section. Select the **Show legends** checkbox.
7. From the **Legends** list, choose **Manual**.
8. In the table, enter the following settings:

Legends
COMSOL

Table Graph 1

1. In the **Model Builder** window, right-click **y-Displacement: Point B** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
3. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
4. Find the **Line style** subsection. From the **Line** list, choose **None**.
5. Click to expand the **Legends** section. Select the **Show legends** checkbox.
6. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Ref. 1`

y-Displacement: Point B

1 In the **Model Builder** window, click **y-Displacement: Point B**.

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

3 From the **Position** list, choose **Lower right**.

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

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Time (s)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Out-of-plane displacement, point B (m)`.

8 In the **y-Displacement: Point B** toolbar, click **Plot**.

Use the following instructions to plot the out of plane displacement of point C shown in Figure 4.

y-Displacement: Point C

1 Right-click **y-Displacement: Point B** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `y-Displacement: Point C` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **y-Displacement: Point C** node, then click **Point Graph 1**.

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

3 Click to select the **Activate Selection** toggle button.

4 Select Point 13 only.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **vC**.

y-Displacement: Point C

1 In the **Model Builder** window, click **y-Displacement: Point C**.

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

3 In the **y-axis label** text field, type `Out-of-plane displacement, point C (m)`.

4 In the **y-Displacement: Point C** toolbar, click **Plot**.

Finally, to generate an animation of the four-bar mechanism, follow these instructions:

Animation 1

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

The default scene contains the displacements which are to be animated, so you do not need to change that setting.

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

3 In the **Number of frames** text field, type `100`.