Created in COMSOL Multiphysics 6.0

Truck Mounted Crane

Introduction

Many trucks are equipped with cranes for load handling. Such cranes have a number of hydraulic cylinders controlling the motion, and several mechanisms.

In this example, a rigid-body analysis of such a crane is performed in order to find cylinder forces and axle forces during an operating cycle.

Model Definition

The crane geometry, which is imported from a CAD model, is shown in Figure 1 and Figure 2. In all, it consists of 14 parts which can move relative to each other.

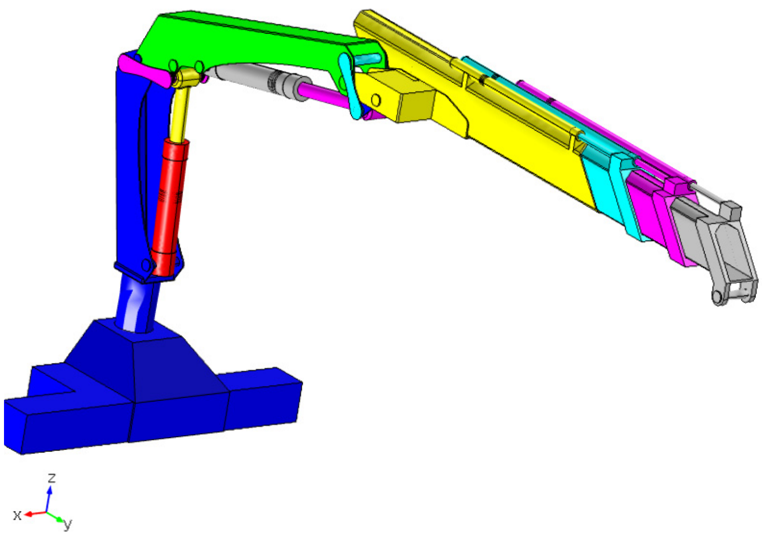

Figure 1: Crane geometry.

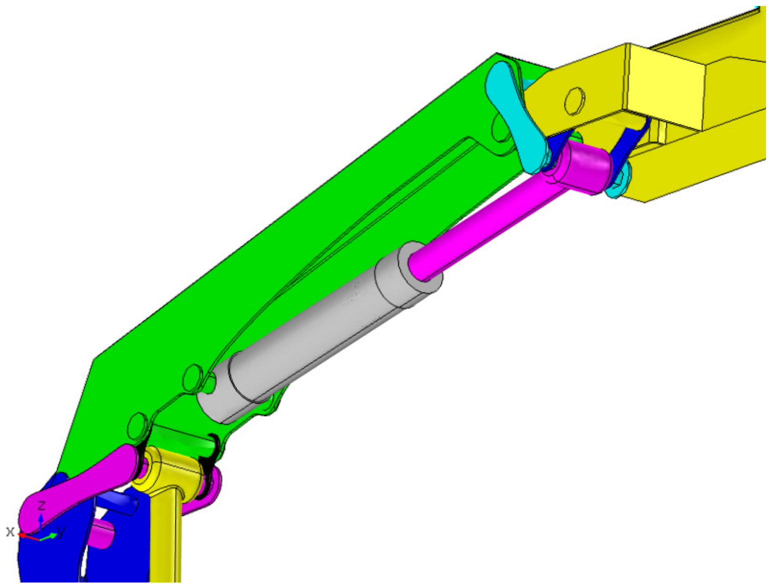

Figure 2: Close-up of link mechanisms

TABLE 1: IDENTIFICATION OF THE CRANE PARTS

PART	NAME IN MODEL	COLOR IN FIGURE
Base	Base	Blue
Inner boom	Boom1	Green
Outer Boom	Boom2	Yellow
Telescopic extensions	Extension1, Extension2, Extension3	Cyan, Magenta, Gray
Boom lift cylinders	Cylinder1, Cylinder2	Red, Gray
Boom lift pistons	Piston1, Piston2	Yellow, Magenta
Inner link mechanism	Link1, Link2	Magenta, Black
Outer link mechanism	Link3, Link4	Cyan, Blue

LOADS

- Self-weight in the negative Z direction.
- A payload of 1000 kg at the tip of the crane.

OPERATING CYCLE

The simulated working cycle consists of lifting the payload from a position far away and below the crane. The crane first moves the load upward and then inward to a position close to the crane. The trajectory of the crane tip during the operating cycle is shown in Figure 3.

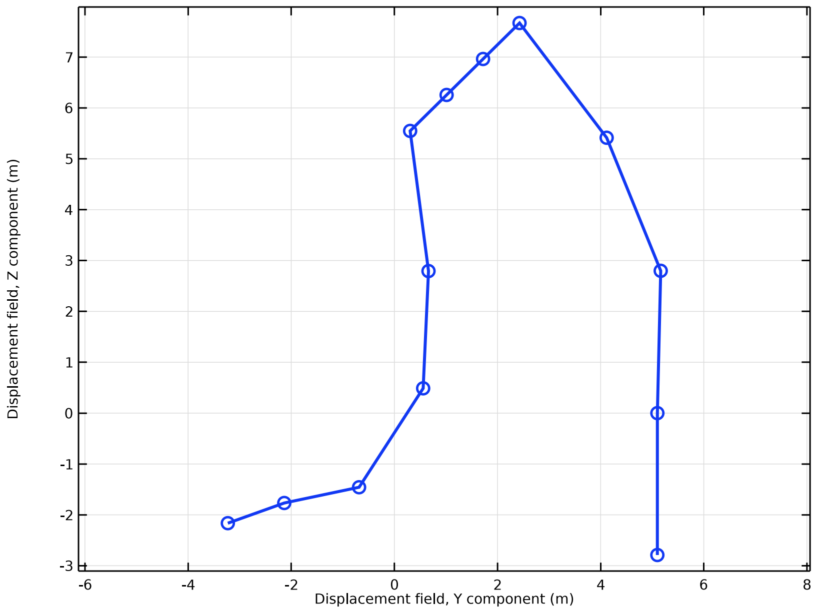

Figure 3: The trajectory of the crane tip during the operating cycle.

In real life, the crane is operated by controlling three cylinder lengths:

- The inner cylinder, which raises the inner boom.
- The outer cylinder, which controls the angle between the inner boom and the outer boom.
- The extension cylinders, which controls how far out the extensions are. The three cylinders are synchronized so that all of them have the same extension.

To prescribe an operating cycle, it is more convenient to use the angles of the booms as parameters instead of cylinder lengths. The parameters chosen to define the operating cycle are given in Table 2.

TABLE 2: OPERATING CYCLE

POSITION	INNER BOOM ANGLE TO HORIZONTAL [DEGREES]	ANGLE BETWEEN INNER AND OUTER BOOM [DEGREES]	TOTAL EXTENSION [M]
1	-15	0	5.5
2	0	0	5.1
3	15	0	5.5
4	30	0	5.5
5	45	0	5.5
6	45	0	4.5
7	45	0	3.5
8	45	0	2.5
9	45	-30	1.5
10	45	-60	1.5
11	45	-90	1.5
12	60	-120	1.5
13	60	-135	1.5

Results and Discussion

The crane in the 9th position of the operating cycle is shown in Figure 4.

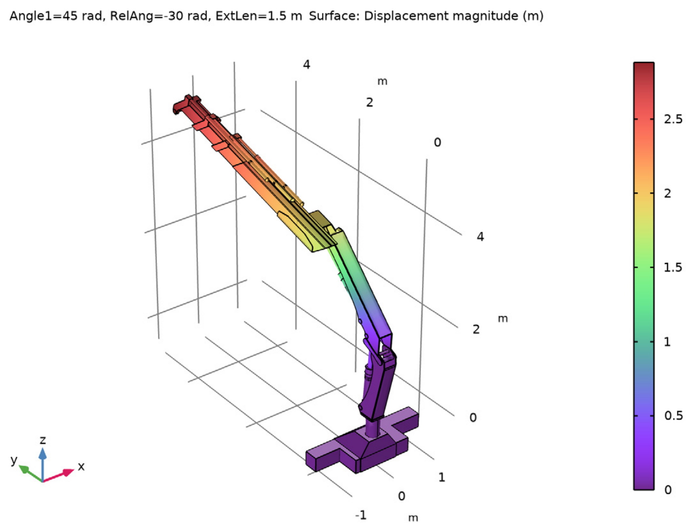

Figure 4: Crane position 9.

The forces in the cylinders controlling the boom are shown in Figure 5. Compressive forces are positive. As can be anticipated, the cylinder forces are large when the payload is far from the crane base, causing large moments around the hinges.

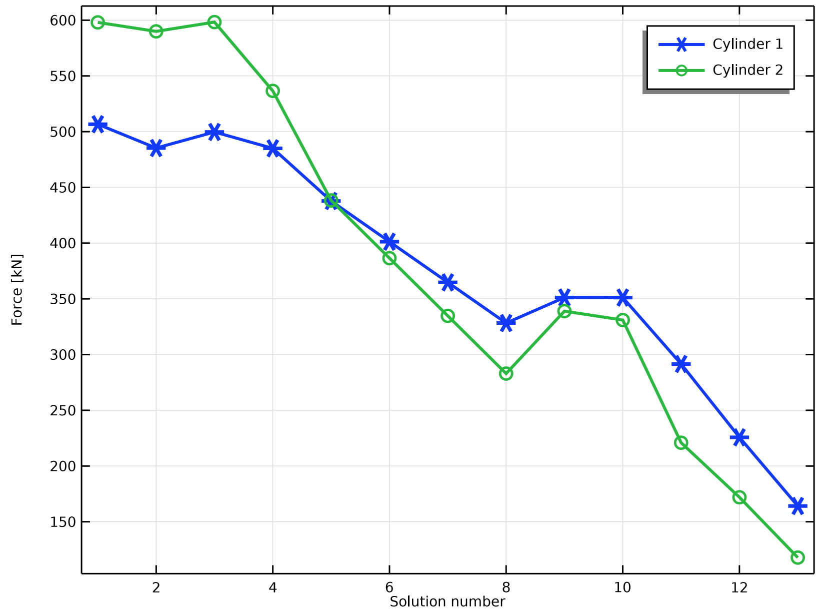

Figure 5: Variation of forces in boom lifting cylinders during the operating cycle.

The forces in the extension cylinders are shown in Figure 6, with a compressive force being defined as positive. When the outer boom is horizontal (position 2), no axial force is needed to maintain the position. The force is higher in the inner cylinders, since they also have to carry the weight of extension segments further out.

The forces acting on the hinge axle connecting the inner and outer booms are shown in Figure 7. In a similar way you can plot the forces acting in the connections between any parts for the crane. These results supply essential information for the structural design of such details.

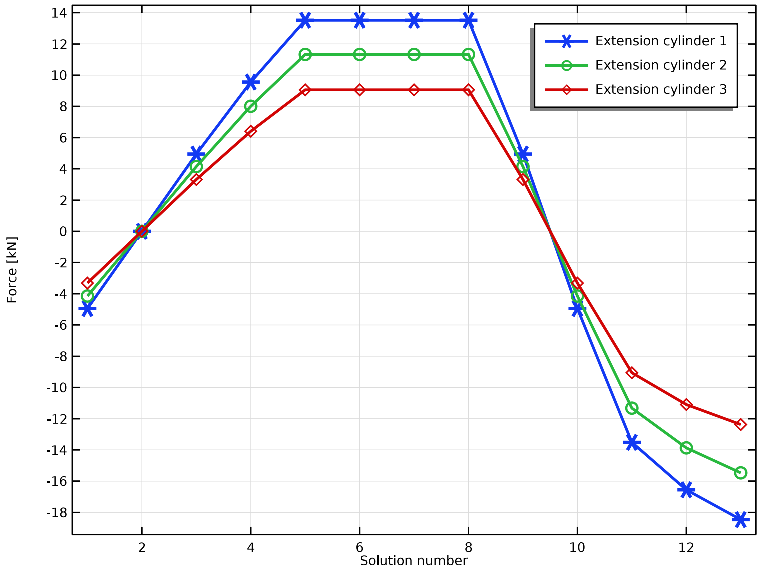

Figure 6: Variation of forces in extension cylinders over the operating cycle.

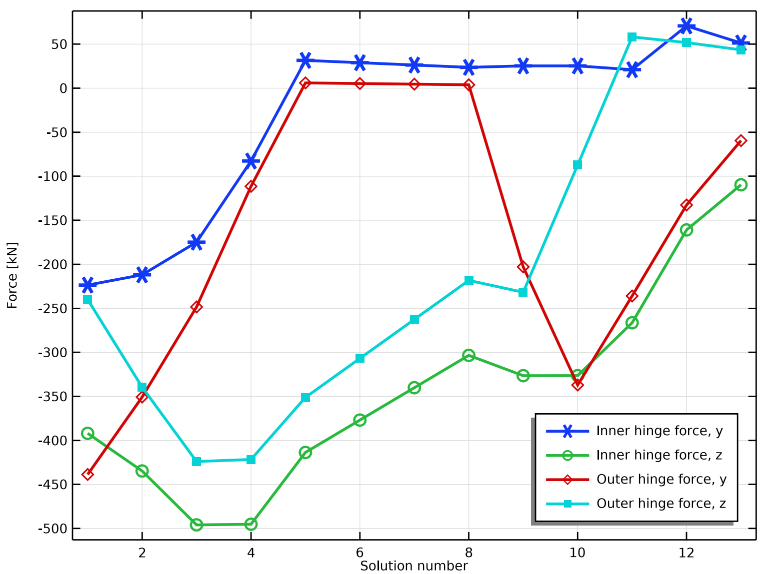

Figure 7: Forces on the hinges between the main crane parts.

Notes About the COMSOL Implementation

The analysis consists of a series of stationary solutions, controlled by a sweep over the 13 parameter combinations. The parameters used are the boom angles and the total displacement of the extension cylinders.

For the boom cylinders, it is the extension of the cylinders which you can enter as input. The known parameters are the boom angles. In this model, you add extra Global Equations where the cylinder extensions are computed based on the required boom angles.

In a model like this, where there are many parts connected to each other, great care must be taken in the selection of joint types to avoid overconstraining the problem. As an example, if only Hinge joints are chosen for one of the link mechanisms, then the translation out of the plane of the crane is prescribed in closed loop. Checking the Rigid Body DOF Summary table gives useful information that helps to avoid such problems. Another possibility is to add some flexibility in the joints in overconstrained directions. This relieves the overconstraint problem.

Application Library path: `Multibody_Dynamics_Module/Machinery_and_Robotics/truck_mounted_crane`

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>Stationary**.

6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 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**.

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)

Base

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multibody Dynamics (mbd)** and choose **Material Models>Rigid Domain**.

2 In the **Settings** window for **Rigid Domain**, type `Base` in the **Label** text field.

3 Select Domains 1, 10, 17, 22, and 23 only.

Data for Rigid Domains

1 Repeat the same operations thirteen times, entering the following data:

Label	**Domain selection**
Boom1	`19-21`
Boom2	`2, 4, 13`
Extension1	`5, 11, 14`
Extension2	`6, 8, 12`
Extension3	`3, 7, 9`
Cyliner1	`29`
Piston1	`27`
Cyliner2	`30`
Piston2	`28`
Link1	`15`
Link2	`18, 25, 32`
Link3	`16`
Link4	`24, 26, 31`

The base is kept fixed.

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

Fixed Constraint 1

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

Now define the connections between various parts of the crane.

Hinge Base-Boom1

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

2 In the **Settings** window for **Hinge Joint**, type `Hinge Base-Boom1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Base**.

4 From the **Destination** list, choose **Boom1**.

5 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Base-Boom1** node, then click **Center of Joint: Boundary 1**.

2 Select Boundaries 404 and 413 only.

Hinge Base-Cylinder1

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

2 In the **Settings** window for **Hinge Joint**, type `Hinge Base-Cylinder1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Base**.

4 From the **Destination** list, choose **Cylinder1**.

5 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Base-Cylinder1** node, then click **Center of Joint: Boundary 1**.

2 Select Boundaries 589 and 598 only.

Hinge Base-Link1

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

2 In the **Settings** window for **Hinge Joint**, type `Hinge Base-Link1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Base**.

4 From the **Destination** list, choose **Link1**.

5 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Base-Link1** node, then click **Center of Joint: Boundary 1**.

2 Select Boundaries 365 and 366 only.

Hinge Boom1-Link2

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

2 In the **Settings** window for **Hinge Joint**, type `Hinge Boom1-Link2` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Boom1**.

4 From the **Destination** list, choose **Link2**.

5 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Boom1-Link2** node, then click **Center of Joint: Boundary 1**.

2 Select Boundaries 414 and 423 only.

Slot Link1-Link2

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

2 In the **Settings** window for **Slot Joint**, type `Slot Link1-Link2` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Link1**.

4 From the **Destination** list, choose **Link2**.

5 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Slot Link1-Link2** node, then click **Center of Joint: Boundary 1**.

2 Select Boundaries 362 and 363 only.

Slot Link1-Piston1

1 In the **Model Builder** window, right-click **Slot Link1-Link2** and choose **Duplicate**.

2 In the **Settings** window for **Slot Joint**, type `Slot Link1-Piston1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Destination** list, choose **Piston1**.

Prismatic Cylinder1-Piston1

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

2 In the **Settings** window for **Prismatic Joint**, type `Prismatic Cylinder1-Piston1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Cylinder1**.

4 From the **Destination** list, choose **Piston1**.

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

6 Select the **Reverse direction** check box.

7 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Prismatic Cylinder1-Piston1** node, then click **Center of Joint: Boundary 1**.

2 Select Boundary 710 only.

Joint Axis 1

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

2 Select Edge 1564 only.

Hinge Base-Boom1, Hinge Base-Cylinder1, Hinge Base-Link1, Hinge Boom1-Link2, Prismatic Cylinder1-Piston1, Slot Link1-Link2, Slot Link1-Piston1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)**, Ctrl-click to select **Hinge Base-Boom1**, **Hinge Base-Cylinder1**, **Hinge Base-Link1**, **Hinge Boom1-Link2**, **Slot Link1-Link2**, **Slot Link1-Piston1**, and **Prismatic Cylinder1-Piston1**.

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

Hinge Boom1-Boom2

1 In the **Settings** window for **Hinge Joint**, type `Hinge Boom1-Boom2` in the **Label** text field.

2 Locate the **Attachment Selection** section. From the **Source** list, choose **Boom1**.

3 From the **Destination** list, choose **Boom2**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Boom1-Boom2** node, then click **Center of Joint: Boundary 1**.

2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 438 and 439 only.

Hinge Boom1-Cylinder2

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Base-Cylinder1.1**.

2 In the **Settings** window for **Hinge Joint**, type `Hinge Boom1-Cylinder2` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Boom1**.

4 From the **Destination** list, choose **Cylinder2**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Boom1-Cylinder2** node, then click **Center of Joint: Boundary 1**.

2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 424 and 433 only.

Hinge Boom1-Link3

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Base-Link1.1**.

2 In the **Settings** window for **Hinge Joint**, type `Hinge Boom1-Link3` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Boom1**.

4 From the **Destination** list, choose **Link3**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Boom1-Link3** node, then click **Center of Joint: Boundary 1**.

2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 391 and 392 only.

Hinge Boom2-Link4

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Boom1-Link2.1**.

2 In the **Settings** window for **Hinge Joint**, type `Hinge Boom2-Link4` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Boom2**.

4 From the **Destination** list, choose **Link4**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Hinge Boom2-Link4** node, then click **Center of Joint: Boundary 1**.

2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 603 and 604 only.

Slot Link3-Link4

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Slot Link1-Link2.1**.

2 In the **Settings** window for **Slot Joint**, type `Slot Link3-Link4` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Link3**.

4 From the **Destination** list, choose **Link4**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Slot Link3-Link4** node, then click **Center of Joint: Boundary 1**.

2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 388 and 389 only.

Slot Link3-Piston2

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Slot Link1-Piston1.1**.

2 In the **Settings** window for **Slot Joint**, type `Slot Link3-Piston2` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Link3**.

4 From the **Destination** list, choose **Piston2**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Slot Link3-Piston2** node, then click **Center of Joint: Boundary 1**.

2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 388 and 389 only.

Prismatic Cylinder2-Piston2

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Prismatic Cylinder1-Piston1.1**.

2 In the **Settings** window for **Prismatic Joint**, type `Prismatic Cylinder2-Piston2` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Cylinder2**.

4 From the **Destination** list, choose **Piston2**.

5 Locate the **Axis of Joint** section. Clear the **Reverse direction** check box.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Prismatic Cylinder2-Piston2** node, then click **Center of Joint: Boundary 1**.
2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.
3 Click **Clear Selection**.
4 Select Boundary 746 only.

Joint Axis 1

1 In the **Model Builder** window, click **Joint Axis 1**.
2 In the **Settings** window for **Joint Axis**, locate the **Edge Selection** section.
3 Click **Clear Selection**.
4 Select Edge 1682 only.

Prismatic Boom2-Extension1

1 In the **Physics** toolbar, click **Global** and choose **Prismatic Joint**.
2 In the **Settings** window for **Prismatic Joint**, type `Prismatic Boom2-Extension1` in the **Label** text field.
3 Locate the **Attachment Selection** section. From the **Source** list, choose **Boom2**.
4 From the **Destination** list, choose **Extension1**.
5 Locate the **Axis of Joint** section. Specify the $\mathbf{e}_0$ vector as

0	x
1	y
0	z

6 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Prismatic Boom2-Extension1** node, then click **Center of Joint: Boundary 1**.
2 Select Boundaries 315 and 316 only.

Prismatic Extension1-Extension2

1 In the **Physics** toolbar, click **Global** and choose **Prismatic Joint**.
2 In the **Settings** window for **Prismatic Joint**, type `Prismatic Extension1-Extension2` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Extension1**.

4 From the **Destination** list, choose **Extension2**.

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

0	x
1	y
0	z

6 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Prismatic Extension1-Extension2** node, then click **Center of Joint: Boundary 1**.

2 Select Boundaries 280 and 284 only.

Prismatic Extension2-Extension3

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

2 In the **Settings** window for **Prismatic Joint**, type `Prismatic Extension2-Extension3` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Extension2**.

4 From the **Destination** list, choose **Extension3**.

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

0	x
1	y
0	z

6 Locate the **Joint Forces and Moments** section. From the list, choose **Computed using weak constraints**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Prismatic Extension2-Extension3** node, then click **Center of Joint: Boundary 1**.

2 Select Boundaries 223 and 224 only.

Check that there are now five remaining degrees of freedom, one for each cylinder.

3 In the **Model Builder** window, click **Multibody Dynamics (mbd)**.

4 In the **Settings** window for **Multibody Dynamics**, click to expand the **Rigid Body DOF Summary** section.

Gravity 1

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

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

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

Body Load 1

1 In the **Physics** toolbar, click **Domains** and choose **Body Load**.

2 Select Domain 3 only.

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

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\text{tot}}$ vector as

0	x
0	y
-1000[kg]*g_const	z

Now, prescribe the position of all cylinders. The operation of three extension cylinders is synchronized.

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
Angle1	0[deg]	0 rad	Angle to horizontal, inner boom
RelAng	0[deg]	0 rad	Angle between booms
ExtLen	0[m]	0 m	Total extension length

MULTIBODY DYNAMICS (MBD)

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

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

3 Click **OK**.

Global Equations 1

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

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
`cyl1Pos`	`Angle1-mbd.hgj1.th*180/pi`	0	0	Inner cylinder extension

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

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

6 Click **Filter**.

7 In the tree, select **General>Displacement (m)**.

8 Click **OK**.

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

10 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (m)	Initial value (u_t0) (m/s)	Description
`cyl2Pos`	`RelAng-mbd.hgj5.th*180/pi`	0	0	Outer cylinder extension

Prismatic Cylinder1-Piston1

In the **Model Builder** window, click **Prismatic Cylinder1-Piston1**.

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 Translational Motion** section.

3 In the u_p text field, type `cyl1Pos`.

The prescribed displacements for the boom cylinders are given by the global equations. They must not couple back into the equation system through reaction forces.

4 Click to expand the **Reaction Force Settings** section. Select the **Apply reaction only on joint variables** check box.

Prismatic Cylinder2-Piston2

In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Prismatic Cylinder2-Piston2**.

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 Translational Motion** section.
3 In the u_p text field, type `cyl2Pos`.
4 Locate the **Reaction Force Settings** section. Select the **Apply reaction only on joint variables** check box.

Prismatic Boom2-Extension1

In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Prismatic Boom2-Extension1**.

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 Translational Motion** section.
3 In the u_p text field, type `ExtLen/3`.

Prismatic Extension1-Extension2

In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Prismatic Extension1-Extension2**.

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 Translational Motion** section.
3 In the u_p text field, type `ExtLen/3`.

Prismatic Extension2-Extension3

In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Prismatic Extension2-Extension3**.

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 Translational Motion** section.

3 In the u_{p} text field, type `ExtLen/3`.

Since this is a rigid body analysis, the only requirement on the mesh is that it can resolve the geometry. Apart from that, it should be as coarse as possible in order to minimize storage and computations.

MESH 1

Free Tetrahedral 1

1 In the **Model Builder** window, expand the **Mesh 1** node.

2 Right-click **Component 1 (comp1)>Mesh 1** and choose **Free Tetrahedral**.

Size

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

2 From the **Predefined** list, choose **Finer**.

Size 1

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

2 Right-click **Free Tetrahedral 1** and choose **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domains 2–4, 10, 15–24, 31, and 32 only.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

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

3 From the **Predefined** list, choose **Fine**.

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domains 5–7, 17–20, 23, and 24 only.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

The positions for the operating cycle are stored on file.

4 Click **Load from File**.

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

6 From the **Run continuation for** list, choose **No parameter**.

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 **comp1.ODE1**.

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

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

6 In the **Scale** text field, type `0.1`.

7 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Stationary Solver 1**.

8 In the **Settings** window for **Stationary Solver**, locate the **General** section.

9 In the **Relative tolerance** text field, type `1e-6`.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

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

RESULTS

Displacement (mbd)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Parameter value (Angle1 (rad),RelAng (rad),ExtLen (m))** list, choose **9: Angle1=45 rad, RelAng=-30 rad, ExtLen=1.5 m**.

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

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

Boom cylinder forces

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Boom cylinder forces` in the **Label** text field.

Global 1

1 Right-click **Boom cylinder forces** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
`mbd.prj1.Fl1`	`kN`	`Cylinder 1`
`mbd.prj2.Fl1`	`kN`	`Cylinder 2`

4 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.
5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
6 From the **Positioning** list, choose **In data points**.

Boom cylinder forces

1 In the **Model Builder** window, click **Boom cylinder forces**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3 Select the **y-axis label** check box.
4 In the associated text field, type `Force [kN]`.
5 Click to expand the **Title** section. From the **Title type** list, choose **None**.
6 In the **Boom cylinder forces** toolbar, click **Plot**.

Extension cylinder forces

1 Right-click **Boom cylinder forces** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Extension cylinder forces` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Boom cylinder forces 1** node, then click **Results>Extension cylinder forces>Global 1**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
mbd.prj3.Fl1	kN	Extension cylinder 1
mbd.prj4.Fl1	kN	Extension cylinder 2
mbd.prj5.Fl1	kN	Extension cylinder 3

4 In the **Extension cylinder forces** toolbar, click **Plot**.

Hinge forces

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

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

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
mbd.hgj1.Fy	kN	Inner hinge force, y
mbd.hgj1.Fz	kN	Inner hinge force, z
mbd.hgj5.Fy	kN	Outer hinge force, y
mbd.hgj5.Fz	kN	Outer hinge force, z

4 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

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

6 From the **Positioning** list, choose **In data points**.

Hinge forces

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

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

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

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

5 In the associated text field, type `Force [kN]`.

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

7 In the **Hinge forces** toolbar, click **Plot**.

Crane tip trajectory

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

2 In the **Settings** window for **1D Plot Group**, type `Crane tip trajectory` in the **Label** text field.

Point Graph 1

1 Right-click **Crane tip trajectory** and choose **Point Graph**.

2 Select Point 350 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `w`.

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

6 In the **Expression** text field, type `v`.

7 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 From the **Positioning** list, choose **In data points**.

Crane tip trajectory

1 In the **Model Builder** window, click **Crane tip trajectory**.

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

3 From the **Title type** list, choose **None**.

4 Locate the **Axis** section. Select the **Preserve aspect ratio** check box.

5 In the **Crane tip trajectory** toolbar, click **Plot**.