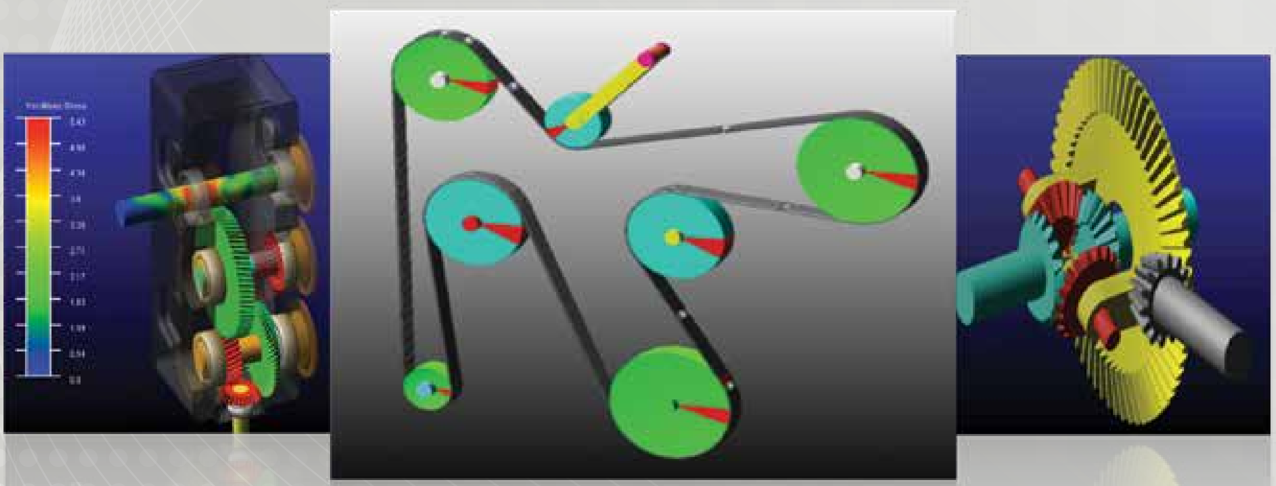
MSC Software

# Adams Tutorial Kit for Mechanical Engineering Courses

(Second Edition)

In Reference to the Textbook
*Design of Machinery by Robert L. Norton*



# Introduction

Dear Professors, Department Chairs, and Deans,

We have received many questions from undergraduate and graduate level mechanical engineering students in recent years, and probably the most common one is:

Are there any Adams tutorials that I can use to help me learn the software?

Adams is the leading multibody dynamics simulation software used extensively by engineers in product development within Automotive and other Industrial sectors worldwide to assess system performance using computer models before investing in physical prototypes.

Companies in the manufacturing industry tell us that multibody dynamics simulations within their engineering departments will increase by 3-5x over the next three years. These same companies tell us they have difficulty finding and hiring trained engineers coming out of universities today with Adams experience.

This is a problem we would like to collaborate with you to solve.

The enclosed Adams tutorial package is designed as a supplemental curriculum kit for undergraduate Mechanical Engineering courses, including ***Design of Machinery, Dynamics, Vehicle Dynamics,*** and ***Mechanical Design.***

There are 44 examples in this Adams tutorial package, including some simple problems like “four-bar linkage”, “spring-damper system”, and also some real industrial examples like “Open differential” or “Gear Train System”, which are created based on a new powerful set of simulation modules in Adams called Adams/Machinery.

Several examples were developed from specific textbook problems, for example, the four problems in section III were developed in reference to the textbook ***Design of Machinery (Fifth Edition) by Robert L. Norton.***

We are asking you to use this Adams tutorial package as supplemental learning material for the aforementioned courses in your mechanical engineering program today, as a way to further develop the skills of your students in engineering simulation, and to prepare them for engineering careers in the future.

We are committed to continuing the development of this supplemental curriculum package. If you have any questions or requests for us, please contact **Yijun.Fan@mscsoftware.com**.

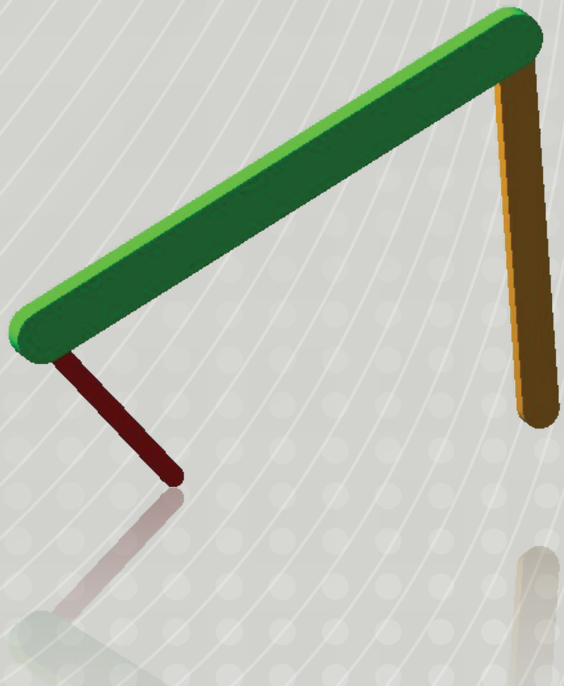
Enjoy,
Adams team at MSC Software

# Table of Contents

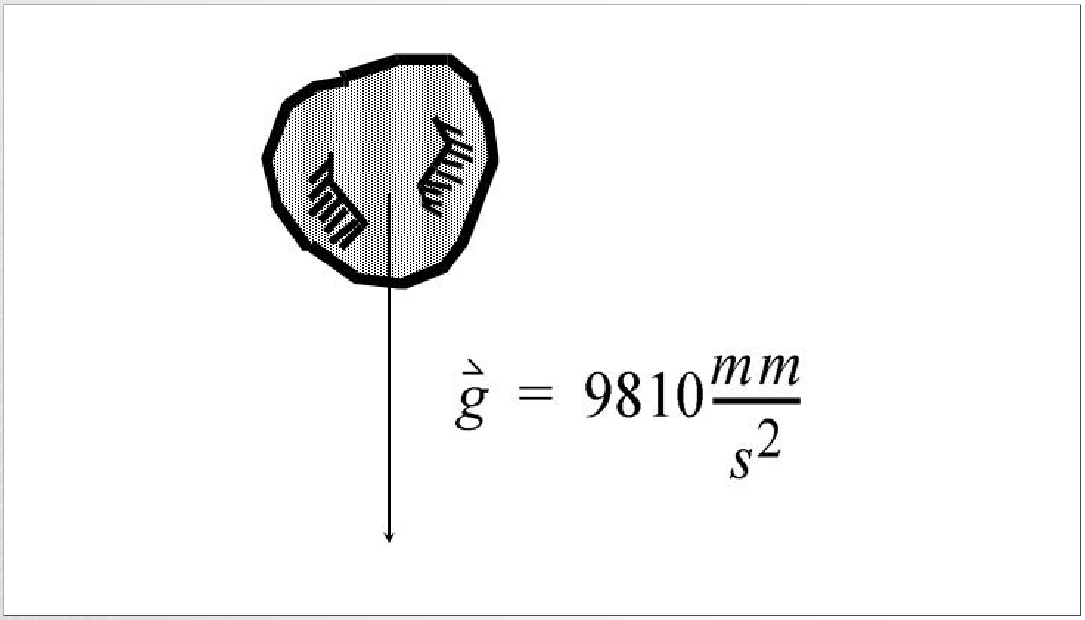
# Section I: Beginner's Level

*This section introduces you the fundamentals of Adams/View with 14 examples. No previous Adams experience is needed to go through this section and detailed guidance is given for each example. You are encouraged to work through this section in sequential order. In this Beginner's level, you will learn:*

- *How to create bodies*
- *How to connect bodies with joints*
- *How to create motions*
- *How to measure displacement, velocity or acceleration*
- *How to view results*



# Example 1: Falling Stone




## Software Version

Adams 2013.2

## Problem Description

Find the displacement, velocity, and acceleration of a stone after one second, when the stone with zero initial velocity, falls under the influence of gravity.

## Step 1. Create a New Adams database.

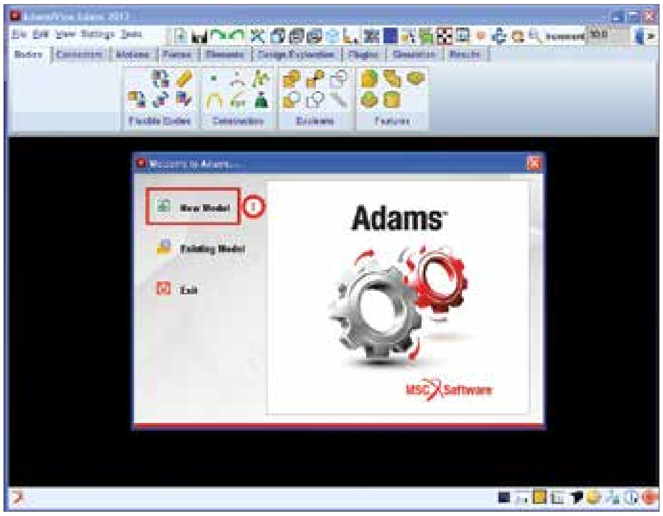
a. Click on **Create a new model**.

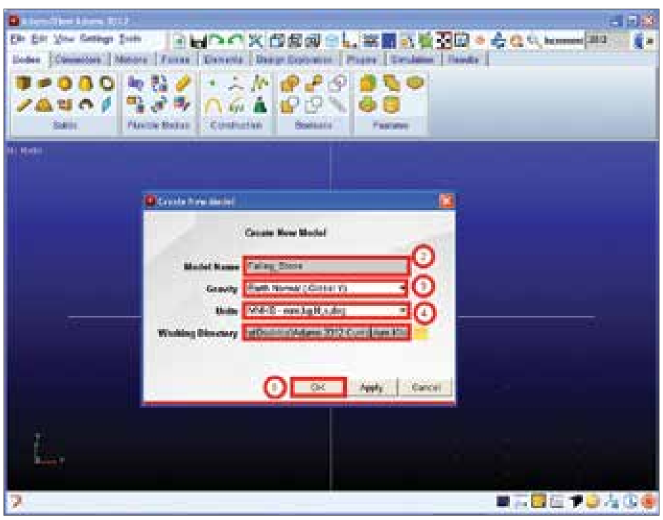
b. For the Model name change it to **Falling_Stone**.

c. For the **Gravity** choose **Earth Normal (-Global Y)**.

d. For the **Units**, set it to **MMKS - mm,kg,N,s,deg**.

e. Then click **OK**.

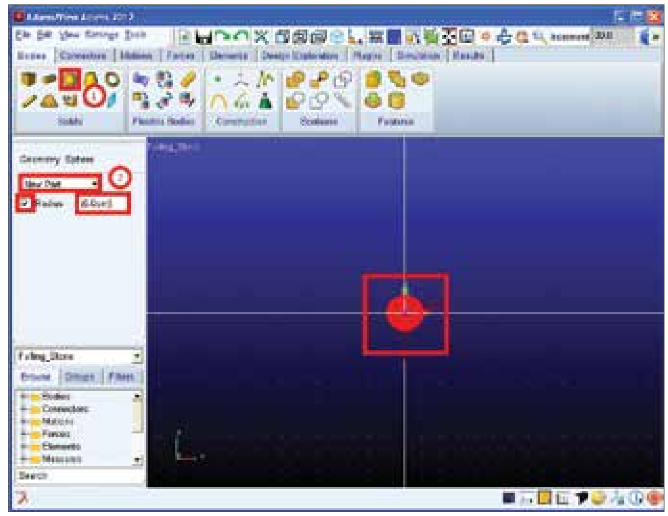




## Step 2. Build the Stone

a. From the **Main** Toolbox, right-click the **Rigid Body** tool stack, and then select the **Sphere** tool.

b. Put a check on **Radius** and set the radius to **5.0cm**.



## Step 3. Renaming the Stone.

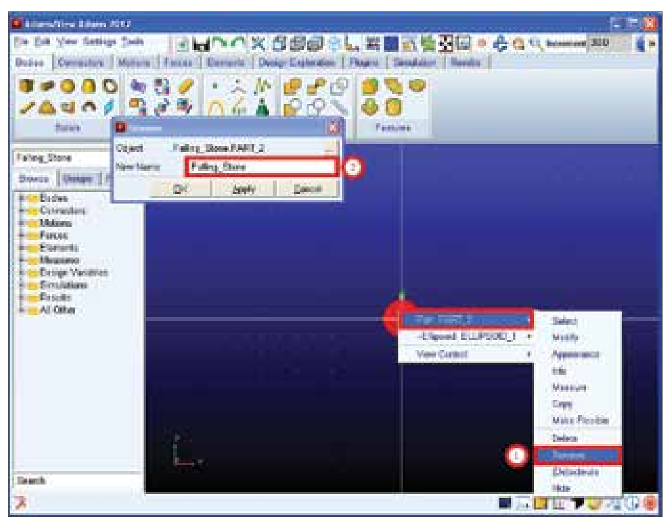
To use the zoom Box shortcut:

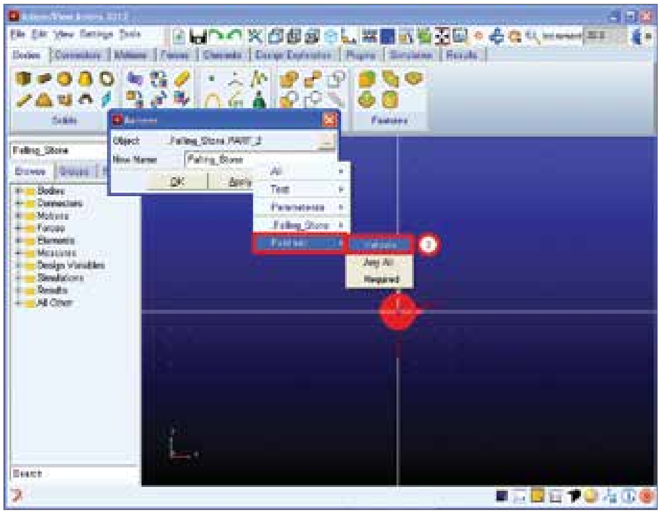
a. First right click on the **Stone** then choose **Part:PART_2** and click **Rename**.

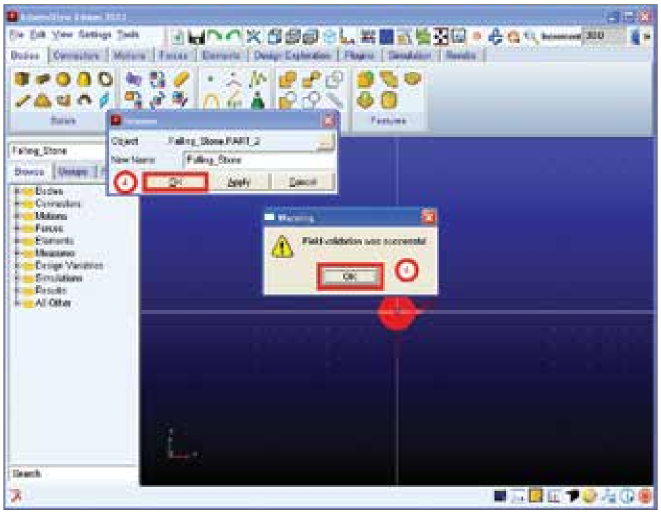
b. For the New Name type in **.Falling.Stone**.

c. Choose **Field Info** and click **Validate**.

d. Click **OK** for **Field validation** was successful and click **OK** again.







## Step 4. Set Mass to 1 kg

a. Right-click the sphere, point to **Part:Stone**, and then select **Modify**.

b. Choose **User Input** on the drop down selection for **Define Mass by**.

c. Type **1.0** for the **Mass** and click **OK**.

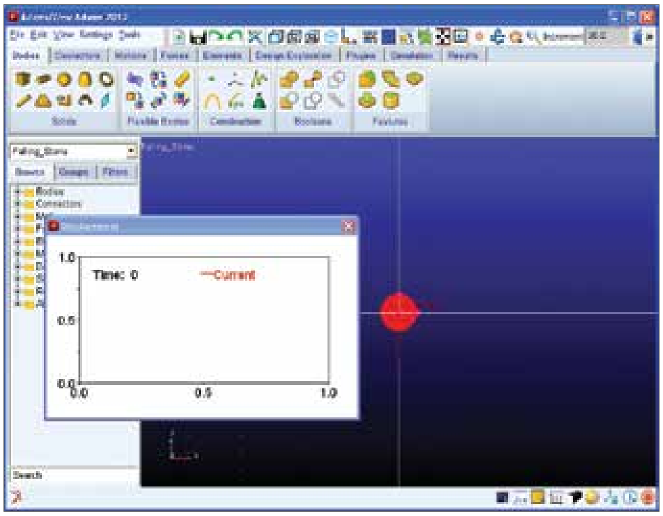
## Step 5. Calculate the Displacement of the Stone

a. Right-click on the **Stone** and choose **Part:Stone** and then click on **Measure**.

b. In the **Measure Name** text box, enter **Displacement** for the **Characteristic**, enter **CM position** for the **Component**, choose **Y**. Make sure that **Create Strip Chart** is Checked then click OK.

c. A measure stripchart appears. It is empty because you need to run a simulation before Adams/View has the necessary information for the stripchart.

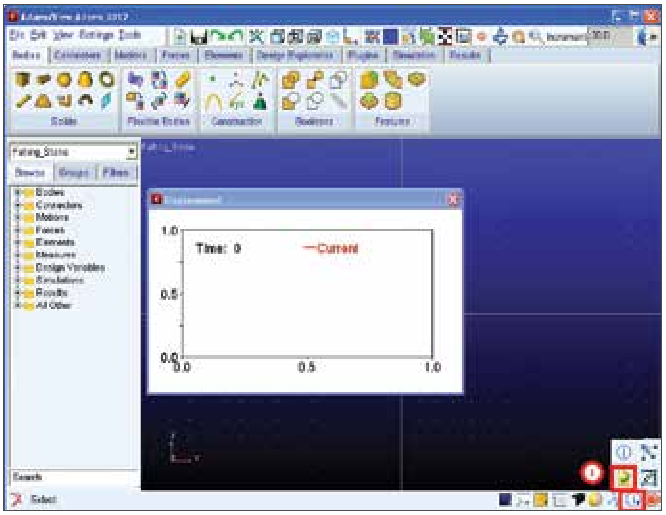
d. For more **Measurements** follow the instructions above and set **Measure Name** to **Velocity, Acceleration,** and **Characteristic** to **CM acceleration**.

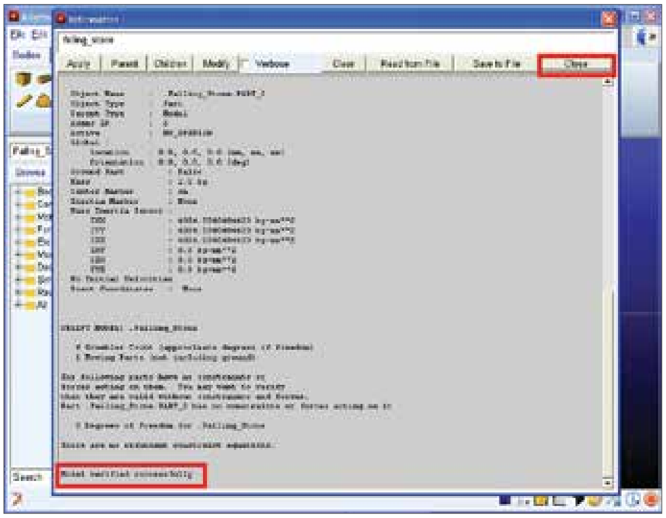


## Step 6. Verify the Model.

a. In the right corner of the **Status** bar, right-click the **Information** tool stack, and then select the **Verify** tool.

b. In the **Information** window, check that the model has verified **Successfully**, then click **Close**.





## Step 7. Set up and Run a Simulation

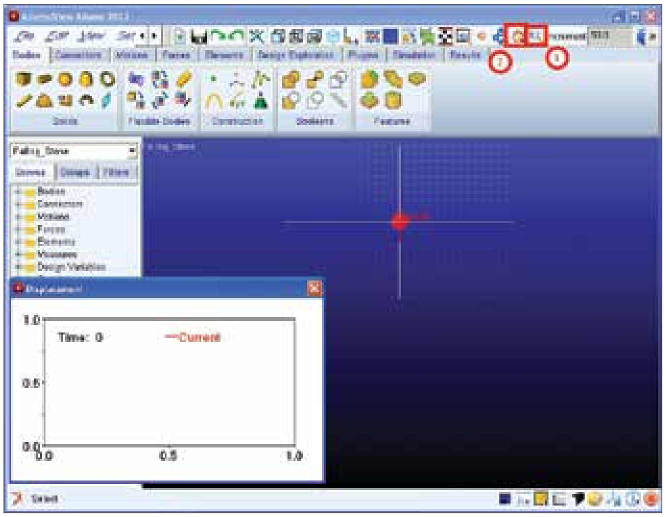
a. Select the **Zoom** tool, and then click and drag the mouse to the zoom out until the entire working grid is visible. Screen click the surface. Click **Apply**.

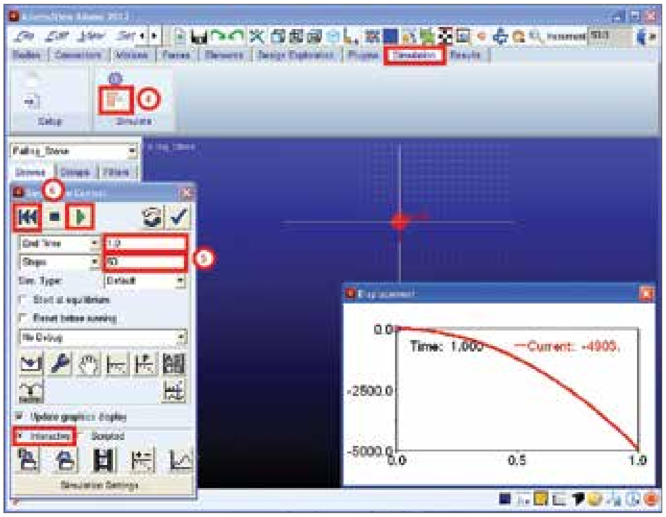
b. Select the **Translate** tool, and then drag the working grid to the top of the screen.

c. In the **Main Toolbox**, select the **Simulation tool**.

d. In the **End Time** text box, enter **1.0** and in the **Steps** text box, enter **50**.

e. Select the **Play** tool and when the simulation ends, reset the model by selecting the **Reset** tool.



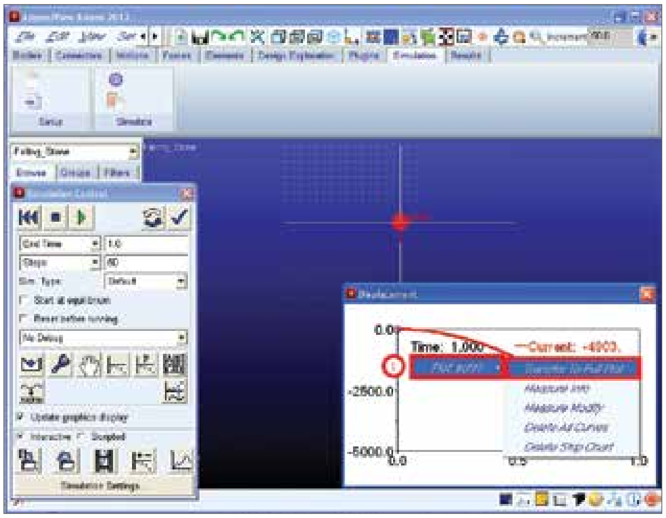


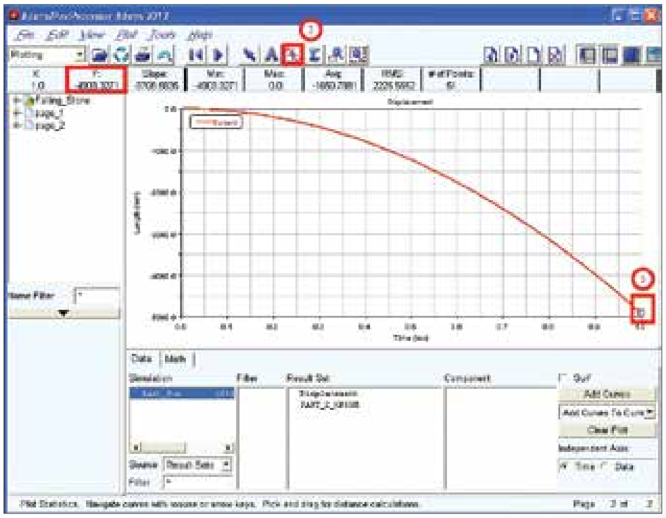
## Step 8. Results

a. To find the Stone's Displacement after 1 second, first right-click the blank area inside the stripchart, then choose **Plot:scht1** then click on **Transfer To Full Plot**.

b. In Adams/Postprocessor, from the main toolbar, select the **Plot Tracking tool**.

c. Because you want to know the final conditions after 1 second, move the cursor over the end point of the plot. In the area below the menu bar, the value of X is displayed as 1. Note the value of Y is -4903.





## Analytical Solution – Verify the results by calculating the analytical solution.

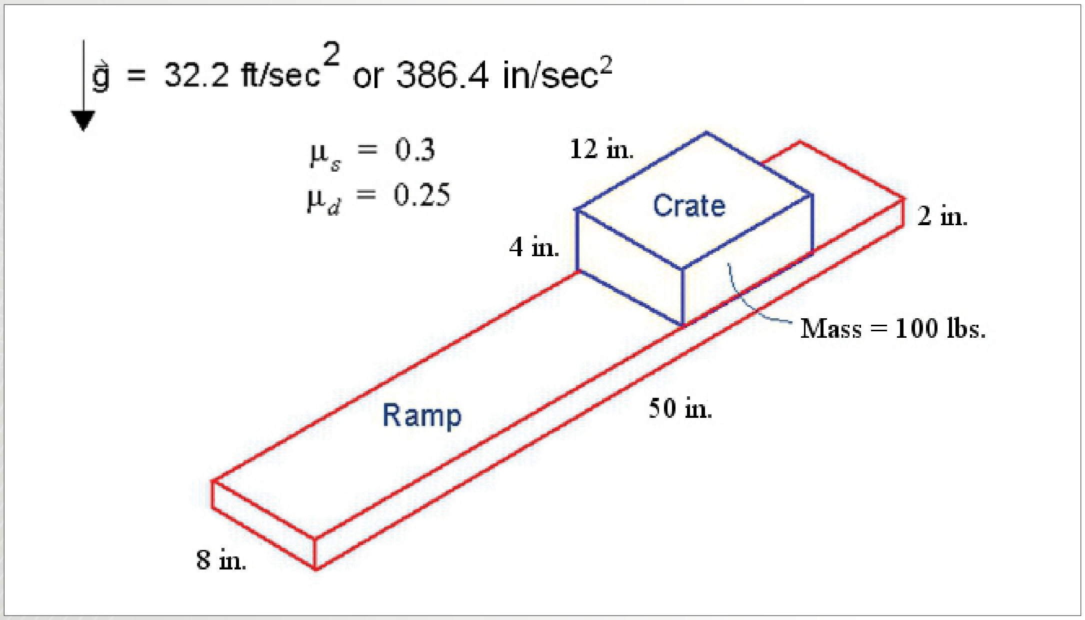
a. To find the distance, use y = - (1/2) gt2

b. Substitute: g = 9810 mm/s2 , t= 1 s, in the above equation.

c. Results: y = - 4905

d. The results produced by Adams View is - 4903, this shows that the stone is traveling 4903 mm in the negative y direction. The hand calculated answer and the Adams/View generated answer has a 0.04% difference.

# Example 2: Inclined Plane




## Software Version

Adams 2013.2

## Problem Description

Find the minimum inclination that will ensure that a crate slides off an inclined plane, the plane has dimensions of 50 in. by 8 in. by 2 in and the crate has dimensions of 12 in. by 8 in. by 4 in. and has a mass of 100 lbs. The coefficient of static friction (μs) is 0.3 and the coefficient of dynamic friction (μd) is 0.25 and gravity is 32.2 ft/sec2.

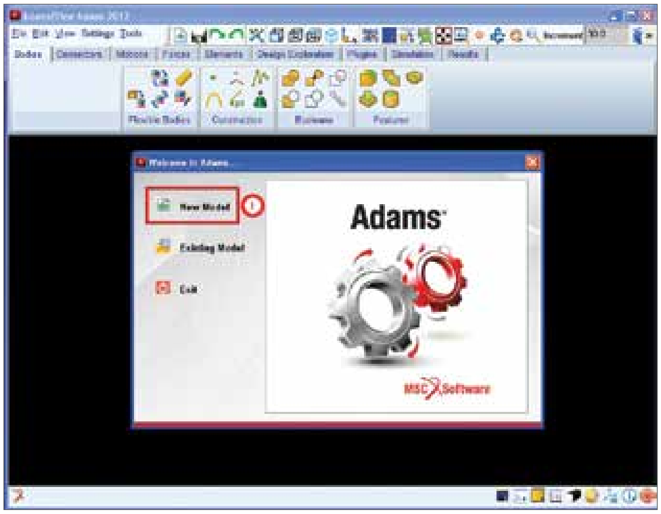
## Step 1. Create a New Adams database

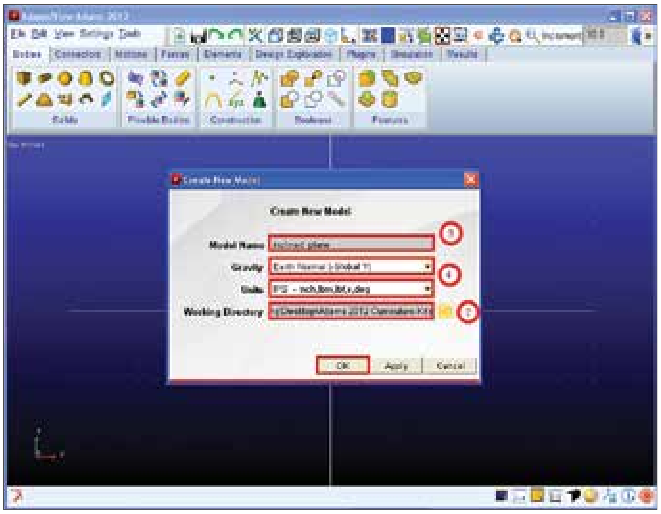
a. Click on Create a new model.

b. Under Working Directory, browse to the folder where you want to save your model.

c. Type the name of the new Model name as inclined_plane and click OK.

d. Make sure that the Gravity is set to Earth Normal (-Global Y) and the Units is set to IPS - inch, lbm, lbf, s, deg.




## Step 2. Adjust the Working Grid.

a. From the Settings menu, select Working Grid.

b. Set Spacing to 1 in. in the x and y direction.

c. Make sure that the working grid is oriented along the Global XY direction (default setting when you open Adams/View). The Set Orientation pull-down menu allows you to choose Global XY, YZ, XZ, or custom orientation. Click Apply and OK.

## Step 3. Constructing the Geometries of the Plane and Crate.

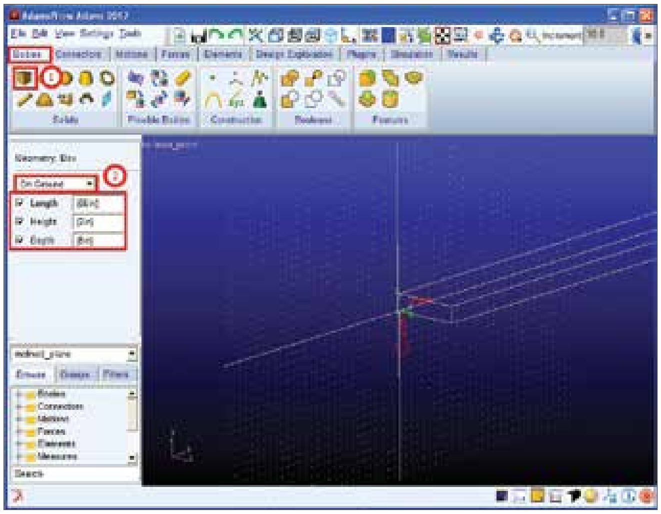
a. To create the plane, right-click on the Rigid Body icon and select Rigid Body: Box.

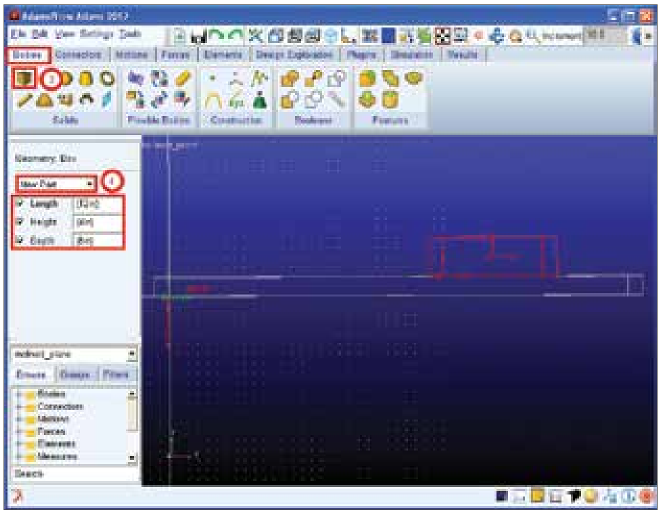
b. Make sure On Ground is selected and enter (50 in) for the Length, (3 in) for the Width, and (8 in) for the Depth.

c. Make sure that the Length, Width, and Depth are all checked. Then click on the center of the coordinate plane and hit Enter to create the plane.

d. To create the crate, right-click on the Rigid Body icon and select Rigid Body: Box.

e. Make sure New Part is selected and enter (12 in) for the Length, (4 in) for the Width, and (8 in) for the Depth. Also make sure that the Length, Width, and Depth are all checked. Then position the crate near the end of the ramp as shown.





## Step 4. Rename the Crate and Ramp Geometry and Assign Physical Properties to the Objects

a. Right-click on the large box (plane), point to Block: BOX_1, and then select Rename.

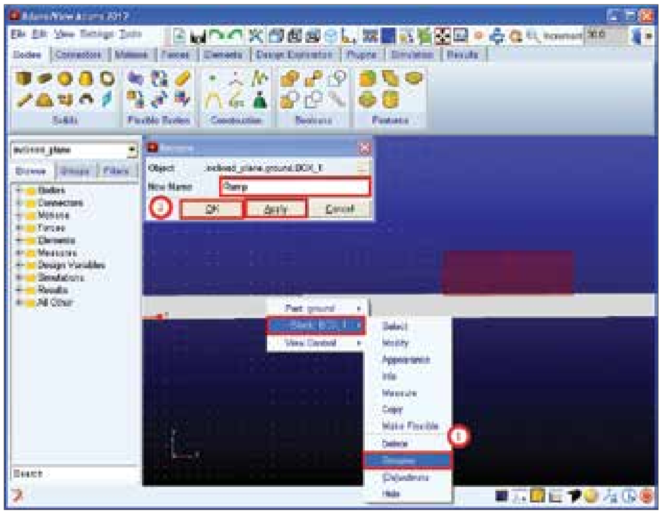
b. Enter Ramp, under New Name, and click Apply and OK.

c. Right-click on the smaller box (Crate), point to Block: BOX_2, and then select Rename.

d. Enter Crate, under New Name, and click Apply and OK.

e. Enter the mass of the crate by right-clicking on crate and going to Part:Crate, and then selecting Modify.

f. Set Define Mass By to User Input and in the Mass text box, enter 100 lbm. Click Apply and OK.



## Step 5. Set the Model's Inclination Angle.

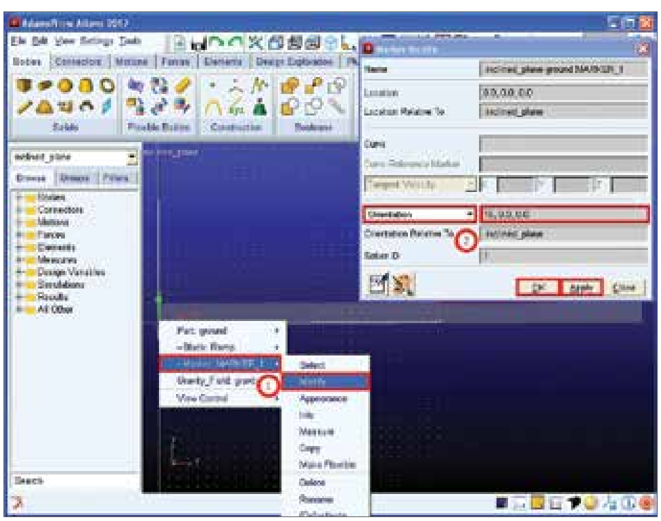
a. On the file tree to the left, under Bodies>ground right click MARKER _1 and select modify.

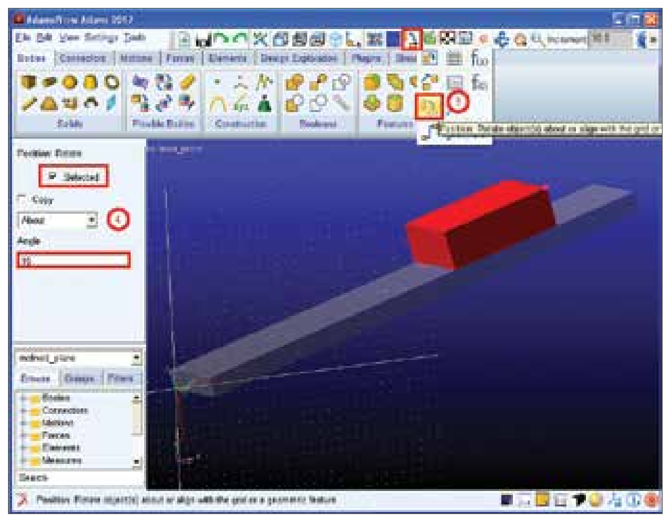
b. Under Orientation, input 15.0, 0.0, 0.0 click Apply and OK.

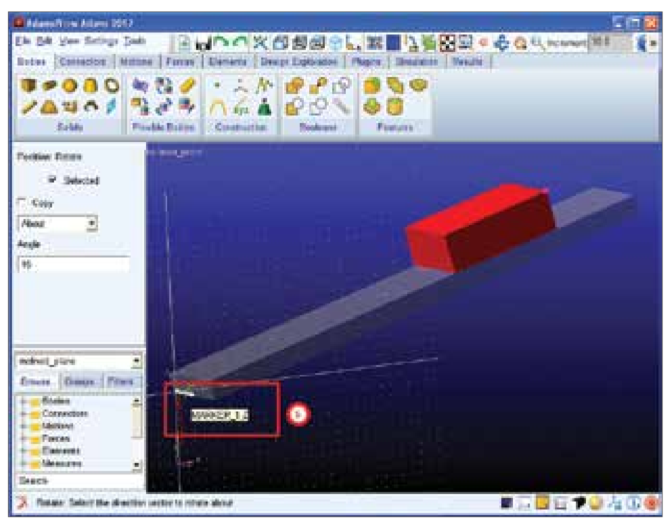
c. Under the Move tool stack, select the Align & Rotate tool.

d. Under Angle, input 15 and press Enter. Then click on the crate to select it as the object that will be rotated.

e. Now select the Z-axis of MARKER_1 (MARKER_1.Z) as the axis of rotation. It may be easier to rotate the view slightly to select the Z-axis.







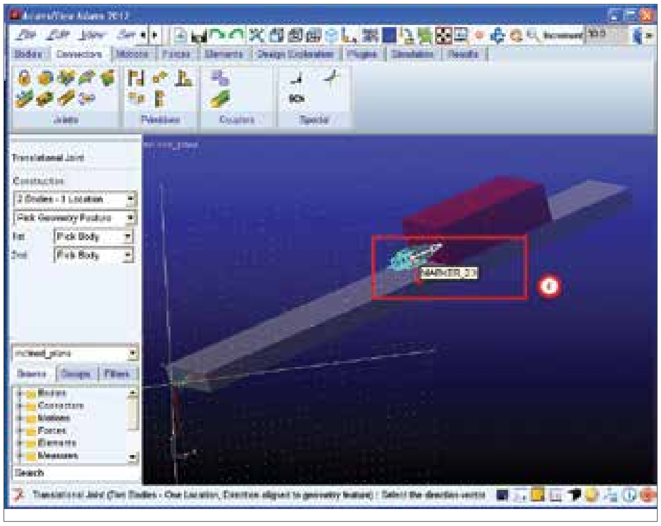
## Step 6. Adding Constraints on the Model.

a. To create a translational joint between the ramp and the crate, first go right-click on the Joint tool stack, and then select the Translational Joint tool.

b. Then select 2 Bod-1 Loc and choose Pick Feature.

c. Then proceed to select the bodies to be constrained by clicking on the crate, then the ramp.

d. Then for location choose Crate.MARKER_2 and then MARKER_2.X with the vector point up the ramp.



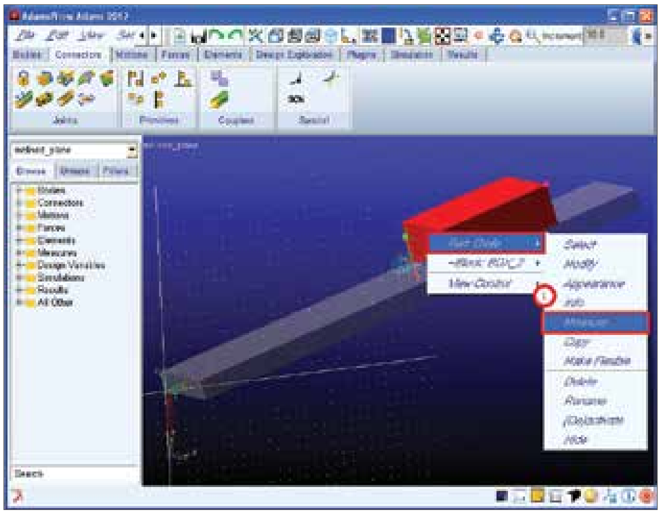
## Step 7. Taking Measurements for the Crate's Acceleration Along the Ramp

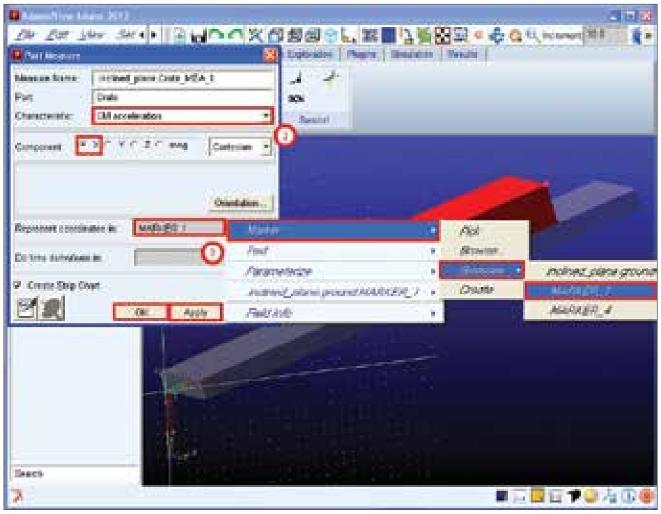
a. Right-click on the crate and go to Part:Crate and then Measure.

b. Under Characteristic select CM acceleration, under Component select X.

c. Under Represent coordinates in: right-click in the gray area and select Marker, then Guesses and then MARKER_1. Alternatively, you can select Pick and then select MARKER_1 in the geometry, which is the corner point at the bottom of the ramp.

d. Click Apply and OK.

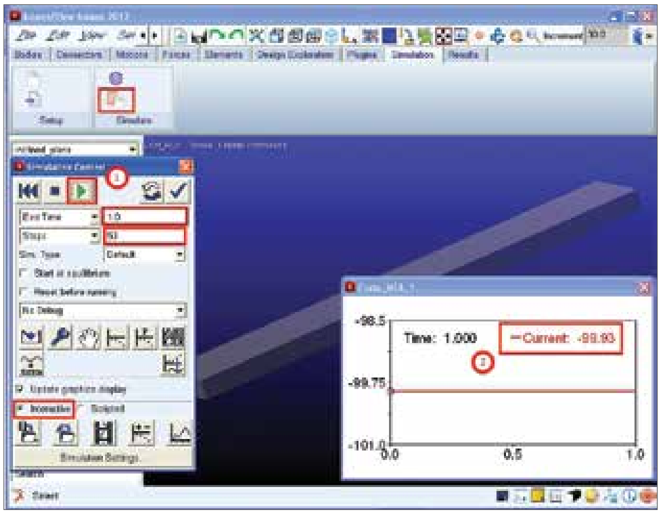


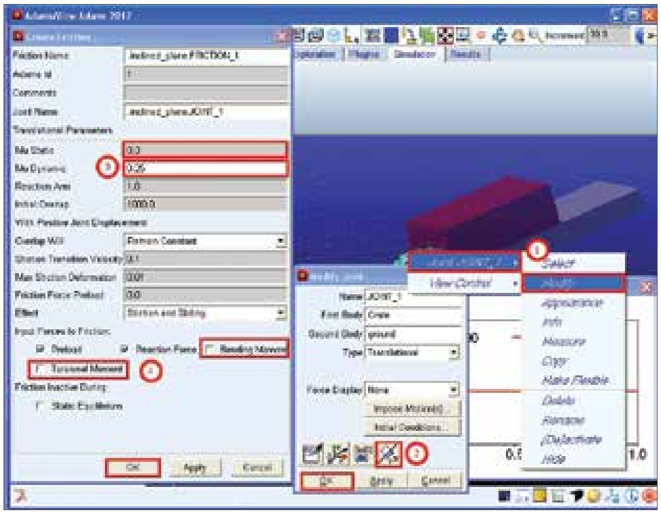


## Step 8. Verify the Mechanism

a. To verify the mechanism, simulate the model by clicking on the "calculator" icon for 1 second and 50 steps.

b. Find the value of the crate's constant acceleration and verify it by checking without friction in the Closed-form solution and making sure the values match.



## Step 9. Refine the model and Add Friction and Simulate

a. Display the joint's modify dialog box by right-clicking on the translational joint and pointing to Joint:JOINT_1, and then select Modify.

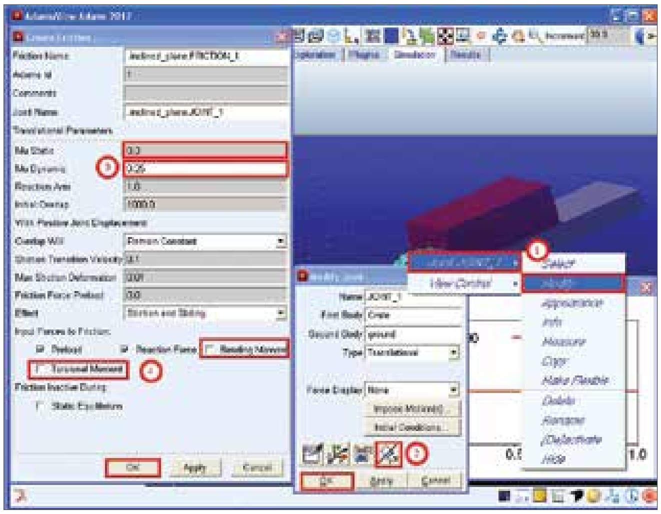
b. In the lower right corner of the Modify dialog box, select the Friction tool.

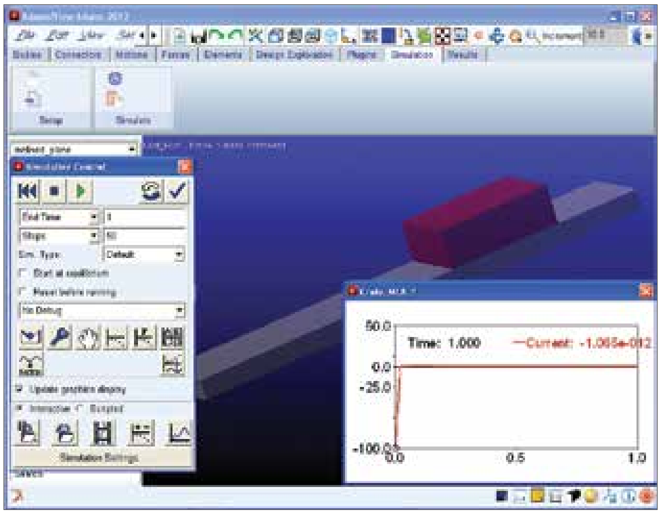
c. Fill in the coefficients of friction (0.3 for the coefficient of static friction and 0.25 for the coefficient of dynamic friction) and leave the remaining friction parameters at their default values.

d. In the Input Forces to Friction section, clear the selection of Bending Moment and Torsional Moment. Click OK on both windows.

e. Simulate the model and note if the create slides off the ramp.

f. Then right-click on the curve in the stripchart, and then select Save Curve.





## Step 10. Refine the Model Again by Changing the Ramp's Rotation Angle to 20°.

a. From the Build menu, select Group and New.

b. Make a group, named rotated_objects, containing: the crate, the joint, and all of the geometry on the ramp (including the markers but not the ground), by right clicking in the Objects In Group text box and going to All and then Browse.

c. This should bring up the Database Navigator, here select the Crate, MARKER_1, MARKER_4 and JOINT_1 (hold CTRL to select multiple entities) and then click OK on both boxes.

d. Now you can rotate the group, by going to the Main Toolbox, and from the Move tool stack, select the Precision Move tool .

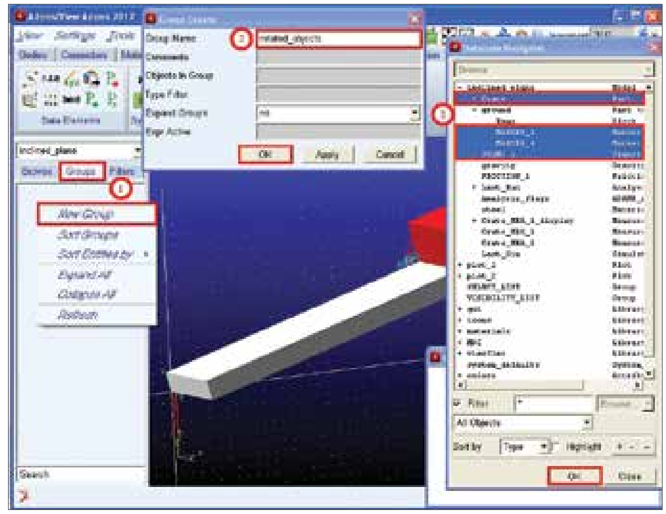
e. In the text box to the right of Relocate the, enter the group name, rotated_objects. Then click OK on the Database Navigator window.

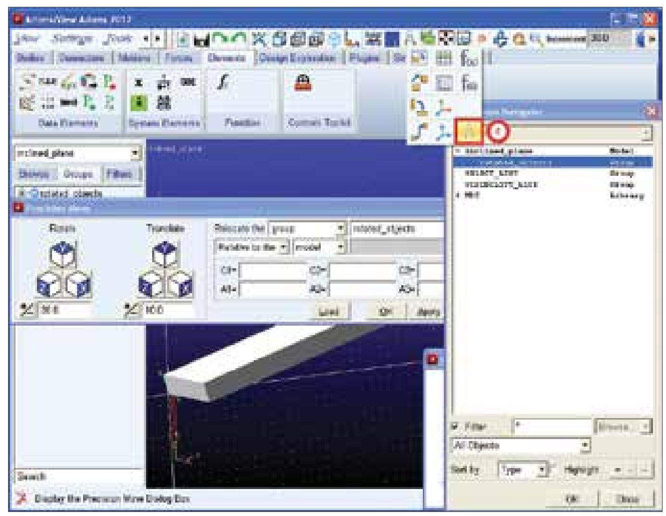
f. Set the menus in the second row to About the and marker.

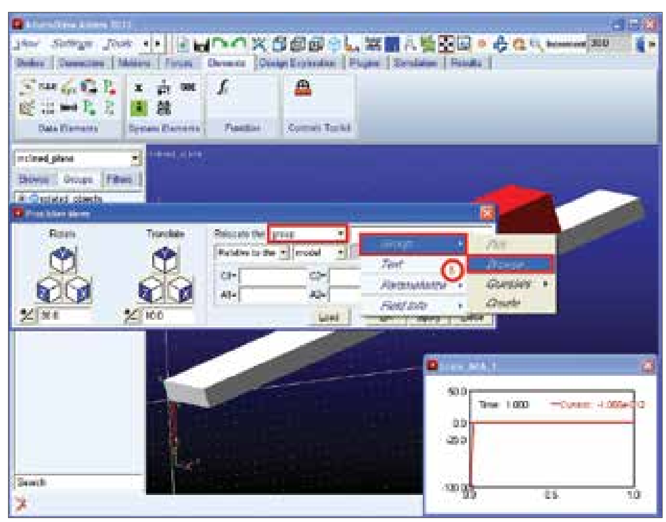
g. In the text box to the right of these menus, enter MARKER_1. The Precision Move tool rotates objects in increments about a specified axis of the marker you just selected.

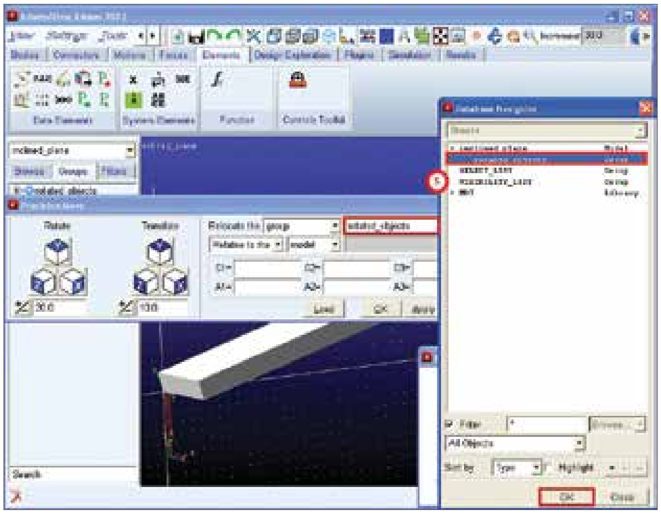
h. In the Plus/Minus text box, enter 5.

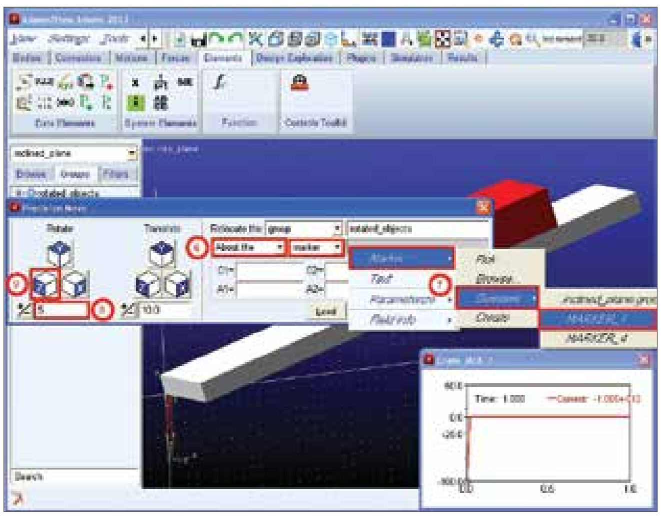
i. Select the Z-axis box. Note that you can select the axis box (X, Y, or Z) to rotate a group to the desired orientation. Now, click Close.







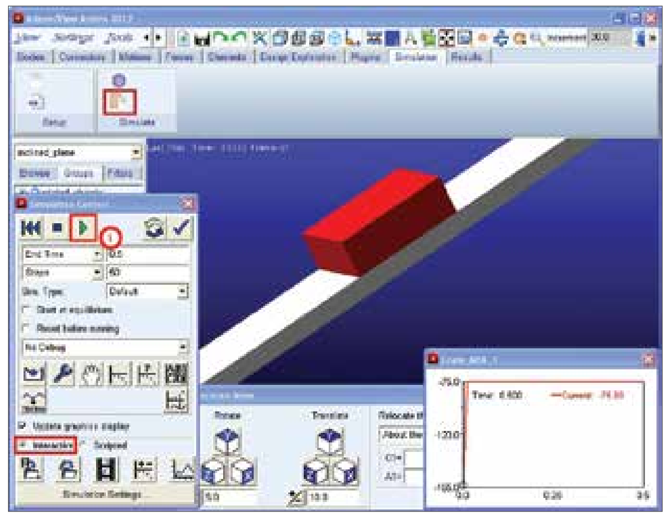




## Step 11. Find the Inclination Angles at which the Crate Starts to Slide.

a. Simulate the model and note if the crate slides off the ramp. For an end time of 0.5 seconds, verify that the create acceleration vs. time graph matches the adjoining figure.

b. Through trial and error, find the approximate angle at which the crate starts to slide off the ramp. Save the curve in the graph plot and compare your results using Adams/PostProcessor.



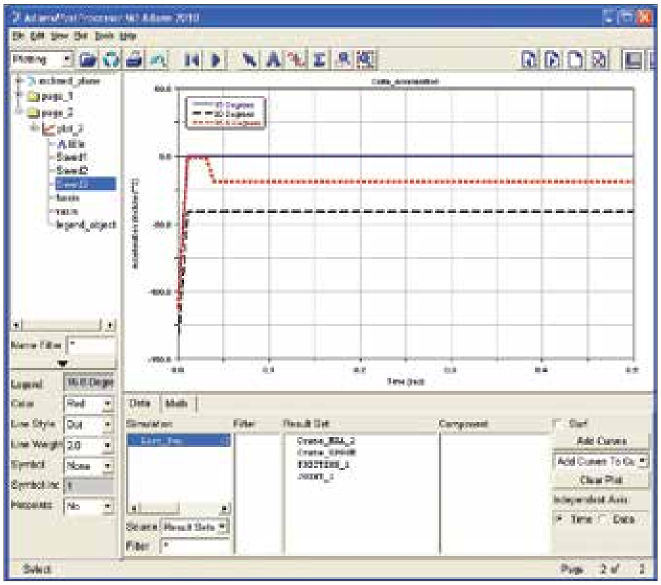
## Analytical Solution

MD ADAMS Simulation Results:

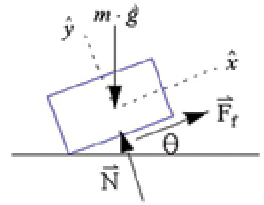
At θ = 15°, a = 0

At θ = 20°, a = -41.35 in/sec2.

Max Angle for Crate to Slip (θmax) = 16.8°. a = -19.19 in/sec2.



### Closed-form solution




**Without friction:**

$$\Sigma F_x = ma_x : -mg \cdot \sin\theta = ma_x$$

$$a_x = -g\sin\theta$$

$$\text{For } \theta = 15°,\ a_x = -32.2\sin(15°)$$

$$a_x = -99.96 \text{ in/sec}^2 \ (-8.33 \text{ ft/sec}^2)$$

**With friction:**

$$\Sigma F_y = 0 : -mg \cdot \cos\theta + N = 0$$

$$N = mg \cdot \cos\theta$$

Maximum angle ($\theta_{max}$) at which the crate will not slide:

$$\Sigma F_x = 0 : F_f - mg \cdot \sin\theta_{max} = 0$$

$$\mu_s \cdot N - mg \cdot \sin\theta_{max} = 0$$

$$\mu_s \cdot mg \cdot \cos\theta_{max} - mg \cdot \sin\theta_{max} = 0$$

$$\mu_s - \tan\theta_{max} = 0$$

$$\theta_{max} = \text{atan}(\mu_s) = \text{atan}(0.30) = 16.7°$$

Once the crate starts sliding,

$$\Sigma F_x = ma_x : F_f - mg \cdot \sin\theta = ma_x$$

$$\mu_k \cdot N - mg \cdot \sin\theta = ma_x$$

$$\mu_k \cdot mg \cdot \cos\theta - mg \cdot \sin\theta = ma_x$$

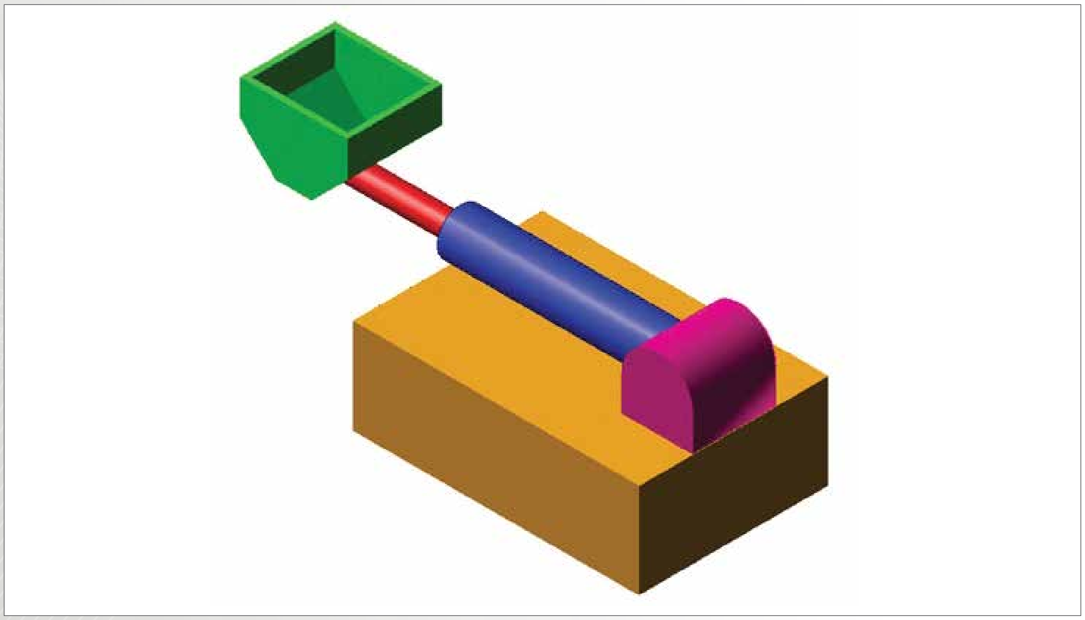
$$\mu_k \cdot \cos\theta - \sin\theta = \frac{a_x}{g}$$

$$a_x = (\mu_k \cos\theta - \sin\theta) \cdot g$$

$$\text{For } \theta = 20°,\ a_x = (0.25 \cdot \cos 20° - \sin 20°) \cdot 32.2 \text{ ft/sec}^2$$

$$a_x = -40.3 \text{ in/sec}^2 \ (-3.45 \text{ ft/sec}^2)$$

# Example 3: Lift Mechanism - Geometry



## Software Version

Adams 2013.2

## Problem Description

Create the geometry of the Lift Mechanism and then set the constraints of the model and then simulate the model.

## Step 1. Create a New Adams database

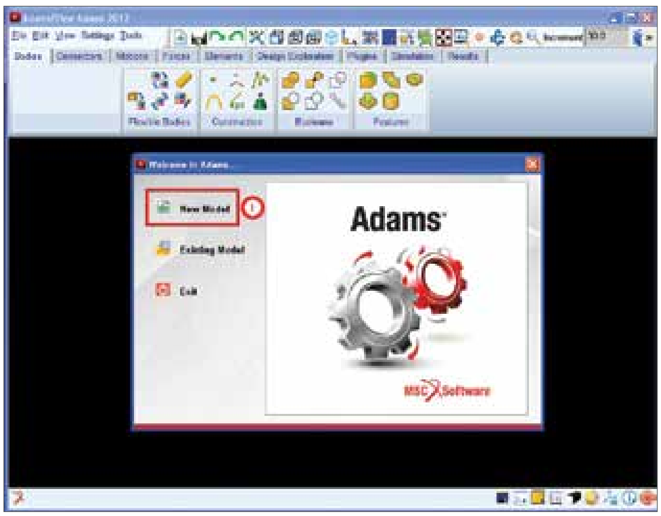
a. To import a file.

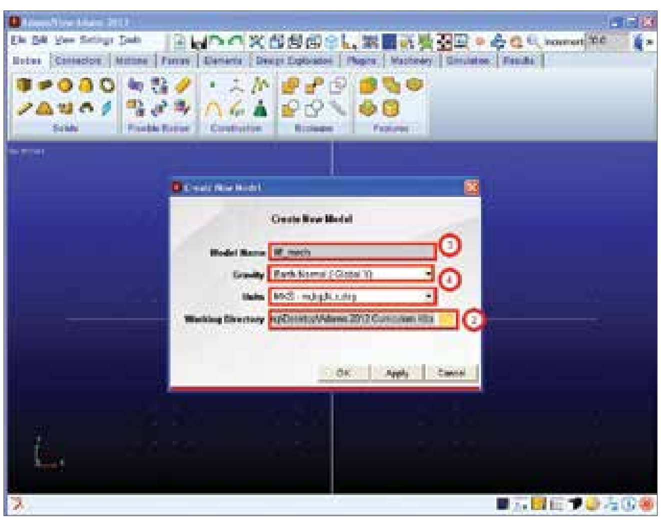
b. Click on **New model**.

c. Under **Working Directory**, browse to the folder where you want to save your model.

d. Type the name of the new **Model name** as **lift_mech** and click **OK**.

e. Make sure that the **Gravity** is set to **Earth Normal (-Global Y)** and the Units is set to **MKS - m,kg,N,s,deg**.



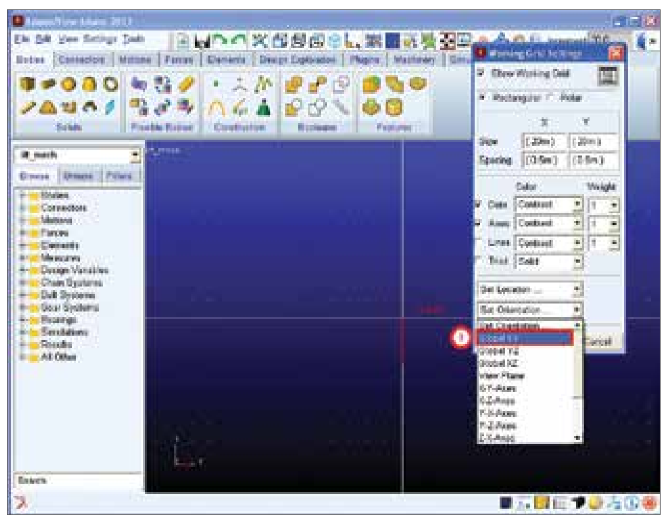




## Step 2. Adjust the Working Grid.

a. From the **Settings** menu, select **Working Grid**.

b. Set the **Size** in the **X direction** to **20 m** and the **Size** in the **Y directio**n to **20 m** and the **Spacing** in the x and y direction to 0.5 m. Since the grid is in meters you will probably need to zoom out to see it.

c. Make sure that the working grid is oriented along the global XY direction (default setting when you open Adams/View). The Set Orientation pull-down menu allows you to choose Global XY, YZ, XZ, or custom orientation. Click **Apply** and **OK**.

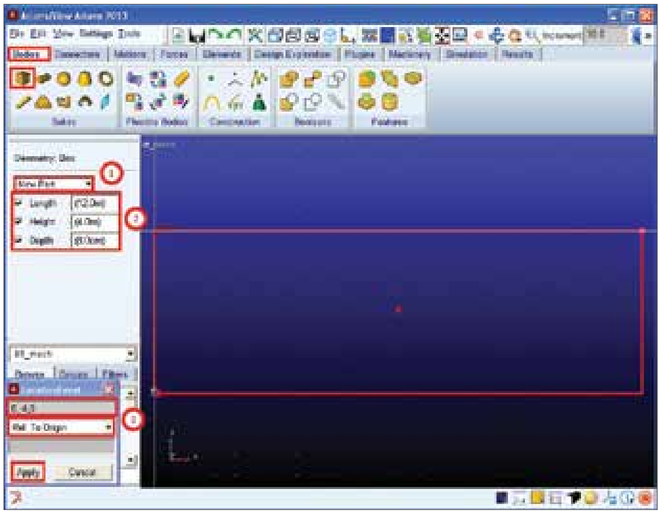


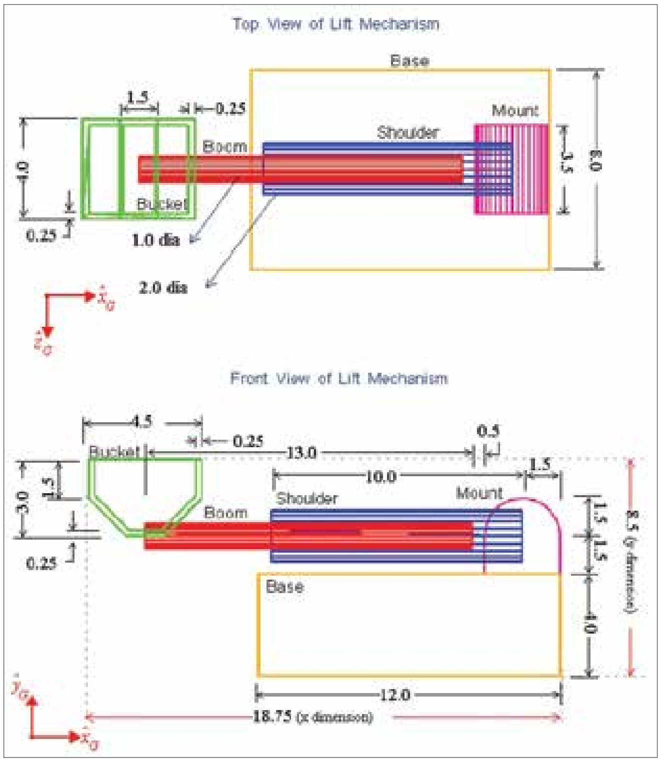
## Step 3. Create the Geometry of the Lift Mechanism: Create the Base

a. Create the geometry of the lift mechanism based on the dimensions on the diagram. For a challenge try to recreate the Lift Mechanism yourself and only use this guide if you are stuck. We're going to start with the Base first. Select the **Rigid Body** toolbox and select Box.

b. Then, under **Length**, enter 12 m, under **Height**, enter **4 m**, under **Depth**, enter 8 m. Make sure all the **Length, Height, and Depth** boxes are checked.

c. Hit Enter and then right-click on the working grid to open the **LocationEvent** box, here enter **0,-4,0** and make sure **Rel. To Origin** is selected then click **Apply**.



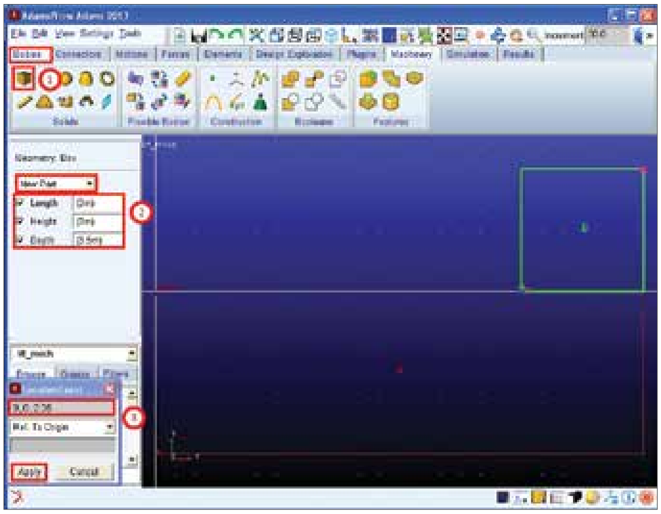



## Step 4. Create the Geometry of the Lift Mechanism: Create the Mount.

a. Select the **Rigid Body** toolbox and select **Box**.

b. Then, under **Length**, enter **3 m**, under Height, enter 3 m, under **Depth**, enter **3.5 m**. Make sure all the **Length, Height, and Depth** boxes are checked.

c. Hit Enter and then right-click on the working grid to open the **LocationEvent** box, here enter **9,0,2.25** and make sure **Rel. To Origin** is selected then click **Apply**.



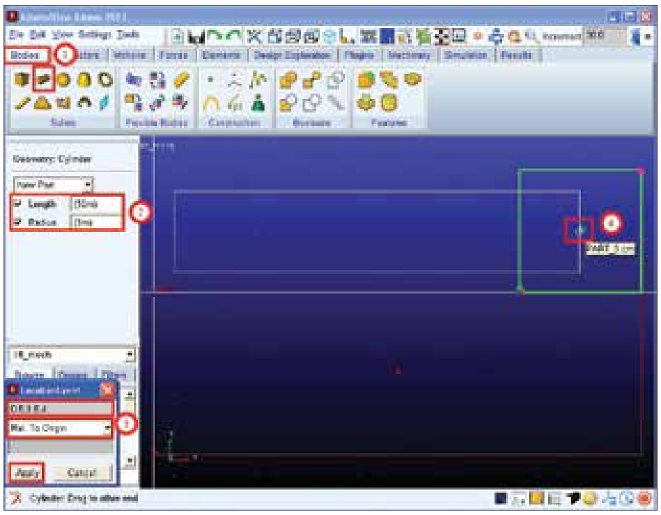
## Step 5. Create the Geometry of the Lift Mechanism: Create the Shoulder.

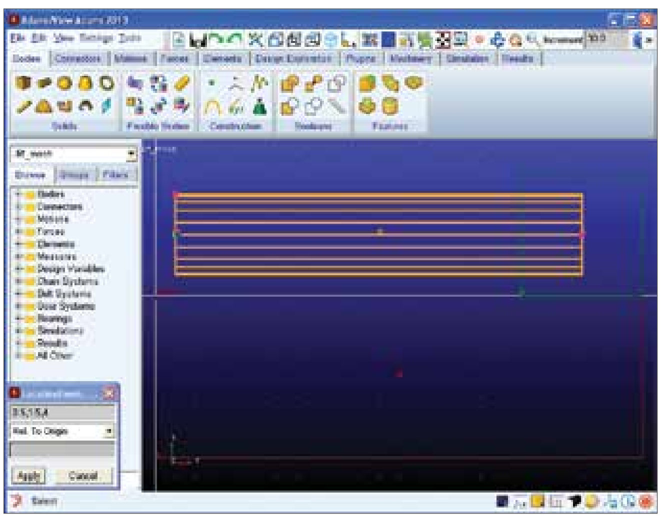
a. Select the **Rigid Body** toolbox and select Cylinder.

b. Then, under **Length**, enter **10 m**, under **Radius**, enter **1 m**. Make sure all the Length and Radius boxes are checked.

c. Hit Enter and then right-click on the working grid to open the **LocationEvent** box, here enter **0.5,1.5,4** and make sure **Rel. To Origin** is selected then click **Apply**.

d. Now click on the center of the **Mount** as shown to define the other endpoint of the cylinder.





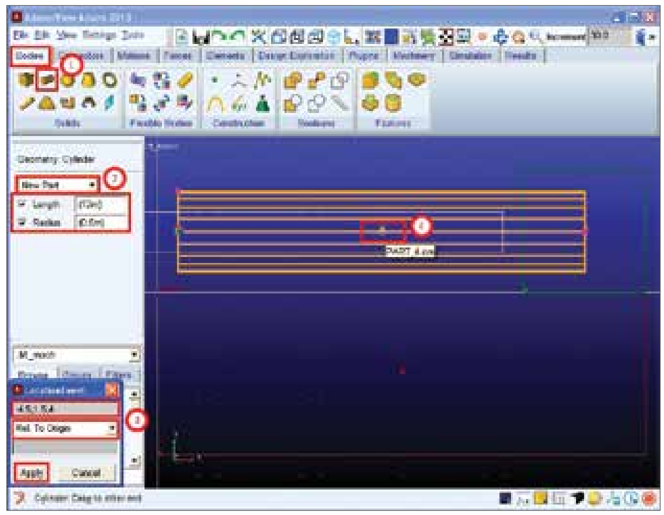
## Step 6. Create the Geometry of the Lift Mechanism: Create the Boom.

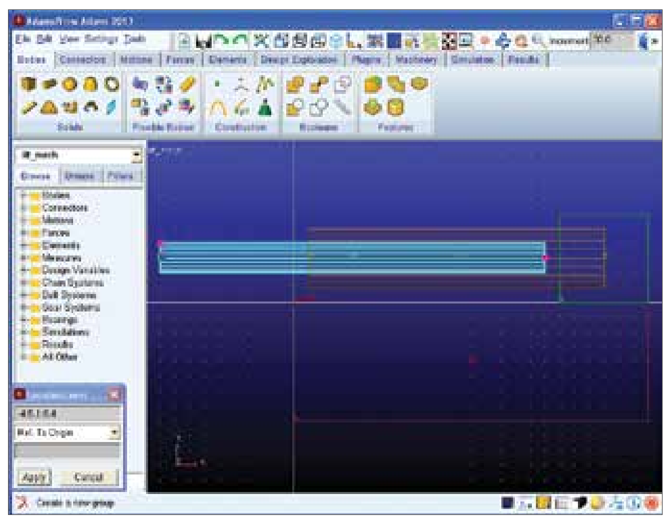
a. Select the **Rigid Body** toolbox and select **Cylinder**.

b. Then, under **Length**, enter **13 m**, under **Radius**, enter **0.5 m**. Make sure all the **Length** and **Radius** boxes are checked.

c. Hit Enter and then right-click on the working grid to open the **LocationEvent** box, here enter **-4.5,1.5,4** and make sure **Rel. To Origin** is selected then click **Apply**.

d. Now click on the center of either the **Shoulder** or the **Mount** as shown to define the other endpoint of the cylinder.



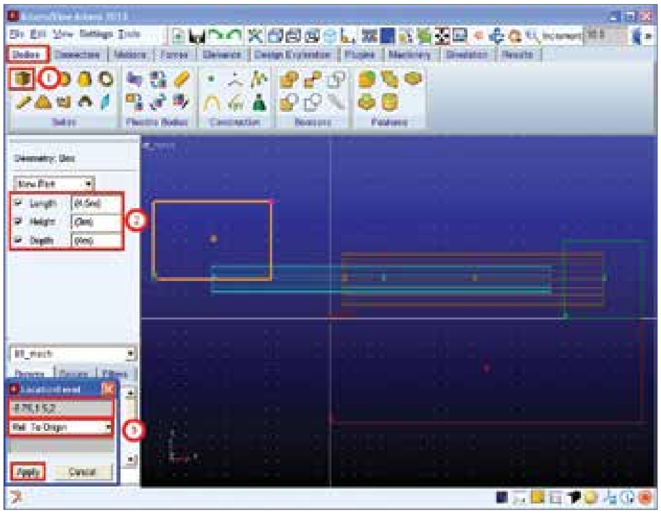


## Step 7. Create the Geometry of the Lift Mechanism: Create the Bucket.

a. Select the Rigid Body toolbox and select Box.

b. Then, under **Length**, enter **4.5 m**, under **Height**, enter **3 m**, under **Depth**, enter **4 m**. Make sure all the **Length, Height, and Depth** boxes are checked.

c. Hit Enter and then right-click on the working grid to open the **LocationEvent** box, here enter **-6.75,1.5,2** and make sure **Rel. To Origin** is selected then click **Apply**.

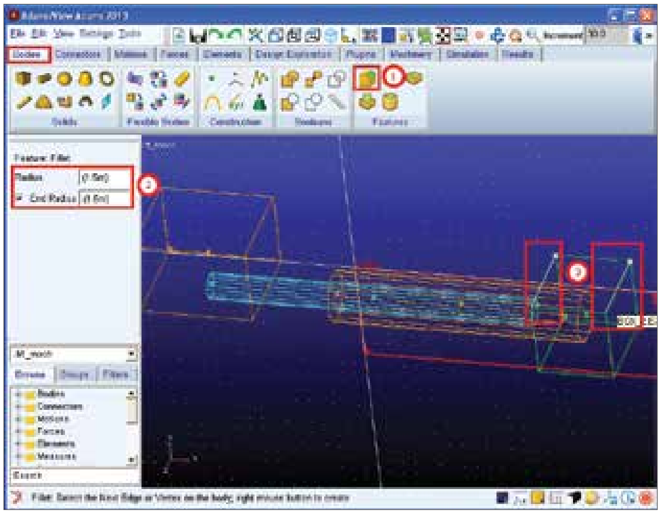


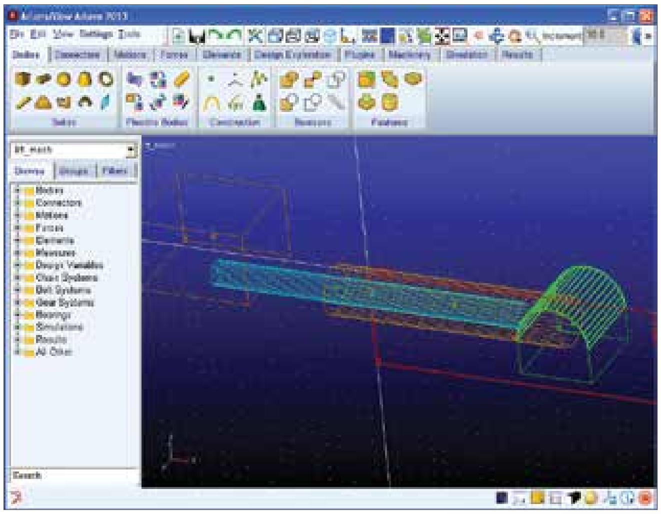
## Step 8. Apply Fillets on the Mount using the Fillet Tool.

a. From the **Rigid Body** toolbox select the **Fillet** tool.

b. Under **Radius**, enter **1.5 m**, and then check the box for **End Radius** and enter **1.5 m**.

c. Then select the edges of the Mount as shown and then right-click on it to create the fillets. You may want to rotate the view to make this task easier.





## Step 9. Modify the Bucket Using the Chamfer and Hollow Tools.

a. From the **Rigid Body** toolbox select the **Chamfer** tool.

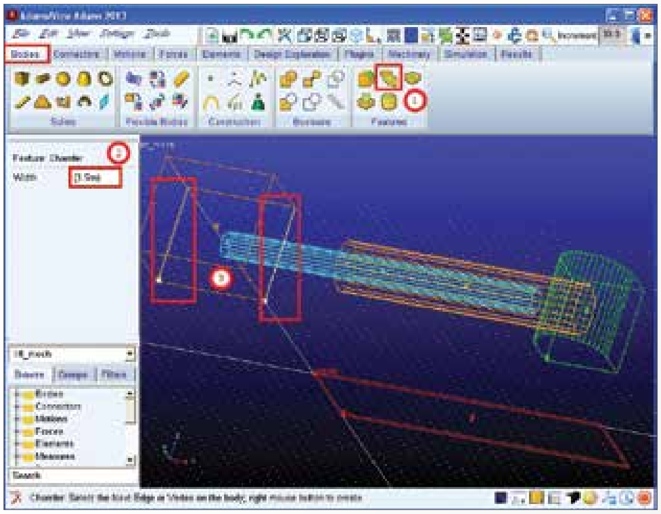
b. Under **Width**, enter **1.5 m**.

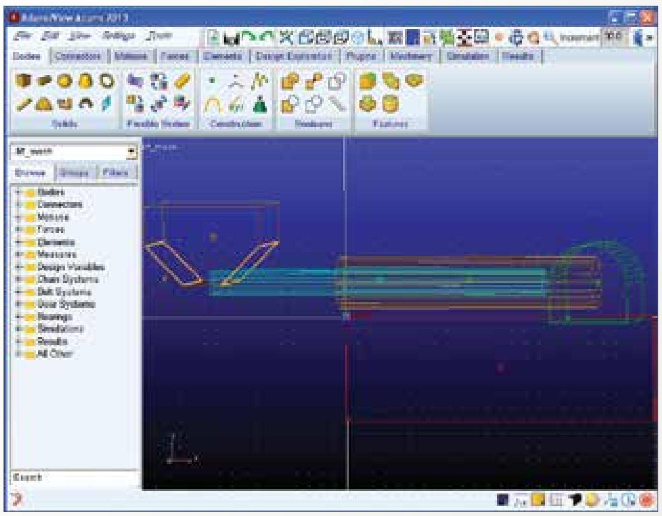
c. Then select the edges of the **Bucket** as shown and then right-click on it to chamfer it. You may want to rotate the view to make this task easier.

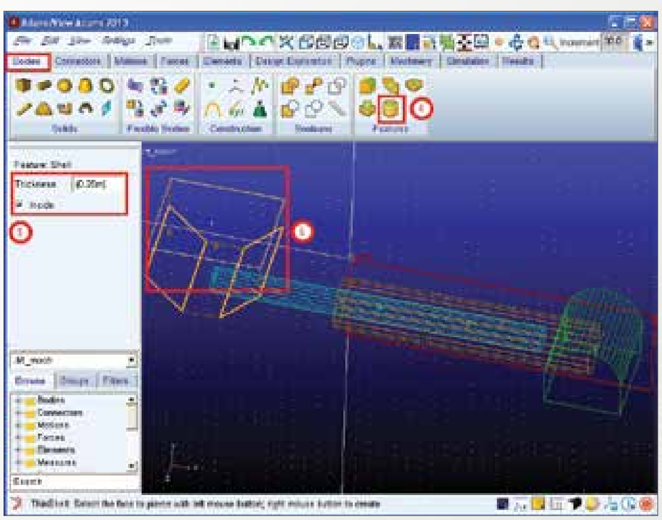
d. Now under the **Rigid Body** toolbox, select the **Hollow** tool.

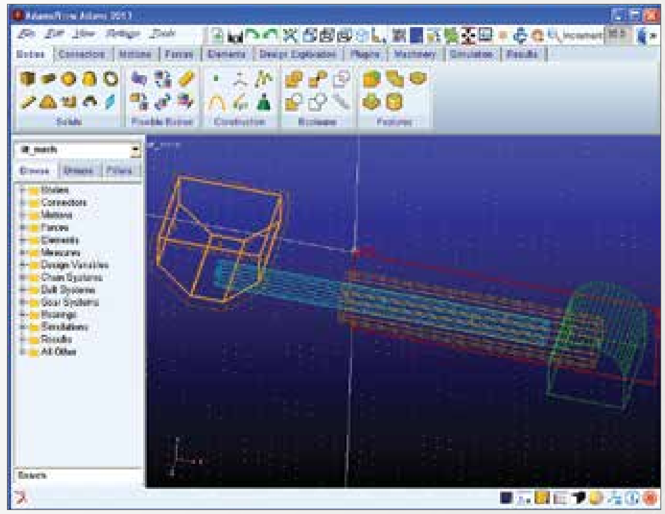
e. Under **Thickness**, enter **0.25 m** and make sure Inside is checked.

f. Then select the top face of the bucket and then right-click to hollow it out. You may want to rotate the view to make this task easier.

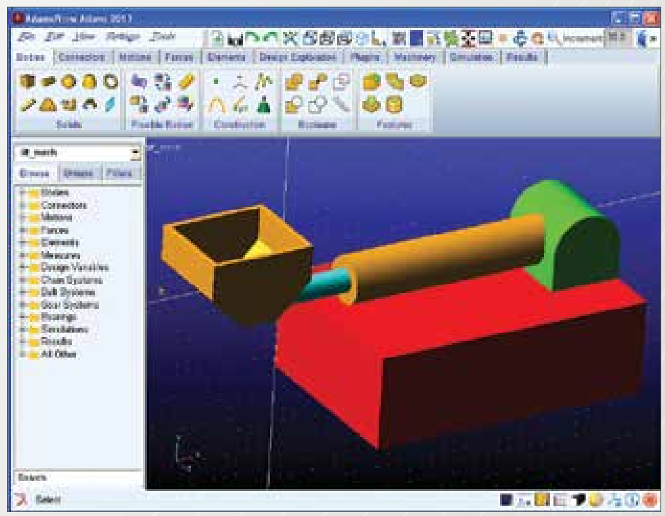


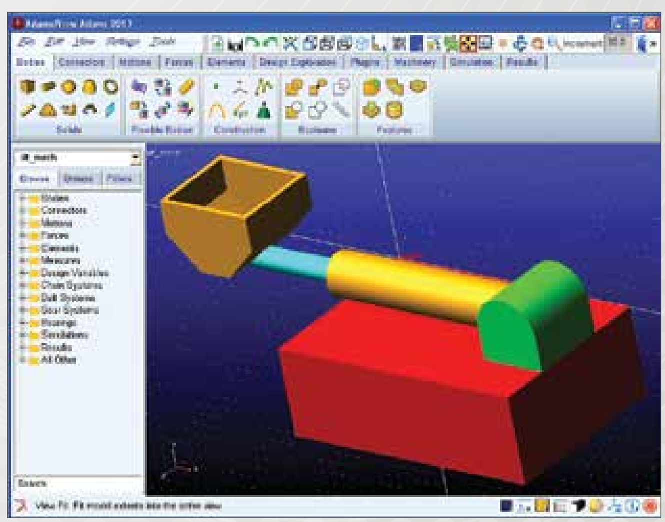




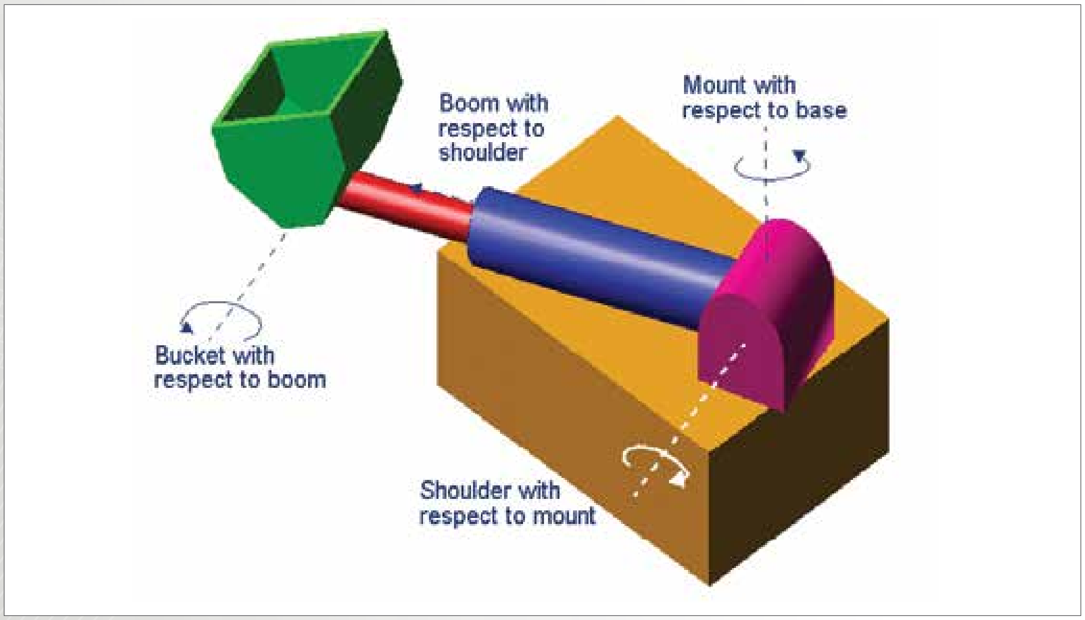


## Step 10. Final Model – Compare your Model





# Example 4: Lift Mechanism - Simulation




## Software Version

Adams 2013.2

## Problem Description

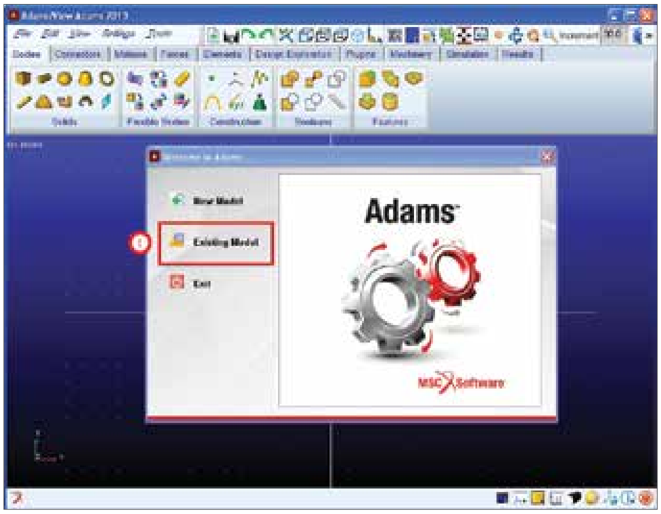
Continuing from the last example where you worked on the construction of the Lift Mechanism, add the proper constraints and joint motions to your model, as shown in the figure below, and successfully run a simulation of your model.

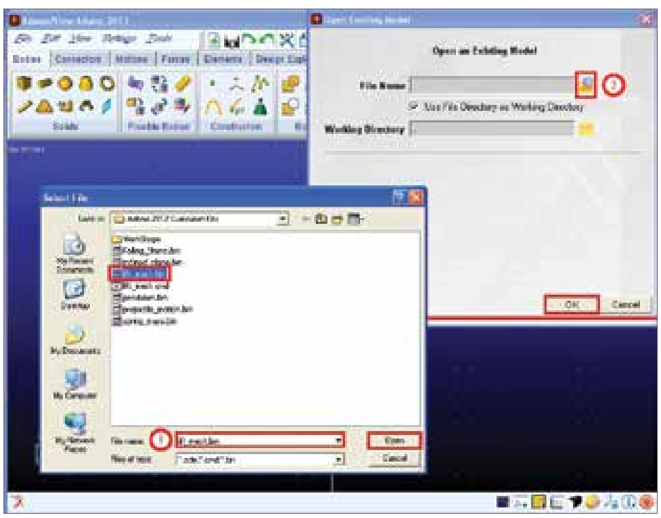
## Step 1. Open the File Containing your Model

a. Click on **Open an existing database**.

b. Under **File Name**, browse to the folder where your model is located, and then click **OK**.

c. Then locate the bin file that contains your model, **lift_mech**, and click **Open**.



## Step 2. Constrain the Base to the Ground.

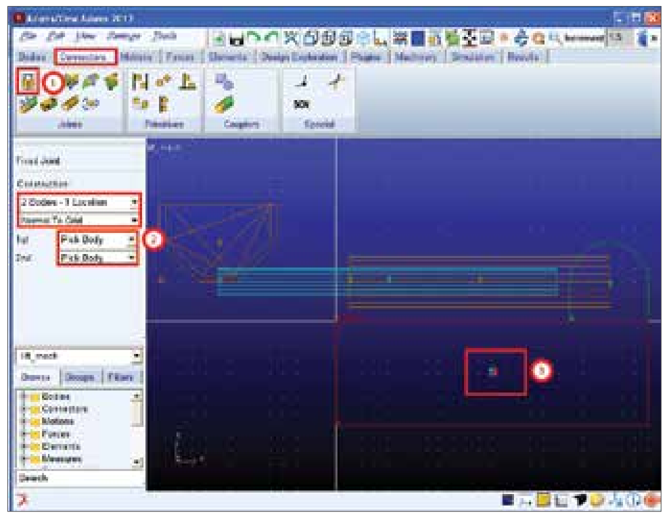
a. From the **Joint** toolbox, select **Fixed**.

b. Under Construction, make sure **2 Bod-1 Loc, Normal to Grid** are selected.

c. Select the **Base** as the **First Body** and the **Ground** as the **Second Body**

d. Select the Midpoint of the **Base** as the **Location**. A Lock icon should appear indicating that you have done this process successful. It may be easier to change the view to Wireframe to complete this process.

- Note: Because of the scale of the model you will need to zoom in to see the Lock icon, if you wish to make the scale of the Lock Icon larger, right-click on Joint:JOINT_1 and then go to Appearance and then increase the Icon Scale to 15 and click OK.



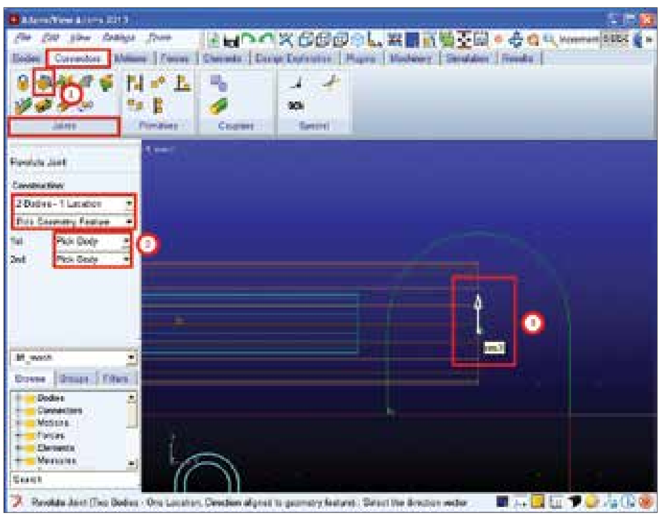
## Step 3. Constrain the Mount to the Base.

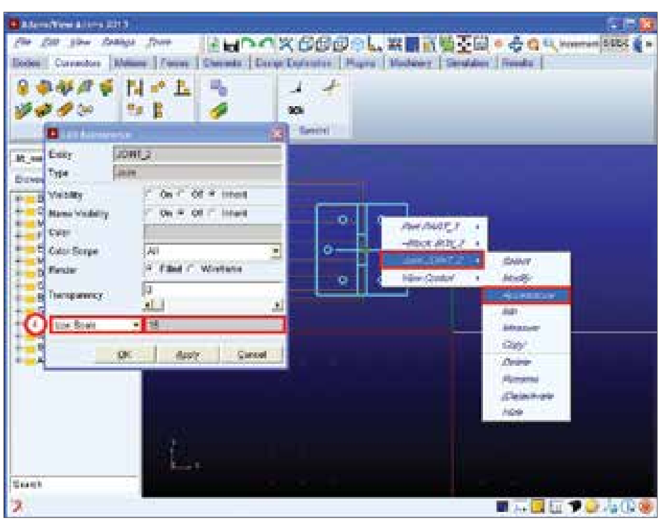
a. From the Joint toolbox, select **Revolute**.

b. Under Construction, make sure **2 Bod-1 Loc, Pick Feature** are selected.

c. Then select the **Mount** as the **First Body** and the **Base** as the **Second Body**, and then select the Midpoint of the **Mount** as the Location. Then select the **Global Y-Direction** as the axis of rotation. A Hinge icon should appear indicating that you have done this process successful.

- Note: Because of the scale of the model you will need to zoom in to see the Hinge icon, if you wish to make the scale of the Hinge Icon larger, right-click on Joint:JOINT_2 and then go to Appearance and then increase the Icon Scale to 15 and click OK.





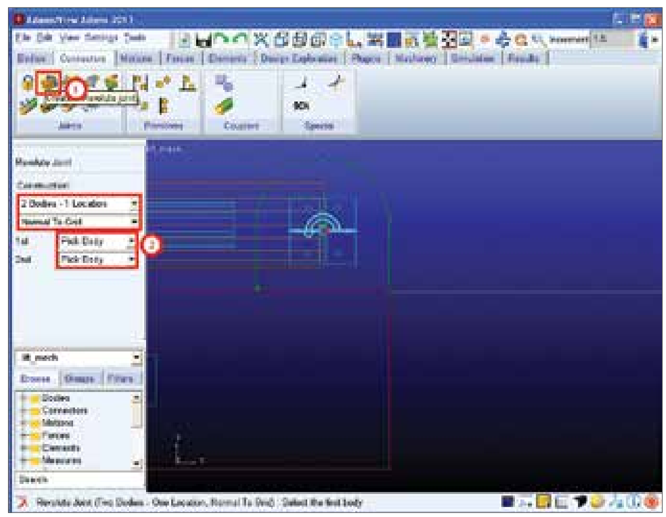
## Step 4. Constrain the Shoulder to the Mount.

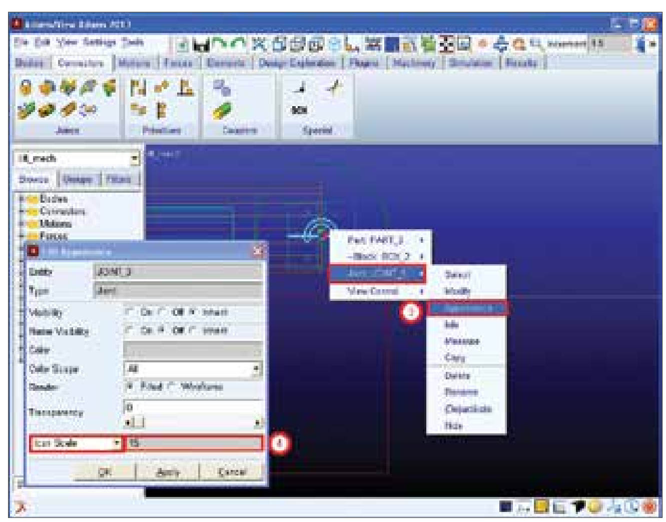
a. From the Joint toolbox, select **Revolute**.

b. Under Construction, make sure **2 Bod-1 Loc, Normal To Grid** are selected.

c. Then select the Shoulder as the **First Body** and the Mount as the **Second Body**, and then select the **Anchor Marker** of the Shoulder as the Location. A Hinge icon should appear indicating that you have done this process successful.

- Note: Because of the scale of the model you will need to zoom in to see the Hinge icon, if you wish to make the scale of the Hinge Icon larger, right-click on Joint:JOINT_3 and then go to Appearance and then increase the Icon Scale to 15 and click OK.





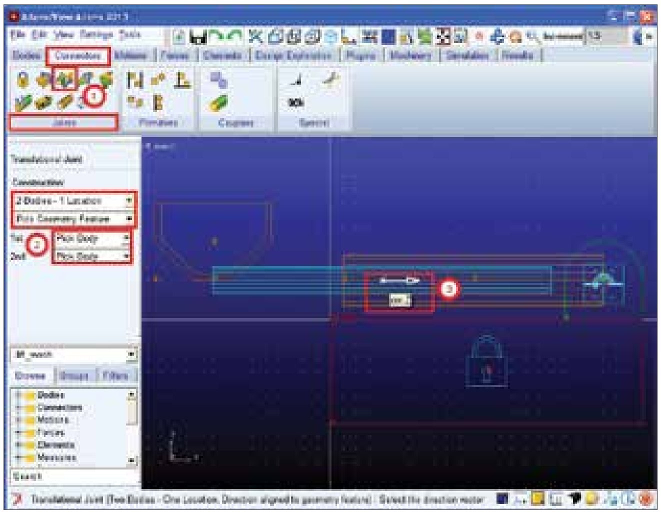
## Step 5. Constrain the Boom to the Shoulder

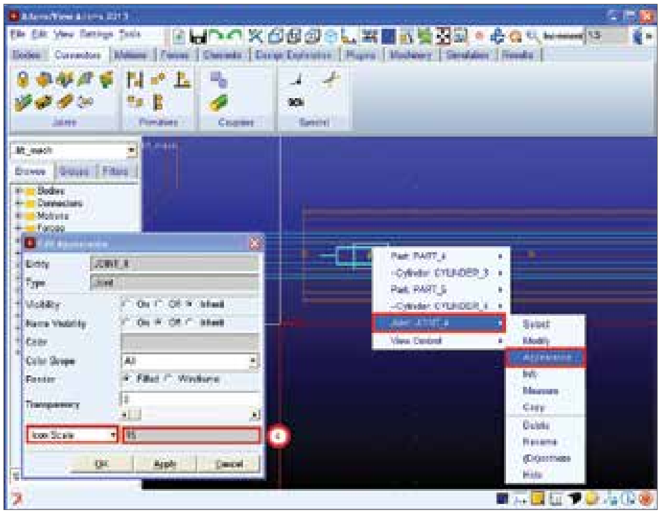
a. From the Joint toolbox, select **Translational**.

b. Under Construction, make sure **2 Bod-1 Loc, Pick Feature** are selected.

c. Then select the **Boom** as the **First Body** and the **Shoulder** as the **Second Body**, and then select the Midpoint of the **Boom** as the **Location**. Then select the **Global X-Direction** as the axis of rotation. A "Translational" icon should appear indicating that you have done this process successful.

- Note: Because of the scale of the model you will need to zoom in to see the Hinge icon, if you wish to make the scale of the "Translational" Icon larger, right-click on Joint:JOINT_4 and then go to Appearance and then increase the Icon Scale to 15 and click OK.





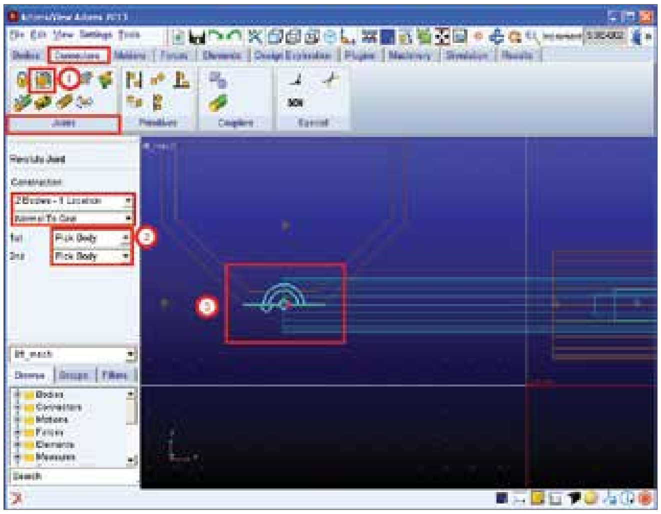
## Step 6. Constraint the Bucket to the Boom.

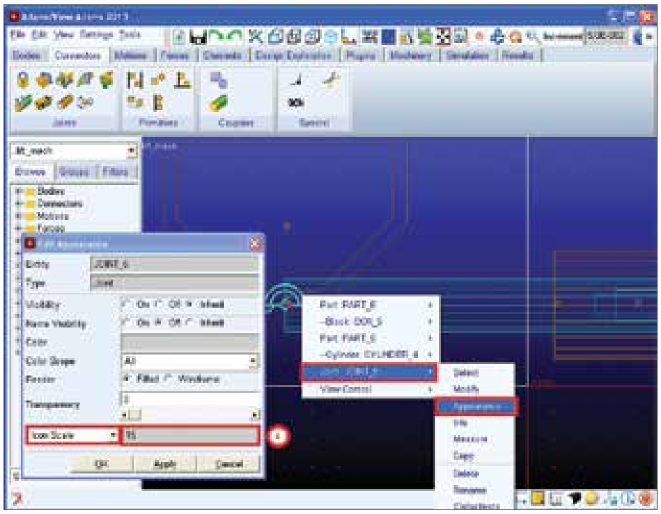
a. From the **Joint** toolbox, select **Revolute**.

b. Under Construction, make sure **2 Bod-1 Loc, Normal To Grid** are selected.

c. Then select the **Bucket** as the **First Body** and the **Boom** as the **Second Body,** and then select the Midpoint of the **Boom** as the **Location**. A Hinge icon should appear indicating that you have done this process successful.

d. Note: Because of the scale of the model you will need to zoom in to see the Hinge icon, if you wish to make the scale of the "Translational" Icon larger, right-click on Joint:JOINT_5 and then go to Appearance and then increase the Icon Scale to 15 and click OK.

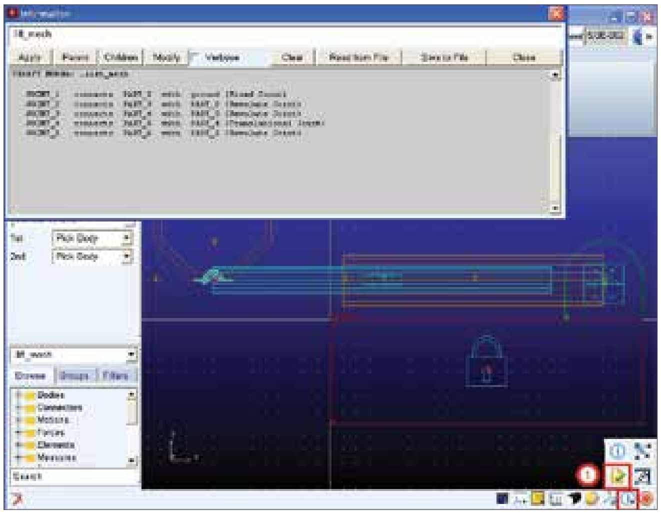


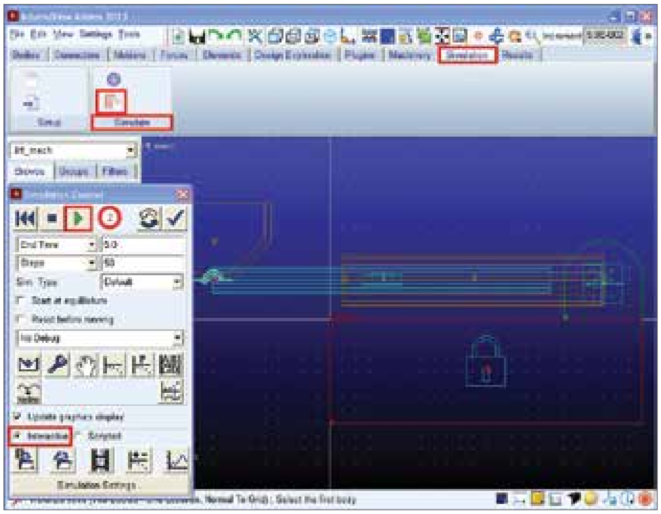


## Step 7. Verify your Model.

a. Check model topology by constraints by going to the **Status** bar and then right-clicking on the Information tool stack. Then select the Model Topology by constraints tool and check to see if everything is constrainted properly.

b. Perform a simulation to visually see if everything is constrained correctly.





## Step 8. Add Joint Motions.

a. First, add a motion to the Mount-to-Base joint by going to the **Motion Driver** tool stack and then select Rotational Joint Motion.

b. Under Speed, enter **360d*time**.

c. Then select the **Mount-to-Base revolulte joint** (JOINT_2) to apply.

d. Click revolute joint motion, then select the Shoulder-to-Mount revolute joint (JOINT_3) to apply. Choose default speed.

e. Now right click this Shoulder-to-Mount joint motion in the model tree and click modify, then enter -STEP(time,0,0,0.10,30d) in the Speed Box (Function(time)).

Now we will add a motion for the Boom-to-Shoulder joint. Under the Motion Driver tool stack, select Translational Joint Motion.

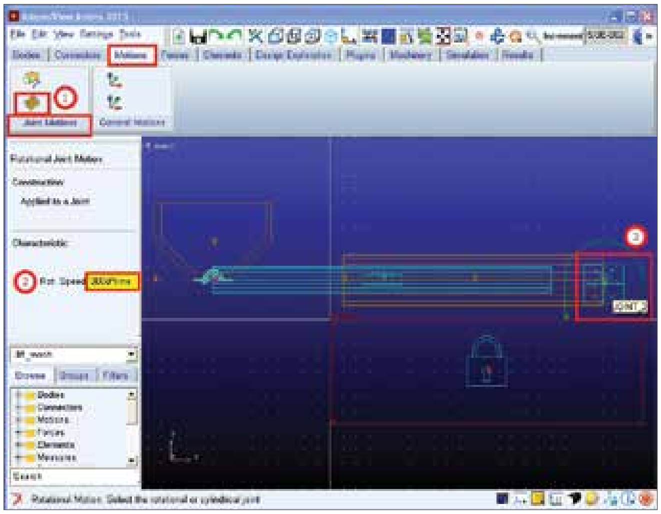
f. Click translational joint, then select the Boom-to-Shoulder translational joint (JOINT_4) to apply. Choose default speed.

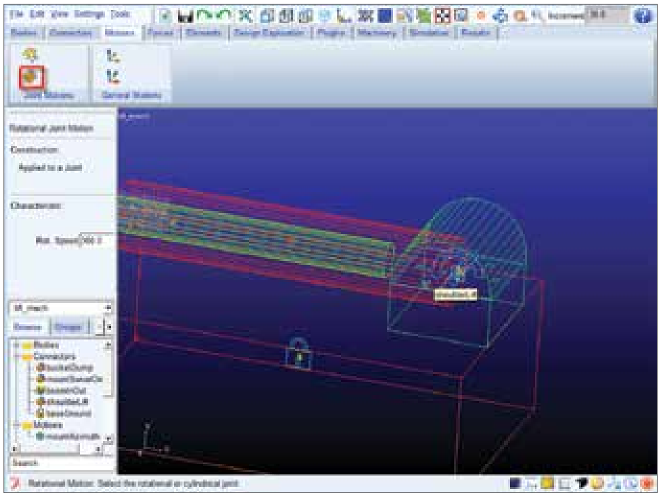
g. Now right click this Boom-to-Shoulder translational joint motion in the model tree and click modify, then enter -STEP(time,0.8,0,1,5) in the Speed Box (Function(time)).

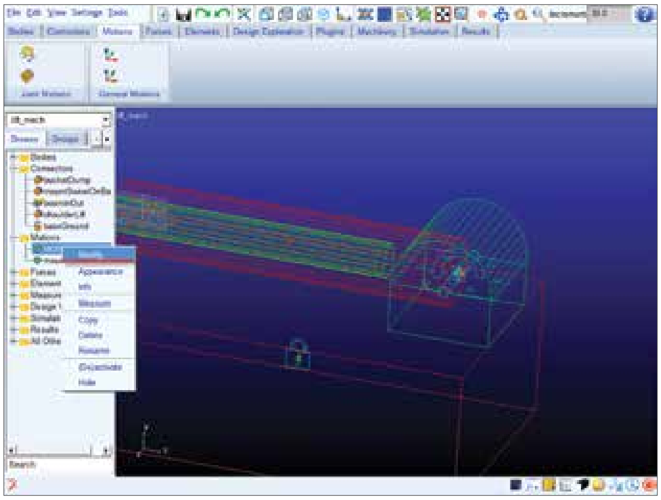
h. Lastly, we will add a motion to the Bucket-to-Boom joint. Once again under the Motion Driver tool stack, select Rotational Joint Motion.

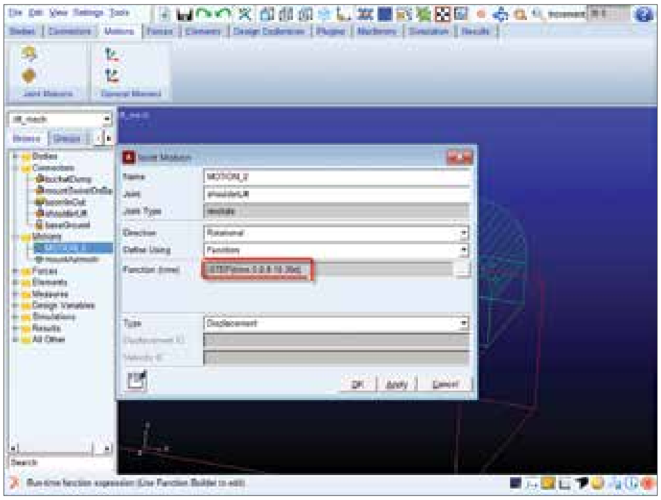
i. Click revolute joint motion, then select the Bucket-to-Boom revolute joint (JOINT_5) to apply. Choose default speed

j. Now right click this Bucket-to-Boom joint motion in the model tree and click modify, then enter 45d*(1-cos(360d*time)). in the Speed Box (Function(time)).









## Step 9. Verify the Joint Motions.

a. Check to see if the functions were properly entered for each joint by going to Modify and then Impose Motion and checking the Function. Also right near the joint and check the motion by going to right-clicking on the **Motion** then **Modify** and checking the **Function(time).** If the function box does not have the correct function, enter it and click **OK**.

b. For example, for the Mount-to-Base joint you can right-click on Joint:JOINT_2 and then click on Modify

c. Then, click on **Impose Motion**.

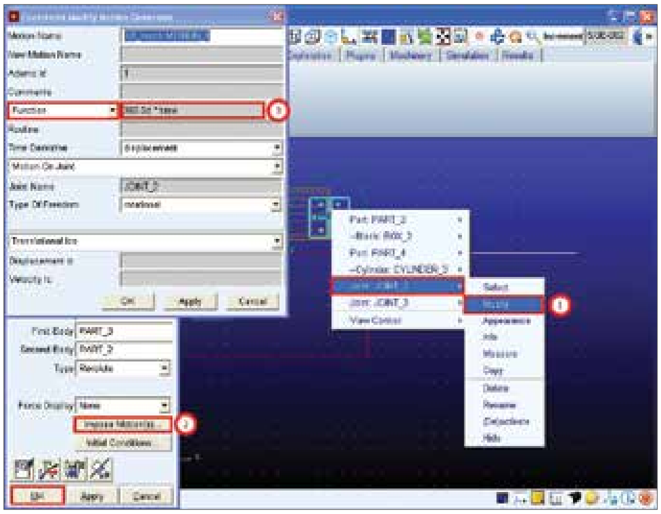
d. Then make sure that the **Function** textbox contains **360d*time**, then click **OK**.

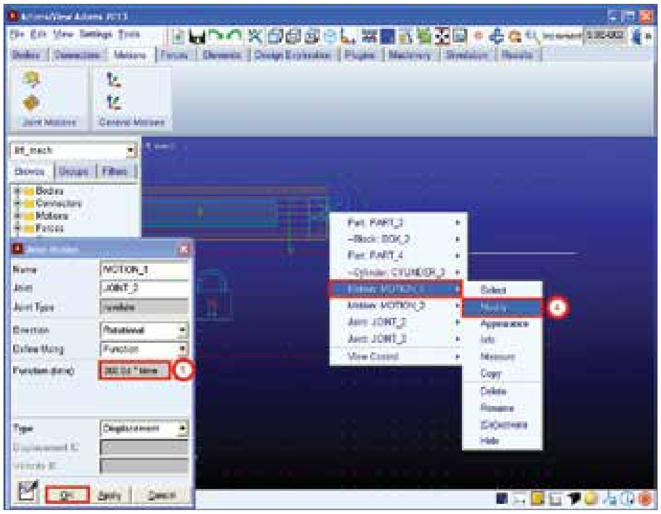
e. Now right-click on Motion: **MOTION_1** and click on Modify.

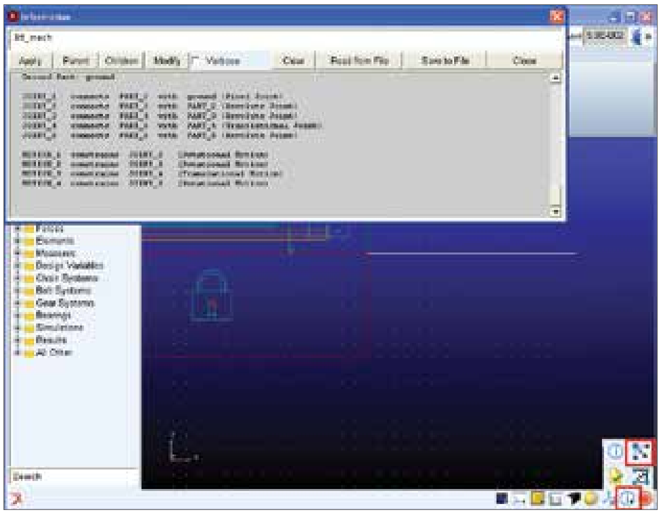
f. Make sure that the Function(time) textbox contains 360d*time, and click OK.

g. Repeat for all joints and motions.

h. Once you have done that, check the model topology by constraints by going to the Status bar and then right-clicking on the Information tool stack. Then select the Model Topology by constraints tool and verify if the joint motions have been applied properly.

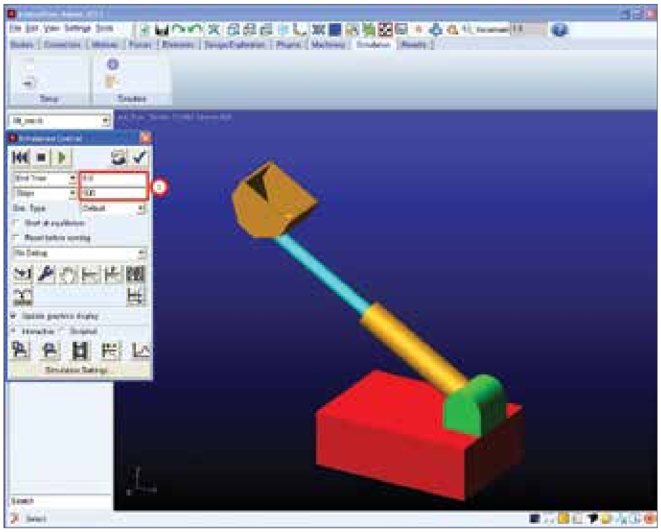




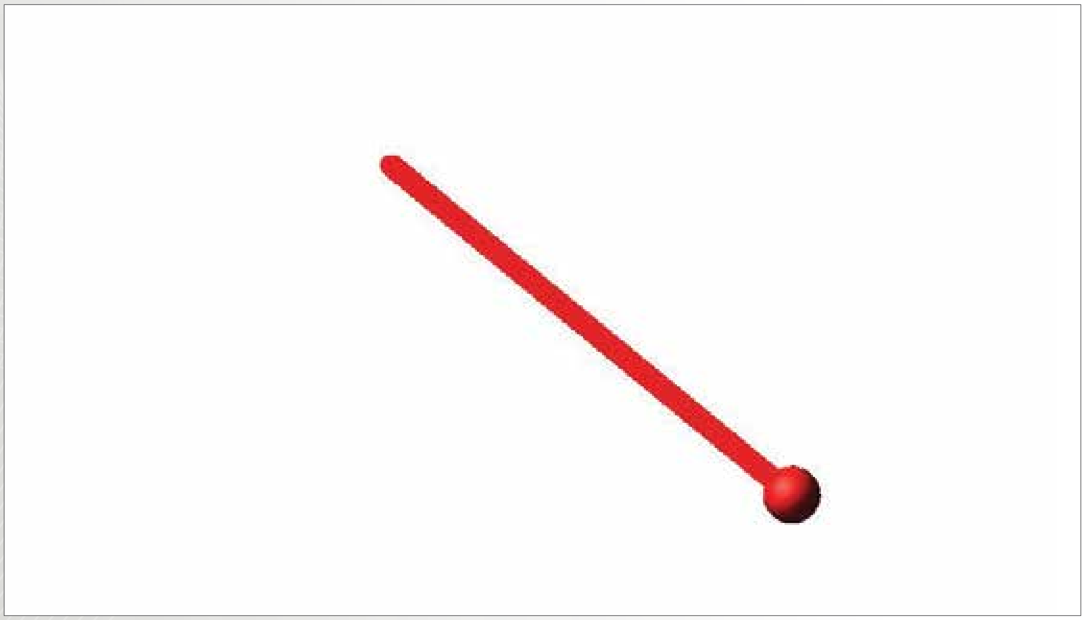


## Step 10. Simulate the Model.

a. Simulate the model for **5 seconds** and **500 steps** and observe the results.



# Example 5: One-degree-of-freedom Pendulum



## Software Version

Adams 2013.2

## Problem Description

Find the initial force supported by the pin at A for a bar that swings in a vertical plane, given the initial angular displacement and initial angular velocity. Also, find the pendulum frequency.

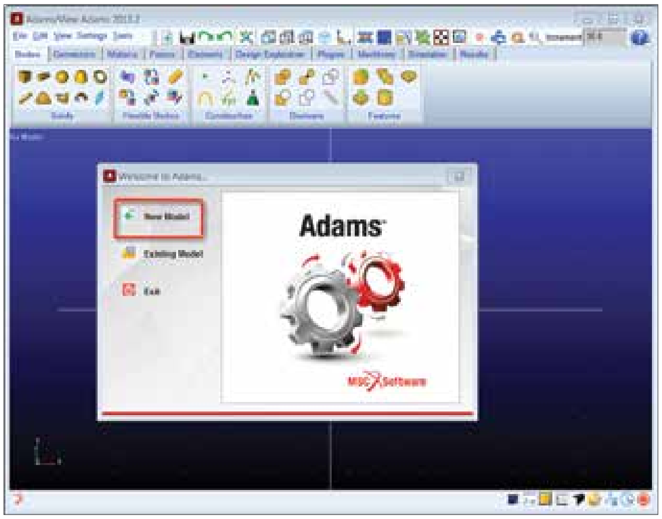
## Step 1. Create a new Adams Database

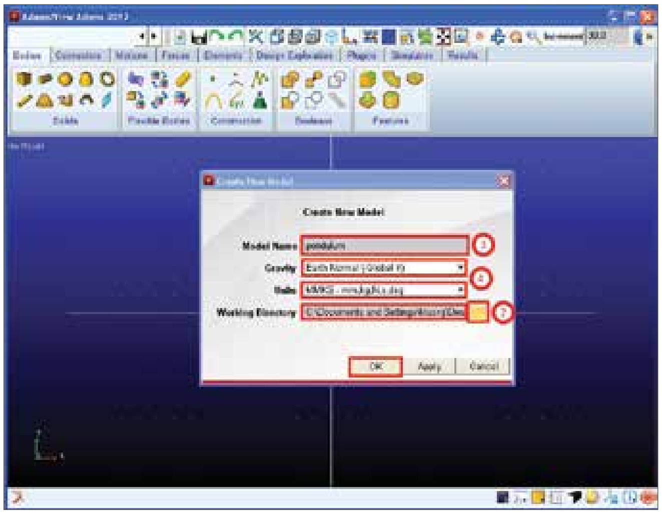
a. Click on Create a new model.

b. Under Start in, browse to the folder where you want to save your model.

c. Type the name of the new Model name as pendulum and click OK.

d. Make sure that the Gravity is set to Earth Normal (-Global Y) and the Units is set to MMKS - mm,kg,N,s,deg.



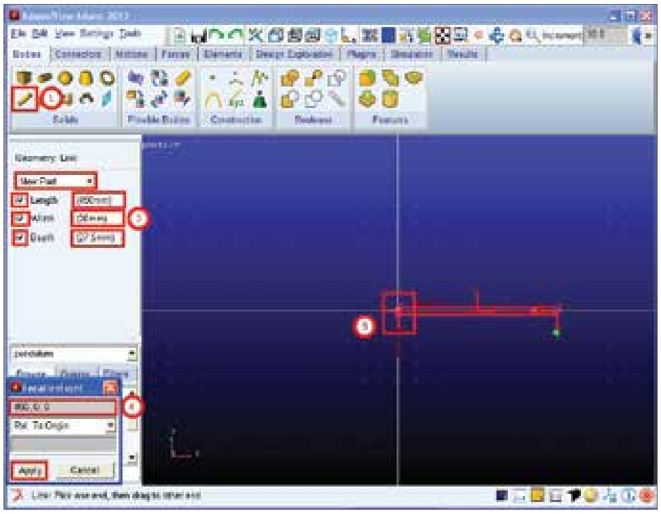
## Step 2. Construct the Pendulum Link.

a. From the Main Toolbox, right-click the Rigid Body tool stack and select the Link tool.

b. Then select New Part and under Length, enter 450 mm, under Width, enter 20 mm, and under Depth, enter 27.5 mm. Make sure the Length, Width, and Depth boxes are checked.

c. Click on the origin on the working grid to place the pendulum at 0,0,0.

d. Right-click anywhere on the working grid and a small window will appear in the bottom left corner of the window, this is called the Location Event window. In the Location Event window, enter 450,0,0 as the other endpoint and click Apply.



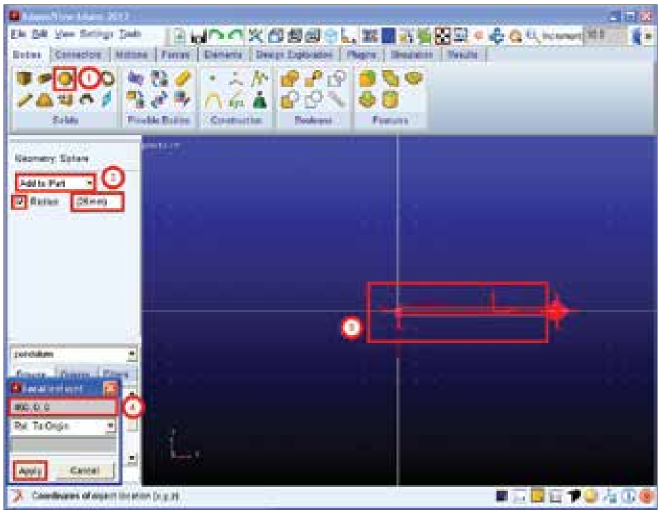
## Step 3. Construct the Bob of the Pendulum

a. From the Main Toolbox, right-click on the Rigid Body tool stack, and then select the Sphere tool.

b. Make sure Add To Part is selected and enter 25 mm for the Radius.

c. Then select PART_2, the link, as the part you are going to add the sphere to.

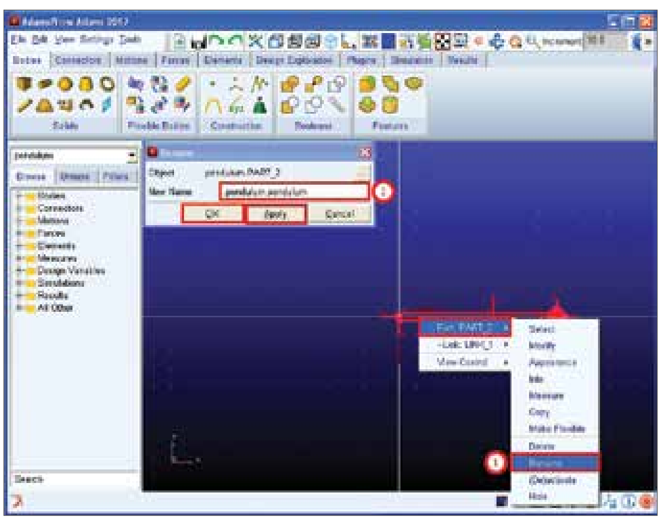
d. Then, in the Location Event window enter 450,0,0 as the center of the sphere and click Apply .



## Step 4. Rename the Pendulum.

a. Right-click on the link and point to Part:PART_2 and then select Rename.

b. In the New Name text box, enter .pendulum.pendulum, and then click OK.



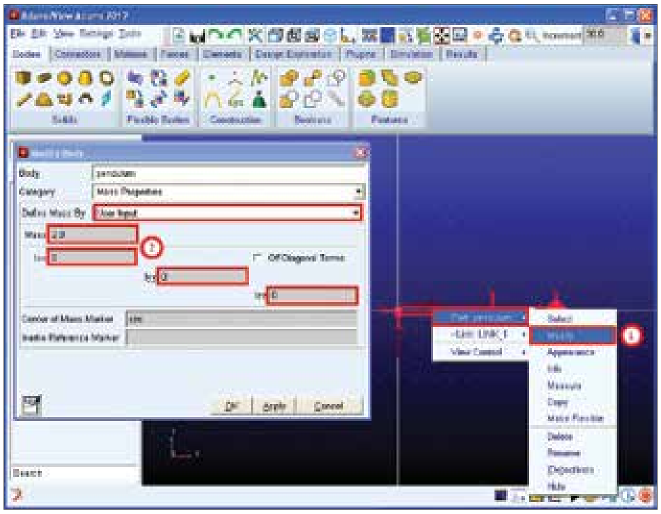
## Step 5. Assign Physical Properties to the Pendulum.

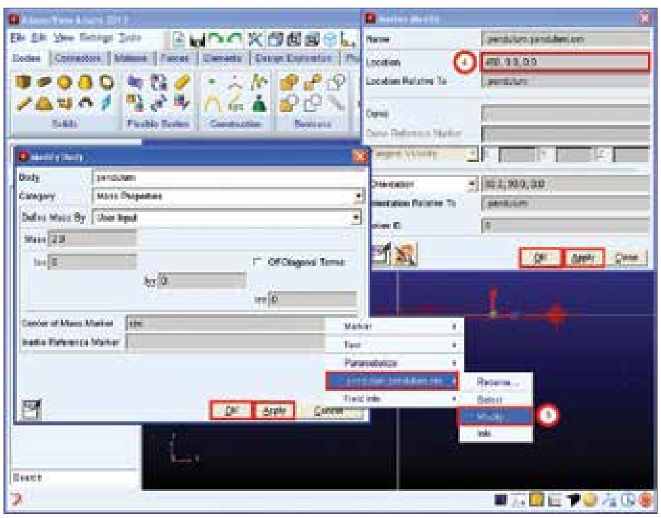
a. Right-click on the pendulum and go to Part: pendulum and then select Modify.

b. Set Define Mass by to User Input and in the Mass text box, enter 2.0. In the Inertia text boxes (Ix, Iyy, Izz), enter 0.

c. Then, right-click the Center of Mass Marker text box, and go to pendulum.pendulum.cm and then go to Modify.

d. In the Location box, enter 450,0,0, then click OK and OK. If you get a warning message about the change in position of your center of mass marker, simply ignore it and click Close.





## Step 6. Build the Pivot.

a. In the Main Toolbox, right-click the Joint tool stack, and then select the Revolute joint tool.

b. In the container, select 2 Bod-1 Loc and Normal to Grid.

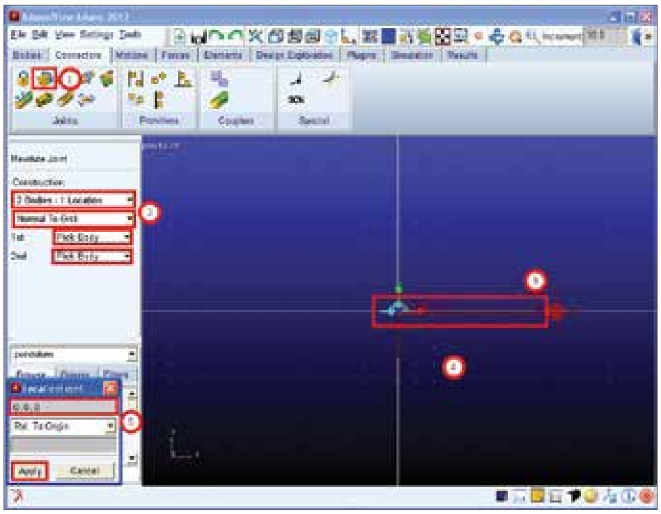
c. Select the pendulum as the first body.

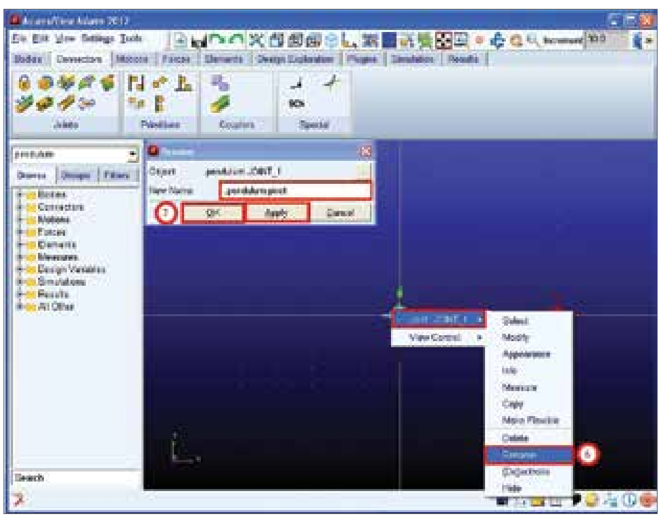
d. The ground as the second body.

e. Then select 0,0,0 as the location in the Location Event Window and click Apply.

f. Right-click on the joint and go to Joint:JOINT_1 and then select Rename.

g. In the New Name text box, enter .pendulum.pivot, and then click Apply and OK.



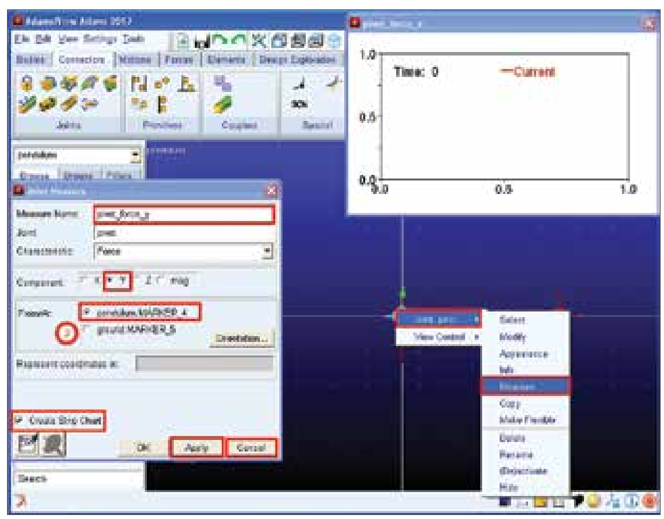


## Step 7. Create Measures for the Pendulum.

a. Right-click on the pivot joint, and go to Joint:pivot, and then select Measure.

b. In the box, where it says Measure Name, enter pivot_force_x. Set Characteristic to Force, and select X as the Component. Make sure .pendulum.MARKER_4 and Create Strip Chart are selected, and click Apply.

c. Again in the box, where it says Measure Name, enter pivot_force_y. Set Characteristic to Force, and select Y as the Component. Make sure .pendulum.MARKER_4 and Create Strip Chart are selected, and click Apply and Cancel.



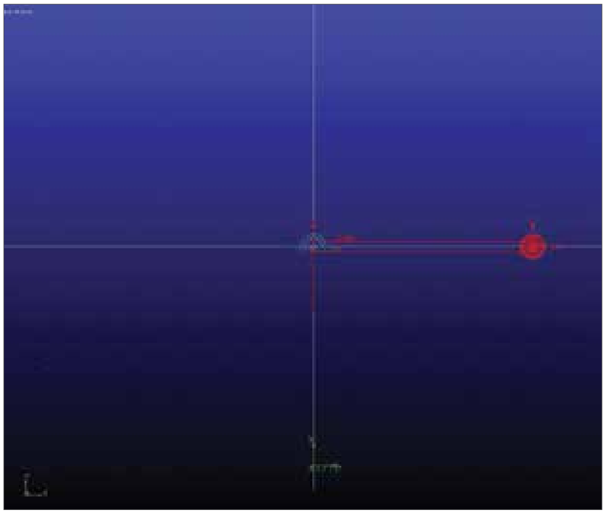
### Step 8. Create a Reference Marker.

a. In the Main Toolbox, right-click on the Rigid Body tool stack, and select the Marker tool.

b. Make sure Add to Ground and Global XY are selected. Right click in the window to invoke the Location Event and select 0,-450,0 as the marker location. The result would look like the first picture below. Notice the green marker beneath the pendulum.

c. With the marker selected, go to Edit and select Rename.

d. In the New Name text box, enter .pendulum.ground.angle_ref, and then click Apply and OK.



### Step 9. Create an Angle Measure.

a. From the Design Exploration menu, go to Measure and then go to Angle and click Advanced.

b. In the Measure Name text box, enter pend_angle.

c. Then, right-click the First Marker text box, and go to Marker, and then go to Pick.

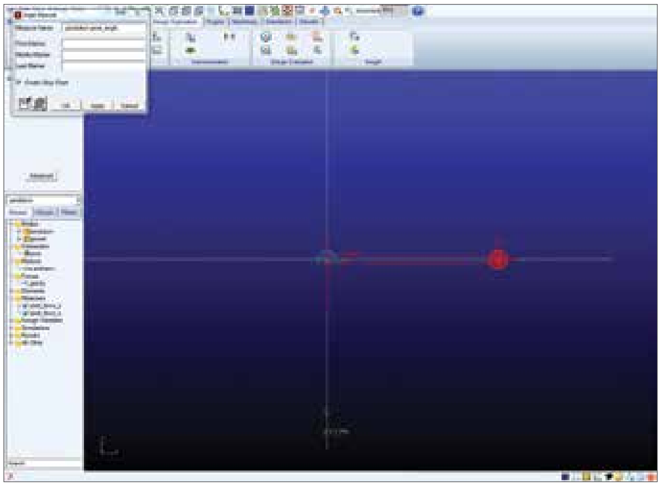
d. Go to the working grid, and pick a marker that is on the pendulum, which is also located at its end, for example, select the cm marker. If multiple markers are coincident, right click at the location and a selection box will be invoked where you can choose among them.

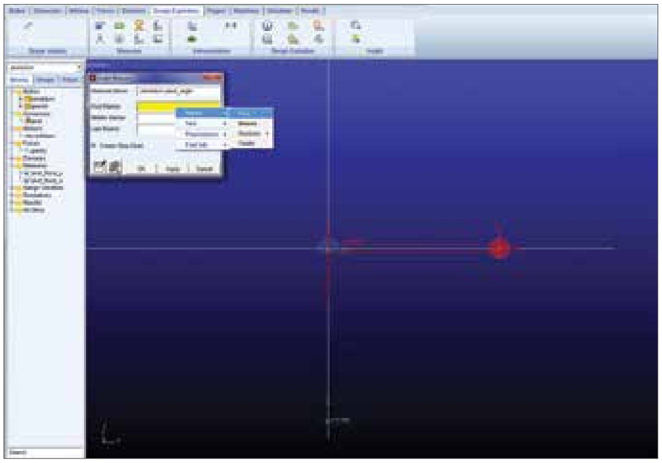
e. Right-click the Middle Marker text box, go to Marker, and then go to Pick.

f. Then, pick a marker that is at the location of the pivot. (Marker_1).

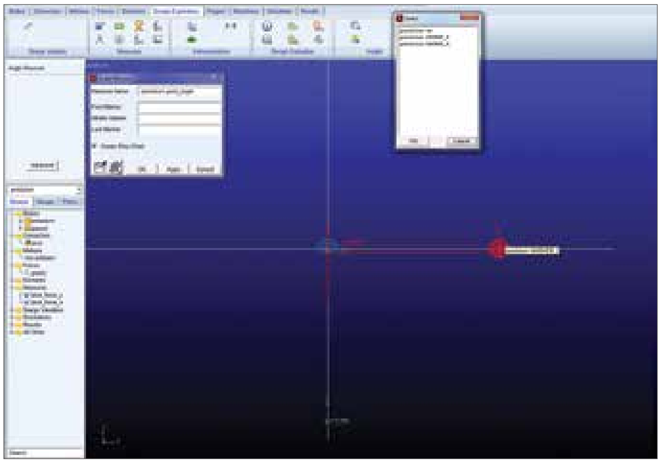
g. Right-click the Last Marker text box, go to Marker, and then go to Pick.

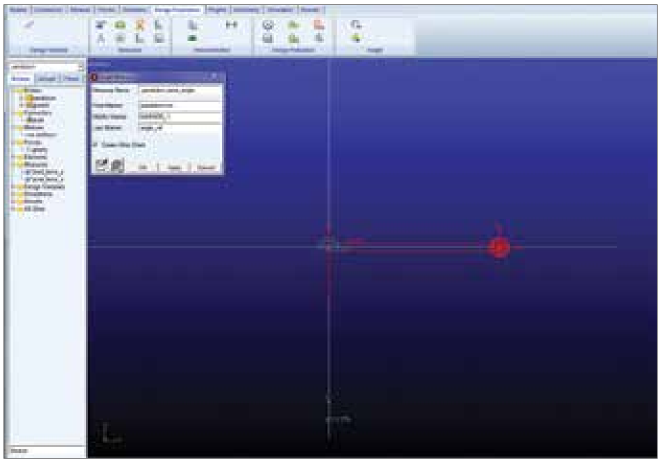
h. Pick the marker that is on the ground and at the end of





the pendulum, the marker that you just created in the previous step, .pendulum.ground.angle_ref. Then click Apply and Cancel.

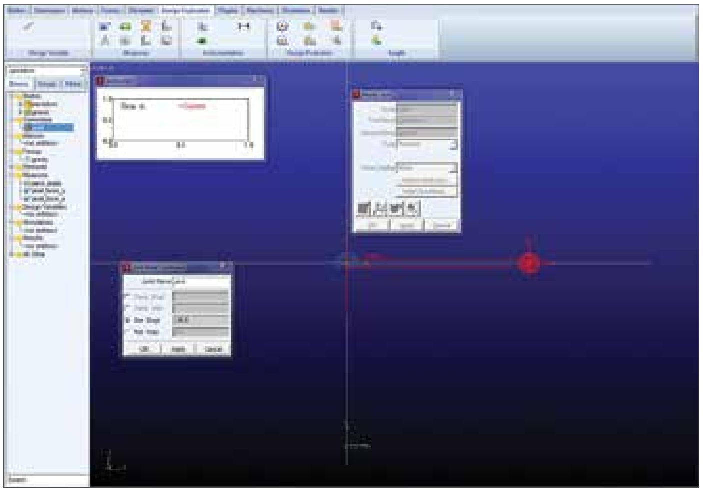




## Step 10. Add the Initial Conditions to the Pendulum Model.

a. Right-click on the pivot joint, and go to Joint:pivot, and then go to Modify.

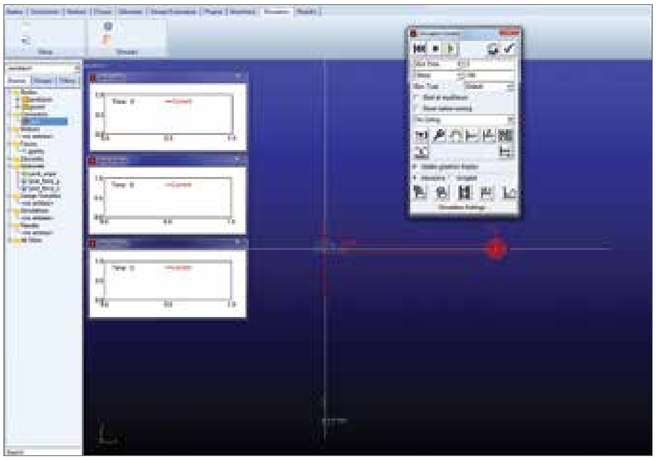
b. Go to Initial Conditions and in the Joint Initial Conditions dialog box, select Rot. Displ and enter -85 in the text box. Then click OK and OK. This will make the pendulum to oscillate with a small displacement of 5 degrees.



## Step 11. Simulate the Model.

a. Verify the model.Refer to Example 1, Step 6 if necessary.

b. Simulate your model for 2 seconds with 100 steps using the Simulation tool.



## Step 12. Using ADAMS/PostProcessor, Determine the Global Components and the Frequency of the Pendulum.

a. Right-click the blank area inside the pend_angle graph, and go to Plot: scht1 and then go to Transfer to Full Plot.

b. You should now be in the Adams/PostProcessor. Now, select the Plot Tracking tool. 

c. To determine the Global Components, move the cursor over the plot to where t=0 and make note of the value of Y.

d. In the dashboard, go to Clear Plot.

e. Set the source to **Measures**, and from the Measure list, select **pivot_force_x** and select **Surf**.

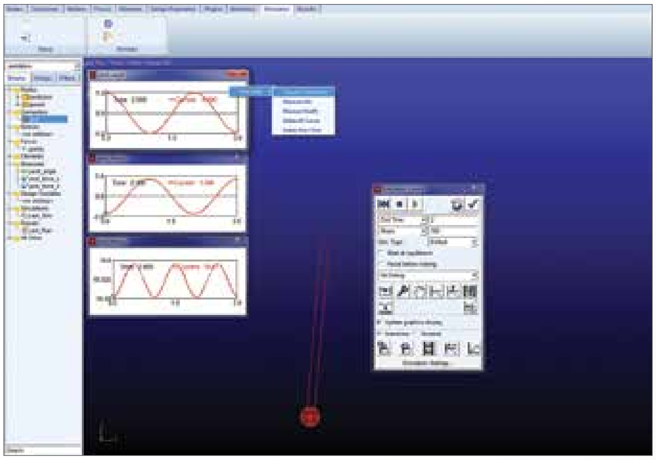
f. Move the cursor over the plot where t=0, and make note of the value of Y.

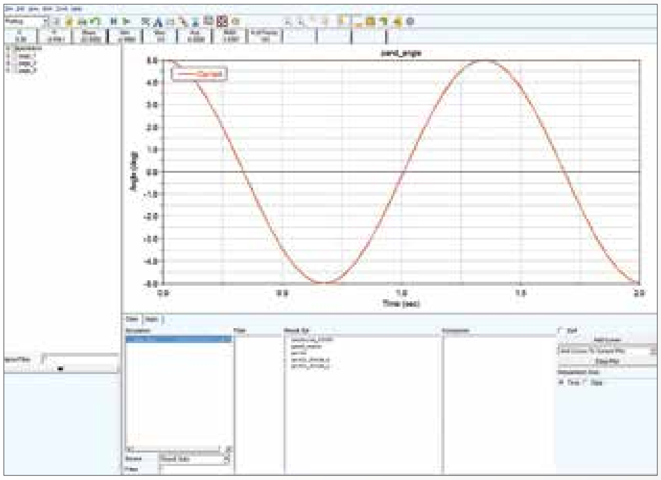
g. From the **Measure** list, select **pivot_force_y**.

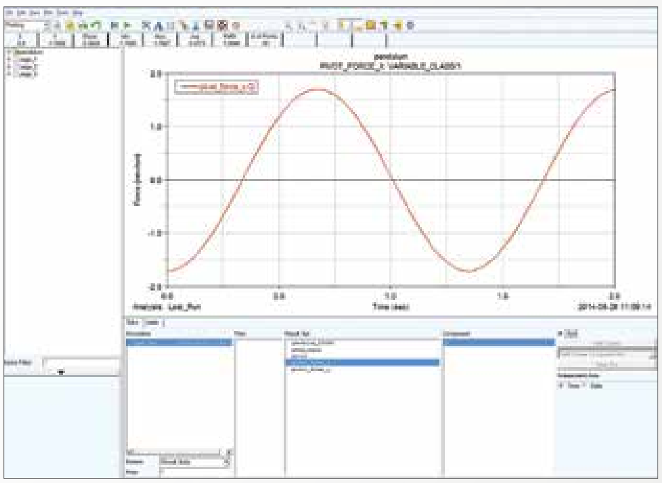
h. Move the cursor over the plot where t=0, and make note of the value of Y.

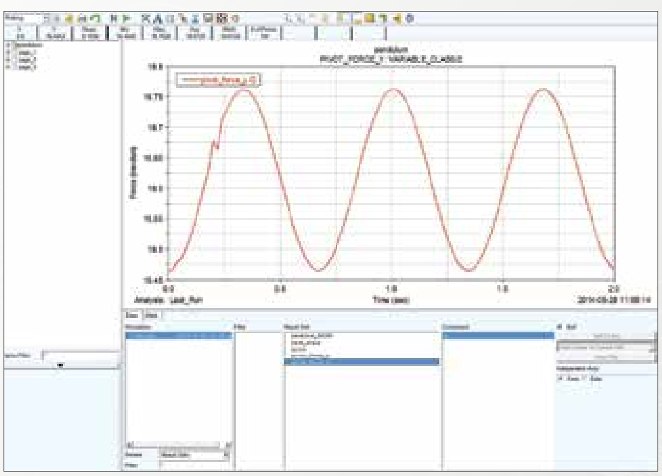
i. To determine the frequency, from the **Measure** list, select **pend_angle**.

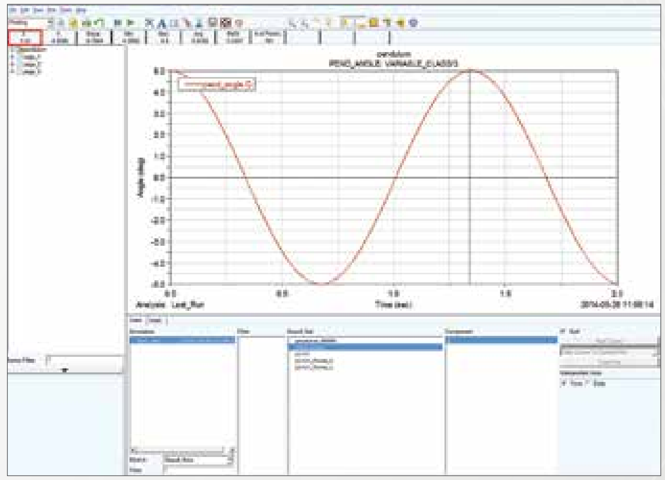
j. Estimate the period of the curve, then find the reciprocal of the period to determine the frequency.







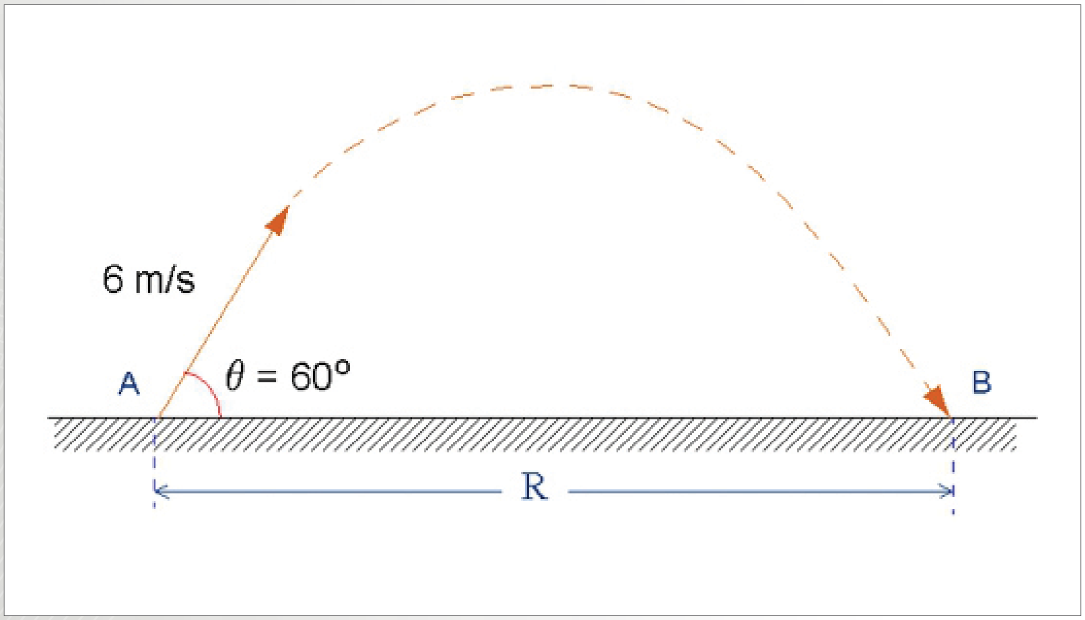




## Analytical Solution – Verify the Results by Calculating the Analytical Solution.

- The period of a simple pendulum is T= 2π(L/g)^0.5. Plug in L=0.45m and g=9.8m/$s^2$, we get T=1.346s which matches the result of Adams simulation (Check the figure above).
- The verification of the forces is left to the student as a practice.

# Example 6: Projectile Motion




## Software Version

Adams 2013.2

## Problem Description

A stone is projected from the ground with initial velocity of 6m/s and 60 degree above the ground.

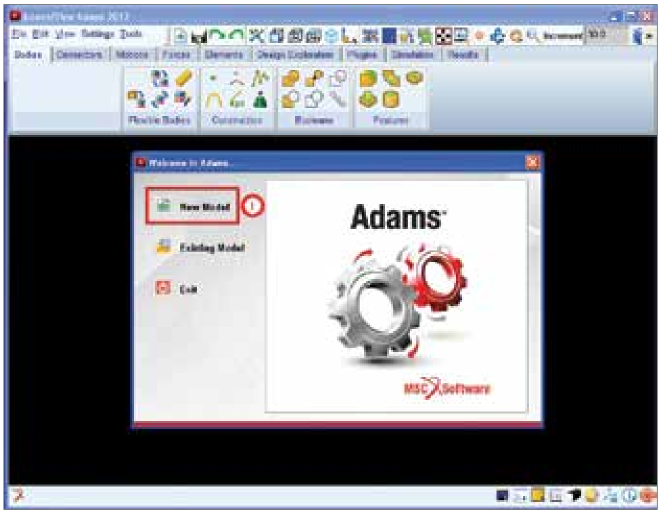
## Step 1. Create a new Adams Database

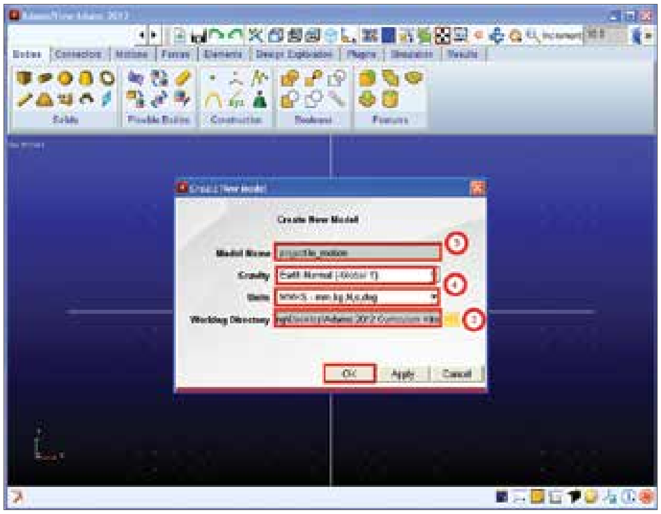
a. Click on Create a new model.

b. Under Working Directory, browse to the folder where you want to save your model.

c. Type the name of the new **Model name** as **projectile_motion** and click **OK**.

d. Make sure that the **Gravity** is set to **Earth Normal (-Global Y)** and the Units is set to **MMKS - mm,kg,N,s,deg**.



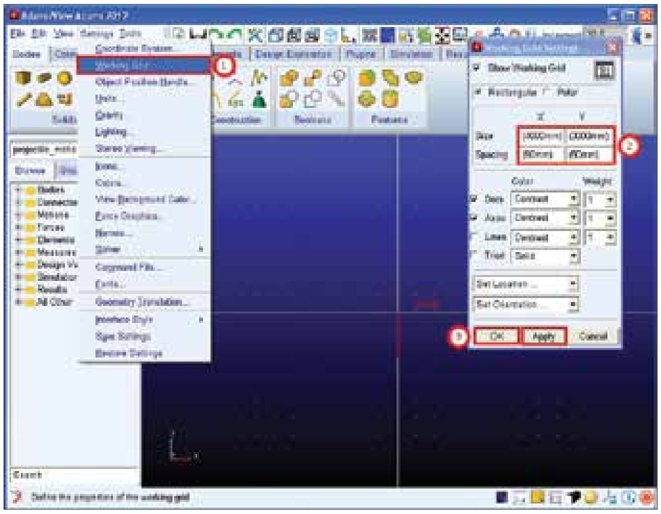



## Step 2. Adjust the Working Grid.

a. From the **Settings** menu, select Working Grid.

b. Set the Size in the X direction to **4000 mm** and the Size in the Y direction to **3000 mm** and the **Spacing** in the x and y direction to **50 mm**.

c. Make sure that the working grid is oriented along the **global XY direction** (default setting when you open Adams/View). The Set Orientation pull-down menu allows you to choose Global XY, YZ, XZ, or custom orientation. Click **Apply** and **OK**.



## Step 3. Constructing the Geometries of the Plane and the Stone.

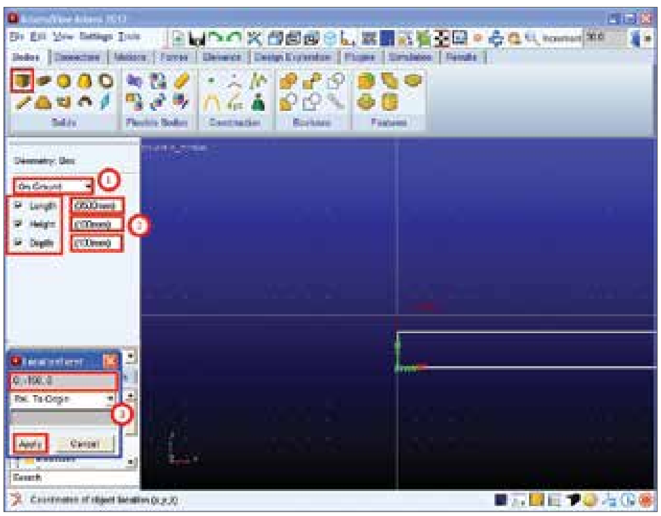
a. To create the plane, right-click on the **Rigid Body icon** and select **Rigid Body: Box**.

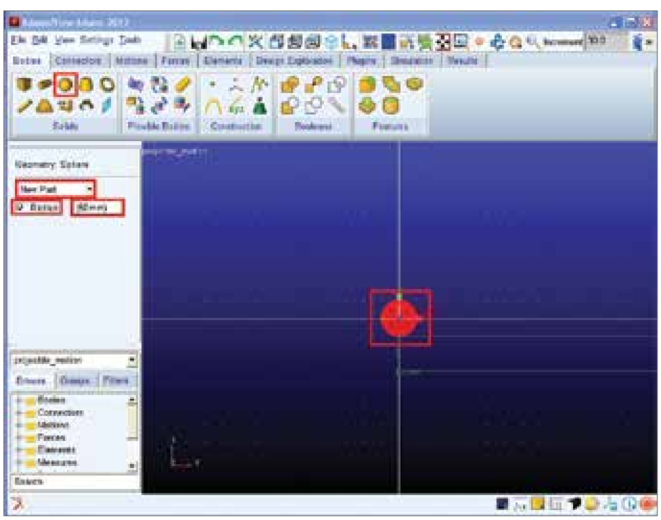
b. Make sure **On Ground** is selected and enter **(3500 mm)** for the **Length, (100 mm)** for the **Height**, and **(100 mm)** for the **Depth**. Also make sure that the **Length, Width, and Depth** are all checked.

c. Then, right-click on the working grid and then enter in the coordinates for the corner of the plane: **0,-150,0** and then click **Apply**.

d. To create the spherical stone, right-click on the **Rigid Body** icon and select **Rigid Body: Sphere**.

e. Make sure **New Part** is selected and enter **(50 mm)** for the **Radius**. Also make sure that **Radius** is checked. Then on the working grid select the origin **(0,0,0)** as the center of the sphere.





## Step 4. Rename the Stone and Plane Geometry and Assign Physical Properties to the Objects.

a. Right-click on the box (plane), point to **Block: BOX_1**, and then select **Rename**.

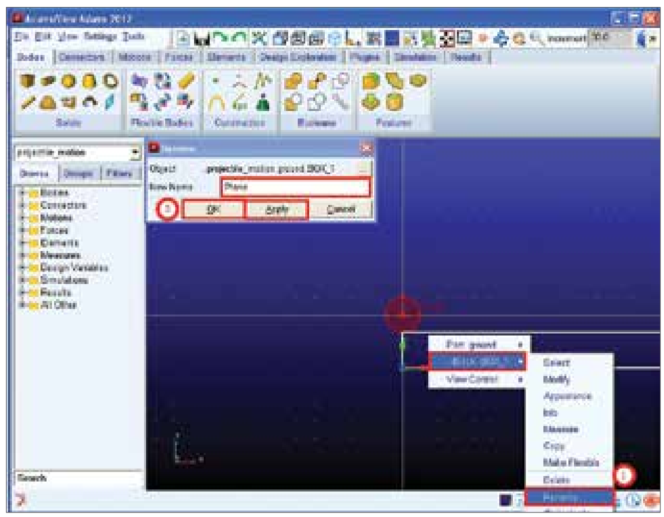
b. Enter Plane, under **New Name**, and click **Apply** and **OK**.

c. Right-click on the sphere (stone), point to **Part:PART_2**, and then select **Rename**.

d. Enter **Stone**, under **New Name**, and click **Apply** and **OK**.

e. Enter the mass of the stone by right-clicking on sphere and going to **Part:Stone**, and then selecting **Modify**.

f. Set **Define Mass** By to **User Input** and in the **Mass** text box, enter **1.0 kg**. Click **Apply** and **OK**.

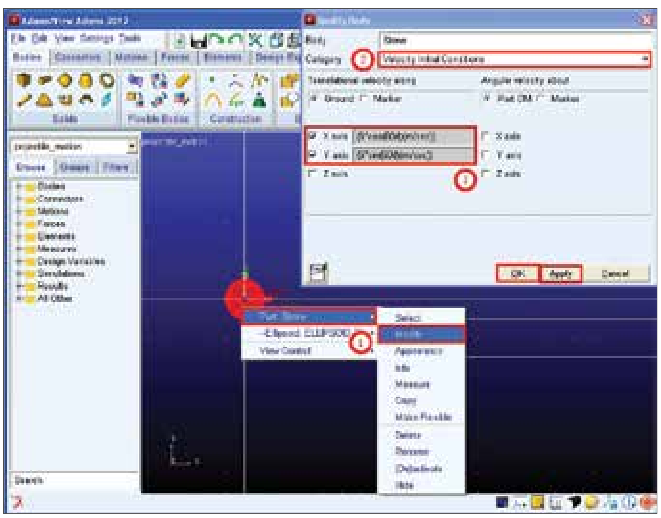


## Step 5. Set Initial Conditions.

a. Right-click on the stone and go to **Part:Stone** and then select **Modify**.

b. Under Category select Velocity Initial Conditions.

c. Check **X Axis** and then enter **(6*cos(60d)(m/sec))**, and then check **Y Axis** and enter **(6*sin(60d)(m/sec)).** Click **Apply** and **OK**.

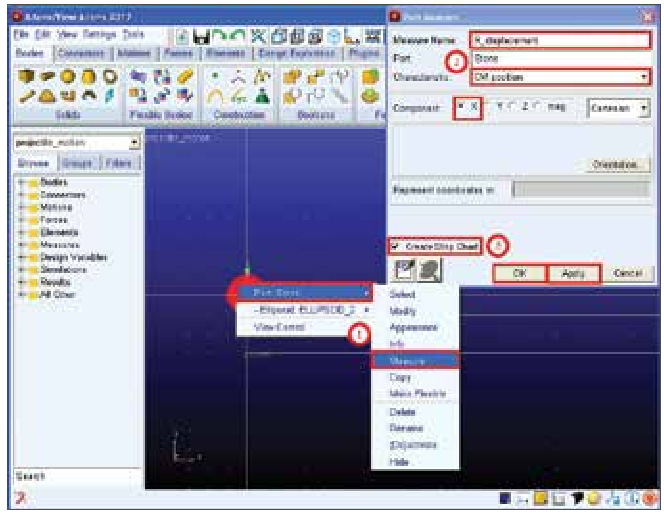


## Step 6. Create Measures for the Projectile Motion.

a. Right-click on the stone and select **Part:Stone** and then select **Measure**.

b. In the **Measure Name** text box, enter **R_displacement**. Set Characteristic to **CM Position** and **Component to X.**

c. Make sure **Create Strip Chart** is checked and select **OK**.

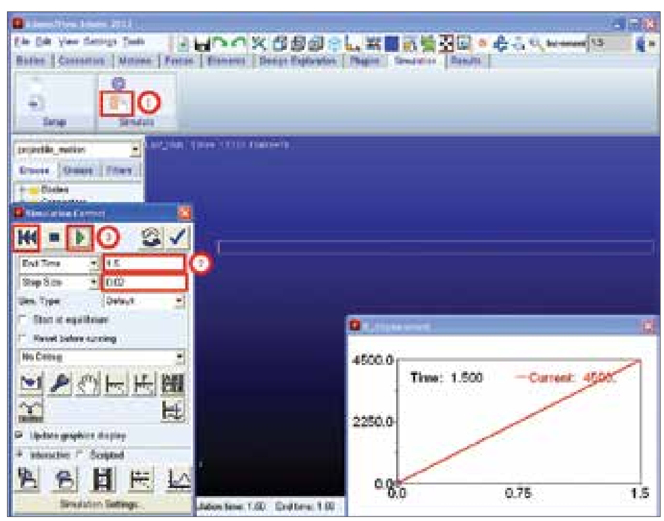


## Step 7. Simulate the Model.

a. From the Main Toolbox, select the **Simulation** tool.

b. In the End Time text box, enter **1.5** and in the **Step Size** text box enter **0.02.** Then click on the **Play** button.

c. After the end of the simulation, click on the **Reset** tool.



### Step 8. Use Animation Tools to Determine the Time at which the Stone Makes Contact with the Plane.

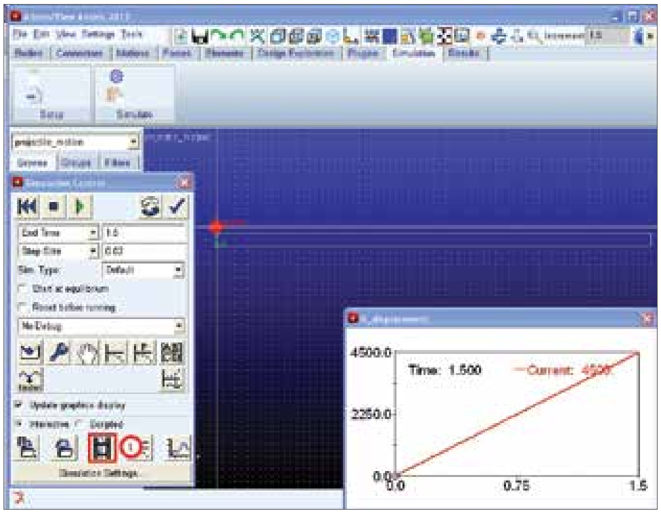
a. From the Main Toolbox, select the **Animation** tool.

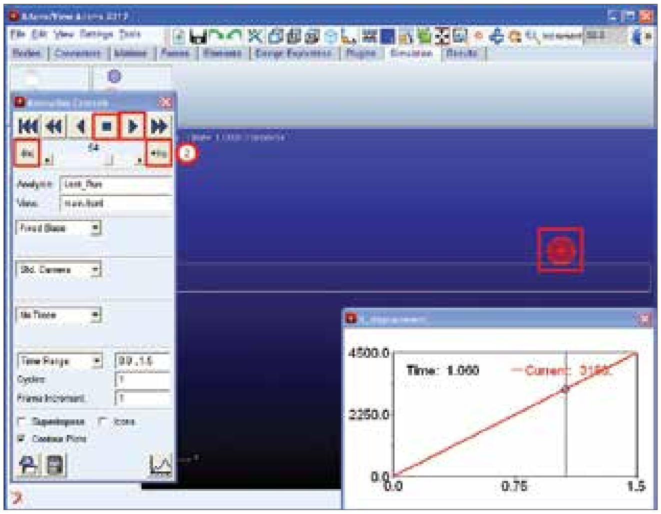
b. Select the **Play** tool and click on **Stop** when the stone makes contact with the plane. Use the Step Forward and Step Backward tools, if needed, to facilitate this step. Make note of the time at which the stone makes contact with the plane on the graph.

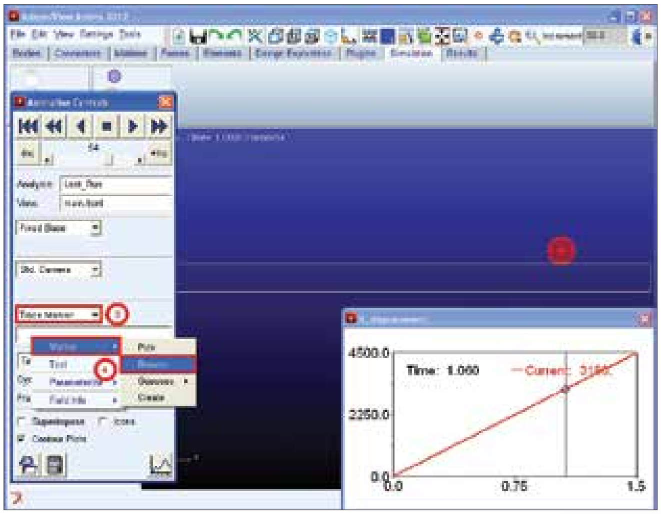
c. Click on the ellipses above the Icons button and then change **No Trace** to **Trace Marker**.

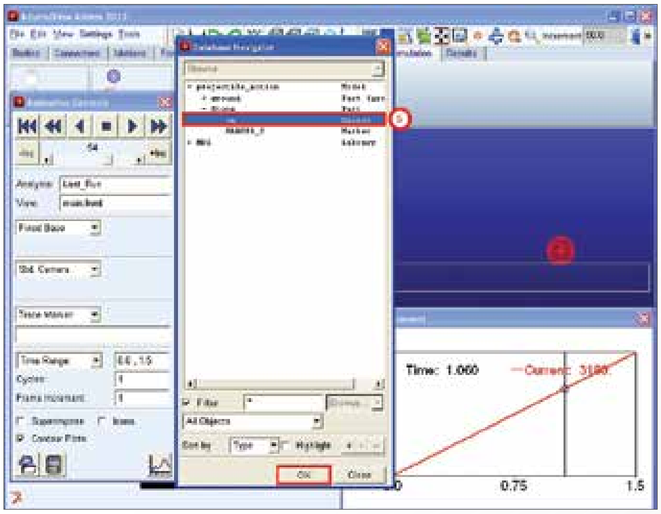
d. In the box, below **Trace Marker**, right-click and go to **Marker** and select **Browse**.

e. In the Database Navigator, under Stone, select cm and then click OK.







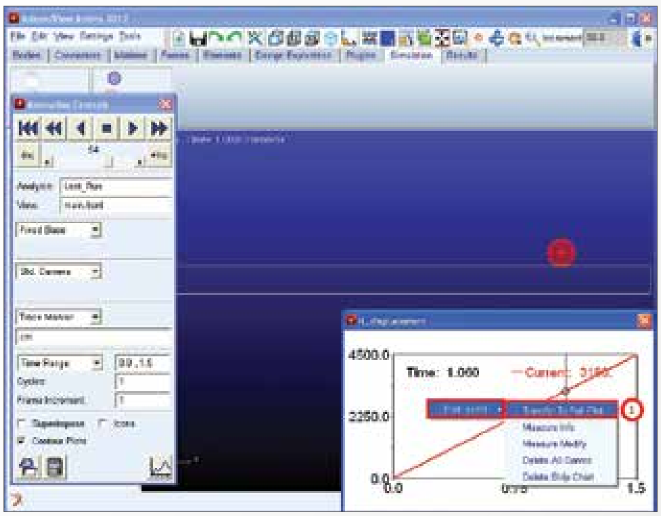


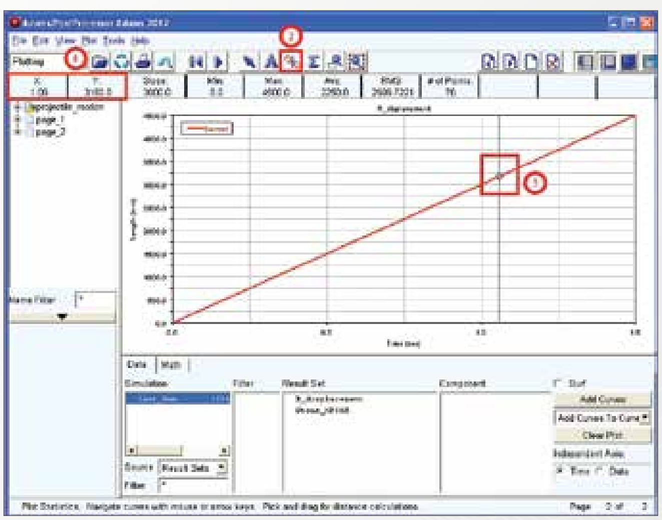
## Step 9. Using ADAMS/Post Processor, determine the range, R.

a. To find the **Stone's Displacement** after 1 second, first right-click the blank area inside the stripchart, then choose **Plot:scht1** then click on Transfer To Full Plot.

b. In Adams/Postprocessor, from the main toolbar, select the **Plot Tracking tool.**

c. Because you want to know the final conditions after 1 second, move the cursor over the end point of the plot. In the area below the menu bar, the value of X is displayed as 1. Note the value of Y is 3000.



## Analytical Solution – Verify the Results by Calculating the Analytical Solution.

$$x_o = 0 \quad x_f = R$$

$$y_o = 0 \quad y_f = 0$$

$$V_{x_o} = 6000 \times \cos 60° = 3000 \text{ mm/sec}$$

$$V_{y_o} = 6000 \times \sin 60° = 5196 \text{ mm/sec}$$

$$y_f = y_o + V_{y_o} t - \frac{1}{2} g t^2$$

$$0 = 0 + 5196t - 0.5 \times 9806 \times t^2$$

$$0 = (5196 - 4905t)t$$

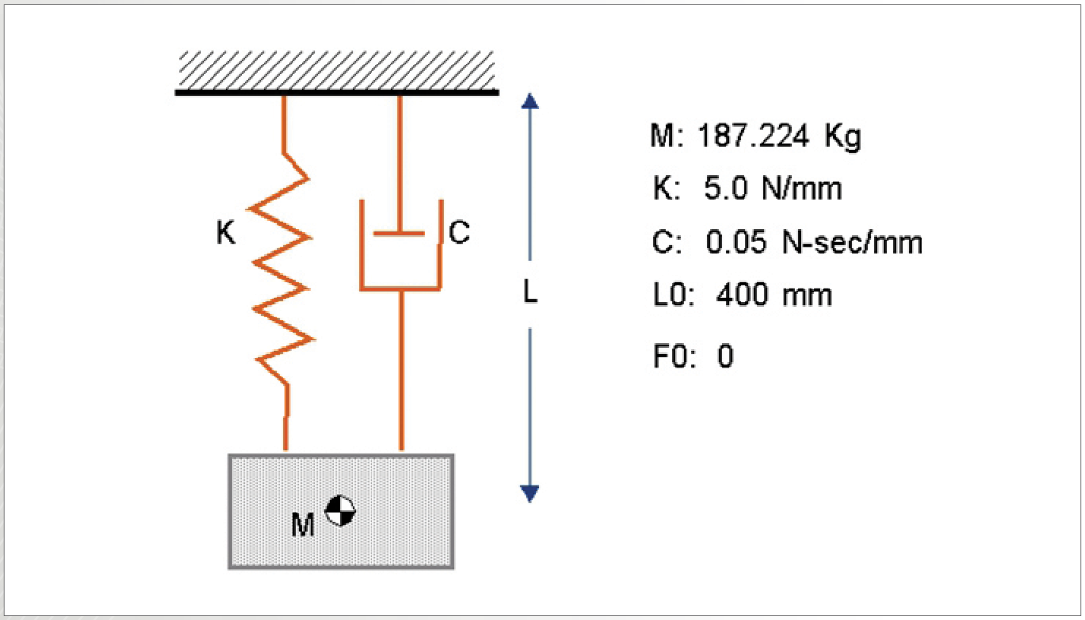
$$t = 1.06 \text{ sec}$$

$$x_f = x_o + V_{x_o} t$$

$$R = 0 + 3000 \times 1.06$$

$$R = 3180 \text{ mm}$$

# Example 7: Spring Damper - Part 1




## Software Version

Adams 2013.2

## Problem Description

Find the force in spring damper at static equilibrium.

## Step 1. Create a new Adams Database

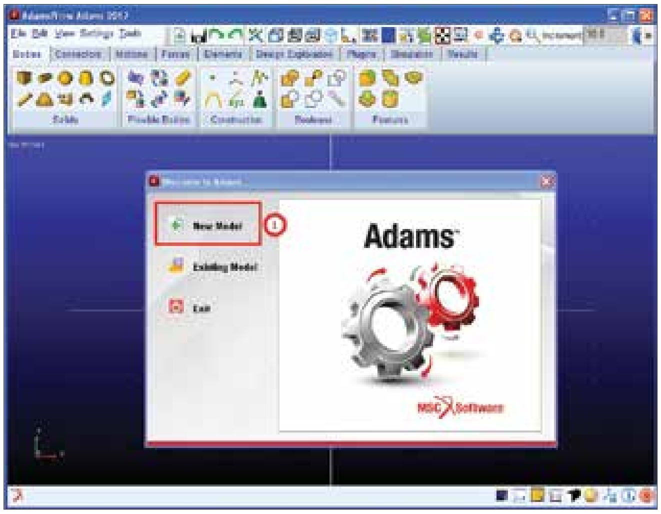
a. Click on **New model**.

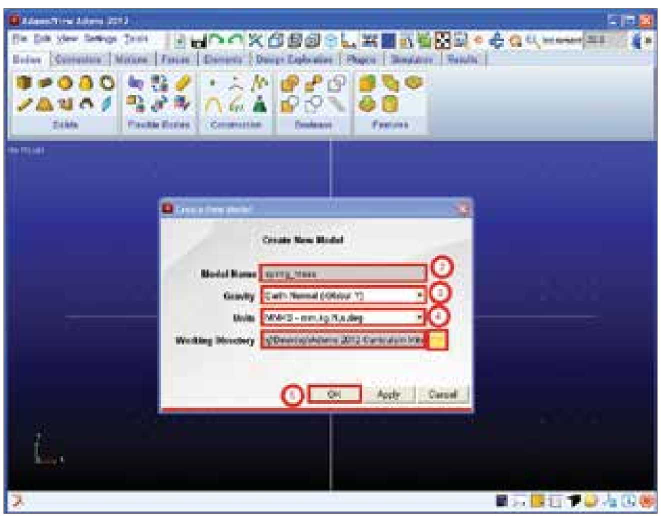
b. For the **Model name** change it to spring_mass.

c. For the **Gravity** choose **Earth Normal (-Global Y)**.

d. For the **Units**, set it to **MMKS - mm,kg,N,s,deg**.

e. Then click **OK**.





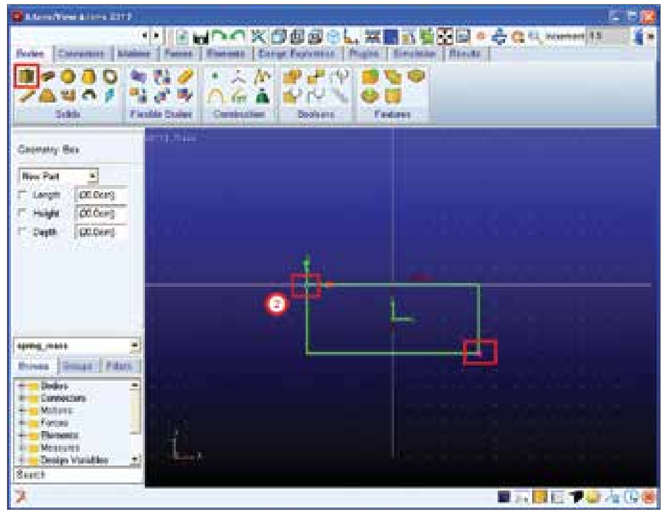
## Step 2. Build the Rigid Body.

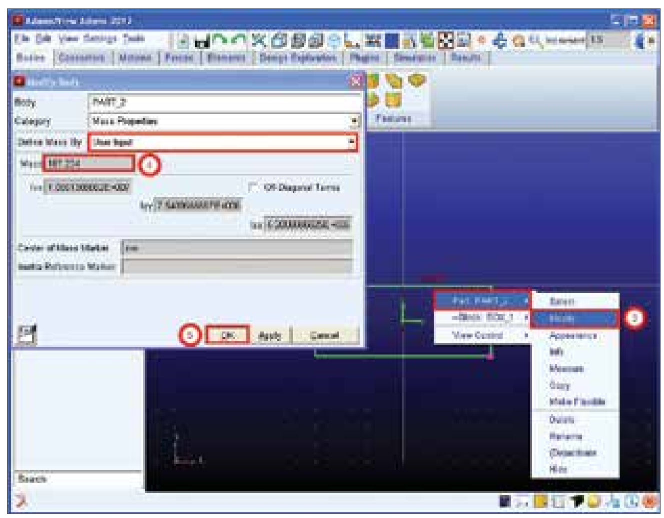
a. From the Main Toolbox, right-click the **Rigid Body** tool stack, and then select the **Rigid Body: Box** tool.

b. Create a **Rigid Body:Box** by clicking on the grid. The dimension of the box is not important, so just create any type of box.

c. Right-Click on the Box and choose **Part:PART_2 : Modify**. Input the **Mass** as **187.224**.

d. After inputing the **Mass**, click **OK**.





## Step 3. Constrain the Block to Move Only in the yg Direction.

a. First right-click on the screen and choose **Rotate XY** then rotate the model until it is similar to the view below. It is best to check the translational joint that will be created by rotating the model to make sure that it is fix in the yg direction.

b. Now click on **Joint: Translational**.

c. Choose the **Rigid Body : Box**, when it says **"Select the first body"** on the bottom of the screen.

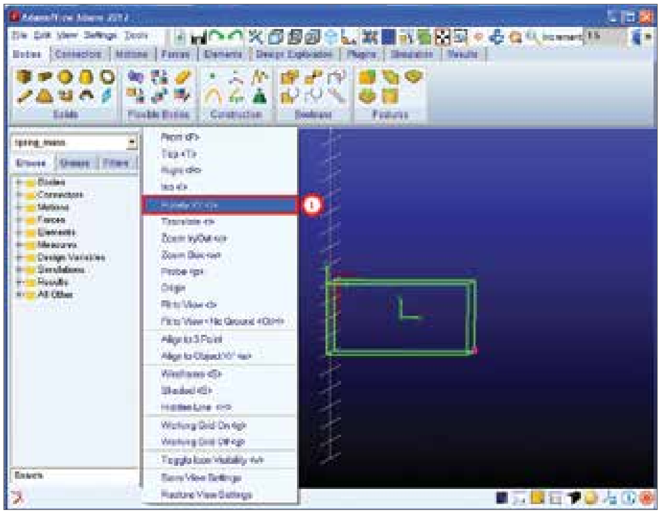
d. Choose the **Ground**, when it says **"Select the second body"** on the bottom of the screen.

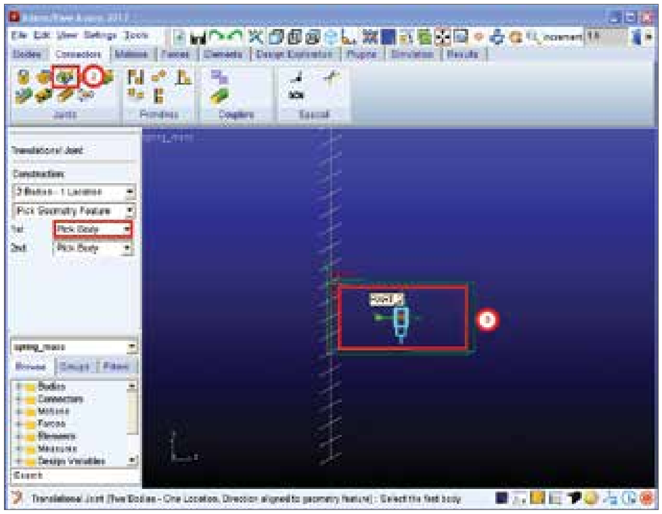
e. Choose the **PART_2.cm**, when it says **"Select the location"** on the bottom of the screen.

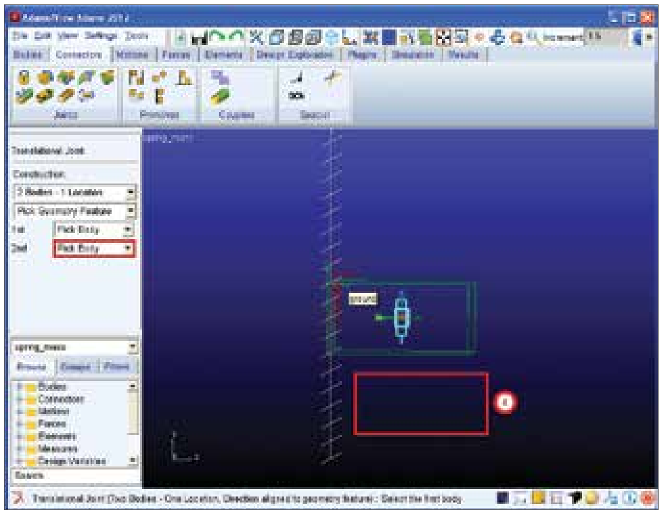
f. Choose the **cm.X,** when it says **"Select the direction vector"** on the bottom of the screen.

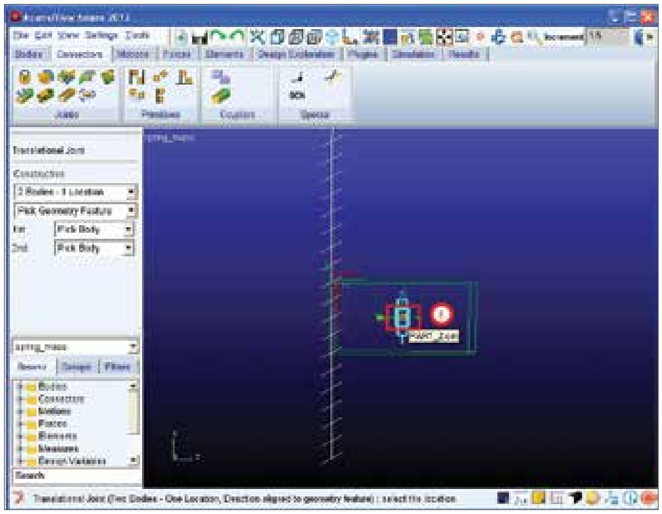
g. To verify the expected behavior, simulate the model by clicking on the Interactive Simulation Controls.

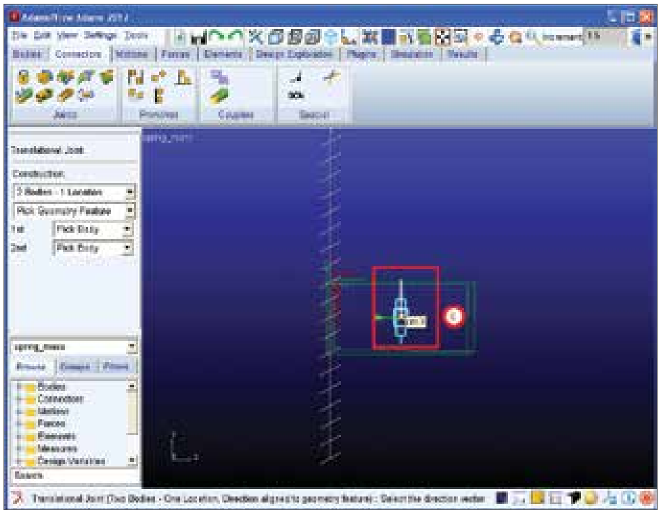
h. Click on the **Play** icon to run the simulation and click on the **Reset** icon.

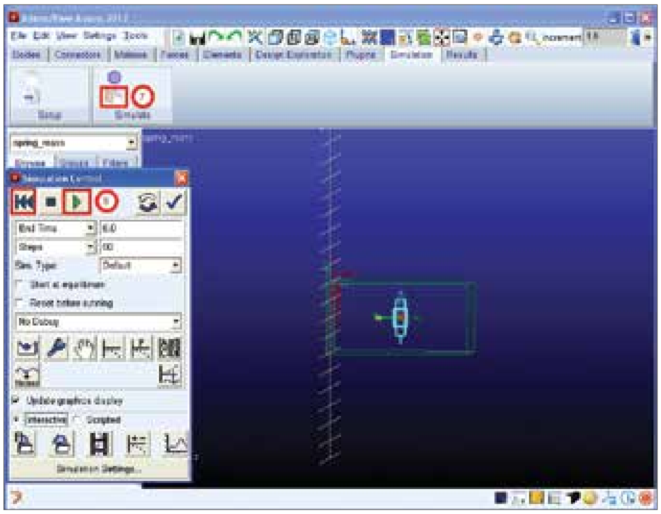












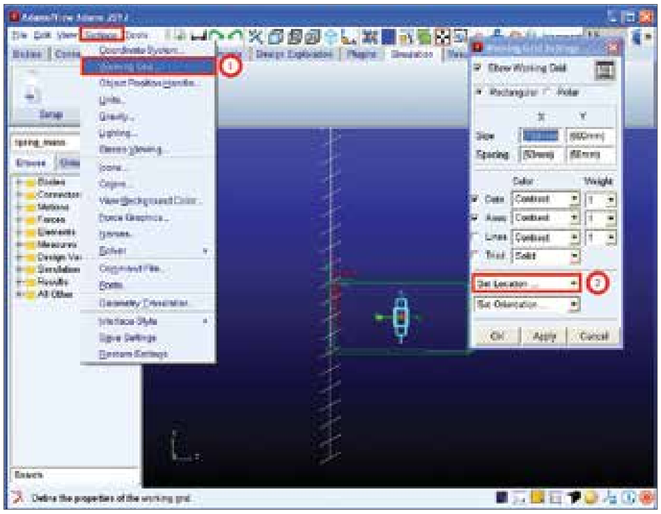
## Step 4. Move the Working Grid.

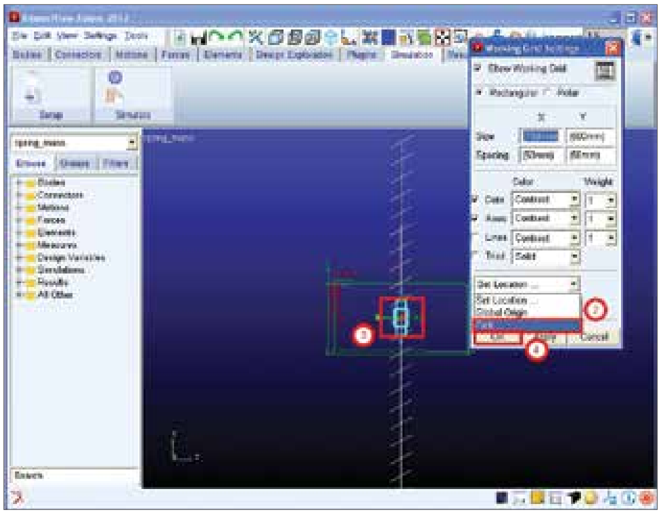
a. To ensure that the spring damper is aligned with the Yg direction, move the working grid to the cm of the Box. First click on **Settings: Working Grid**....

b. Change **Set Location** to **Pick**.

c. Pick on the cm of the Box.

d. Click **OK**. Now the working grid is in the center of the box.





## Step 5. Add the Pre-Defined Spring Damper

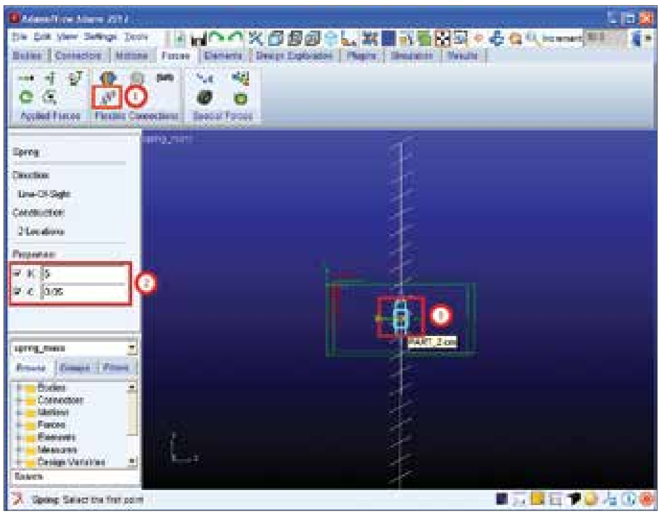
a. Click on the **Translational spring damper**.

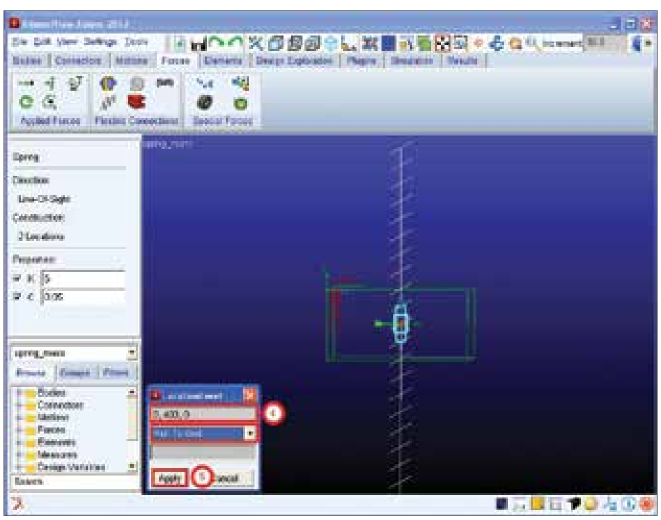
b. Input the **K** value of **5** and the **C** value of **0.05**.

c. Choose the **PART_2.cm**, when it says **"Select the first point"** on the bottom of the screen.

d. Right-click anywhere on the ground to display the **Location Event.** Enter **0, 400, 0**, and change **Rel. to Origin to Rel. to Grid**.

e. Click **Apply**.





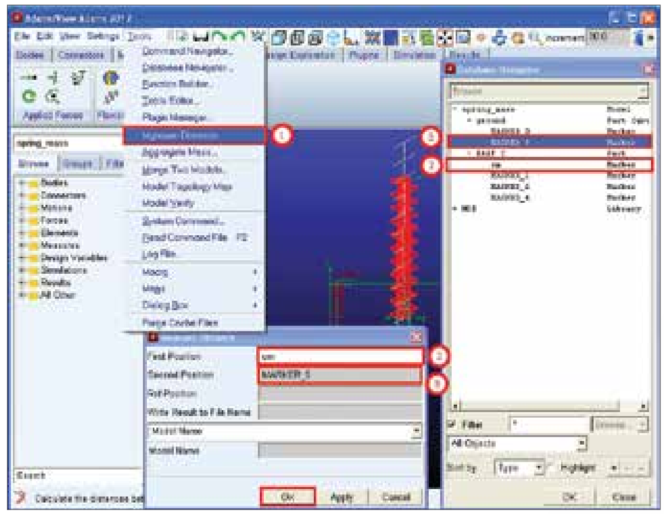
## Step 6. Verify the Distance of the Spring Damper.

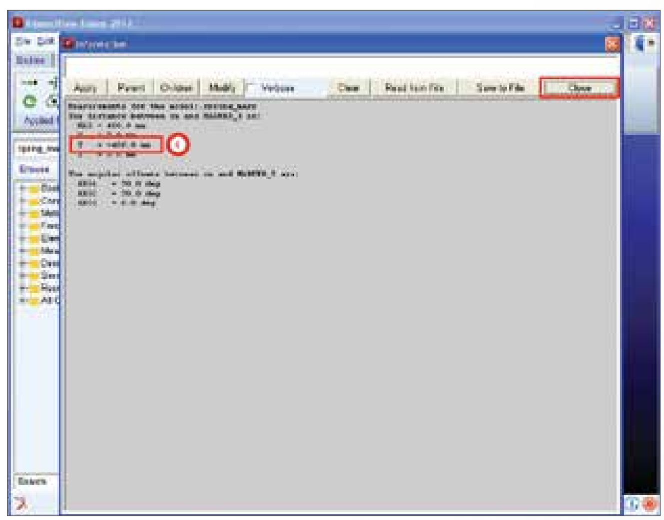
a. Click on **Tools** and choose **Measure Distance**....

b. Click on **First Position** and choose **cm**, because this is the position where one of the spring end is located.

c. Click on **Second Position** and choose **MARKER_5**, because this is the position where the other spring end is located. Then click **OK**.

d. Verify the value of Y.





## Step 7. Finding the Force in Spring Damper at Static Equilibrium

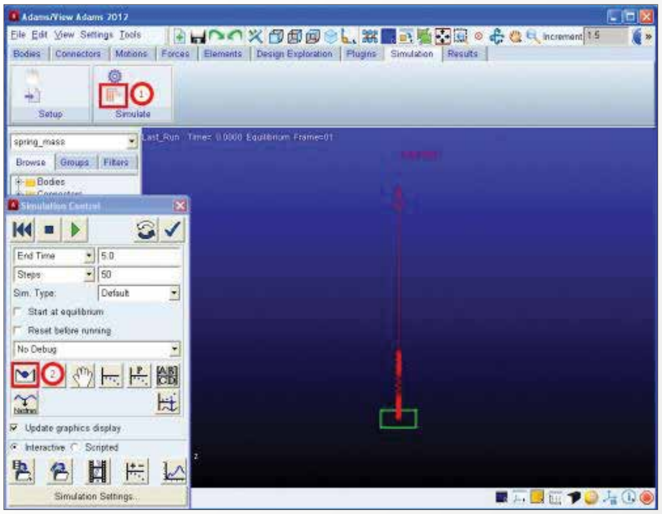
a. Select **Interactive Simulation Controls** on the Main Toolbox.

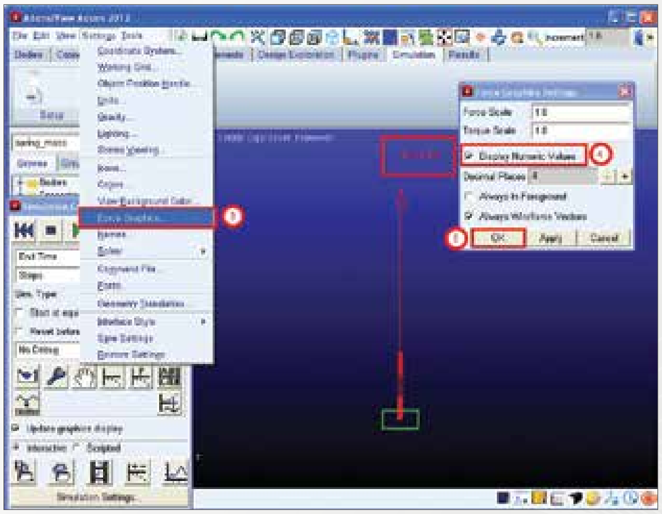
b. Select the **Static Equilibrium** tool.

c. Select **Force Graphics**... under **Settings** on the Main Menu.

d. Put a check on **Display Numeric Values** on the **Force Graphics Settings**.

e. Click **OK**. Zoom out until you can see the force value. As shown the force in the spring damper at static equilibrium is 1836.04 N.

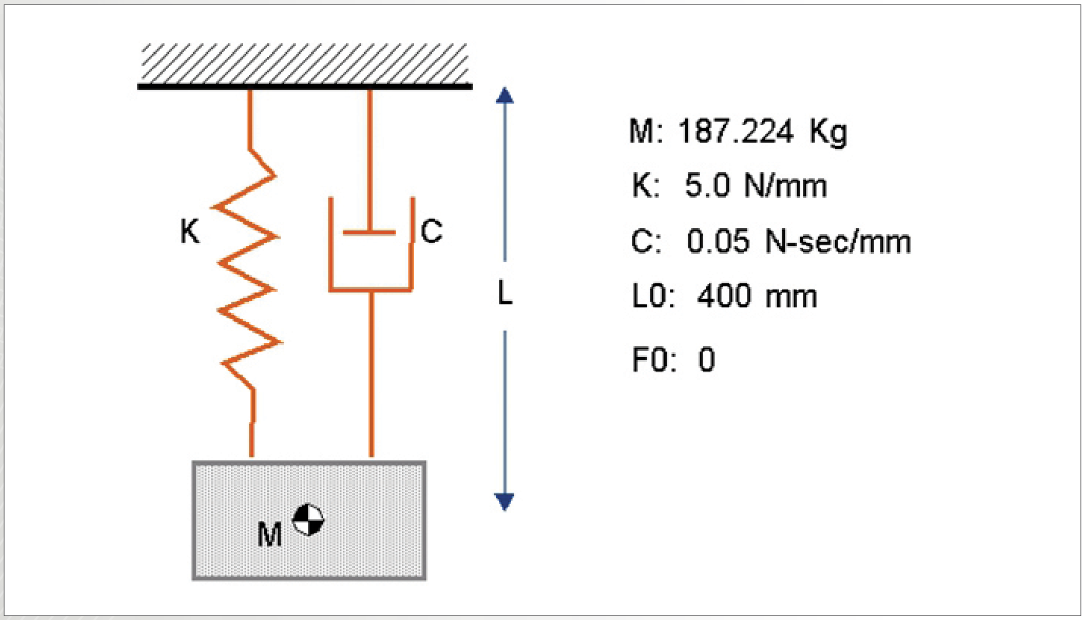




## Analytical Solution – Verify the Results by Calculating the Analytical Solution.

- The block's mass is 187.224 kg.
- Therefore, to balance the force of gravity, the spring damper must generate:
- 187.224 kg * 9806.65 mm/s2 = 1836.04 N
- The results produced by Adams View are the same as the hand calculated answer.

# Example 8: Spring Damper - Part 2




## Software Version

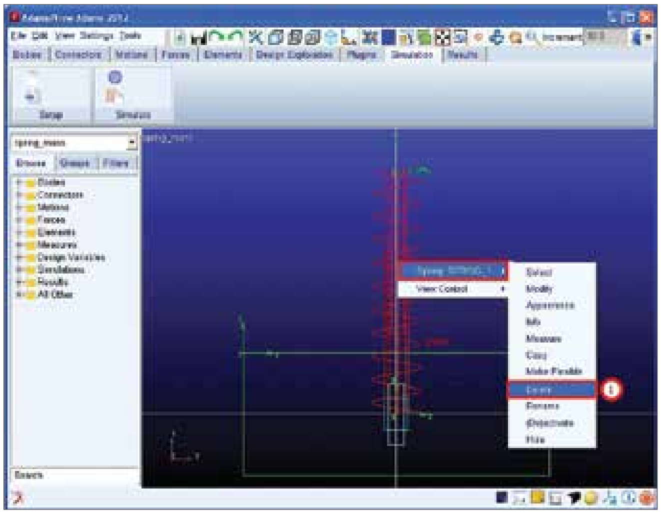
Adams 2013.2

## Problem Description

Replace the Spring Damper with a Single-Component Force. Create a Length vs Force Plot. Find the Static Equilibrium using the Single-Component Force.

## Step 1. Replace the Spring Damper

a. Right Click on the **Spring**, choose **Spring: SPRING_1**, and click on choose **Delete**.
b. Click on the **Forces** Tab and go to **Applied Force: Force (Single-Component)**.
c. Change the **Run-time Direction** to **Two Bodies,** for the **Characteristic** choose K and C and input **K=5.0, C=0.05**.
d. Then click on **PART_2** for the action body.
e. Then click on **ground** for the reaction body.
f. Then click on **PART_2.cm** for the action point.
g. Then click on any point on the global y-axis for the reaction point. The user can right click in the window to invoke LocationEvent or simply snap on a point using the working grid.

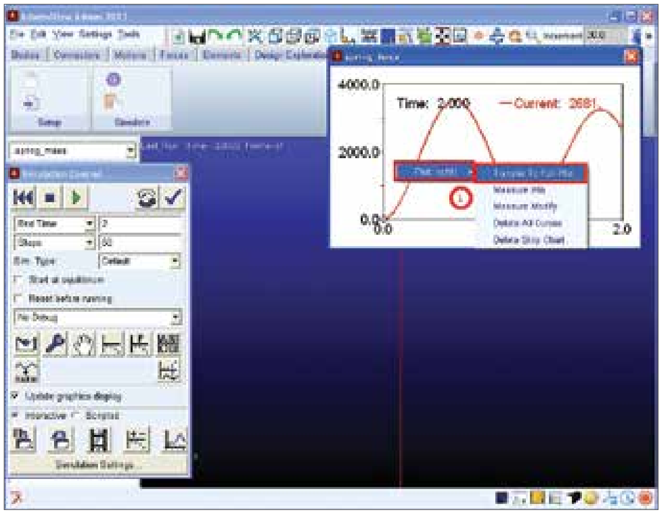


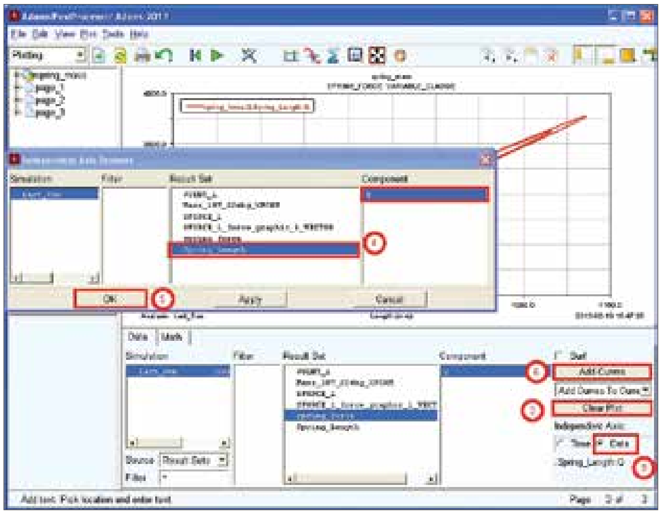
## Step 2. Simulate the Model.

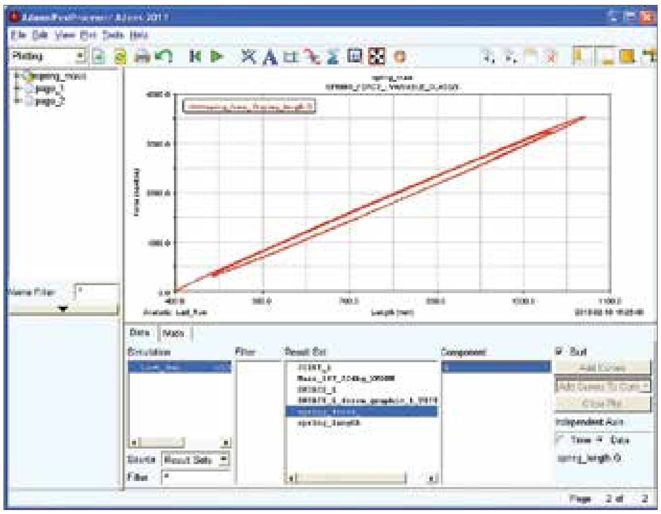
a. Right-click on the **Force: SFORCE_1** and select **Measure**.
b. Change the Measure Name to **spring_force**.
c. Change the **Characteristic** to **Force**.
d. Change the **Component** to **mag** then click OK.
e. Follow similar procedures to create a displacement measure of SFORCE. Change the characteristic to **displacement**. Change the Measure Name to **spring_length**.
f. Go to the **Simulation Tab** and click on **Run a Scripted Simulation** (Calculator Icon).
g. Click on **Interactive**.
h. Change the **End Time** to **2**.
i. Change the **Steps** to **50**.
j. Then click on **Start** or continue simulation.

## Step 3. Creating Length (mm) vs. Force (N) Plot.

a. First right-click **Plot**, choose **Plot: scht1** then click on **Transfer To Full Plot.**
b. Click on **Clear Plot.**
c. Click on **Data**.
d. In the Independent Axis Browser, click on **Spring_length** in the **Results Set** and **Q** in the **Component**.
e. Click **OK**.
f. Click **Add Curves**.



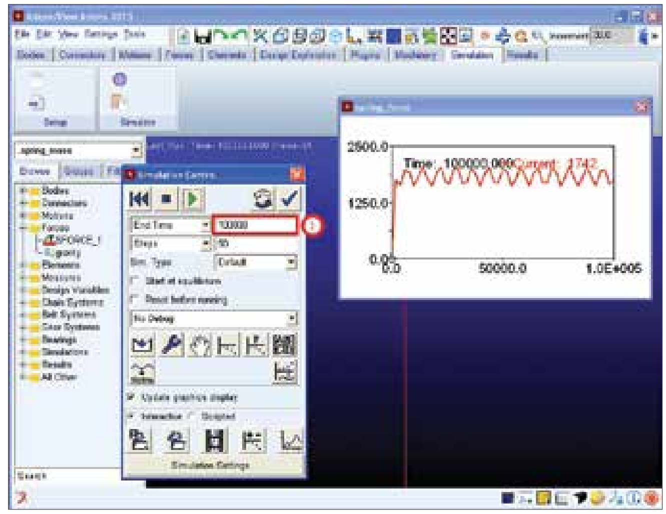


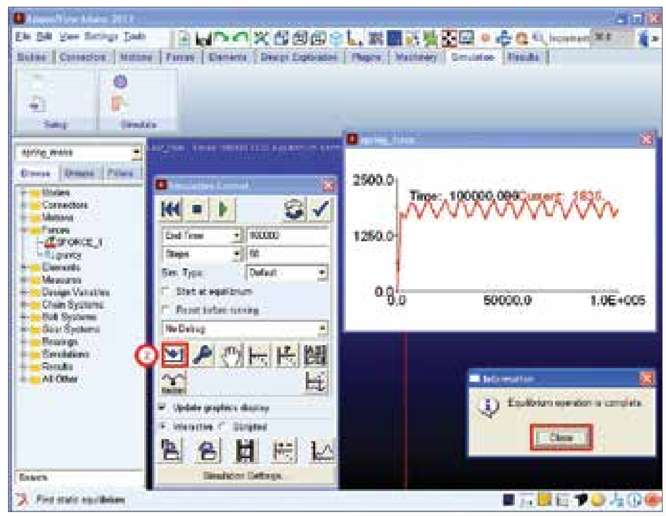


## Step 4. Finding the Static Equilibrium of the Single-Component, Action-Reaction Force.

a. Finding the **Static Equilibrium** of the **Single-Component, Action-Reaction Force**

b. To view the force at static equilibrium click on the Static **Equilibrium** tool. As you can see the value of the Force generated is the same as the **Force** generated by the **Spring Damper**.

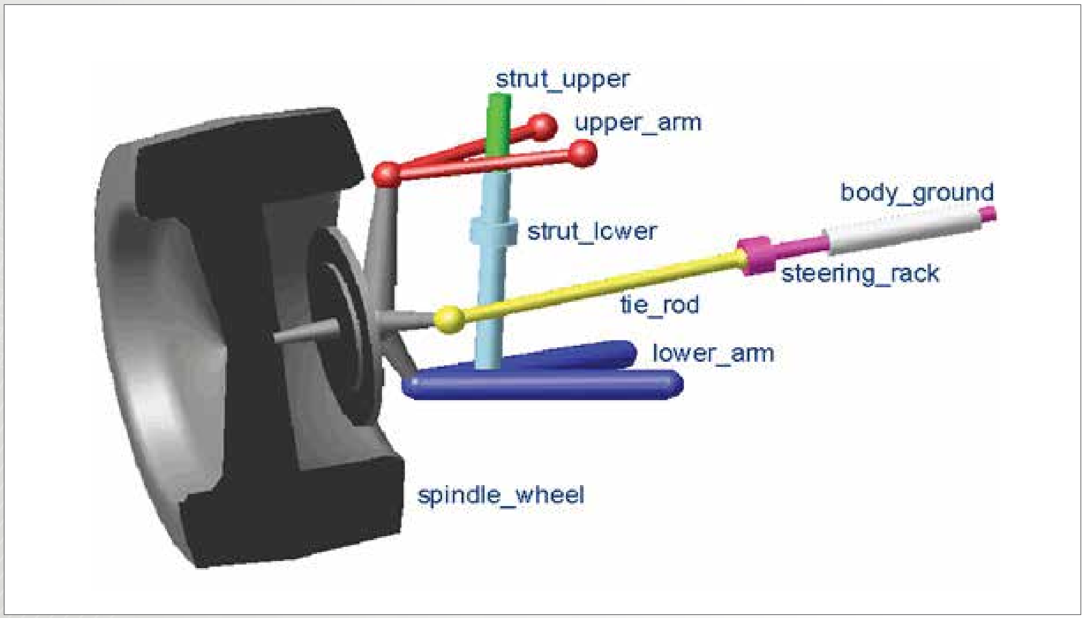




## Analytical Solution – Verify the Results by Calculating the Analytical Solution.

- The block's mass is 187.224 kg.
- Therefore, to balance the force of gravity, the spring damper must generate:
- 187.224 kg * 9806.65 mm/s2 = 1836.04 N
- The results produced by Adams View is the same as the hand calculated answer.

# Example 9: Suspension System 1




## Software Version

Adams 2013.2

## Files Needed

- suspension_parts_starts.cmd
- Located in the directory exercise_dir/Example 9

## Problem Description

Inspect the toe angle that the wheel exhibits throughout its vertical travel of 80 mm in jounce and rebound. The given model is a geometric representation of a short-long arm (SLA) suspension subsystem.

## Step 1. Creating a New Database.

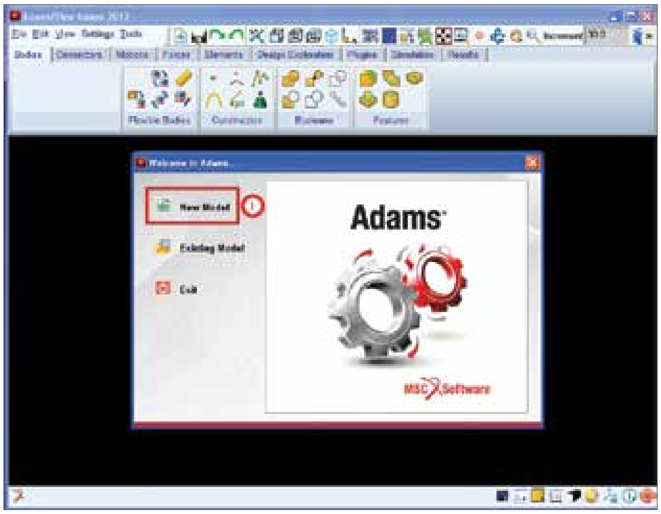
a. Click on **Create a new model.**

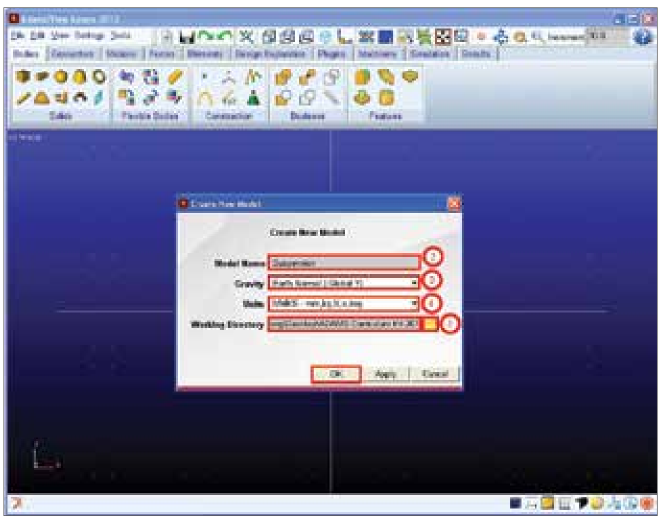
b. First, change the **Model** name to **Suspension**.

c. For the **Gravity** choose **Earth Normal (-Global Y).**

d. Change the units to **MMKS-mm,kg,N,s,deg**.

e. Choose the directory where you want to save the model and then click OK.



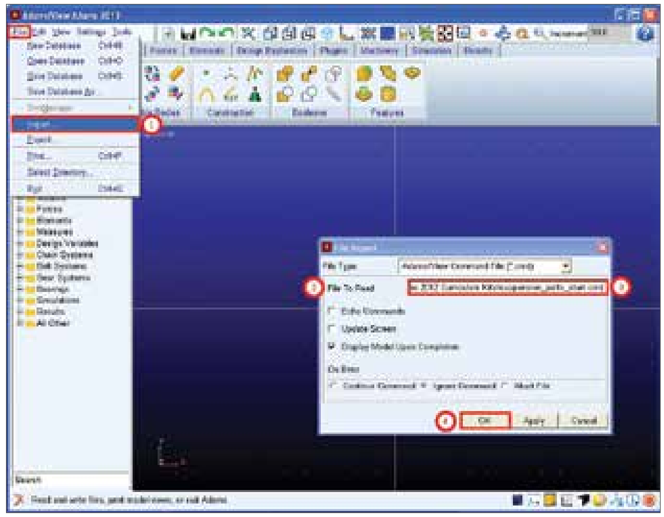
## Step 2. Import the Model.

a. To import the model, first click on File and then choose **Import**.

b. Now click on the **File To Read**.

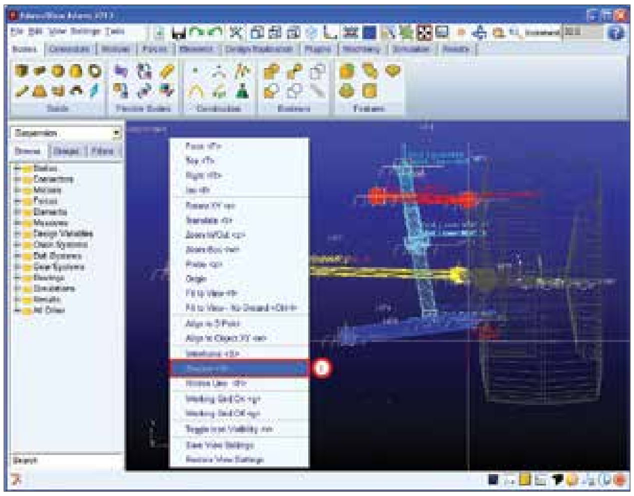
c. For the file choose and Open suspension_parts_starts.cmd.

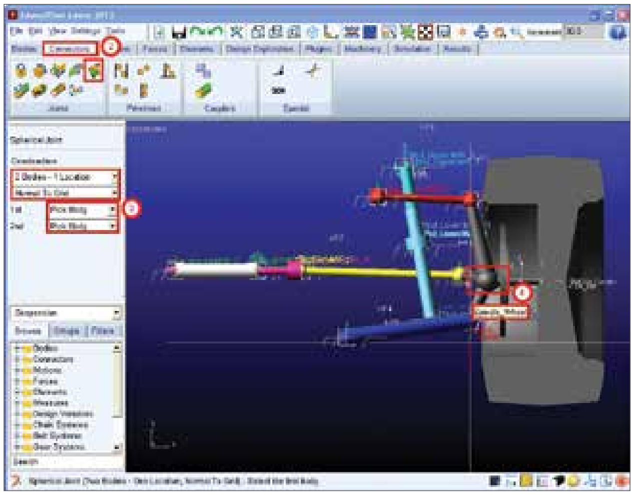
d. Then click **OK**.

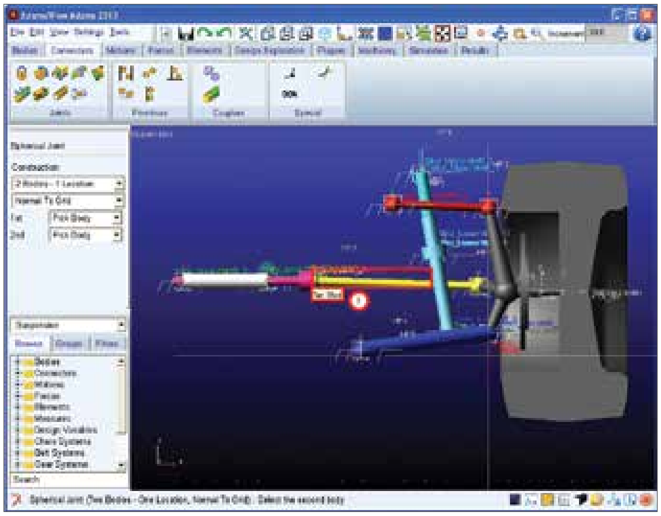


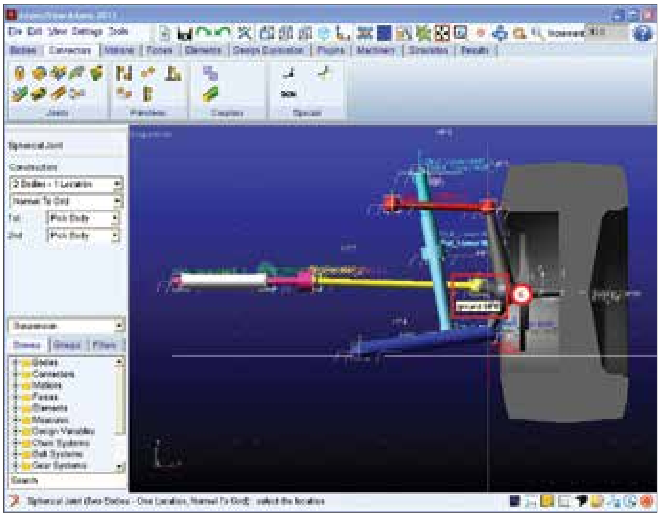
## Step 3. Create a Spherical Joint.

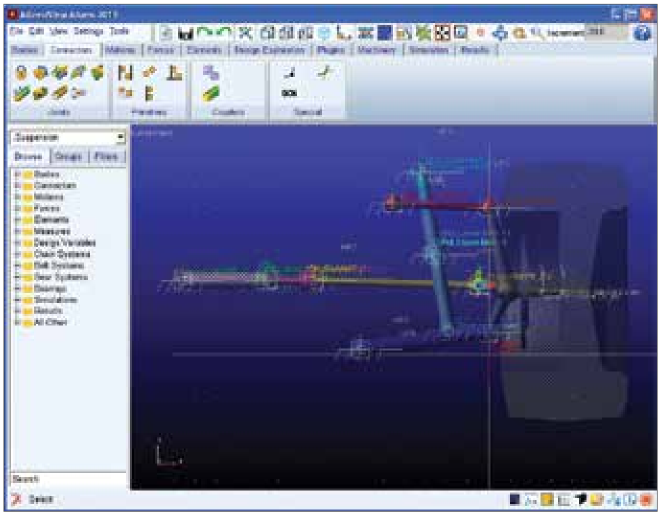
a. First, right-click on the screen, choose **Shaded**.
b. Click on Joint and choose **Joint:Spherical.**
c. For the **Construction** pick **2 Bod-1 Loc** and choose **Normal To Grid** for the **First Body** choose **Pick Body** and the **Second Body** choose **Pick Body**.
d. Choose the **Spindle_Wheel** for the first body.
e. Pick the **Tie_Rod** for the second body.
f. For the location choose **ground.HP8.**
g. If you are not sure where ground.HP8 is, go to the model tree, expand the ground folder and highlight HP8. The hard point will be hilighted in the view window. The proceed to step e. when you are prompt to select the marker, right click at the area where HP8 is and select it from the select list window that pops out.











## Step 4. Create a Hooke Joint.

a. Click on Joint and choose **Joint:Hooke**.

b. For **Construction** choose **2 Bod-1 Loc** and choose **Pick Feature**. For the **First Body** and **Second Body** choose **Pick Body**.

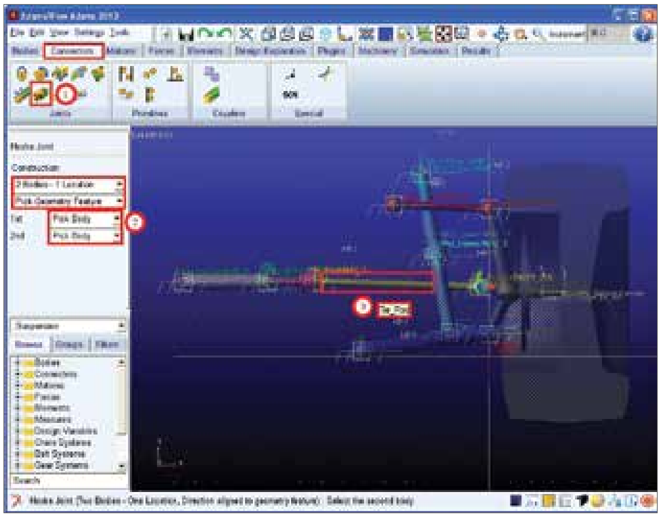
c. Click on the **Tie_Rod** when selecting the first body.

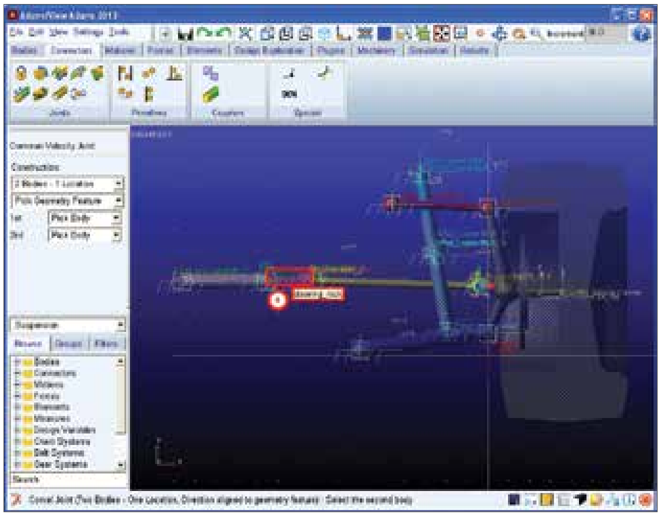
d. Click on the **steering_rack** when selecting the second body.

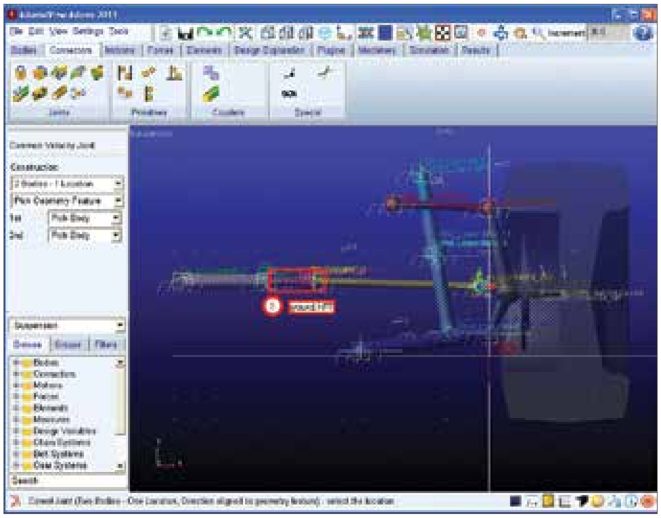
e. Click on the **ground.HP7** when selecting the location.

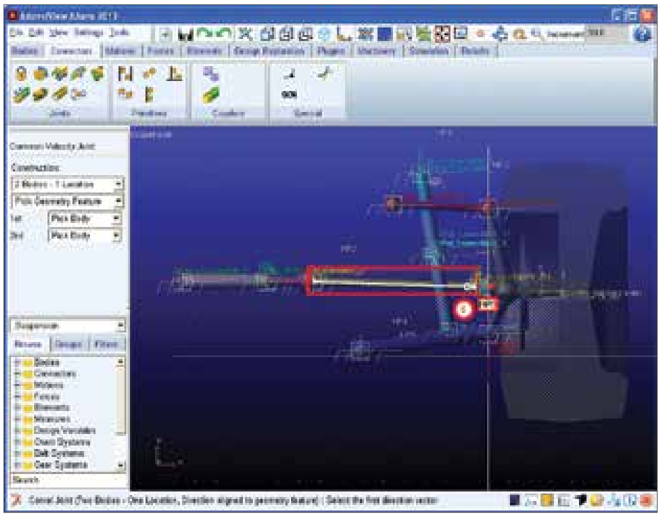
f. Click on **HP8** when selecting the first direction vector.

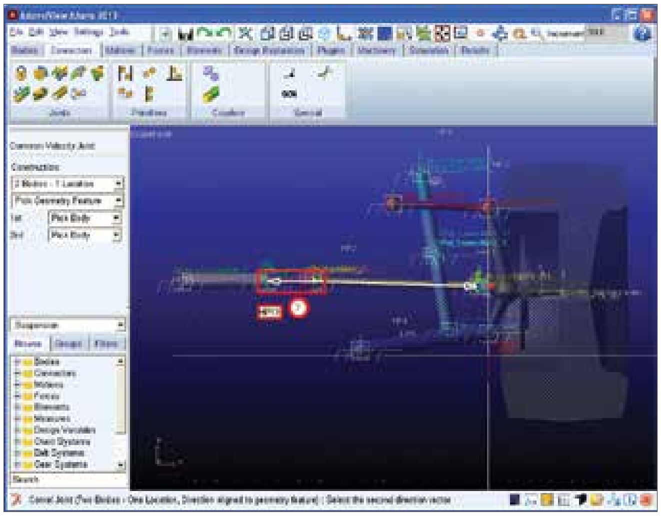
g. Click on **HP13** when selecting the second direction vector.

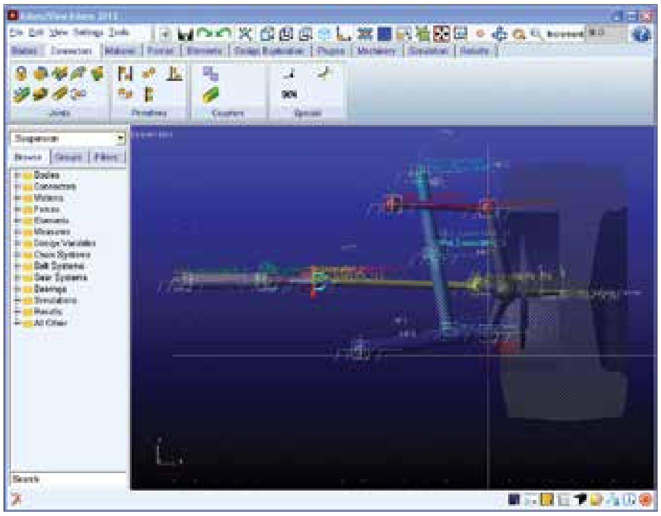












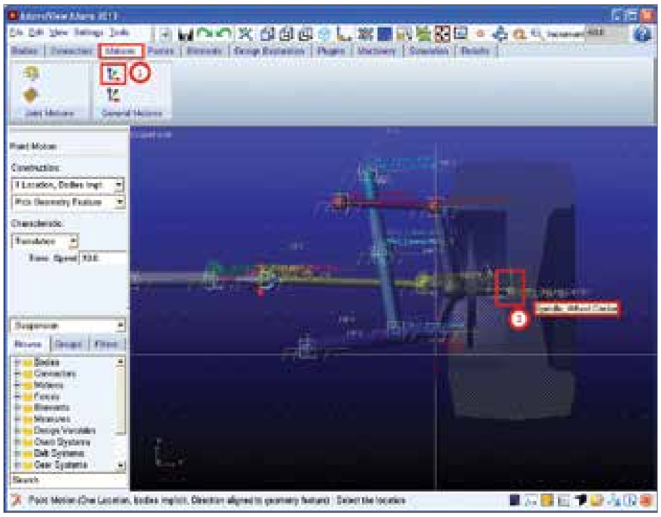
## Step 5. Create a Point Motion.

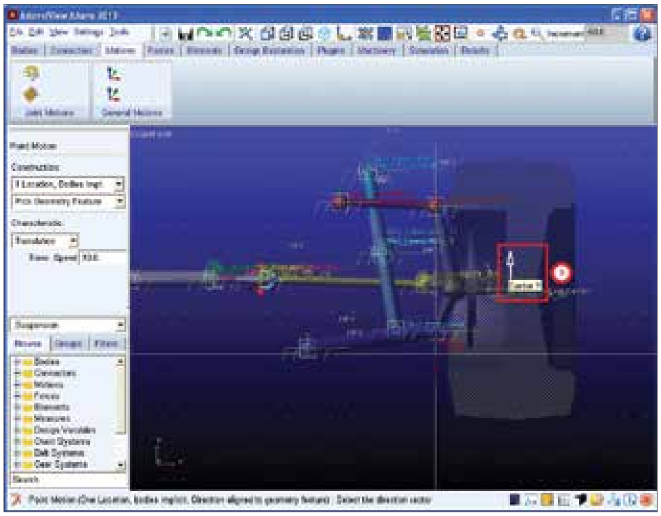
a. First, click on **Motion** ribbon and choose **Point Motion**.

b. Under **Construction**, choose **1 Location, Bodies implied**

c. Choose the **Spindle_Wheel.Center** when selecting the location.

d. Choose the **Center.Y** when selecting direction vector.



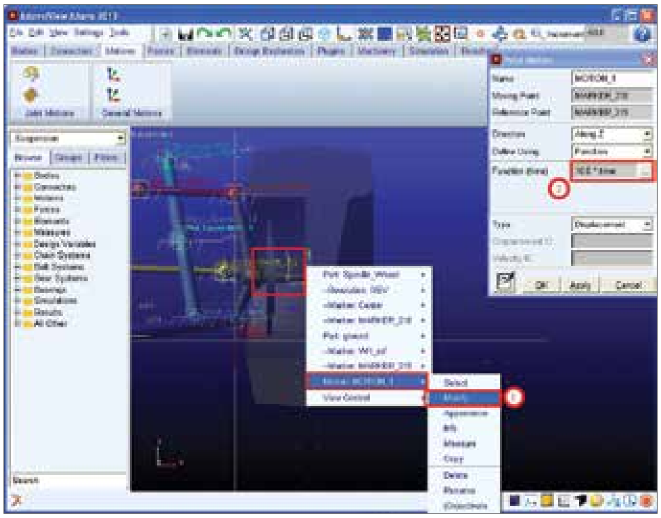


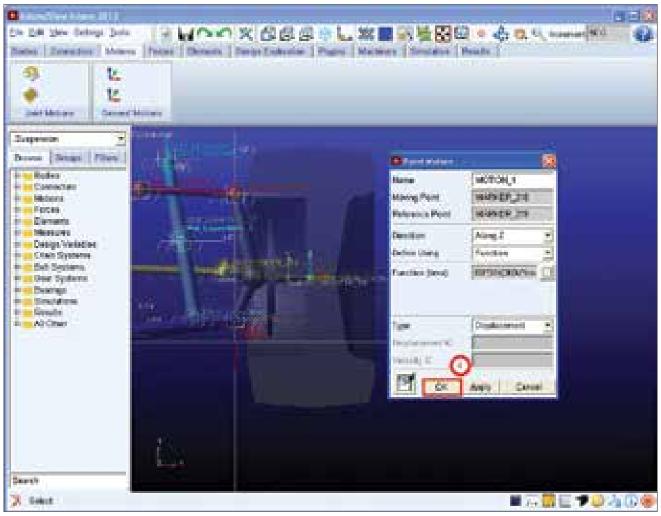
## Step 6. Modify the Motion to a Specific Function.

a. Right-click on the **Wheel.Center** choose **Motion:MOTION_1** and then click on **Modify**.

b. Click on the **Function (time)**.

c. Modify the **"Define a runtime function"** to **80*SIN(360d*time)**. Click on **SIN** under the Math Functions when inputting a SIN function. Then click **OK**.



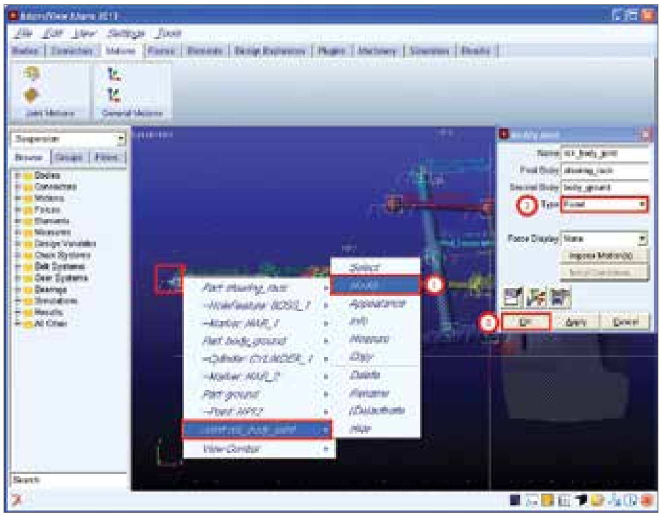


## Step 7. Modify the Translational Joint to be a Fixed Joint.

a. Right-click on the Joint: **rck_body_joint** then click on **Modify**.

b. Change the Type to **Fixed**.

c. Click **OK**.

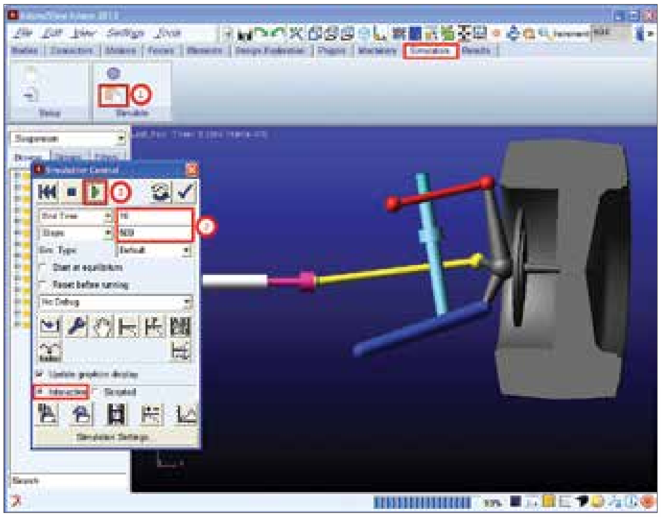


## Step 8. Verify and Simulate the Model.

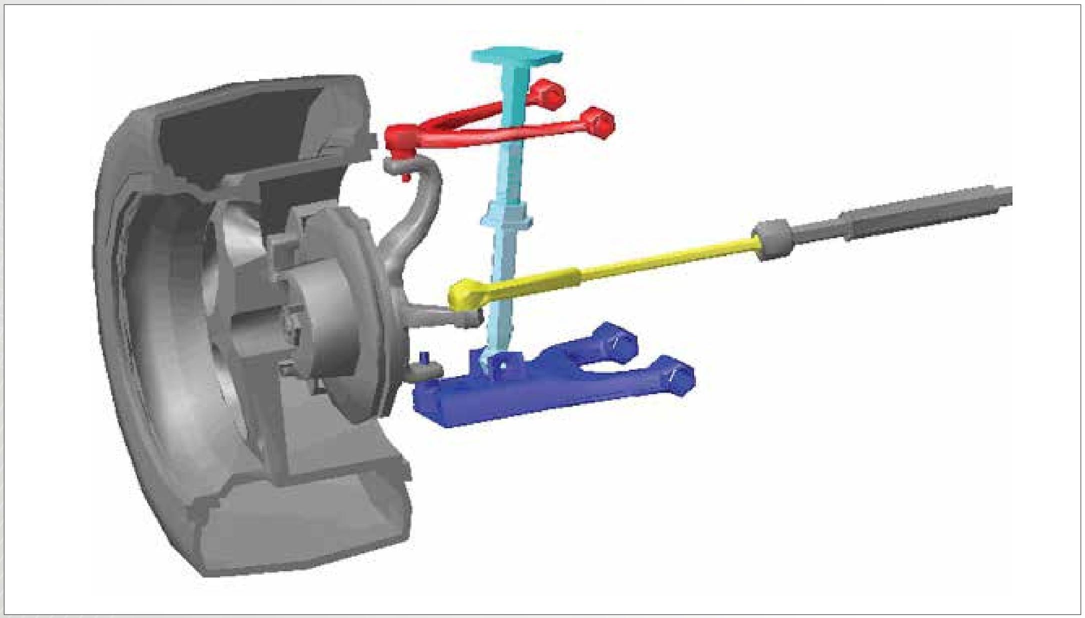
a. First, click on the Interactive **Simulation Control.**

b. Change the **End Time** to **10** and change the **Steps** to **500**.

c. Then click on **Start Simulation.** Now you have completed creating a **Spherical Joint, Hooke Joint and Point Motion** on this suspension subsystem.



# Example 10: Suspension System 2



## Software Version

Adams 2013.2

## Files Needed

- **wheel.slp**
- Knuckle.slp
- Located in the directory **exercise_dir/Example 10**

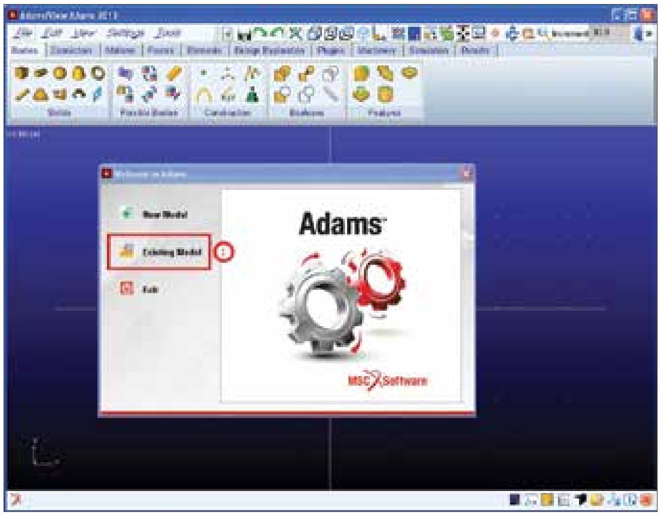
## Problem Description

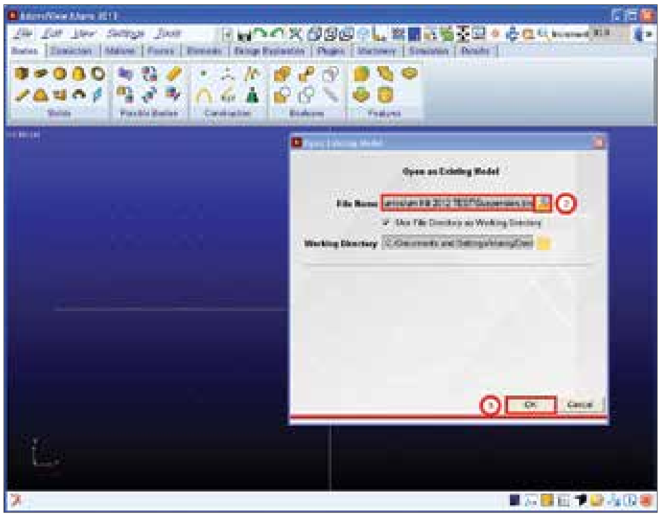
Use the model you built in the previous workshop (Suspension System I) to inspect the toe angle that the wheel exhibits throughout its vertical travel of 80 mm in jounce and rebound. Note: you can either use your own model created in Example 9 or use the start file in the directory.

## Step 1. Open an Existing Database.

a. First, choose **Existing Model.**

b. Under File Name, locate the **suspension_start.cmd** file.





c. Then click **OK**.

## Step 2. Create Point-to-Point Measure.

a. To find the relative wheel displacement in the Yg direction, click on the **Design Exploration** tab, then choose **Measure**, pick **Point-to-Point** and click on **New**....

b. Click on **Advanced**.

c. Now change the **Measure Name** to **.suspension. Wheel_Height.**

d. Change the **"To Point"** to Center. This can be typed in or double click on it to choose it from the Database Navigator. As for the **"From Point"** double click, choose **WH_ref** from the **Database Navigator** and click **OK**.

e. Then choose **Y** for the **Component**.

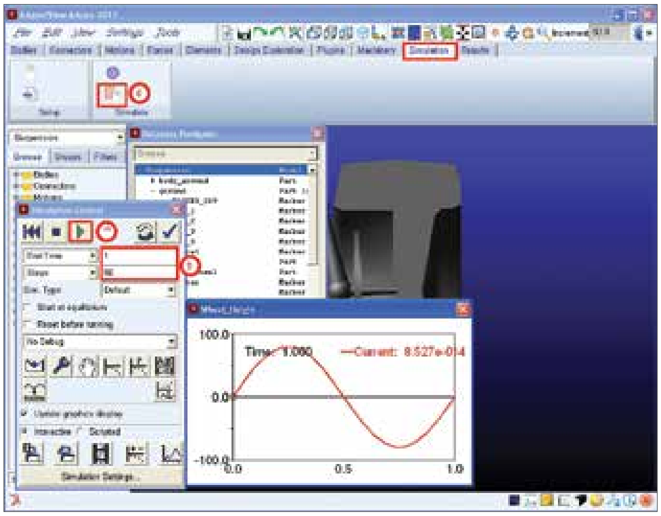
f. Choose **Cartesian**.

g. Click **OK**.

h. Click on the **Interactive Simulation Controls**.

i. Change the **End Time** to **1.0** and the **Steps** to **50**.

j. Click on **Start Simulation**. As you can see a plot of **Time vs Displacement** in the Yg direction has been

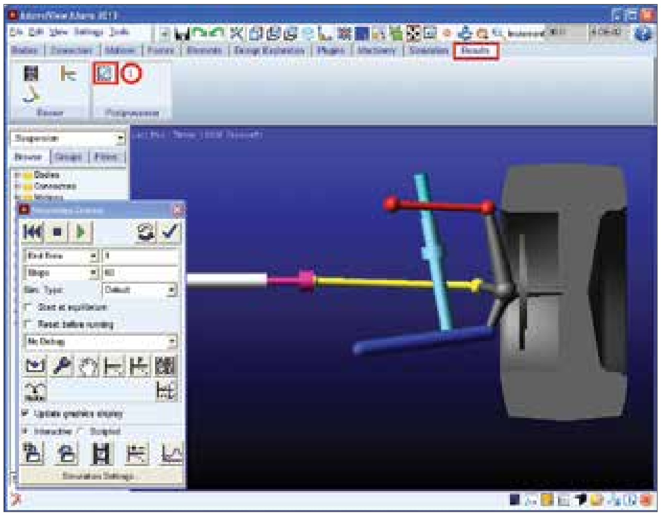


created.

## Step 3. Use a Function Measure to Create a Toe Angle.

a. Using an Adams/Solver function measure, create a toe angle measure using the markers **Spindle_Wheel.Center** and **Spindle_Wheel.TA_ref.** First click on **Build**, choose **Measure**, click on **Design Exploration** and then click on **Create a New Function Measure.**

b. Input **ATAN(DZ(Center,TA_ref)/DX(Center,TA_ref)** for **Create or Modify a Function Measure,** choose



**ATAN** under the **Math Functions.**

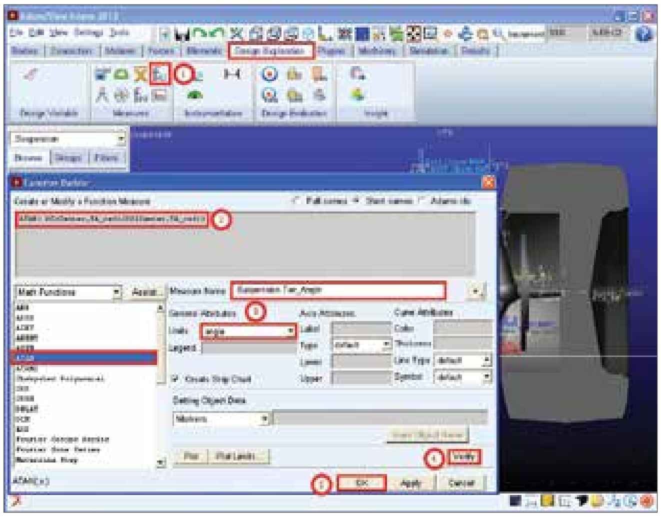
c. Change the **Measure Name** to **.suspension.Toe_Angle** and change the **Units** to angle.

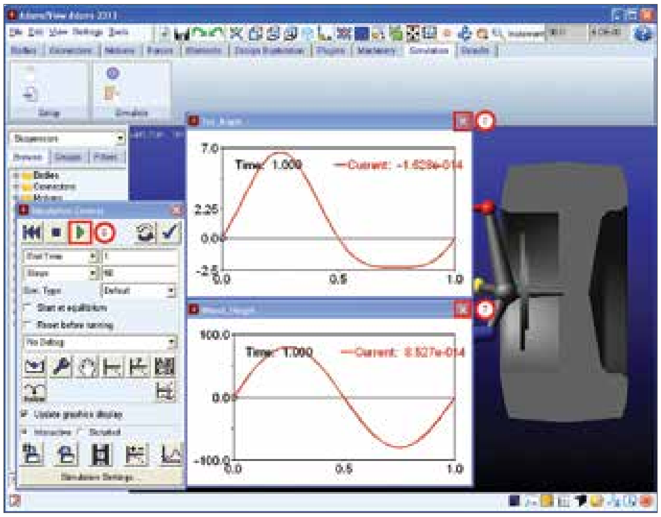
d. Click on **Verify** then click **OK** when the **Function syntax** is correct.

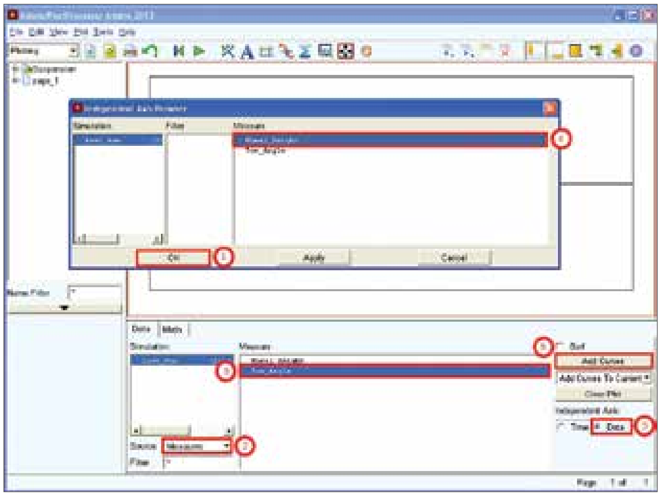
e. Click **OK**.

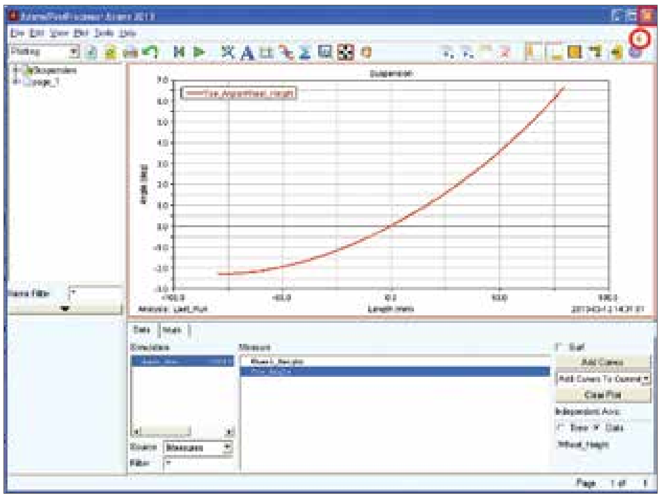
f. Click on the **Start Simulation**.

g. Click on **Close** to close the plots.









## Step 4. Plot Toe Angle versus Wheel Height.

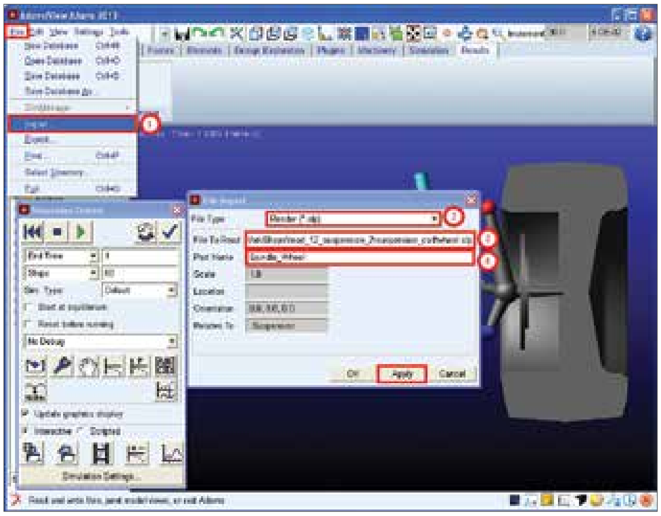
a. Click on Results and go to Opens Adams/ PostProcessor.

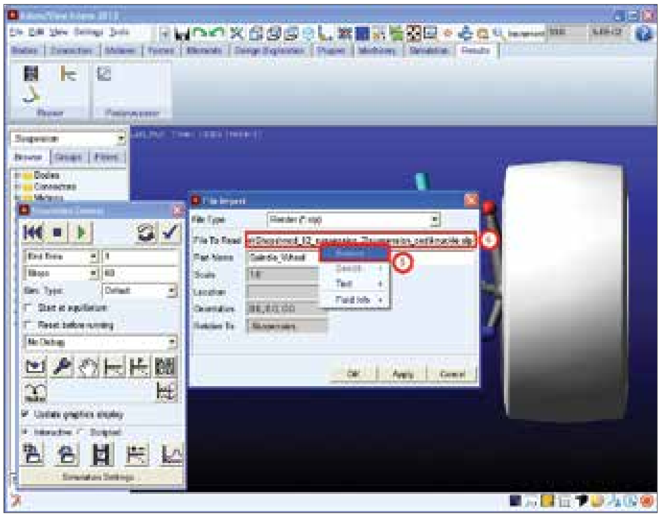
b. For the Dependent Axis under Measure choose Toe_ Angle and then click on Data for the Independent Axis.

c. Choose Last_Run for Simulation and choose Wheel_ Height for Measure.

d. Click OK then click on Add Curves.

e. Close the plotting window.





## Step 5. Importing the Knuckle and Wheel.

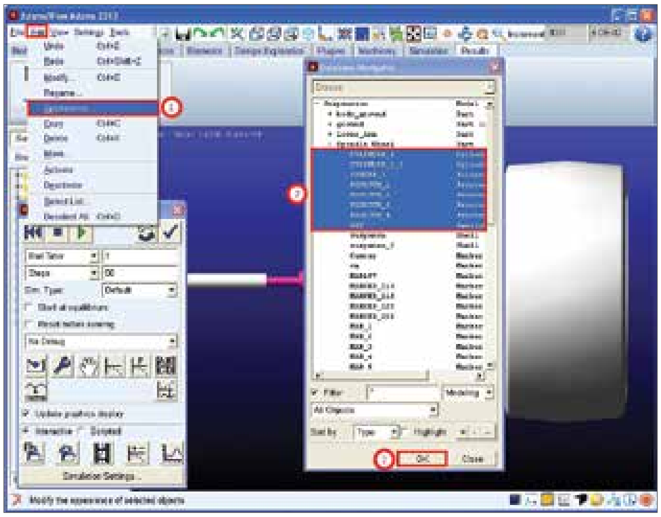
a. Now, you'll import more realistic, CAD-based spindle/ wheel geometry. First click on **File** and choose **Import**.

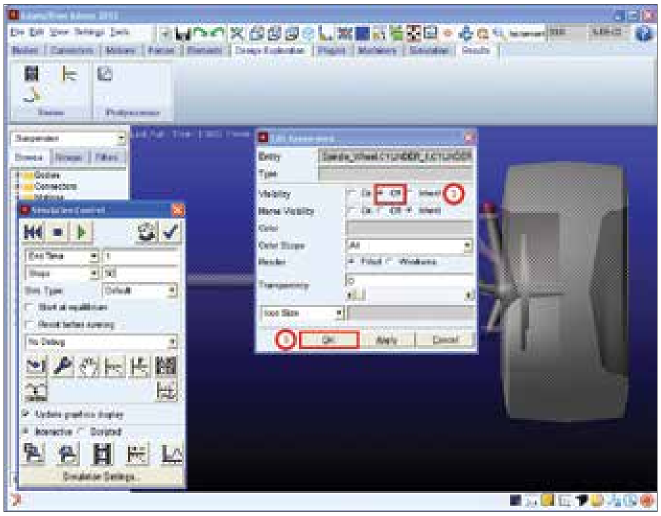
b. Choose **Render(*.slp)** for **File Type**.

c. Choose the appropriate location by clicking on **File to Read**. Choose the file **wheel.slp** then click Open.

d. Change the **Part Name** to **Spindle_Wheel** you can screen pick this by right-clicking and choose **Part** and click on **Guess**. Then click **Apply**.

e. Change **File to Read** by right-clicking and choose **Browse....**

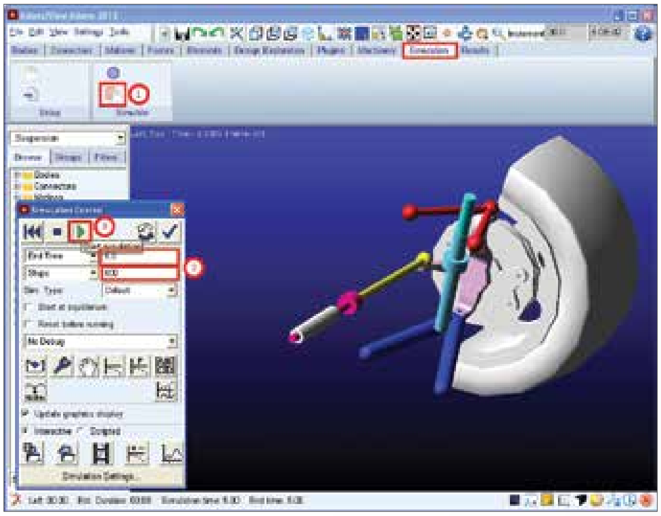




f. Choose **knuckle.slp** then click **Open**.

## Step 6. Turn off Spindle Geometry.

a. Turn off the appearance of Adams/View spindle geometry so that only the CAD geometry is visible. First click on **Appearance**... under the **Edit** menu.



b. Hold on the Shift key and choose **CYLINDER_1, CYLINDER_1_2, SPHERE_1, FRUSTUM_1, FRUSTUM_2, FRUSTUM_3, FRUSTUM_4, REV** and click **OK**.

c. Click on **Off** for **Visibility**, then click **OK**.

d. To rotate the model, click on R on the keyboard or right-click on the screen and choose Rotate.

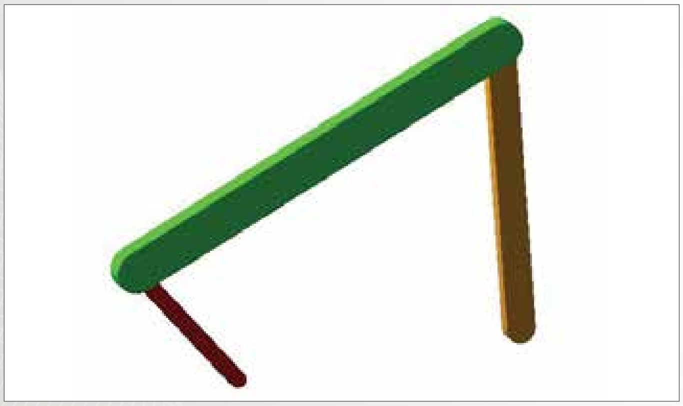
## Step 7. Simulate the Model.

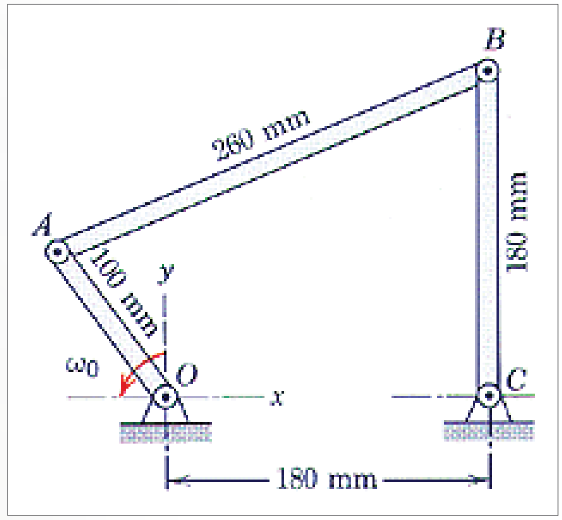
a. First, click on the **Interactive Simulation Control.**

b. Change the **End Time** to **5.0** and the **Steps** to **500**.

c. To simulate the model click on **Start Simulation**.

# Example 11: Four Bar Velocity






## Problem Description

Use Adam/View to

- Create a marker
- Change angle units
- Add motion

Use Adams/PostProcessor to

- Create center of mass angular velocity measurements

## Software Version

Adams 2013.2

## Problem Description

In the four-bar linkage shown, control link OA has a counterclockwise angular velocity omega = 10 rad/s during a short interval of motion. When the link CB passes the vertical position shown, point A has coordinates x = -60 mm and y = 80 mm. By means of vector algebra, determine the angular velocity of AB and BC.

This problem asks for the rotational velocity of segment BC when it is in the pictured position given a constant and known rotational velocity for segment OA. We will use ADAMS to create a model with the given conditions and collect the data needed.

## Step 1. Creating the Model

a. Start **Adams/View.**

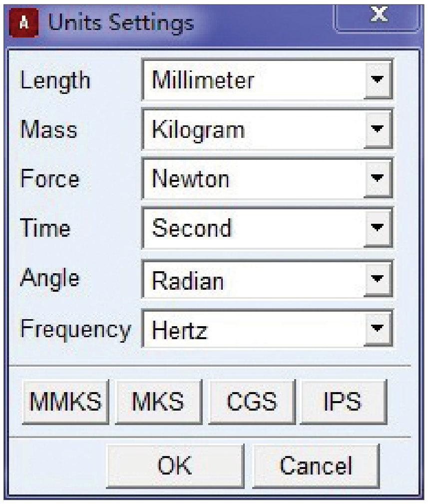
b. Create a new model. **(Model Name = Fourbar, Units = mmks, Gravity = none)**

c. Modify the spacing of the **Working Grid (X = 10mm, Y= 10mm)**

d. Click **Units** from Settings menu

e. Select **Radian** from **Angle** pull down menu

f. Click **OK**

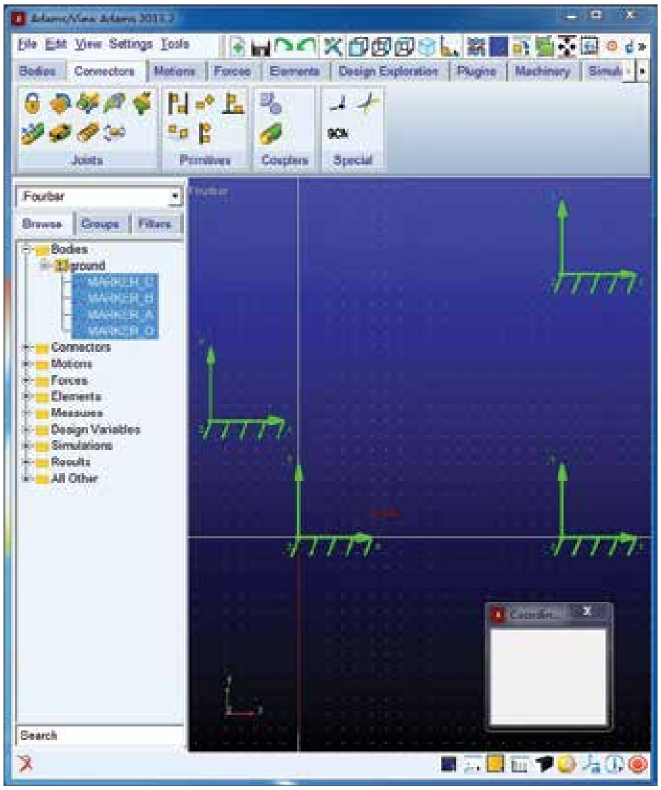




## Step 2. Create a Marker

a. Press **F4** to Open **Coordinate** Window

b. From **Bodies** ribbon, select **Construction Geometry: Marker**

c. Create a marker at each of the following coordinates: **O (0, 0, 0); A (-60, 80, 0); B (180, 180, 0); C (180, 0, 0)**



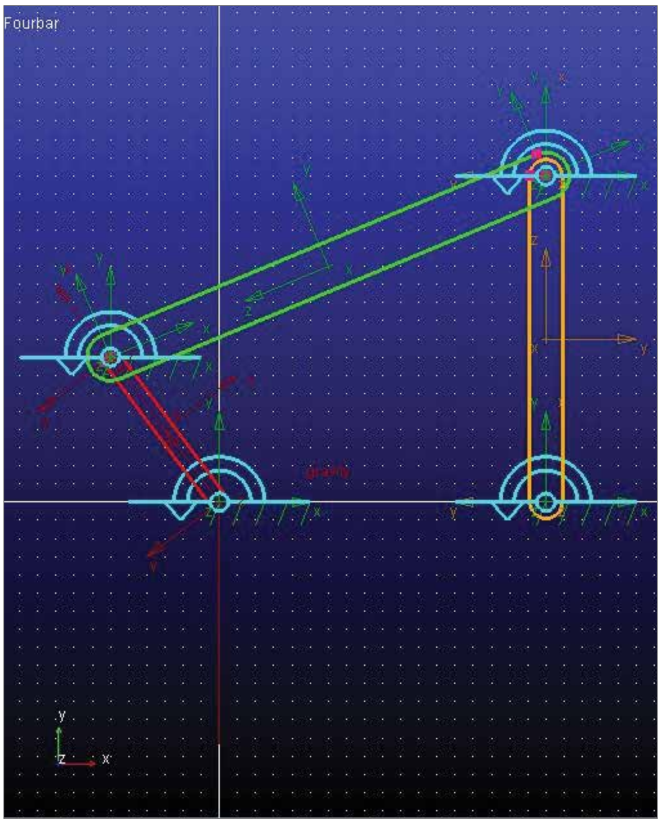
## Step 3. Create Links and Joints

a. From **Bodies** ribbon, double click **RigidBody: Link** 

b. Create links OA, AB, and BC, using the markers as end points.

c. From **Connectors** ribbon, double click **Create a Revolute joint** 

d. Make revolute joints between two links at points **A** and **B**, and between link and ground at **O** and **C**.

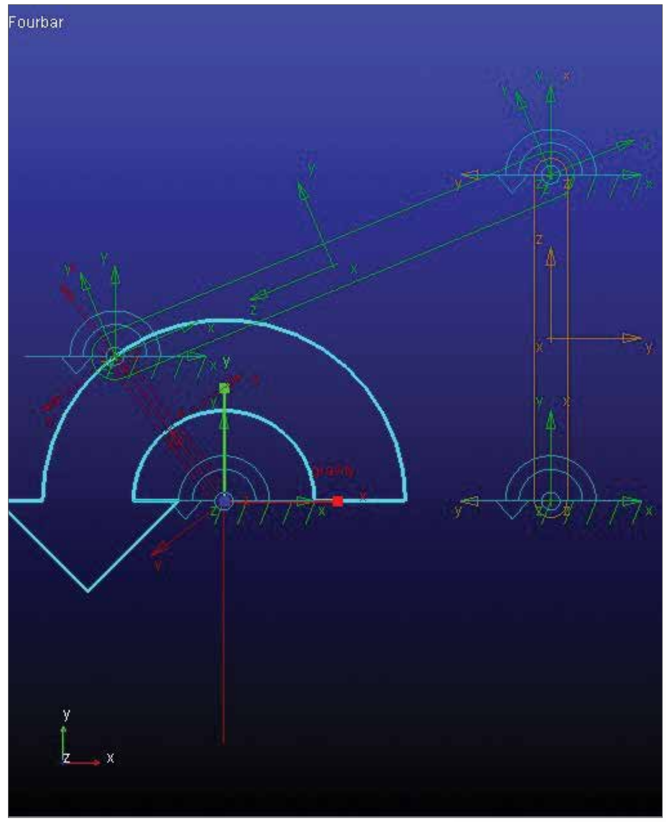



## Step 4. Add Motion

a. From Motions ribbon, select Rotational Joint Motion 

b. Enter **(1rad)** in **Rot.Speed** text field

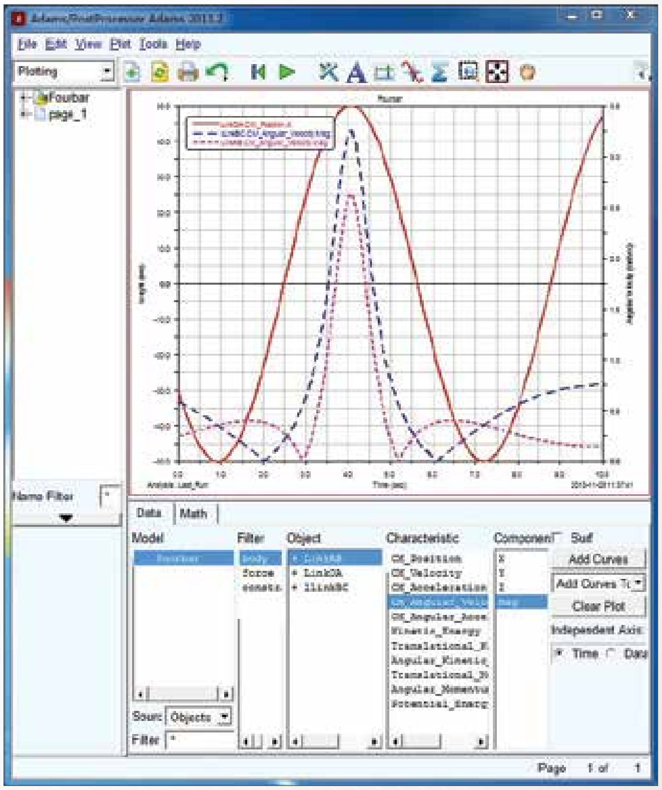
c. Select joint at point **O**



## Step 5. Testing the Model

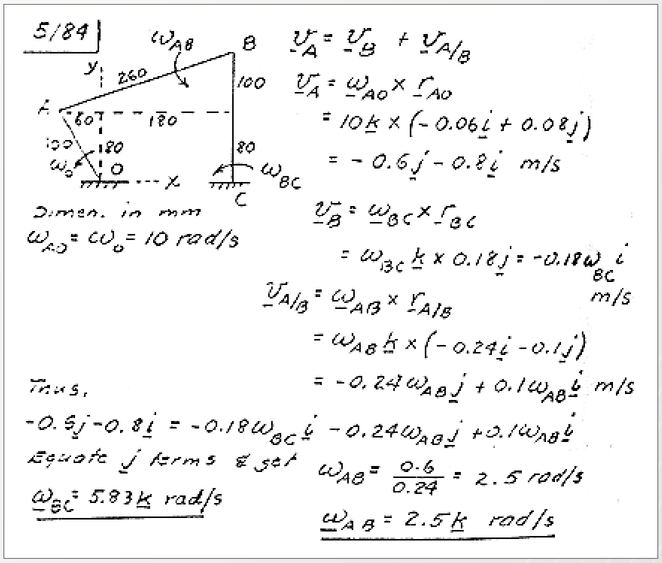
a. From **Simulation** ribbon, select **Run an Interactive Simulation**
b. Set **End Time** to **10** and **Step Size** to **0.1**
c. Click **Start,**
d. Click **Plotting**
e. Create a **CM position** plot for link **OA** in **X** component
f. Create a **CM angular velocity** plot for **LinkAB** and **LinkBC** in **mag** component

g. Use the **Plot tracking tool**
h. Follow the plot curve. Find the angular velocity at X = 0.0




## Results

Theoretical Solution

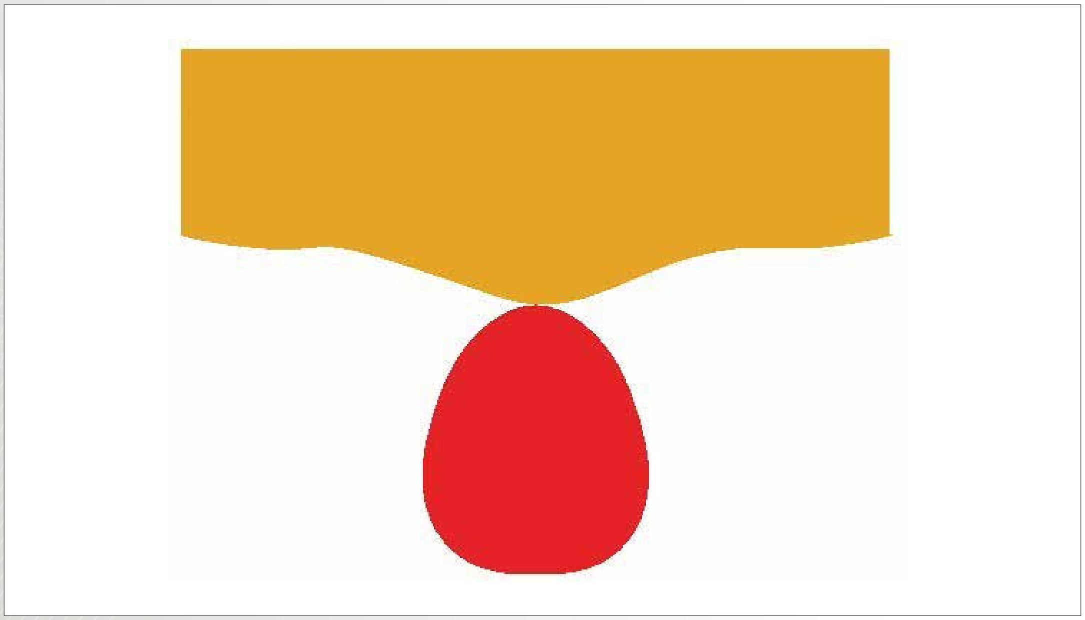



Adams solution

| X: | Y: |
|---|---|
| 0.0 | 0.25 |

| X: | Y: |
|---|---|
| 0.0 | 0.5833 |

# Example 12: Cam-Follower



## Workshop Objectives

Use Adams/view to

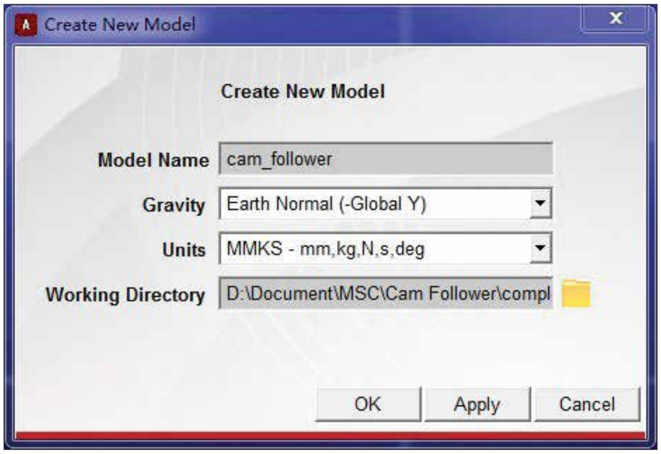
- Create different shapes using the open and closed splines
- Add constraints (joints): revolute joint, translational joint and a 2D curve-curve constraint
- Create a rigid body: box
- Measure

## Software Version

Adams 2013.2

## Step 1. Getting Started:

a. Start **Adams/View**
b. Select **New model** button.
c. Enter **cam_follower** as **Model Name**
d. Choose a Location to save your files
e. Verify the **Gravity** text field is set to **Earth Normal (-Global Y).**
f. Verify that the **Units** text field is set to **MMKS - mm,kg,N,s,deg.**
g. Select **OK**.




## Step 2. Settings Grid Size:

a. Click **Settings** menu, then **Working Grid**…
b. The **Working Grid Settings** window will appear
c. Change the **Spacing** text fields in X and Y to (**10mm)**
d. Click **OK**.


## Step 3. Closed Body Spline

a. Under the **Bodies** ribbon, click on **Spline** 
b. Select **New Part** from **Spline** pull down menu
c. Turn on checkbox next to **Closed**.
d. Click on the 13 points in the table below.
e. Right click to create a closed spline

- *Note that the first point and the last point have the same coordinates to create a closed spline.

f. An alert box will appear warning you that the part has no mass. Close the box.

- *If your part's geometry does not match the illustration, it can be fixed by clicking and dragging any of the "hot points" (rectangular boxes) to its proper location

| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| X | 0 | -50 | -70 | -80 | -70 | -50 | 0 | 50 | 70 | 80 | 70 | 50 | 0 |
| Y | 0 | -30 | -70 | -120 | -160 | -180 | -190 | -180 | -160 | -120 | -70 | -30 | 0 |
| Z | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

### Step 4. Create Revolute Joint

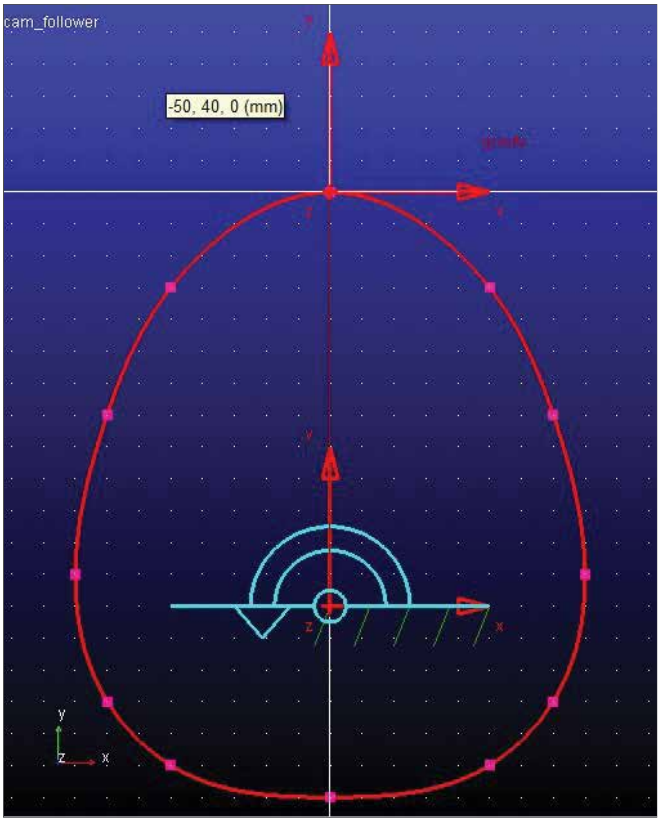
a. Under the **Connector** ribbon, select **Revolute Joint** 

b. Verify that the **Construction** text fields read **2 Bod-1 Loc.**

c. Left-click on any blank area in the working window (ground)

d. Left click on your cam

e. Click on the position **(0,-130,0)**




### Step 5. Open Body Spline

a. Select the **Spline** tool 

b. Turn on checkbox next to **Closed**.

c. Click on the 27 points in the table below.

d. Right click to create a closed spline

- *Note that the first point and the last point have the same coordinates to create a closed spline.

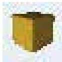
| Points | 1 | 2 | 3 | 4 | 5 | 6 | 7 |
|---|---|---|---|---|---|---|---|
| X | -250 | -200 | -150 | -100 | -50 | 0 | 50 |
| Y | 50 | 40 | 40 | 30 | 10 | 0 | 10 |
| Z | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

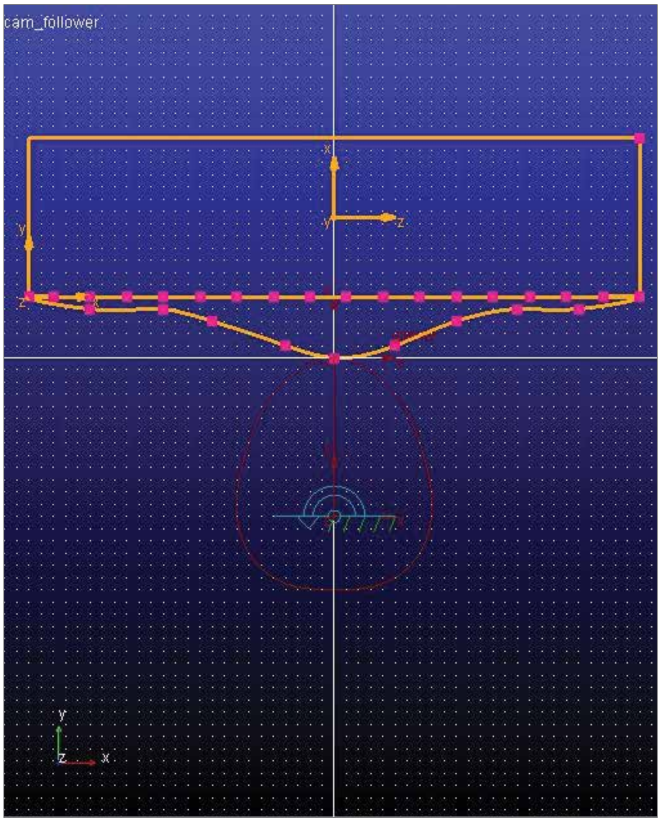
| Points | 8 | 9 | 10 | 11 | 12 | 13 | 14 |
|---|---|---|---|---|---|---|---|
| X | 100 | 150 | 200 | 250 | 220 | 190 | 160 |
| Y | 30 | 40 | 40 | 50 | 50 | 50 | 50 |
| Z | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

| Points | 15 | 16 | 17 | 18 | 19 | 20 | 21 |
|---|---|---|---|---|---|---|---|
| X | 130 | 100 | 70 | 40 | 10 | -20 | -50 |
| Y | 50 | 50 | 50 | 50 | 50 | 50 | 50 |
| Z | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

| Points | 22 | 23 | 24 | 25 | 26 | 27 | 28 |
|---|---|---|---|---|---|---|---|
| X | -80 | -110 | -140 | -170 | -200 | -230 | -250 |
| Y | 50 | 50 | 50 | 50 | 50 | 50 | 50 |
| Z | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

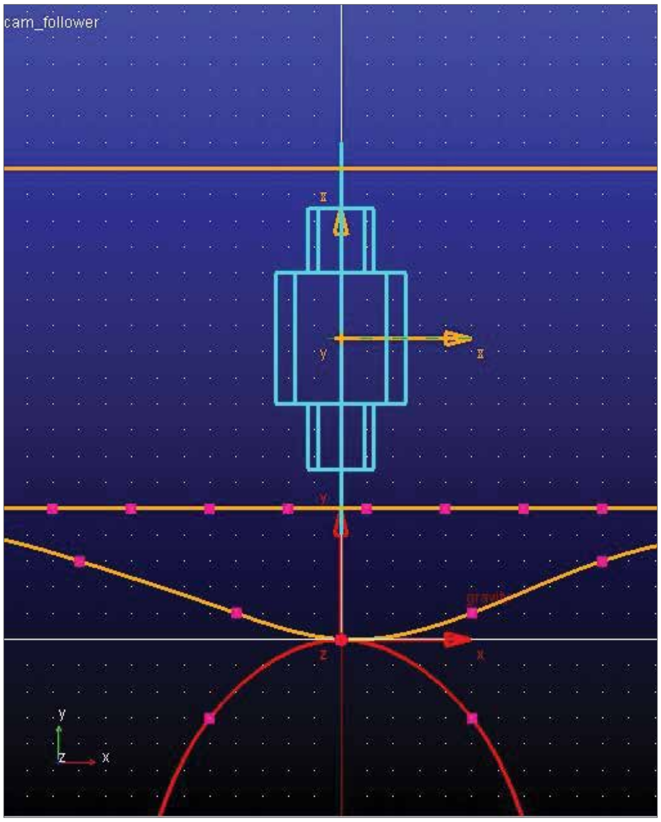
## Step 6. Create Box

a. Select the **Box**. 
b. Select **Add to Part** from Box menu
c. Click on the **Open Body Spline** in the working area to select the part to add to.
d. Click on the left end of the open spline **(-250,50,0).**
e. Click on **(250,180,0).**




## Step 7. Create Cylindrical Joint

a. Select the **Joint:Cylindrical** tool. 
b. Set that the Construction text fields to
**1 Location**
c. Click on **PART_3.cm**
d. Move the cursor in the positive **Global Y** axis until an arrow pointing straight up appears. Click once.
e. Make sure the arrow is parallel to the Y axis. This arrow determines the direction of the translational joint.



## Step 8. Create Curve-on-Curve Contact

a. Select the **Cam 2D Curve-Curve Constraint** tool 

b. Click on the cam part

c. Click on the follower

## Step 9. Add Rotational Joint Motion

a. Select the **Rotational Joint Motion** 

b. In the **Speed** text field, enter **(360d)** to set the motion displacement to be 360 degrees/second.

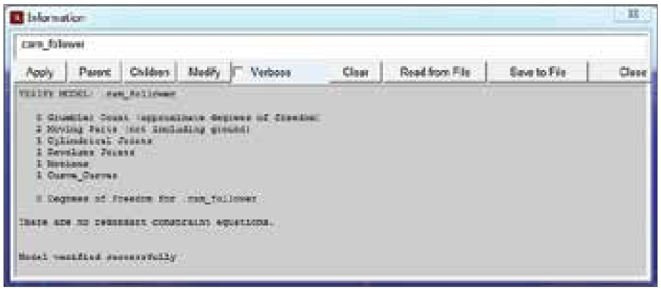
c. Left-click on the revolute joint.

## Step 10. Verify

a. Right-Click on the **Information** Icon in the bottom right corner of the Working Window 

b. Left-click on the **Verification** Icon

c. After seeing that the model has verified successfully, click on the cose button.




## Step 11. Measure

a. Right click the follower part and choose **measure**.

b. The **Part Measure** dialog box appears.

c. Select **CM Position** from **Characteristic** pull down menu and select **Y** for the **Component** entry to measure the displacement in the Y direction.

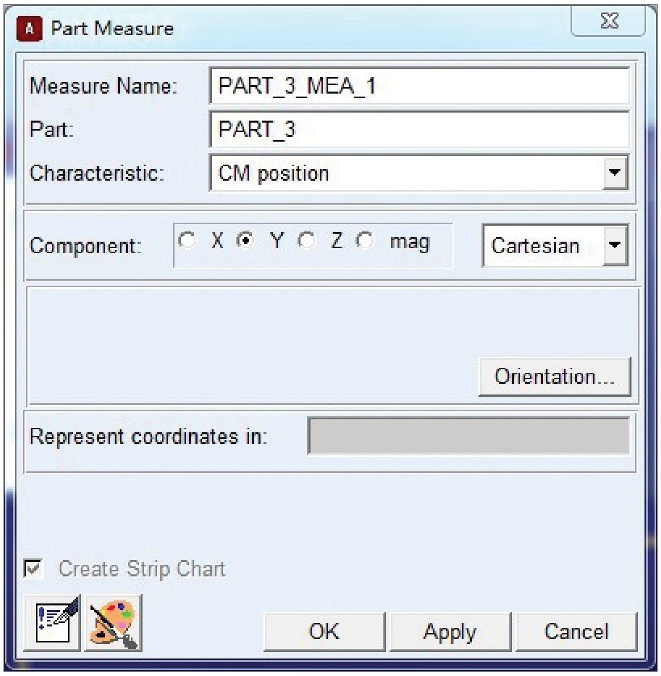
d. Click **Apply**.

e. A graph window appears. This is where data will be displayed.

f. Repeat, step b & c, except use **CM Velocity** for **Characteristic**.

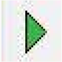
g. Repeat, step b & c, except use **CM Acceleration** for **Characteristic**. A new graph window will appear for each new measure.

h. After the three graph windows are created, click Cancel to close the dialog box

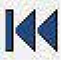


## Step 12. Simulation

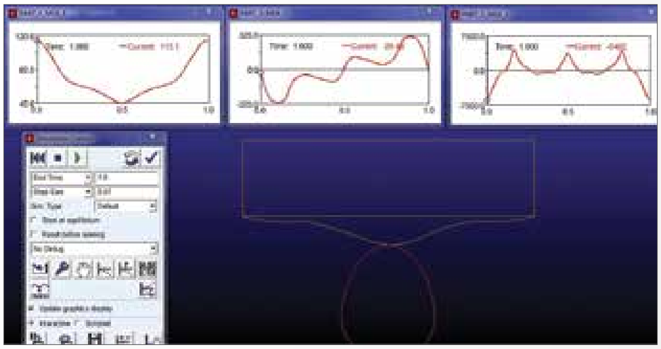
a. Click on the Simulation tool in the Toolbox.
b. Enter 1 in End Time text field
c. Change Steps to Step Size, enter .01 in the text field



d. Click on the Play icon.
e. You should see the cam rotate about the pivot and the follower slide along its translational joint.



f. When the simulation ends, click on the Rewind icon



## Step 13. Plotting

a. To get a closer look at a plot, click on a blank area inside the small plot window with the right mouse button and follow the pull right menu. Select **Transfer to Full Plot.**
b. The ADAMS Plot Window will open, replacing the modeling window. To return to the modeling window, go to the File pull-down menu and select Close Plot Window or press F8 or click on the Return to modeling environment button

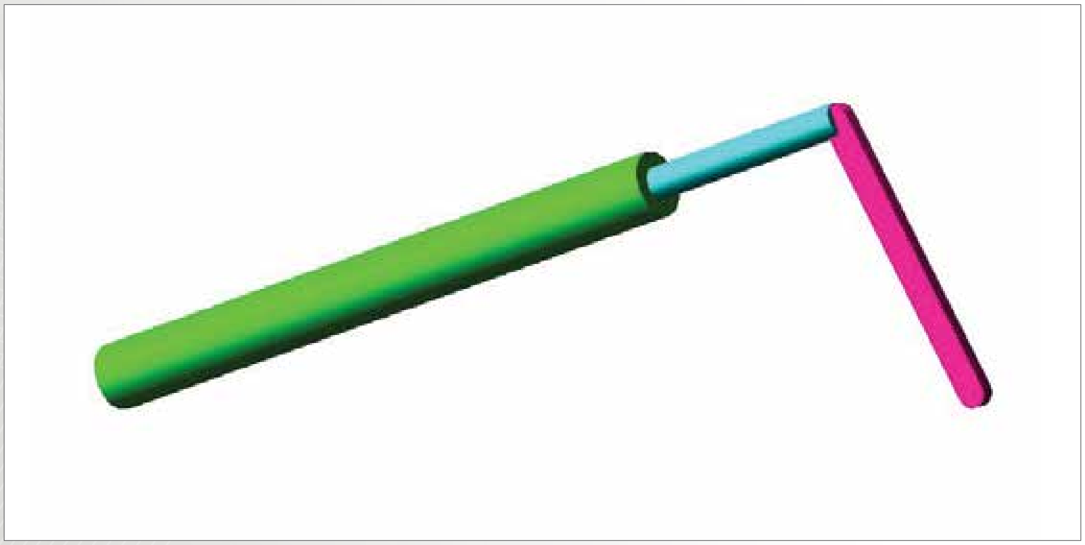
## Step 14. Viewing Plots

a. Select Objects for the source text field
b. Choose a Filter **(Body, Force. Constraint)**
c. Choose an **Object**
d. Choose a **Characteristic**
e. Choose a **Component**
f. Select **Surf** if you would like to replace the curve in the **Plot Window**, or select **Add Curves** to add more curves to the window

## Step 15.Saving

a. Return to ADAMS modeling window
b. Under the File pull-down menu, select **Save Database As...**
c. In the text field next to **File Name**, enter the name you wish to give this model, for example, **cam**.
d. Select **OK**.
e. A binary file (.bin) has been created in the folder you choose when opening ADAMS

# Example 13: Crank Slider



## Workshop Objectives

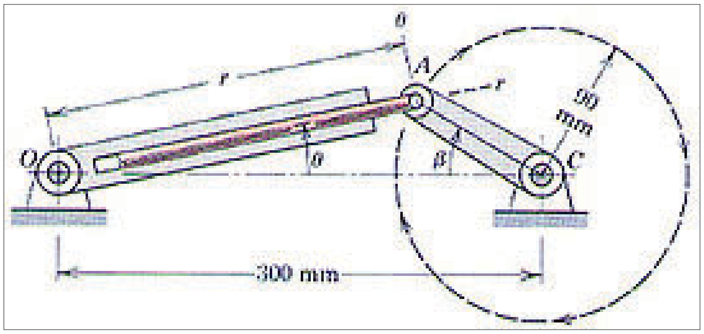
Use Adams/View to

- Create a revolution
- Create a Point-to-Point measure
- Create a measure about an axis
- Create an angular velocity measure about an axis
- Create an angular acceleration measure about an axis

## Software Version

Adams 2013.2

# Problem Description




Pin A moves in a circle of 90-mm radius as crank AC revolves at a constant rate beta-dot = 60 rad/s. The slotted link rotates about point O as the rod attached to A moves in and out of the slot. For the position beta=30 degrees, determine r-dot, r-double dot, theta-dot, theta-double dot.

This problem asks for the translational speed and acceleration of the slider rod and the angular speed and acceleration of the slider assembly at a given crank angle of 30 degrees and crank angular velocity of 60 radians per second. To solve this, we will build an ADAMS model of the crank and slider assembly based on the information given and measure the data we want using an ADAMS simulation of the model.

*Problem 2/163 from J. L. Meriam and L. G. Kraige, Engineering Mechanics: Volume 2, Dynamics 3rd edition. John Wiley & Sons, Inc.*



### Step 1. Creating the Model

a. Start **Adams/View**

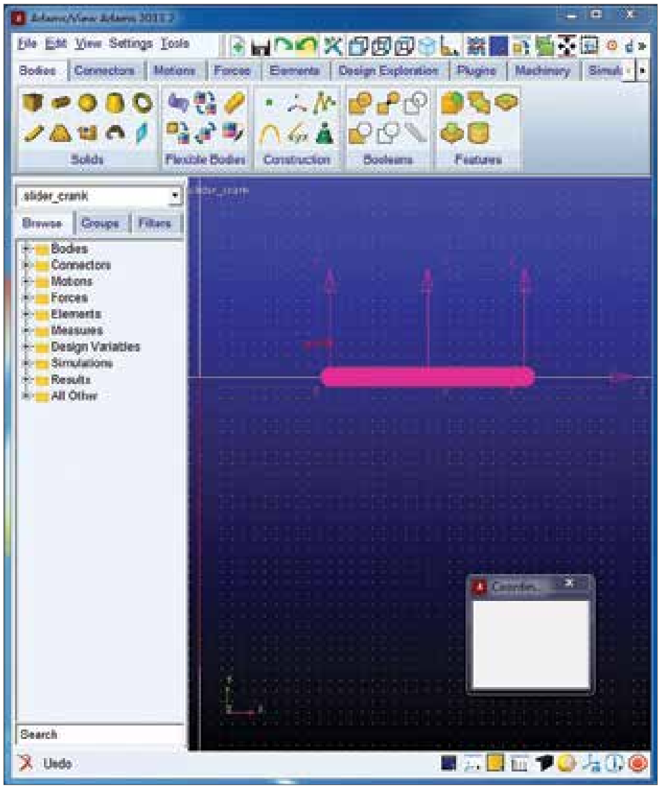
b. Create a new model. **(Model Name = slider_crank, Units = mmks, Gravity = -y earth)**

c. Resize the working grid, **Size = X – 375mm, Y – 250mm, Spacing X – 5mm, Y – 5mm**

d. Open **Coordinate Window**

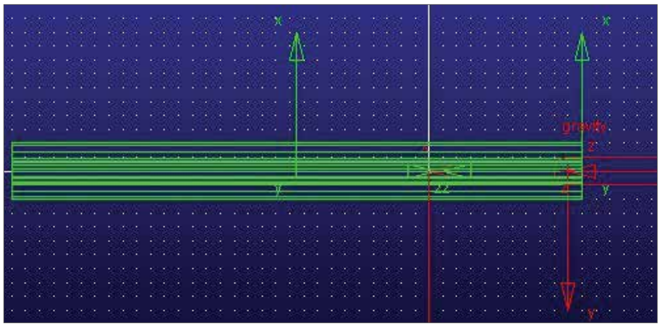
e. Create crank part from point **(60, 0, 0) to (150, 0, 0)**

f. Rename **.slider_crank.crank**



## Step 2. Creating Revolution

a. Select **Rigid Body:Revolution**
b. Click points: **(55 0 0), (-150 0 0), (55 -5 0), (55 -10 0), (-150 -10 0), (-150 -5 0), (55 -5 0)**
c. Right-click to close
d. Rename **.slider_crank.cylinder**



## Step 3. Creating Joints

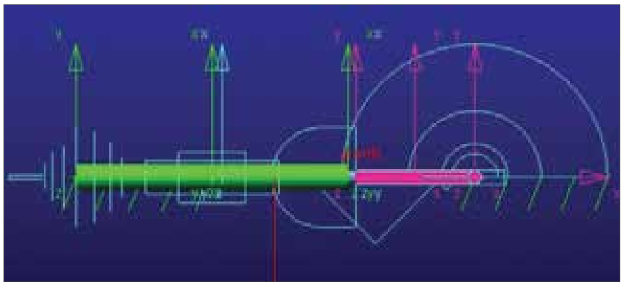
a. Select **Rigidbody:Cylinder**
b. Create piston part. **(cylinder, length = 200 mm, radius = 5 mm, from (60, 0, 0) to (-140, 0, 0)),**
c. Rename **.slider_crank.piston**

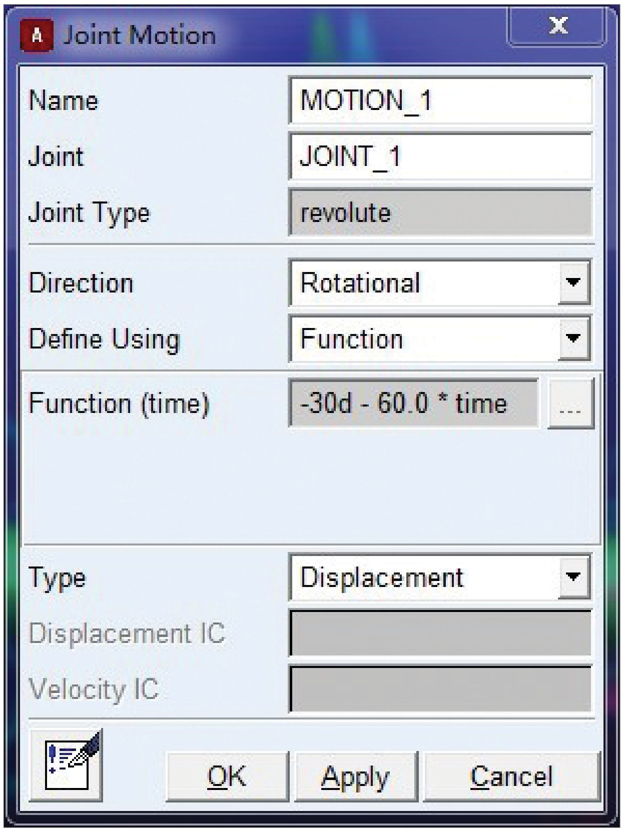
d. Create **revolute joints** between **crank** and ground

e. Create **spherical joint** between **cylinder** and ground

f. Create **translational joint** between **piston** and **cylinder**.

g. Create **Hooke joint** between **crank** and **piston**
h. Add **Rotational joint motion** to revolute joint with **function = -30deg - 60 * time.**



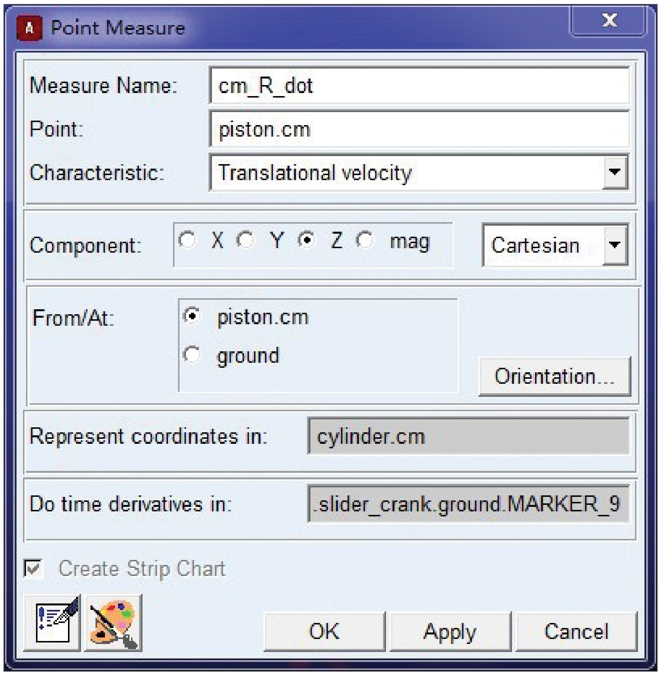


## Step 4. Create Point-to-Point Measure

a. From **Design Exploration** ribbon, select **Point-to-Point** Measure
b. Select **Displacement** as **Characteristic**
c. Select **GloabalZ** as **Component**.
d. Select the **Marker** at the left end of **cylinder** and the **Marker** at the left end of **crank**
e. Rename it as **MEA_PT2PT_R**

## Step 5. Create Point Measure

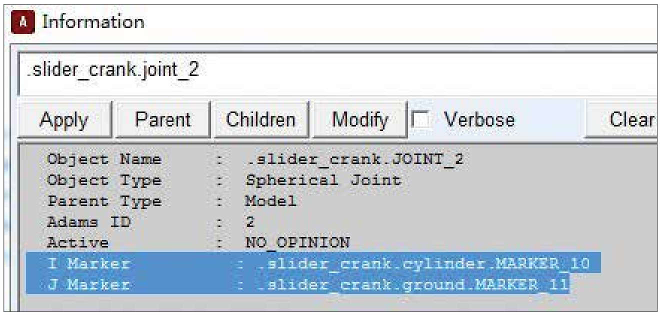
a. Under the **piston** tree in the **Model Browser,** right-click cm and select **Measure**
b. Select **Translational Velocity** and select **Z Component.**
c. Enter **cylinder.cm** as **Represent coordinates**
d. Select any **Marker** belongs to the ground as **DO time derivatives in.**
e. Repeat the above steps to create a **translational acceleration.**

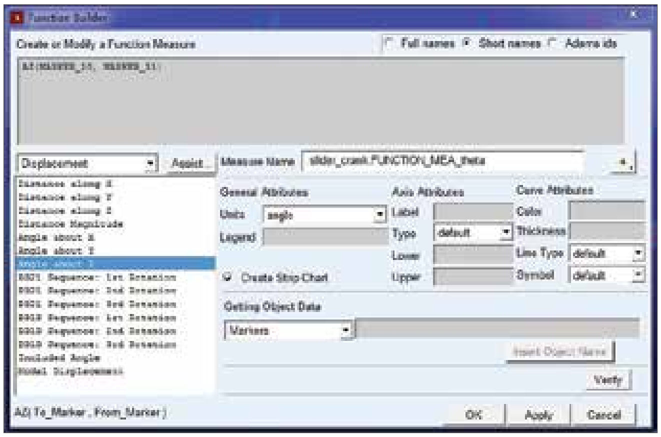



## Step 6. Create Angle about Axis Measure

a. In the **Bodies** tree, right-click the **spherical joint** between **cylinder** and the ground
b. Select **info** and remember the name of **I Marker** and **J Marker.**
c. Close the **info** window.
d. Select **Function Measure**

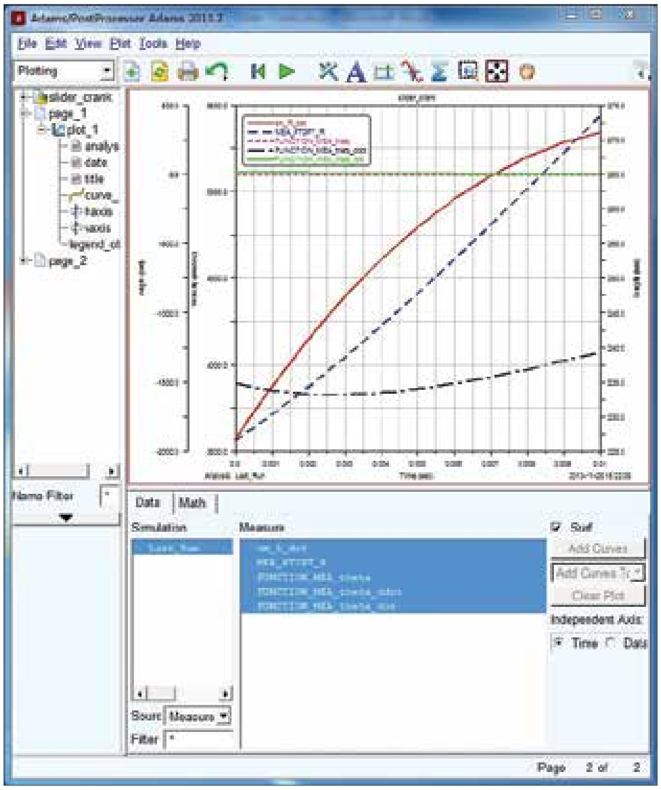
e. Select **Angle about Z** under **Displacement** and enter the marker name in **Step b**.
f. Select **angle** as **units**



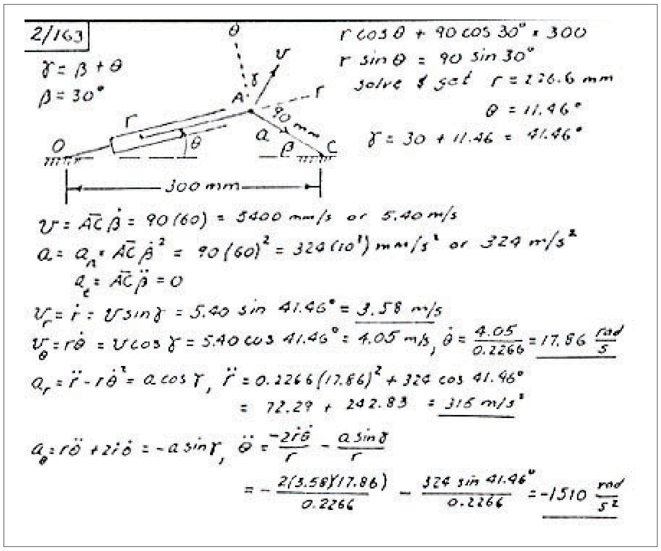
## Step 7. Testing the Model

a. From **Simulation** ribbon, select **Run an Interactive Simulation**
b. Set **End Time** to **0.01** and **Step Size** to **0.001,** and then click **Start**
c. Click **Plotting**
d. Use the **Plot tracking tool**
e. Follow the plot curve. Find the size measurement at X = 0.0



## Results

Theoretical Solution




ADAMS solution

r = 2.266m

r-dot = 3.58 m/s,

r-double dot = 316 m/s^2,

Theta = 11.46deg

theta-dot = 17.86 rad/s,

theta-double dot = -1510 rad/s^2

# Example 14: Controls Toolkit in ADAMS/View

## Workshop Objectives

Use Controls Toolkit in Adams/View

- Create an input-signal block
- Create a summing-junction block
- Create a gain block
- Modify torque function

## Software Version

Adams 2013.2

## Files Required

- **Lift_Mechanism_start.cmd**
- Located in the directory **exercise_dir/Example 14**

## Problem Description

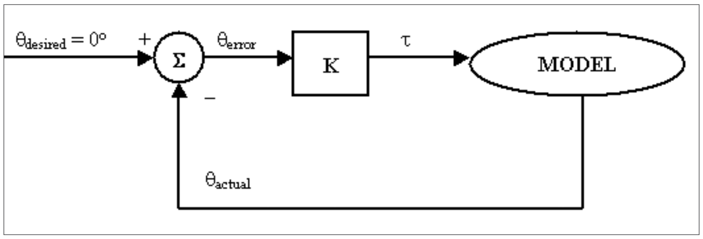
This example provides a simple introduction to the Controls Toolkit that is integrated into ADAMS/View. This example closely follows the process outlined in the 'Using the Control Toolkit' section of the ADAMS/View guide. Model consists of two moving parts, one imposed motion, and one single-component torque.

Boom - is constrained to the ground with a Revolute Joint and a Joint Motion that makes it oscillate.

Bucket - is constrained to the Boom with a Revolute Joint. There is also a TORQUE between the Bucket and the Boom that has a magnitude of 0 right now. This is where we will be giving the output of our controls blocks.

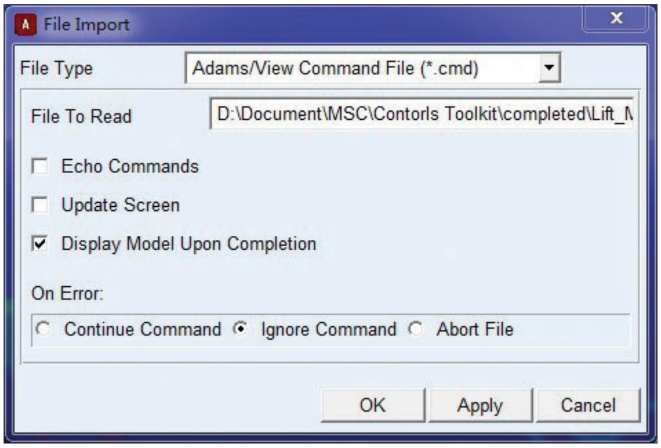
Notice as you run a simulation the Boom rotates according to the function on the joint motion, while the Bucket just randomly oscillates. We are going to use the Controls Toolkit to keep the bucket at a horizontal orientation with respect to the ground.

Our Controls Block Diagram will look like this:




### Step 1. Importing a command file (.cmd)

a. Start with **New Model**
b. Select **File**, and then select **Import**.
c. Right-click **File To Read** text field, select **Browse**
d. Locate saved file **Lift_Mechanism_start.cmd**
e. Click **Open**
f. Click **OK**



## Step 2. Creating Input-Signal Block

a. From **Element** ribbon, click **Controls Toolkit** 

b. Click **input-signal** block tool 

c. Enter **.Lift_Mechanism.theta_desired** in **Name** text field

d. Enter **0.0** in **Function** text field, and then click **Apply**

e. Click **input-signal** block tool again

f. Enter **.Lift_Mechanism.theta_actual** in **Name** text field

g. Click **Function Builder** button 

h. Select **Displacement** from pull down arrow

i. Click **Angle about Z**, and then click **Assist**

j. Right-click in **To Marker** text field click **Marker → Browse**

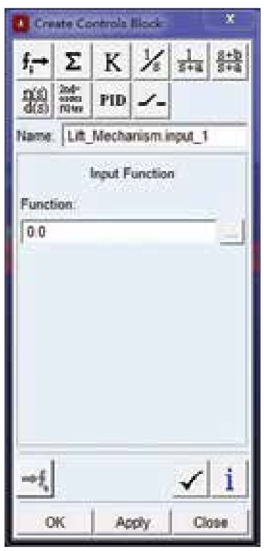
k. Click **Torq_I_mar,** and then click **OK**

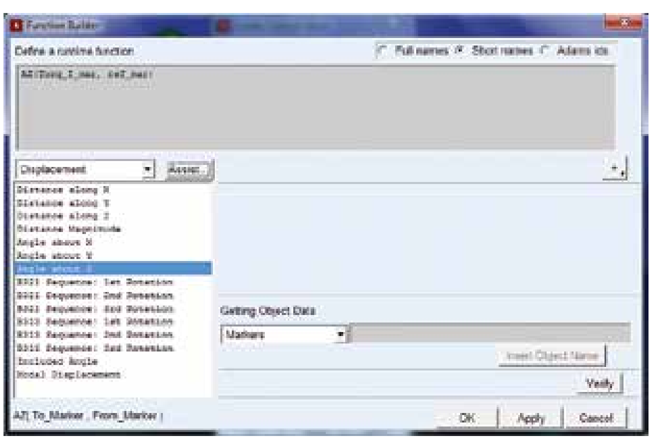
l. Right-click in **From Marker** text field click **Marker → Browse**

m. Click **ref_mar**, and then click **OK**

n. Make sure the **Define a runtime function** text field reads **AZ(Torq_I_mar, ref_mar)**

o. Click **OK**




## Step 3. Create a Summing-Junction Block

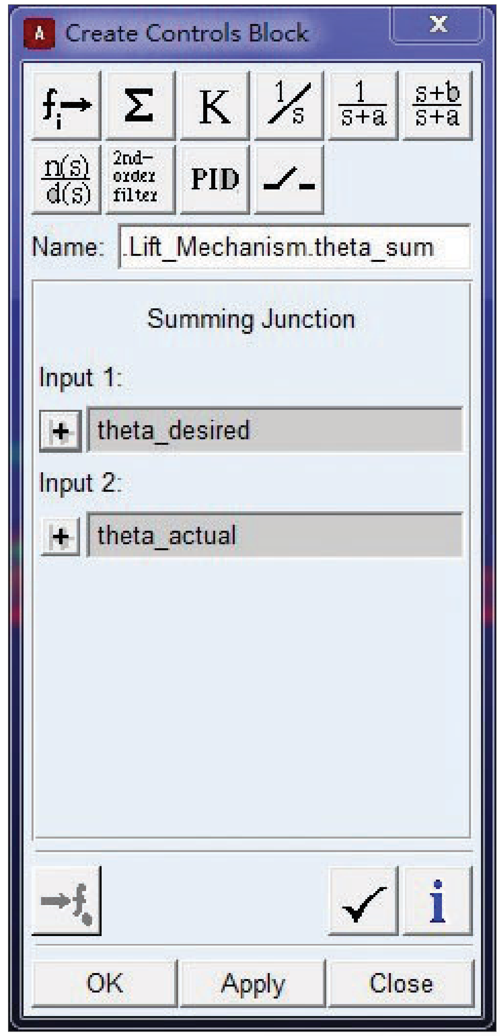
a. Click **summing-junction** block tool 

b. Enter **.Lift_Mechanism.theta_sum**

c. Right-click in Input 1 text field, select **controls_input → Guesses → theta_desired**

d. Right-click in Input 2 text field, select **controls_input → Guesses → theta_actual**

e. Click **OK**




## Step 4. Create a Gain Block

a. Click **gain** block tool 

b. Enter **.Lift_Mechanism.theta_gain**

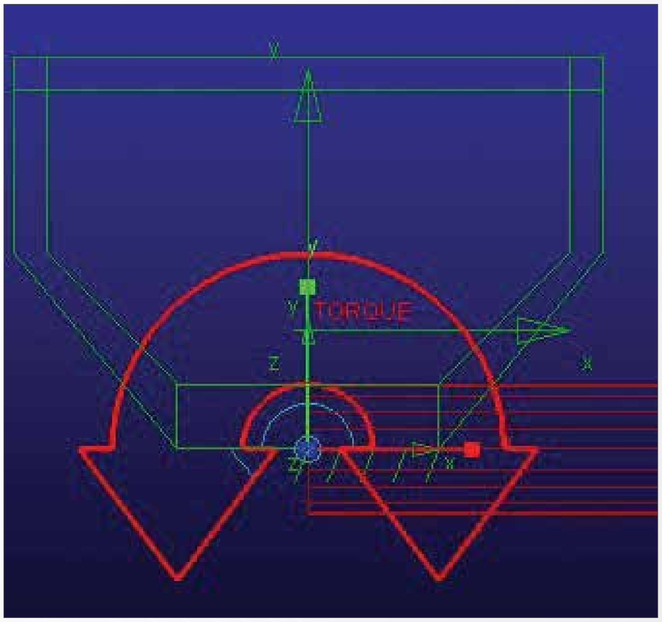
c. Right-click in **Input** text field, select **controls_sum → Guesses → theta_sum**

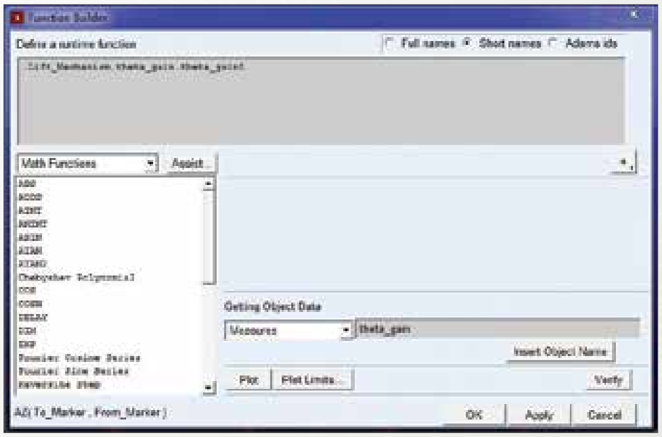
d. Enter **1e9** in the **Gain** text field

e. Click **OK**

## Step 5. Modify Torque Function

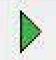
a. Right click on torque icon, select **Torque: TORQUE → Modify**
b. Click **Function Builder** button next to **Function** text field
c. Select **Measure** from **Getting Object Data** pull down arrow
d. Right click in text field, select **Runtime_Measure → Guesses → theta_gain**
e. Click **Insert Object Name**
f. The name of the measure should appear in the editor above
g. Click **OK**

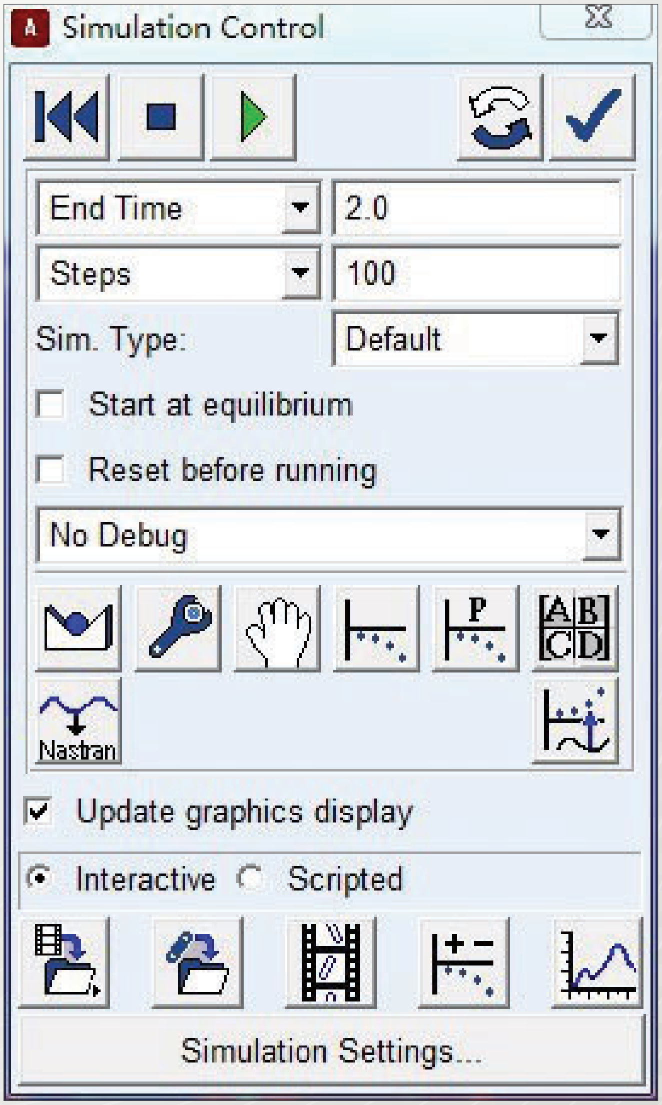



## Step 6. Verify and run Simulation


a. Click **Simulation** tool
b. Click **verify**
c. Make sure there are no redundant constraints and only 1 Degree of Freedom
d. Click **Close**
e. Select **Duration** from pull down menu, and enter **2**
f. Select **Steps** from pull down menu, and enter **100**
g. Click **Play** button



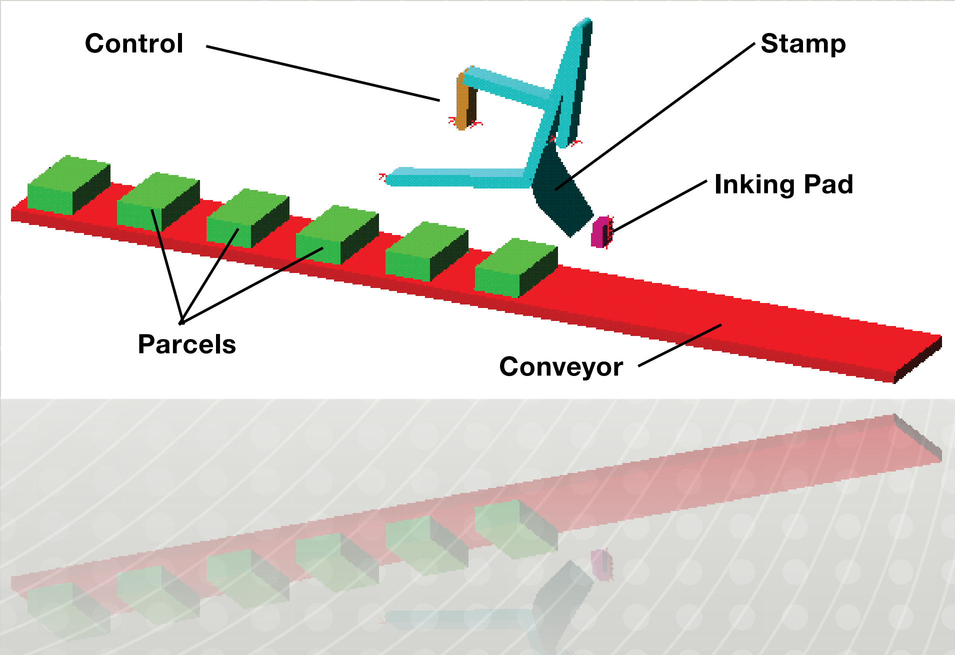


# Section II: Intermediate Level

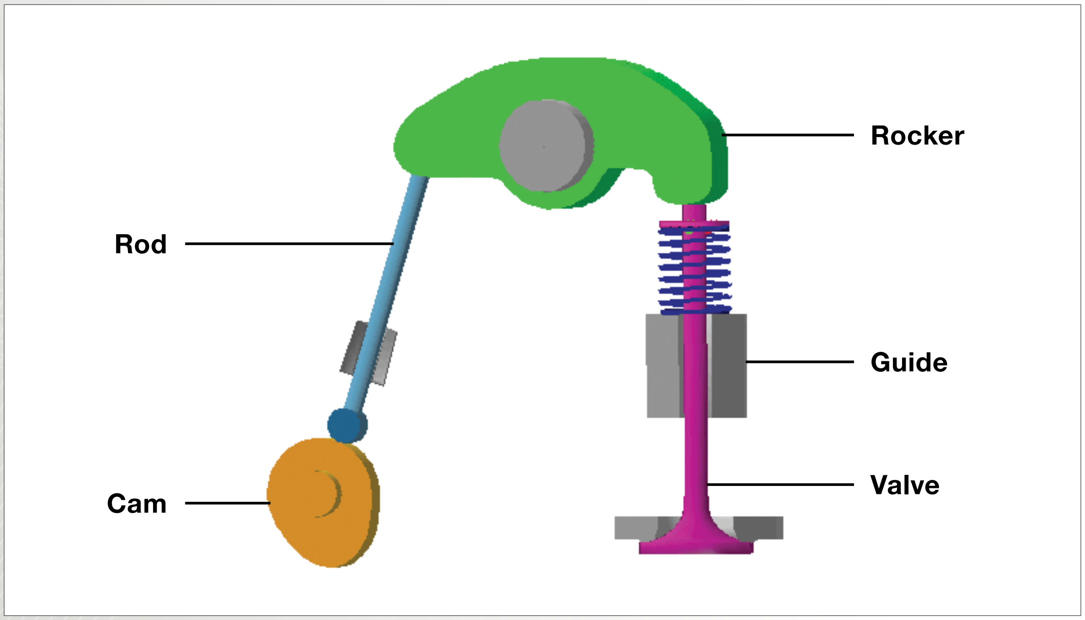
*In this section, you'll work on four more complex Adams examples compared to the first section.*

*The purpose of this section is to reinforce what you have learned in Section I. If you are an experienced Adams user, you can start from this section to get familiar with the new interface and to learn some more advanced skills in Adams/View, for instance:*

- *How to create contacts*
- *How to use function measurement*
- *Optimization analysis*



# Example 15: Valvetrain Mechanism




## Workshop Objectives

Use Adams/View to manipulate, inspect, simulate, and animate the valvetrain mechanism.

## Software Version

Adams 2013.2

## Software Version

- **valve.cmd**
- Located in the directory **exercise_dir/Example 15**

# Problem Description

- The model represents a valvetrain mechanism.
- The cam is being rotated at a given velocity.
- The rod (follower) moves translationally based on its constraint to the cam.
- The rocker pivots about a pin attached to the engine block.
- The spring is always in compression to try and keep the rod in contact with the cam.
- The valve moves vertically as the rocker rotates.
- When the valve moves, it lets small amounts of air into the chamber below it (not modeled here).

## Step 1. Import File

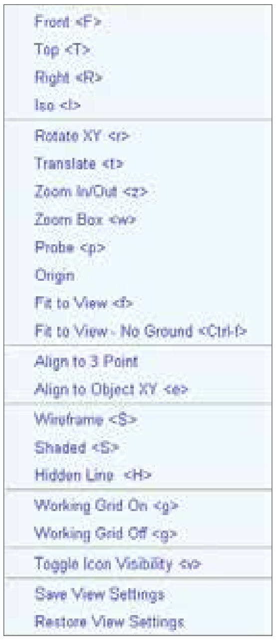
To import a file.

a. Start Adams/View.
b. From the Welcome dialog box, select **Existing Model.**
c. Click the **file folder** icon, and the Select Directory dialog box appears.
d. Find and select the directory **Exercise_dir/mod_2_aview_interface.**
e. Click **OK**.
f. Click on the file folder icon of the **File Name**, select the **file valve.cmd** and click **Open**.
g. Click **OK** on the **Open Existing Model** dialog box.

## Step 2. View the List of Keyboard Shortcuts

To view the list of keyboard shortcuts:

a. Move the cursor away from the model and then right-click in the Adam/View window. A menu appears listing the keyboard shortcuts.
b. To close the menu, left-click away from the menu.
c. In the space below, write the shortcut keys for performing the following view operations.
   a. Rotate: ..............................
   b. Translate: ..............................
   c. Zoom with a box: ..............................
   d. Zoom into a specific Area: ..............................
   e. Fit: ..............................
   f. Front View: ..............................



## Step 3. Use the Zoom Box Shortcut

a. To use the zoom Box shortcut:

b. Zoom into the cam area by using the shortcut <w>.

c. Notice the instructions in the status bar instruct you to select the area.

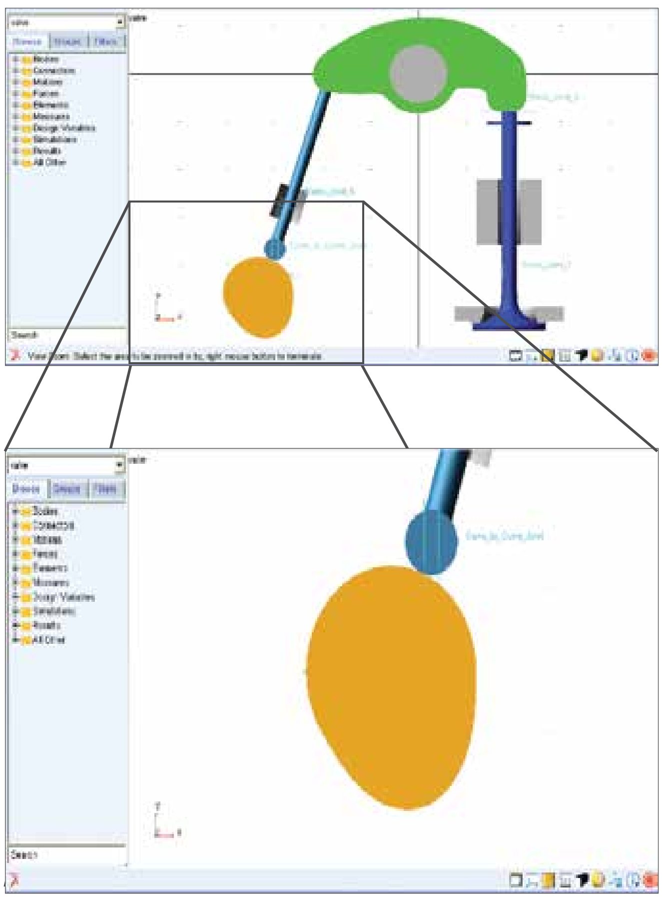
d. Click the left mouse button in the place where you want the top left corner of your zoomed in rectangle to be.

e. Now the status bar instructs you to: drag to select size of view.

f. Draw a rectangular box around the cam.

g. You should now be zoomed into the cam area.

h. Use the fit shortcut <f> to return to the original view.



## Step 4. View the Model from Different Angles

To view the model from the top:

a. Use the Top shortcut <T> and the view changes to a top view.

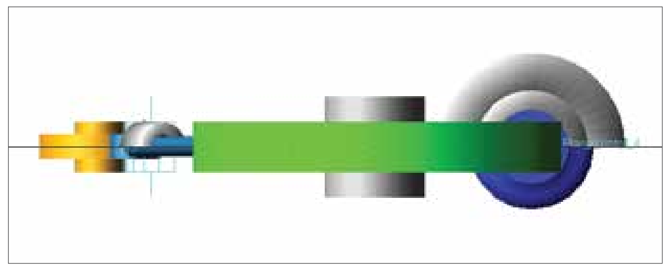
To view the model from the right:

b. Use the Right shortcut <R> and the view changes to the right view.

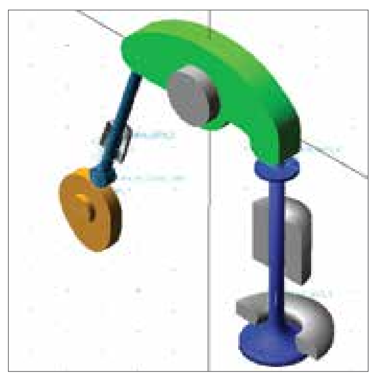
To view the model in an isometric view:

c. Use the Iso shortcut <I> and the view changes to an isometric one.

If you wish you may continue to try the other shortcut keys.



**Top View**



**Isometric View**

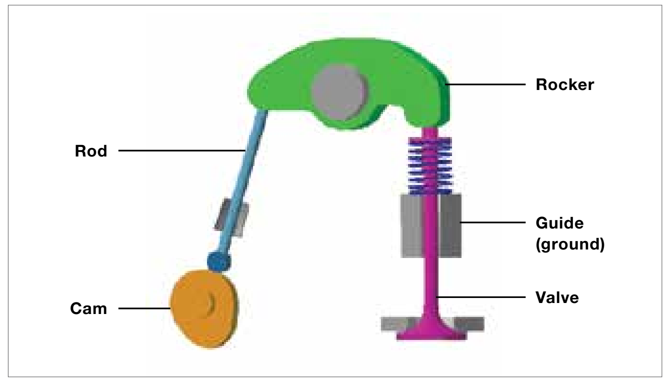
**Right View**

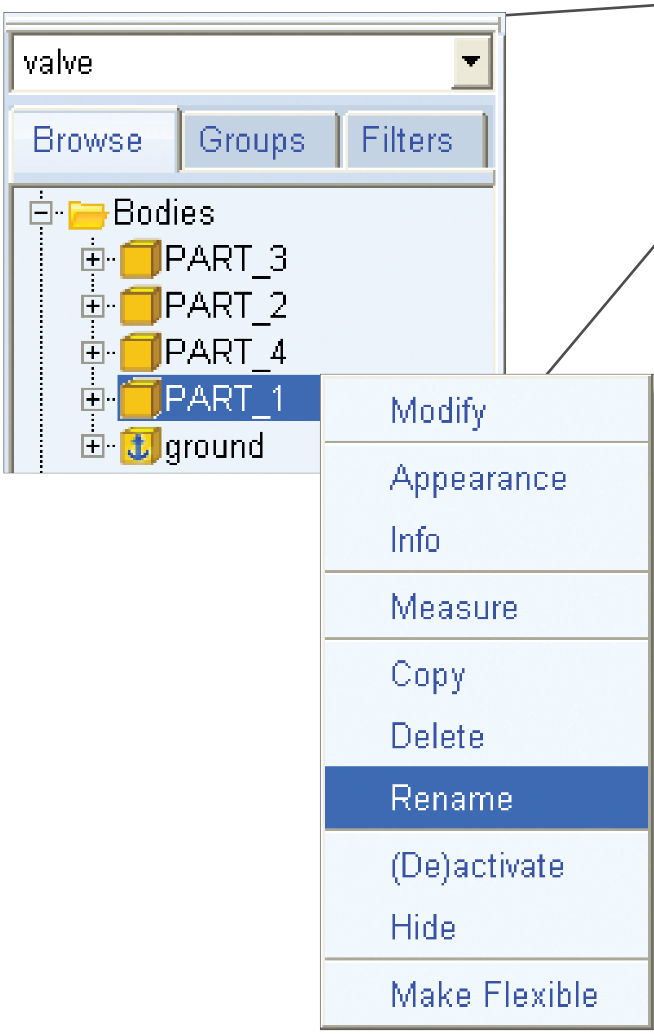
## Step 5. Rename the Parts

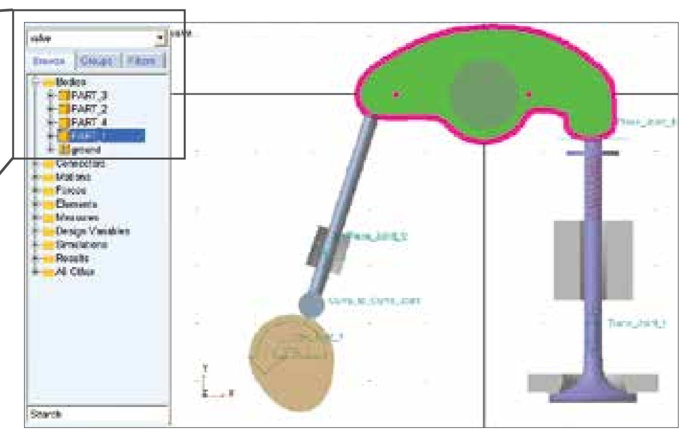
As you go through these instructions notice that right-clicking always give you a list of choices while left clicking selects an object.

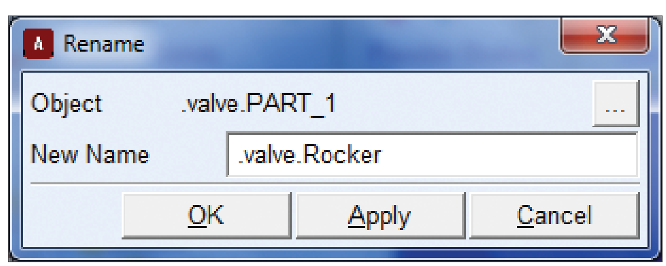
To rename the parts to match the ones given in the diagram to the right:

a. From **Model Browser**, select the part displayed under the Bodies tree. Same part will be selected and highlighted.
b. Right click and select **Rename** from the displayed menu.
c. In the **Rename** dialog box, change the name according to the given diagram.
d. Click **OK** to change the part name.
e. Repeat the above steps a through e for the Rod, Cam, Guide, and Valve.



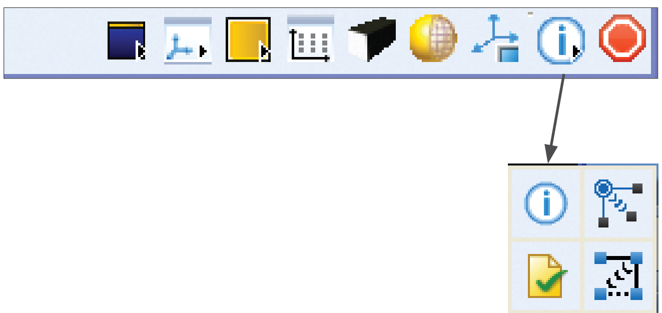



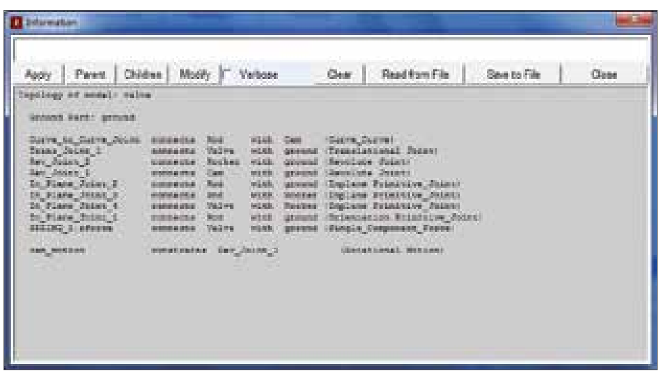


## Step 6. Inspect the Model

To inspect the model to determine the number and type of constraints:

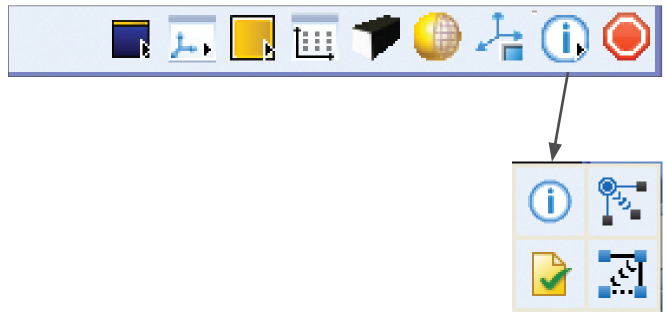
a. Right-click the small arrow on the Information tool stack on the right side of the Status Bar at the bottom of the screen.
b. Select the **Model topology by constraints** tool.
c. From the Information window that appears, note the number and type of constraints and use them to answer Question 1 in the **Workshop 2, Review** section, page WS2-19
d. Close the Information window.

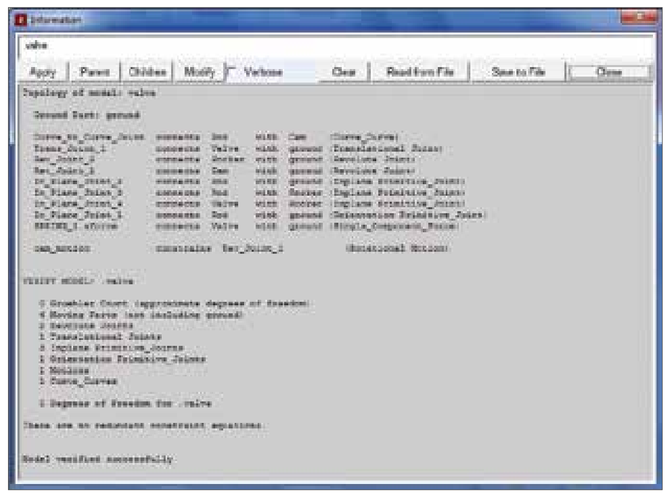





To inspect the mode to check if the model verified successfully:

e. Right-click the small arrow at the bottom of the information tool stack.
f. Select the **verify** tool.
g. From the Information window that appears, notice that the model verified successfully.
h. Close the Information window.

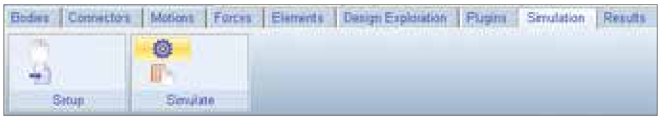


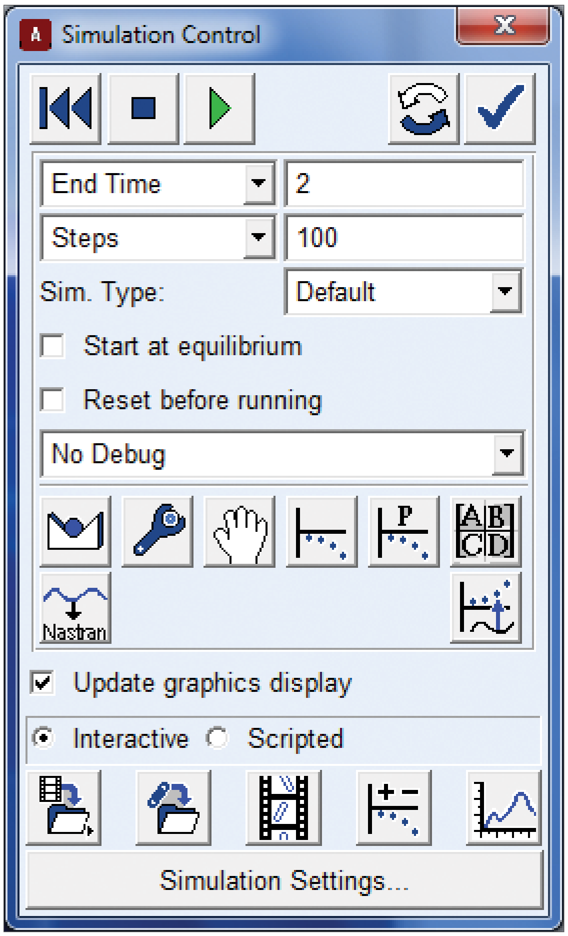


## Step 7. Simulate the Model

To run a simulation:

a. Select the ribbon **Simulation**.
b. From the options available select **“Run an Interactive Simulation.”**
c. In the **Simulation Control** dialog box select **End Time.**
d. In the text box adjacent to End **Time**, enter **2**.
e. In the text box adjacent to **Steps** enter **100**.
f. Click on the Play tool.
g. When the simulation is complete, click the **Reset** tool.

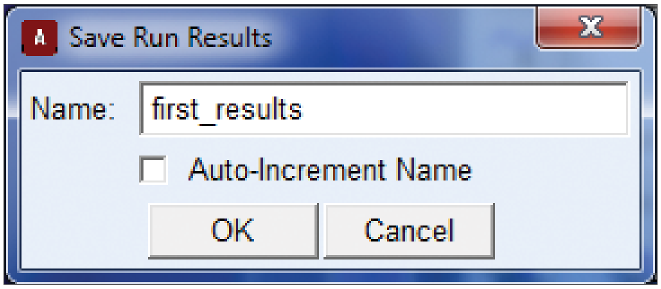


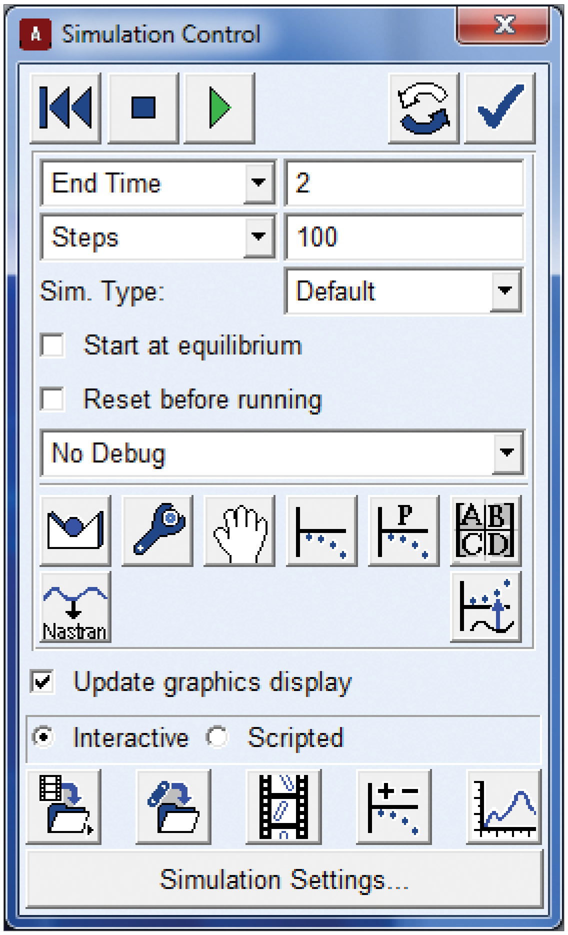




## Step 8. Save the Simulation

To save the simulation:

a. To save the last simulation results to the database under a new name, select the **Save simulation** tool. The **Save Run Results** dialog then appears
b. In the **Name** text box, enter a name for the simulation results, such as **first_results**.
c. Click **OK**.
d. Close the Simulation Control dialog box.



## Step 9. Animate the Results

To animate the results in the default mode with icons off:

a. Switch to **Animation Controls** from **Simulation Control.**

b. To see the animation, click the **Play** button.

c. When the animation is complete, click the **Reset** tool.

d. To see the animation in incremental steps click either the **+Inc** to move forward or the **-Inc** to rewind the animation.

e. The step number will be listed in the center between these two buttons.

f. When finished, click the **Reset** tool.

To animate the model with icons turned on:

g. At the bottom of the Animation Controls dialog box, check **icons**.

h. Repeat the step from b. to f.

i. Close the Animation Controls dialog box.

## Step 10. Save Your Work

To save your work so that the saved file contains only the model information:

a. From the File menu, select **Export**.

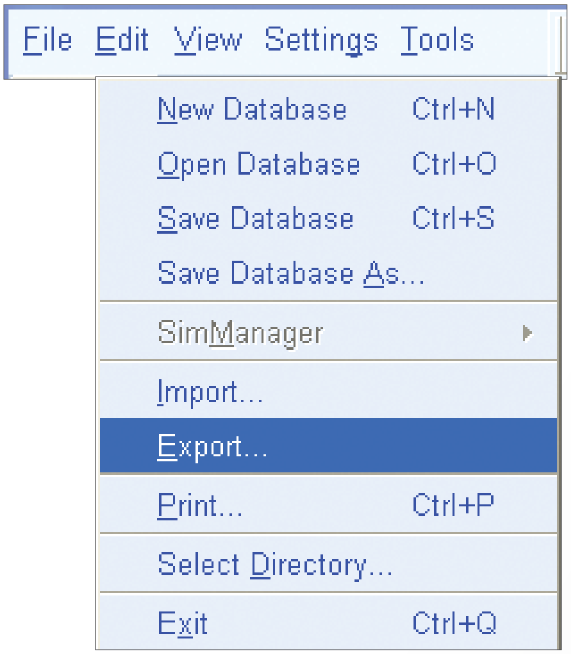
b. Set File Type to **Adams/View Command File**.

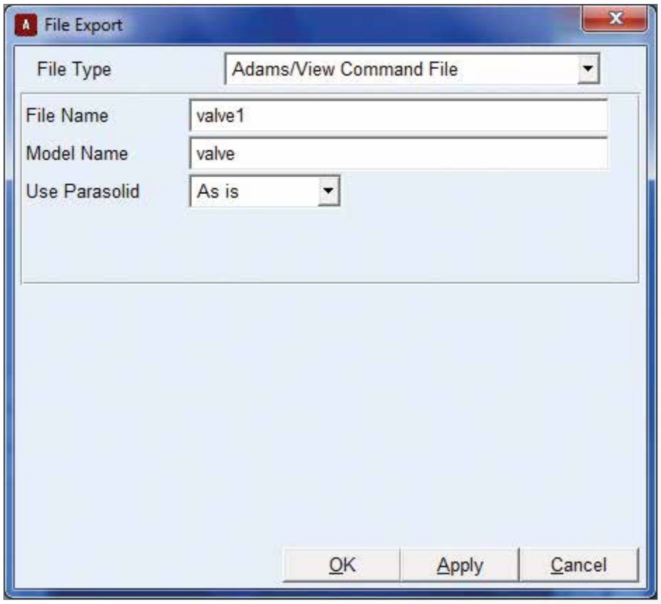
c. In the **File Name** Text box, enter **valve1**.

d. In the **Model Name** text box, enter **valve**.

e. Click **OK**.

Since this is the last step for the workshop, you may manipulate the model and experiment with it as time permits.



## Workshop Questions

How many constraints are there in this system? What type of constraints are they?

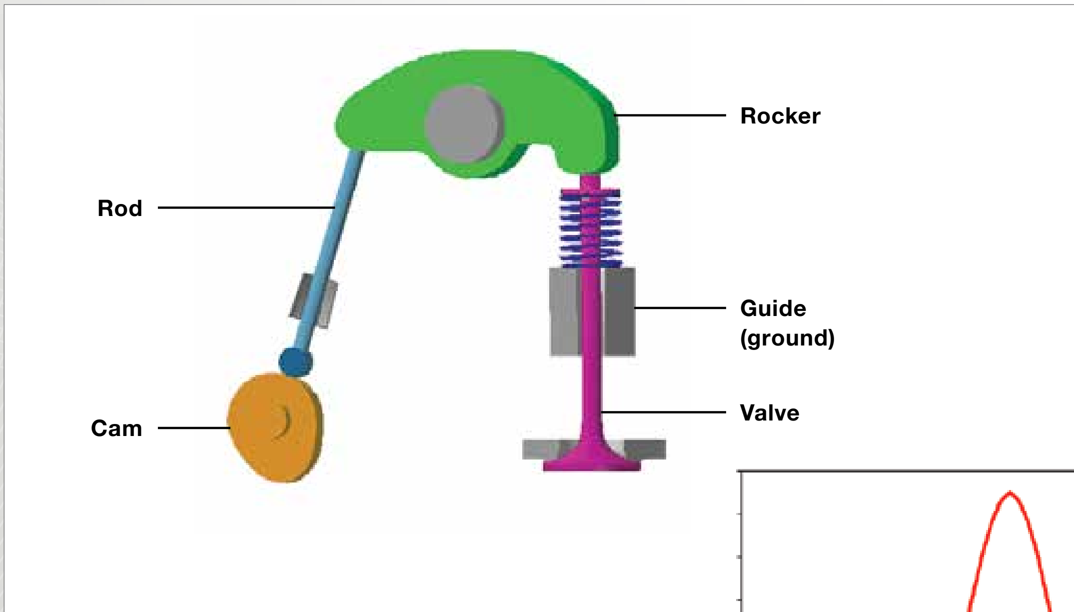
Is it possible to have more than one model in a database?

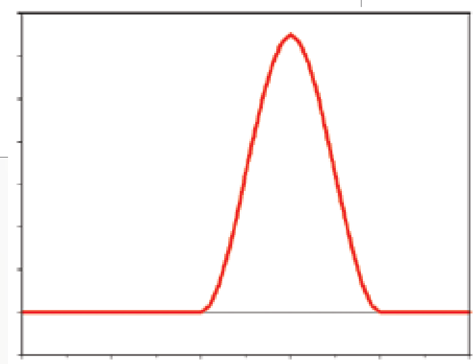
Is geometry a direct child of a model? If not, what is geometry a child of?

If you are in the middle of an operation and you are not sure what input Adams/View wants next, where should you look?

If you are working with our technical support staff and you want them to look at one of your files, what file format would you send them, a .cmd or .bin? Why?

# Example 16: Cam-rocker-valve




## Software Version

Adams 2013.2

## Files Required

- **valve_train_start.cmd**
- **Located** in **exercise_dir/Example 16**

## Workshop Objectives

Design a cam profile based on desired valve displacement, and ensure that there is no follower liftoff when the cam is rotated at 3000 rpm.

## Problem Description

- The model represents a valvetrain mechanism.
- The cam is being rotated at a velocity of 1 rotation per second.
- The rocker pivots about a pin attached to the engine block (ground).
- The valve displaces up and down as the rocker moves.
- When the valve moves, it lets small amounts of air in the chamber below it (not modeled here).

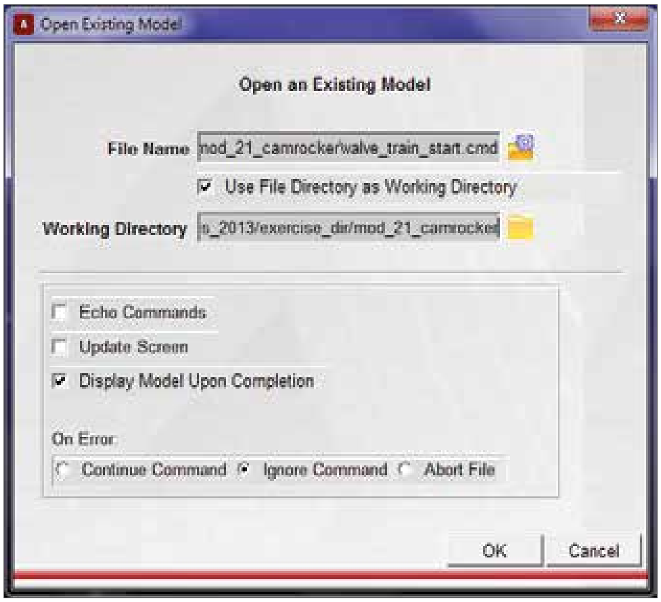
### Step 1. Import File

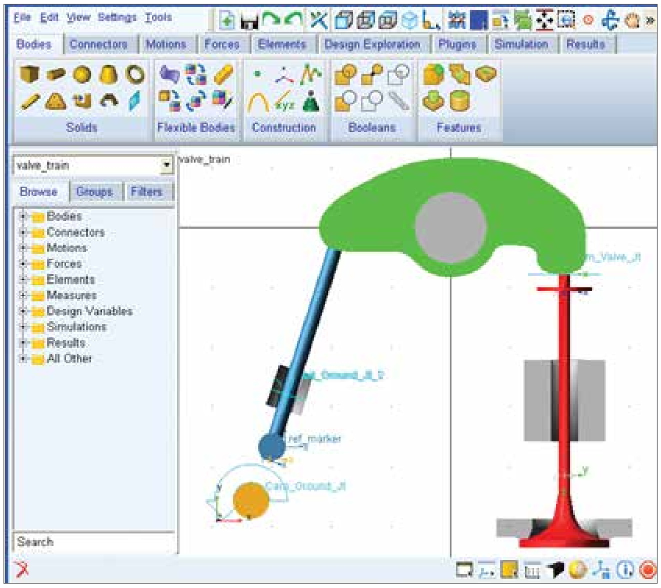
To import a file.

a. Open Adams/View from the directory **exercise_dir/ Example 16.**

b. From the directory **exercise_dir/Example 16**, import the model command file **valve_train_start.cmd.**

c. The file contains a model named **valve_train.**



## Step 2. Apply a Motion

a. From the ribbon **Motion** select **Translation Motion** tool to add a motion to the joint, **Valve_Ground_Jt.**

b. Use the STEP function below to define the displacement. Add the two STEP functions together such that the final function looks as follows:

   a. **STEP(time, .4, 0,.6,13) + STEP(time,.6,0,.8,-13).**

   b. Enter this function in the **Function(time)** textbox, on the Joint Motion dialog.

c. From ribbon **simulation**, select **Interactive Controls.**

d. From the **simulation control** Run a **1-second, 100-step** simulation to verify that the valve displaces as a result of the joint motion.

## Step 3. Create a Cam Profile

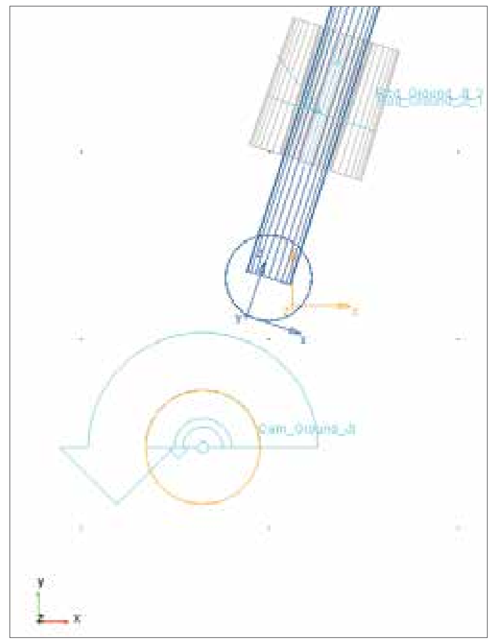
Use a point trace to create a cam profile:

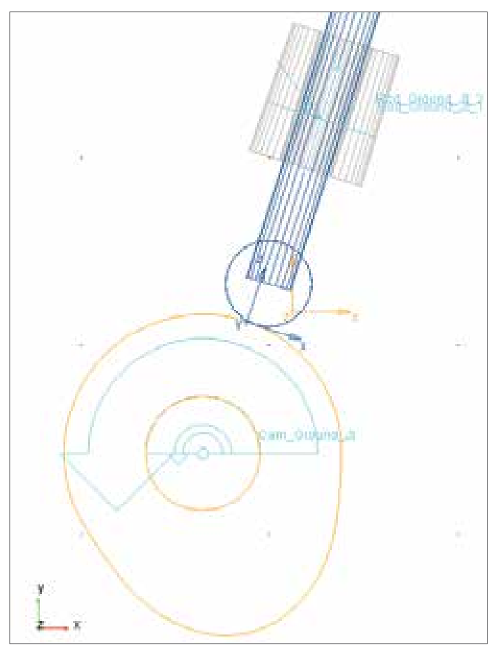
a. To use a point trace: From the ribbon **Results**, select **Create Trace Spline.**

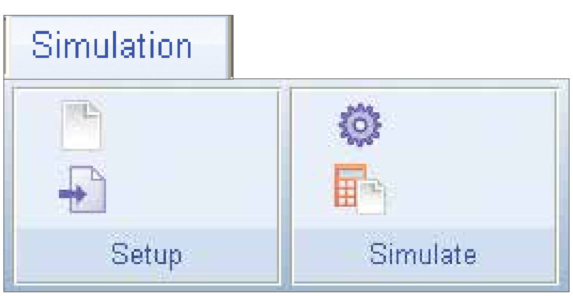
b. Select the circle on the rod, **rod.CIRCLE_1** and then the part named **cam**.

c. Verify that you now have a spline representing the cam profile.

d. Run a simulation to verify that the **Rod** appears to move along the surface of the **Cam**.





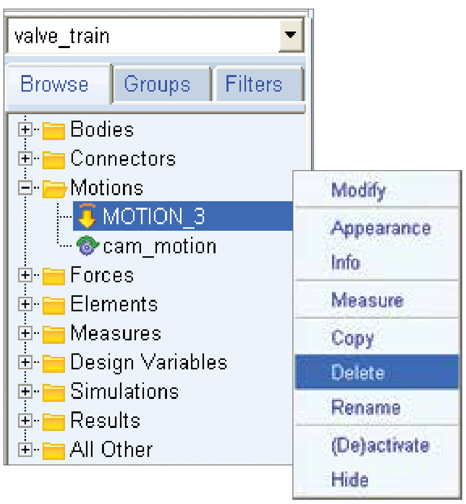
## Step 4. Constrain the Rod to the Cam

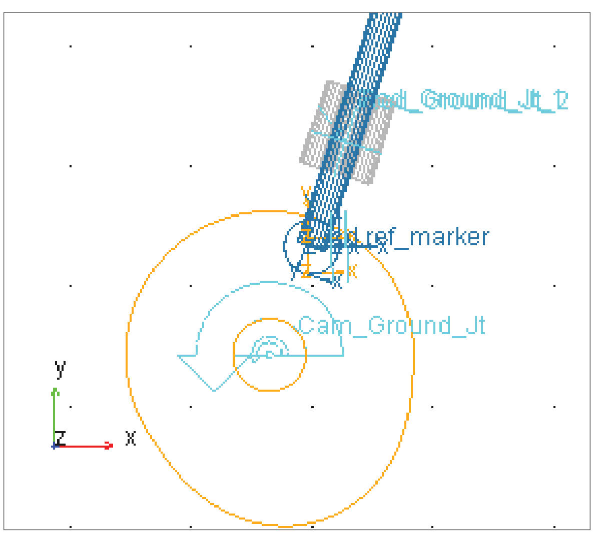
To constrain the rod:

a. Delete the joint motion on the joint, **Valve_Ground_Jt.**

b. From the ribbon **Connectors**, select **Curve-Curve Constraint** tool to create a curve-on-curve constraint between the circle on the **Rod** (CIRCLE_1) and the cam profile on the **Cam.** (GCURVE_232) Note that the number may vary.

c. Run an **interactive simulation** to verify that the new constraint works.


## Step 5. Measure the Force

Measure the force in the curve-on-curve constraint. To measure the force:

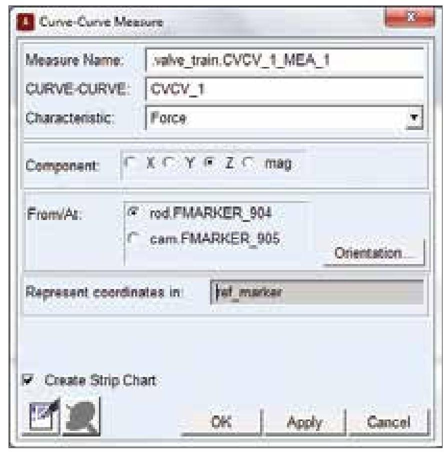
a. Create a force measure for the curve-on-curve constraint. Right-click the constraint and then select **Measure**.

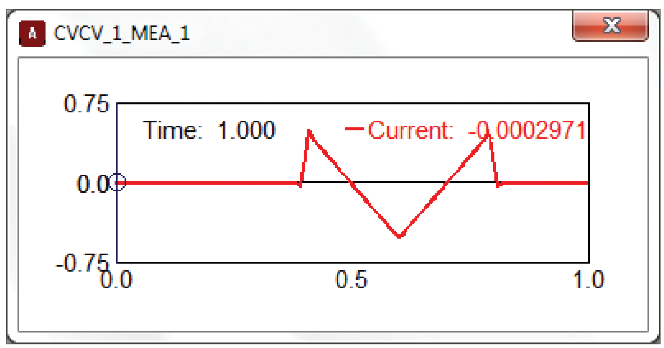
b. Measure the force along the z-axis of **ref_marker**, which belongs to the rod:

- **Characteristic: Force**
- **Component: Z**
- **Represent coordinates in: ref_marker**

c. A strip chart for the measure will be displayed.

**(Note:** The curve-on-curve constraint applies a negative force that keeps the rod follower on the cam, avoiding any liftoff.)

## Step 6. Replace the Curve-On-Curve Constraint

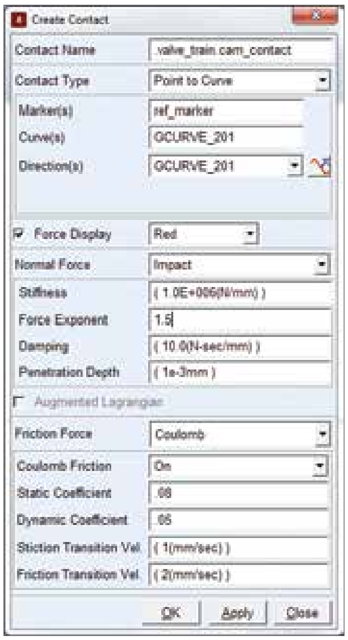
Make the cam-to-rod contact more realistic by replacing the curve-on-curve constraint with a Point-to-curve contact force. To replace the curve-on-curve constraint:

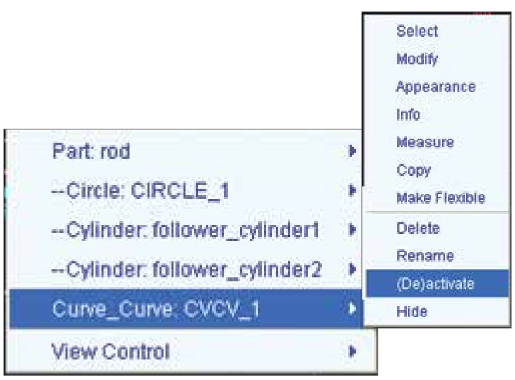
a. Deactivate the curve-on-curve constraint you created in Step 4 on page WS21- 9.

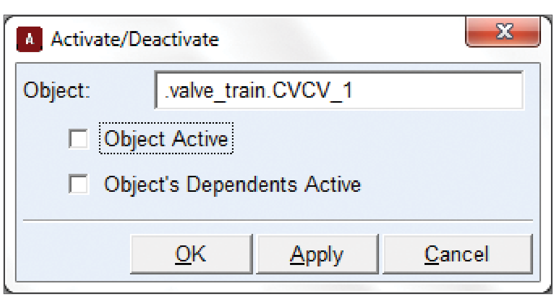
b. From the ribbon **Force**, select **create a contact**.

c. Use the following contact parameters:

- Contact Name:cam_contact
- Contact Type: Point to Curve
- Marker: ref_marker
- Curve: GCURVE_201
- Normal Force: Impact
- Stiffness (K): 1e6 (N/mm)
- Force Exponent (e): 1.5
- Damping (C): 10 (N-sec/mm)
- Penetration Depth (d): 1e-3 mm
- Friction Force: Coulomb
- Coulomb Friction: On



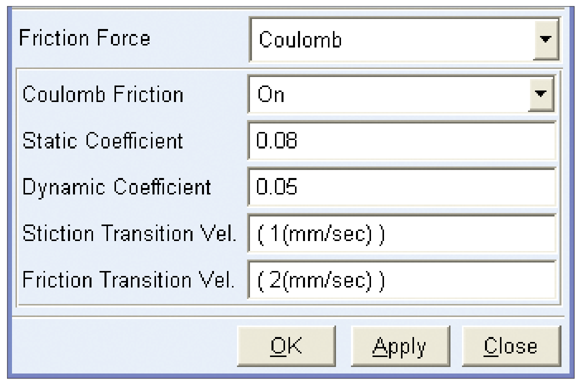



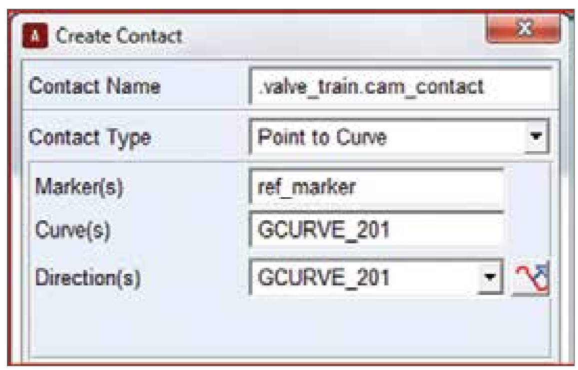

d. Use the following contact parameters continued:

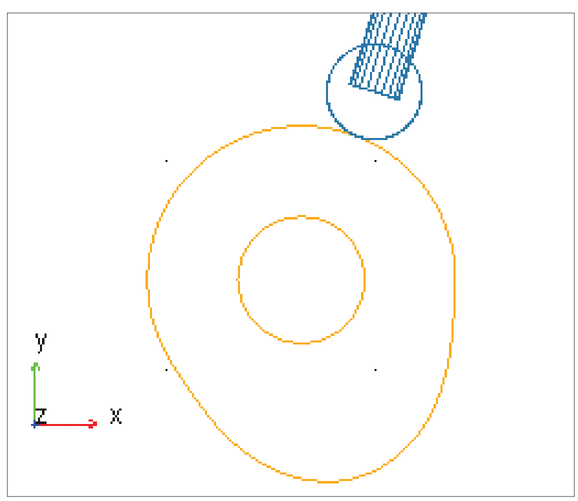
- Static Coefficient (μs): 0.08
- Dynamic Coefficient (μd): 0.05
- Stiction Transition Vel. (vs): 1 (mm/sec)
- Friction Transition Vel. (vt): 2 (mm/sec)

e. Use the **Change Direction** tool next to the Directions textbox, to make sure that the normal arrow points outward from the curve (GCURVE_232) as shown to the right.

f. Run an **Interactive simulation** to check if liftoff occurs.



## Step 7. Create a Spring

Since lift off still occurs, to prevent it create a spring damper:

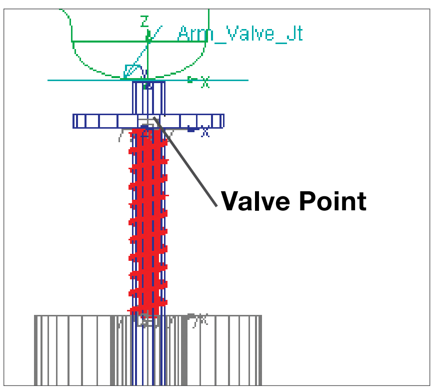
a. To add a marker on the valve at the location, **Valve_ Point:**
From ribbon **Bodies**, select **Construction Geometry: Marker**

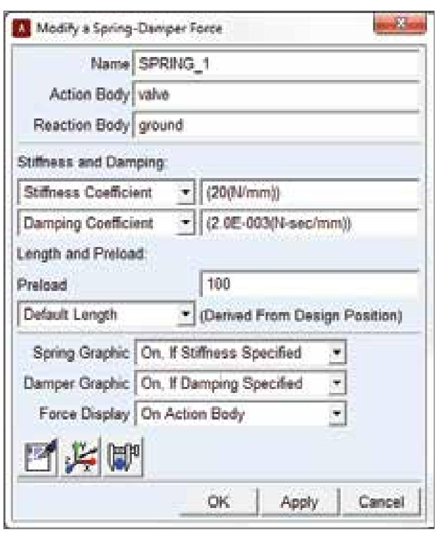
- Add to Part
- From the screen, select valve and the location Valve_ Point.

b. From the ribbon **Forces**, select **create Translational Spring-Damper.** Add a spring damper between the marker you just created and the point, **Ground_Point** (which is a point on ground, at the top of the guide), using the following parameters:

- Stiffness (K): 20 (N/mm)
- Damping (C): 0.002 (N-sec/mm)

c. To add a preload to the spring you must modify the spring, use a pre-load of **100 N.**




## Step 8. Find Static Equilibrium

To find the static equilibrium of the model:

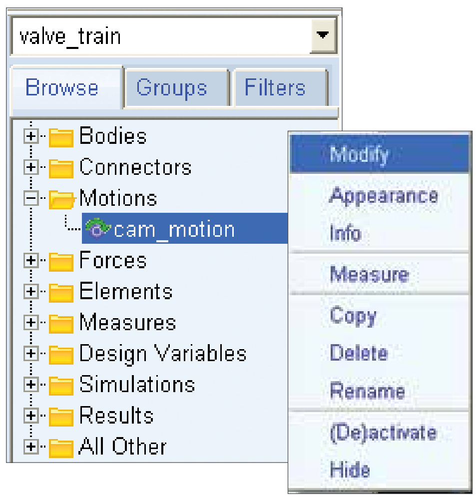
a. From the ribbon **simulation**, select **Interactive Simulation.** Click **Find Static Equilibrium**. Do not reset the model before going on to the next step.

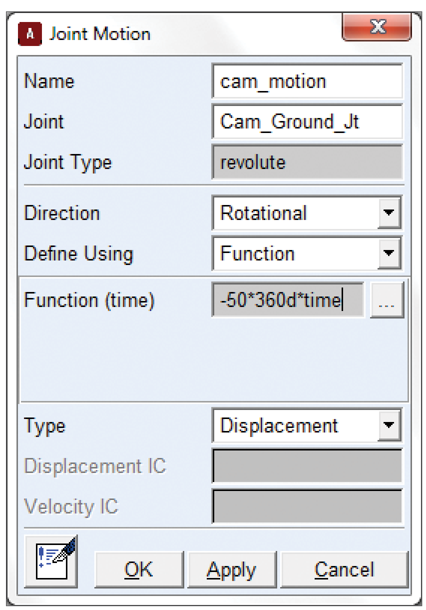
b. Run a dynamic simulation to view the effects of the spring starting from static equilibrium.

c. Modify the rotational motion on the cam.

d. The speed should be 3000 rpm, so enter the displacement function as **-50*360d*time.**

e. To view only one rotation of the cam, run a static equilibrium followed by a dynamic simulation for **end=1/50 seconds, steps=100**. Note: an easy way to run this simulation sequence is to create a simulation script.



## Step 9. Create a Measure on the Contact Force

To create a measure on the contact force:

a. From the ribbon **Design Exploration**, select **Create newFunction Measure**

b. Change the **units** to **force**.

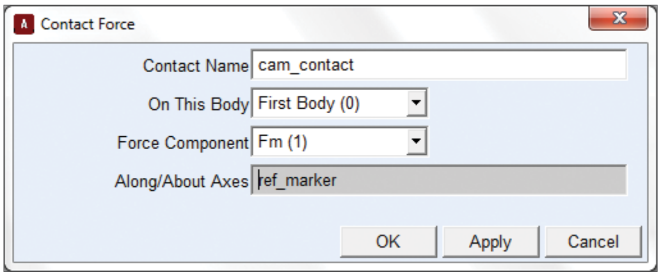
c. Use the category **Force in Object**, select **Contact force** and click on **Assist** tab.

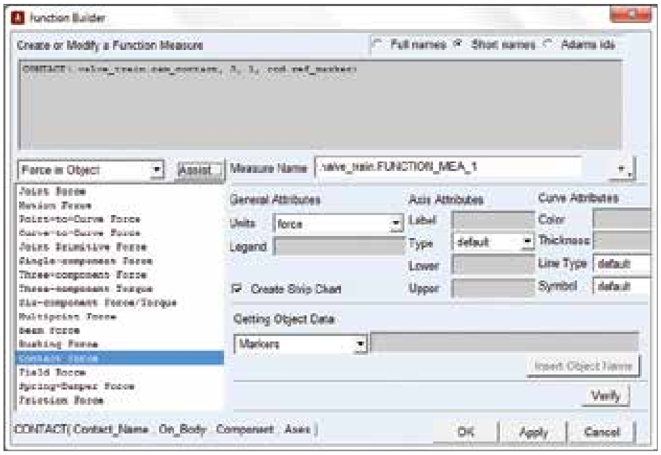
d. Fill out the contact Force dialog as shown below.

e. Your function should look like the one shown below in the Function Builder.

f. Remember to **Verify** the function before clicking OK.

g. Rerun the simulation to populate the new measure strip chart.

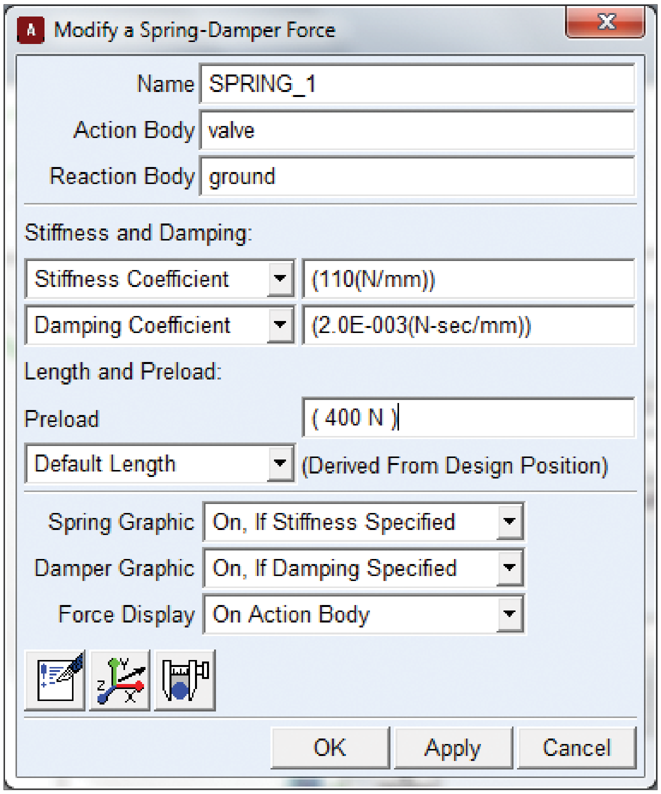



## Step 10. Modify the Spring Damper to Prevent Liftoff

a. Modify the spring-damper characteristics (stiffness, damping, and preload) to prevent liftoff based on the new rotational speed of the cam. **Note**: Experiment with different values until the no-lift criteria is met.

b. Save the model.

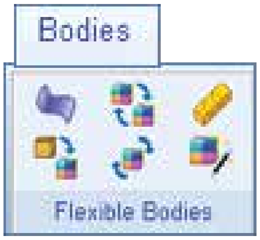


## Step 11. Create and Swap the Flexible Part using ViewFlex

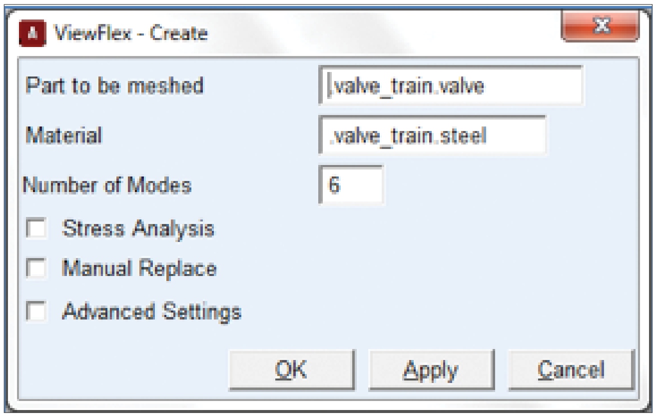
You will use the ViewFlex utility to convert the rigid valve part to a flexible valve part and transfer the constraints acting on the rigid body to the flexible body.

To create and swap the flexible part:

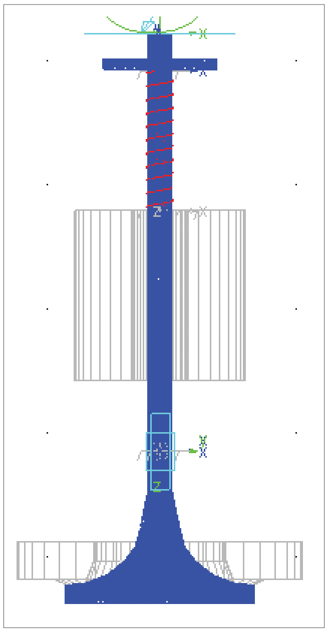
a. From the ribbon **Bodies**, select **Rigid to Flex.**
b. From the **Make Flexible** select **Create New**
c. Right-click in the **Part to be meshed** field and select the **Valve** part.
d. Check **Advanced Settings** to open more settings

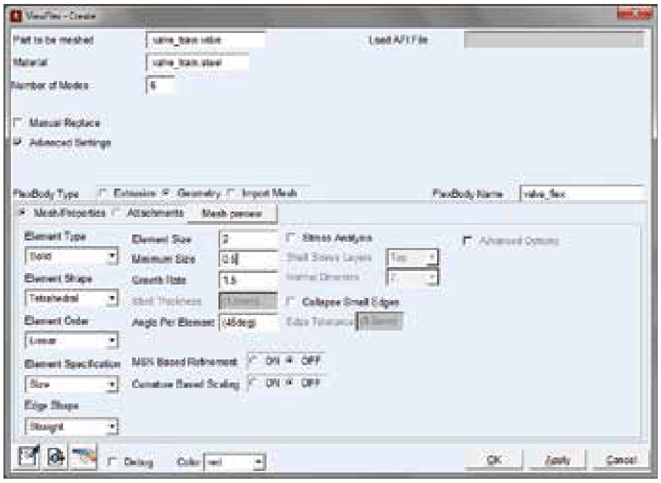







e. Select **Size** option in the **Element Specification**
f. Set the **element size =2** and **minimum size = 0.5**
g. Click **OK**.
h. The Rigid valve is now replaced by Flexible valve as shown below



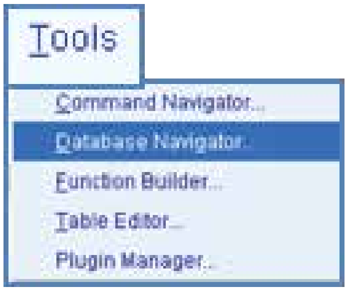


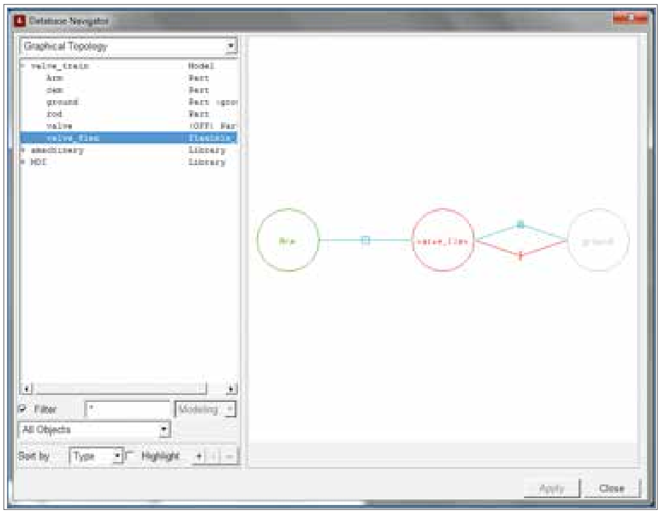
i. From the Tools menu, select **Database Navigator.**

j. Change **Browse** to **Graphical Topology**.

k. Highlight **Valve_flex** part.

l. Notice that the joints and spring are now attached to the flexible valve part.



## Step 12. Run a Simulation and Save

a. To view only one rotation of the cam, run a static equilibrium followed by a dynamic simulation for **end=1/50 seconds, steps=100**.

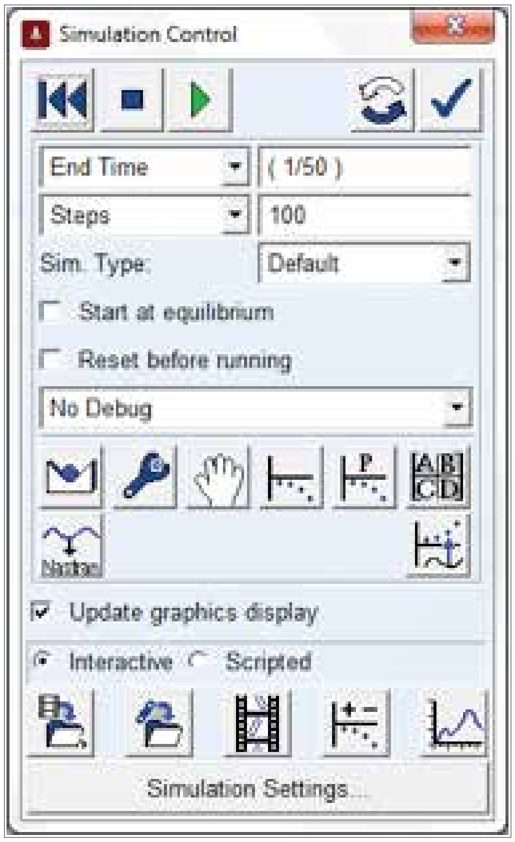
b. Use Adams/PostProcessor to investigate how the flexible body affects the model.

   a. Does lift off occur in the model now?

   ..........................................................................................................................................

c. Save the model

d. dIf you want to further explore the model, as suggested in the next section, leave the model open. Otherwise, Exit Adams/View.

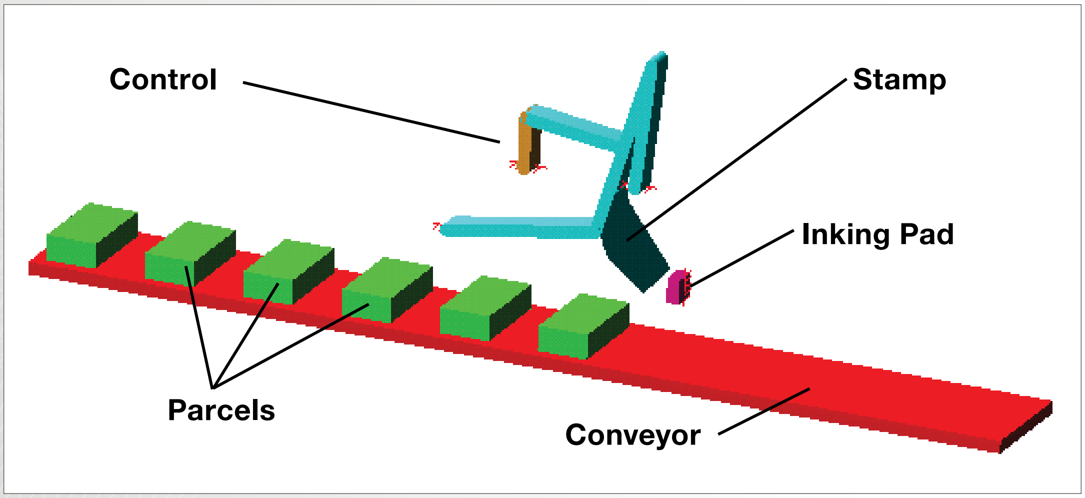


## Workshop Questions

How many DOF are removed by adding a curve-on-curve constraint?

How many DOF are removed by a curve-to-curve force?

# Example 17: Stamping Mechanism




## Workshop Objectives

To understand the virtual prototyping process by improving the design of the stamping mechanism.

## Software Version

Adams 2013.2

## Files Required

- **aview.cmd**
- Located in the directory **exercise_dir/ Example 17**

## Problem Description

- This model represents a mechanism for stamping parcels that are moving along a conveyor belt.
- During the work cycle, the stamp does not contact the parcels that it is supposed to label.
- To fix this design flaw, modify the length of the control link.

## Step 1. Import File

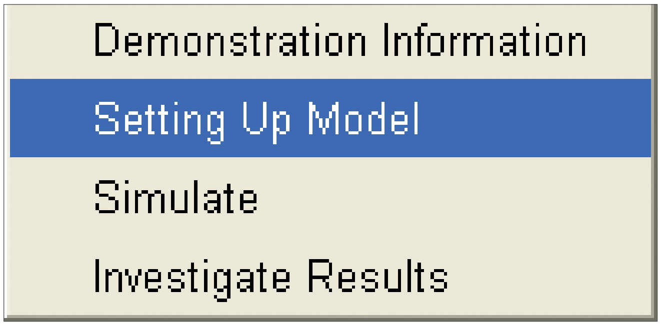
To import a file.

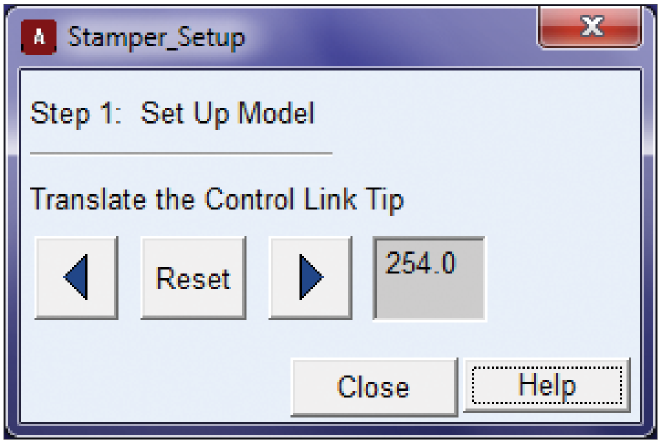
a. Start Adams/View.
b. From the Welcome dialog box, select **Existing Model.**
c. Click the **file folder** icon, and the Select Directory dialog box appears.
d. Find and select the directory **Exercise_dir/ Example17.**
e. Click **OK**.
f. Click on the file folder icon of the **File Name**, select the file **aview.cmd** and click **Open**.
g. Click **OK** on the **Open Existing Model** dialog box.

## Step 2. Change the Length of the Control Link

To change the length of the control link:

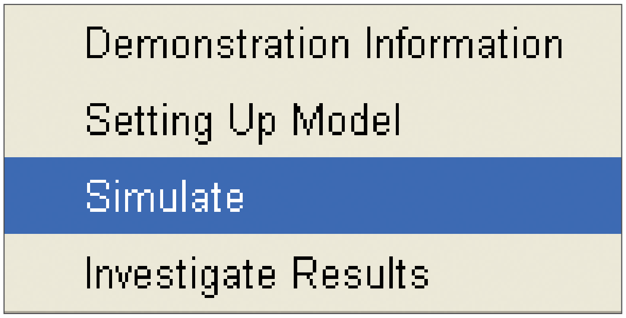
a. From the **Stamper** menu, select **Setting Up Model.** The Stamper_Setup dialog box appears.
b. Use the left and right arrow buttons to modify the length of the control _link.
   a. The buttons shift the location of the top of the control_link upward and downward 3 mm at a time.
   b. The parts connected to the control link are parameterized in such a way as to move the appropriate amount automatically whenever you adjust the length of the control link.
c. Watch the model change as you press these buttons.
d. To reset your model to the original configuration, select **Reset**. Leave the Stamper_Setup dialog box open, and continue with the next step.

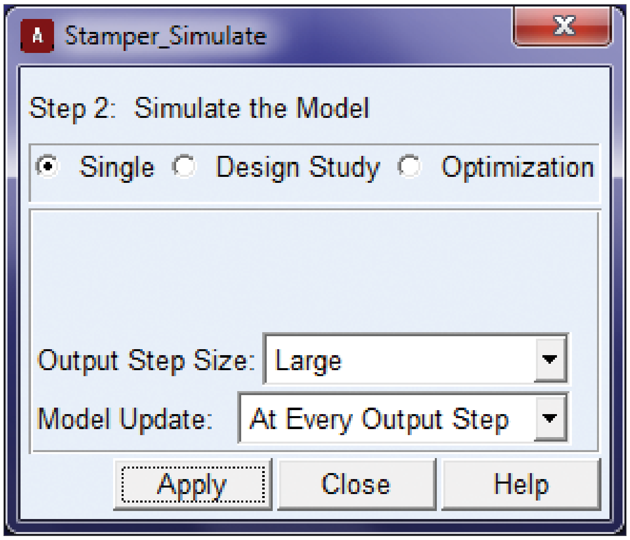



## Step 3. Simulate the Model

a. To simulate the model:
b. From the **Stamper** menu, select **Simulate**. The Stamper_Simulate dialog box appears.
c. To simulate the current design variation, ensure that Single is selected.
d. **Note:** The default setting for Model Update is set to Never. If you were to change **Model Update** from **Never** to **At Every Output Step** the model would update on the screen but would not solve faster.
e. To solve the equations of motion for the current design, select **Apply**.
f. When a single simulation is completed, Adams/ View tells you what the penetration was during the simulation. A positive number indicates penetration. To continue, click **OK**.
g. Leave the Stamper_Simulate dialog box open, and continue with the next step.





## Step 4. Investigate the Results

To investigate the results:

a. From the **Stamper** menu, select **Investigate Results**. The Stamper_Investigate dialog appears.

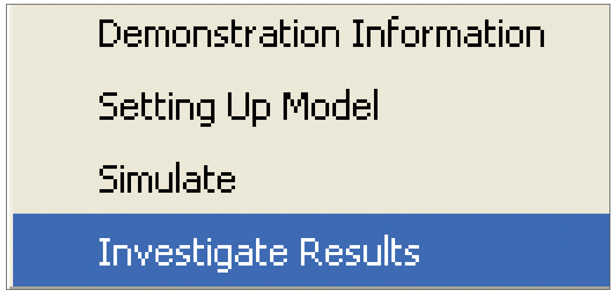
b. To see the motion resulting from the last simulation, select **Animate Results.**

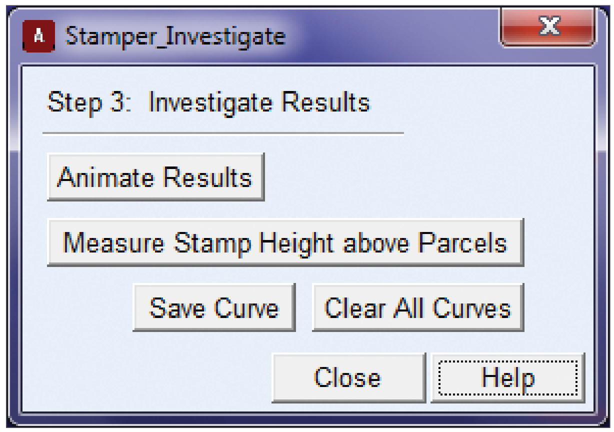
c. If necessary, use the stop sign in the lower right corner of the window to stop an animation before it has completed.

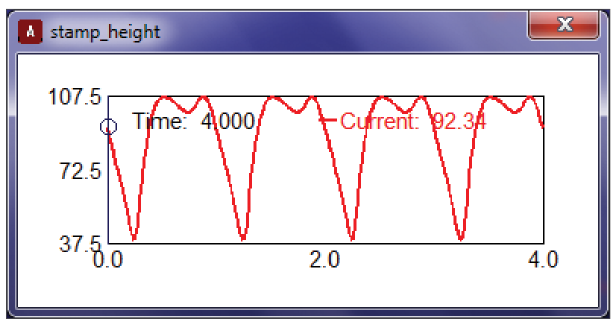
d. To plot the vertical travel of the stamper with respect to the parcel tops versus time, as calculated from your last simulation, select **Measure Stamp Height above Parcels.**

e. A stripchart appears, which shows a plot of the height of the stamp above the parcels. Note, your stripchart may look different depending on the value you used in the Stamper_Setup dialog. In this example that value was 254 (see WS1-8).

f. To save an existing curve so that the next simulation will not overwrite the exiting curve but will be superimposed on the saved curve, select **Save Curve.**



## Step 5. Manually Find the Correct Height

To manually find the correct Height:

Repeat the steps on the previous pages using 3 mm increments until you can identify the control_link length at which the stamp makes contact with the parcels. Use this value to answer Question 1.

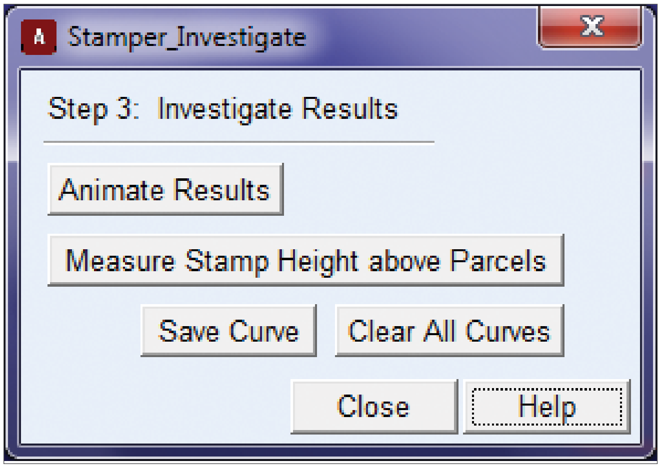
Helpful hint:

- If the stamp_height > 0, the stamper does not make contact with the parcels
- If the stamp_height < 0, the stamper makes contact with parcels.

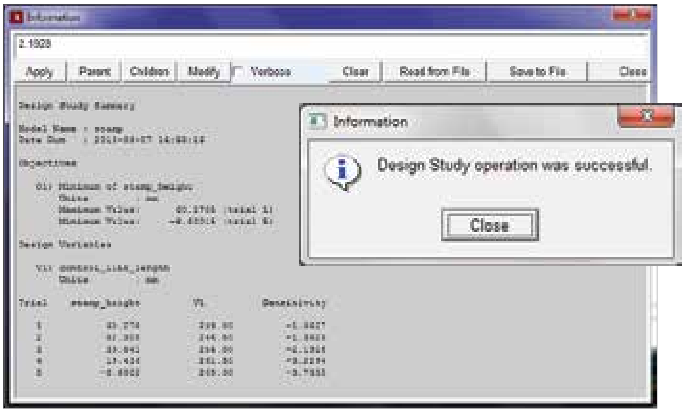
## Step 6. Perform a Design Study

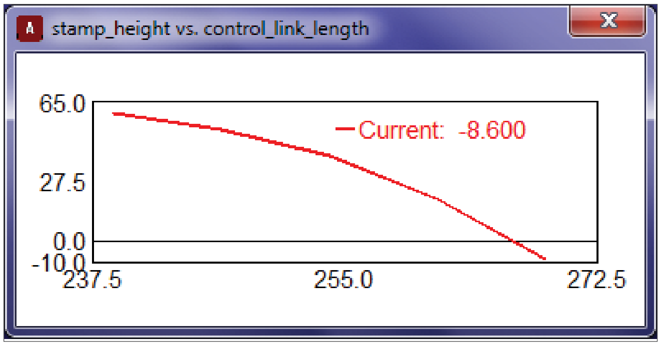
The design study automatically analyzes the model using the specified upper and lower limits for control_link length and the specified number of runs. To perform a design study:

a. On the Stamper_Simulate dialog box, select **Design Study.**
b. Default values for the upper and lower limit are given, but you can modify these if you wish.
c. In this case, leave the number of **Runs** at **5**.
d. To speed up the simulation, set the **Model Update** to **Never**.
e. Click **Apply** to submit the design study.




f. The design study automatically analyzes the model. Click **Close** on the Information Dialog that informs you that the design study was successful.
g. After the study is complete a stripchart and information window appear.
h. From the information window, identify the range of the control_link length values within which the stamp makes contact with the parcels. Use this range to answer Question 2.
i. **Close** the information window.



## Step 7. Perform an Optimization Study

During an optimization study, Adams/View systematically varies the control_link length and runs a number of simulations until the specified penetration is achieved to within a set tolerance. To perform an optimization study:

a. On the Stamper_Simulate dialog box, select **Optimization**.

b. Set the **Desired Penetration** to **4 mm**. You do not have to enter the units, Adams/View will automatically use the default units set for the model.

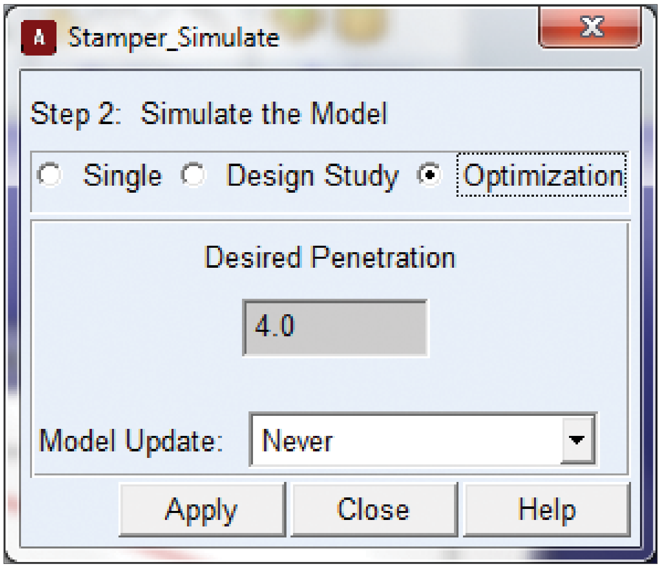
c. Set **Model Update** to **Never**.

d. Click **Apply** to submit the optimization study.

e. The information window appears displaying the control_ link length for maximum penetration of 4mm.

f. Use this displayed value of the control link length to answer Question 3.

g. Click **OK** to close the information window.




## Workshop Questions

Using 3 mm increments, at what control link length do you first notice penetration?

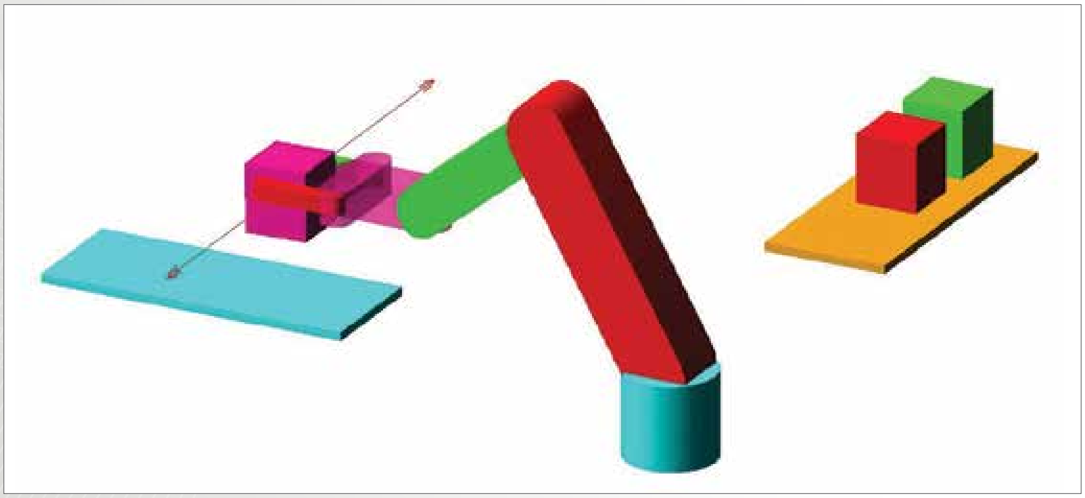
From the design study, what control link length results in penetration? How does this compare with your previous results?

If you specify a maximum desired penetration of 4 mm, what is the optimal length of the control link? How close is the maximum actual penetration to the maximum desired penetration?

How many moveable parts does the model consist of?

How many joints does the model consist of?

# Example 18: Robot Arm



## Workshop Objectives

- Construct a robot arm in Adams
- Manipulate the working grid for use with multi-planar part layouts
- Create a gear constraint between revolute joints
- Use a SFORCE to apply griping torque to a robot manipulator
- Define 3D object contact and friction
- Synchronize joint motions and motor toques to perform a complex task

## Software Version

Adams 2013.2

## Step 1. Build the Lower Links

a. Start Adams/View.

b. Create a new model. **(Model Name = robot_arm, Units = mmks, Gravity = -y earth)**

c. Create a **Link** from **(0, 0, 0)** to **(0, 150, 0)** and rename it **lower_link**.

- Note that the working grid, by default, snaps to 50mm spaced grid points, which make it easy to select the specified points.

d. Build another from **(0, 150, 0)** to **(0, 250, 0)**. Rename it **middle_link.**

e. Modify the link geometries of **lower_link** to have a **width** and **depth** of **40**. For **middle_link**, set its link geometry to a **width** and **depth** of **30**.

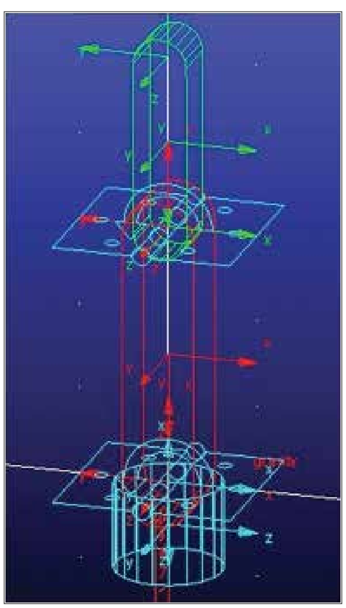
f. Build a **Cylinder** with a start point at **(0, 0, 0)** and an end point at **(0, -50, 0).** Rename it **base**.

g. Modify the cylinder **radius** to be **30**.

h. Select the **Revolute Joint** follow the instruction in the status bar to create the following joints.

| First Body | Second Body | Location |
|---|---|---|
| lower_link | base | 0,0,0 |
| middle_link | lower_link | 0,150,0 |

- The order of the body selections is important because it determines which direction is positive for applied motion. After all of the joints are created, the model will be tested to make sure motions act in the correct direction. If not, this can easily be changed.



## Step 2. Change the Working Grid

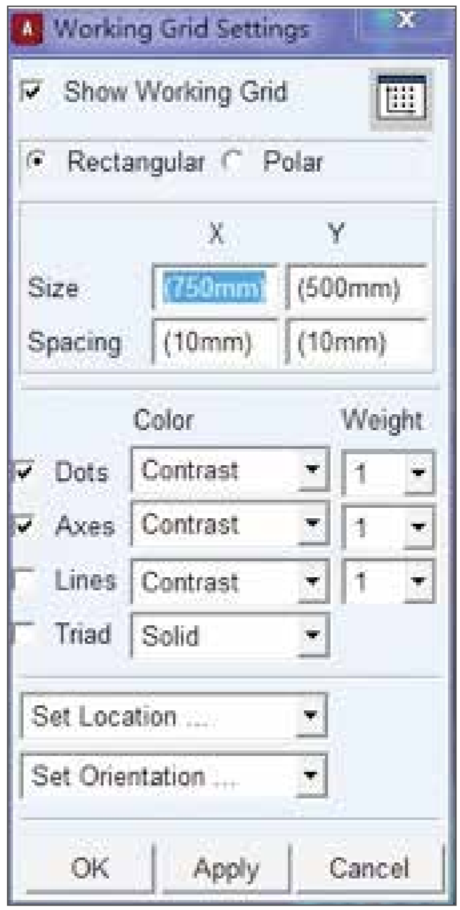
a. Go to **Settings >> Working Grid**

b. Set **Orientation** to **Global YZ** plane and spacing to **10mm x 10mm.**

c. Select **OK**.

d. Press **Shift + R** to change to **right view**.

- The XY working grid was ideal for creating the lower links and base, but the YZ is much more convenient for building the manipulator. Not only does this cause the cursor to snap to points in the appropriate plane, but also allows the use the use to the default **Create Normal to Grid** option on Joints and other entities.



## Step 3. Build the Manipulator I

a. Decrease the size of the icons to make working near them easier:

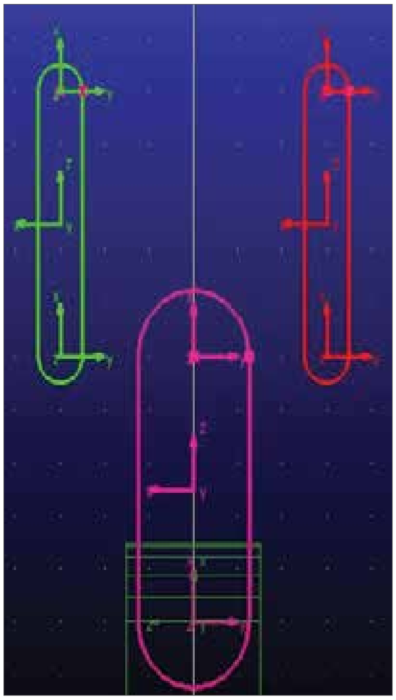
- Go to **Settings >> Icons**… and type **10** in the **New Size** box

b. Build and resize links for the manipulator as shown below.

c. Referring to the image below, (your colors may differ) rename the **MAGENTA** link **manipulator_base**, the **RED** link **gripper_right** and the **GREEN** link **gripper_left.**

d. Save your work.

| Link | End Points | Width | Depth |
|---|---|---|---|
| manipulator_base | (0, 300, 0) (0, 250, 0) | 20mm | 20mm |
| gripper_right | (0, 350, -30) (0, 300, -30) | 10mm | 10mm |
| gripper left | (0, 350, 30) (0, 300, 30) | 10mm | 10mm |



## Step 4. Build the Manipulator II

For this simulation, the robot will be grasping 40mm square cubes. Use precision move to correctly space the grippers.

a. Select **gripper_left.**

b. Select the **Position:Move** icon in the Main Toolba 

c. Type **5mm** in the **distance** box.

d. Select **Vector** and select any vector in the global z direction

e. Repeat for **gripper_right**, moving it left **5mm** instead.

Finish building the geometry for the manipulator base part

f. Select the **Rigidbody:Link** icon.

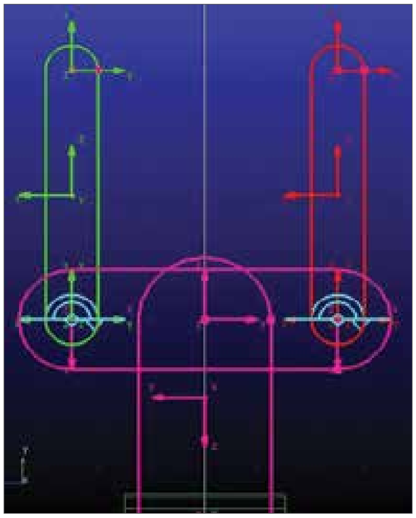
g. Change from **New Part** to **Add to Part** in the drop-down menu

h. Select the check boxes **Width** and **Depth**, and enter **20** into each field

i. Select the **manipulator_base** part, then select the lower markers of each gripper part to define the new link geometry

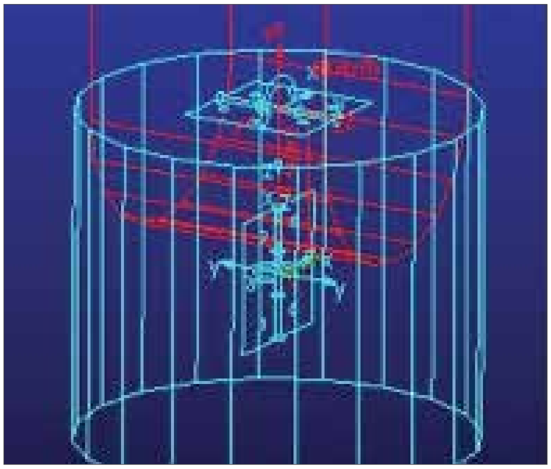
Note that this is a new geometry added to the manipulator_base part, not a new part.

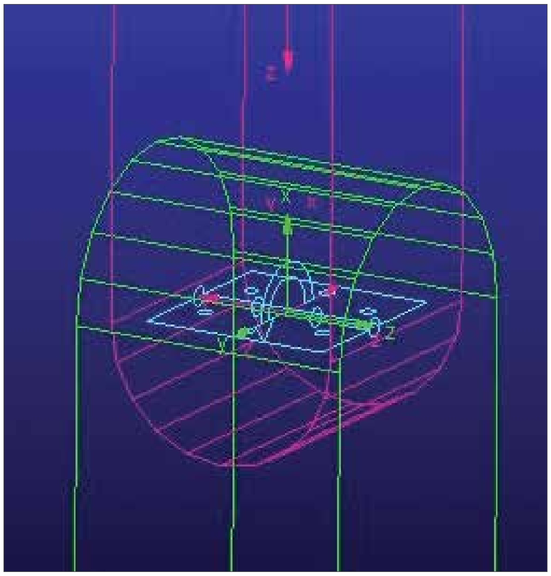
j. Build **revolute joints** between each of the **grippers** and **manipulator_base** as shown, selecting the grippers at the first bodies



## Step 5. Define the Remaining Joints

a. Switch the working grid orientation to the **Global XY**.
b. Build a revolute joint between **manipulator_base** and **middle_link** at **(0, 250, 0)**
c. Switch the working grid orientation to **Global XZ.**
d. Build a revolute joint between **ground** and **base** at the **bases .cm marker**, which is at **(0, -25, 0).**

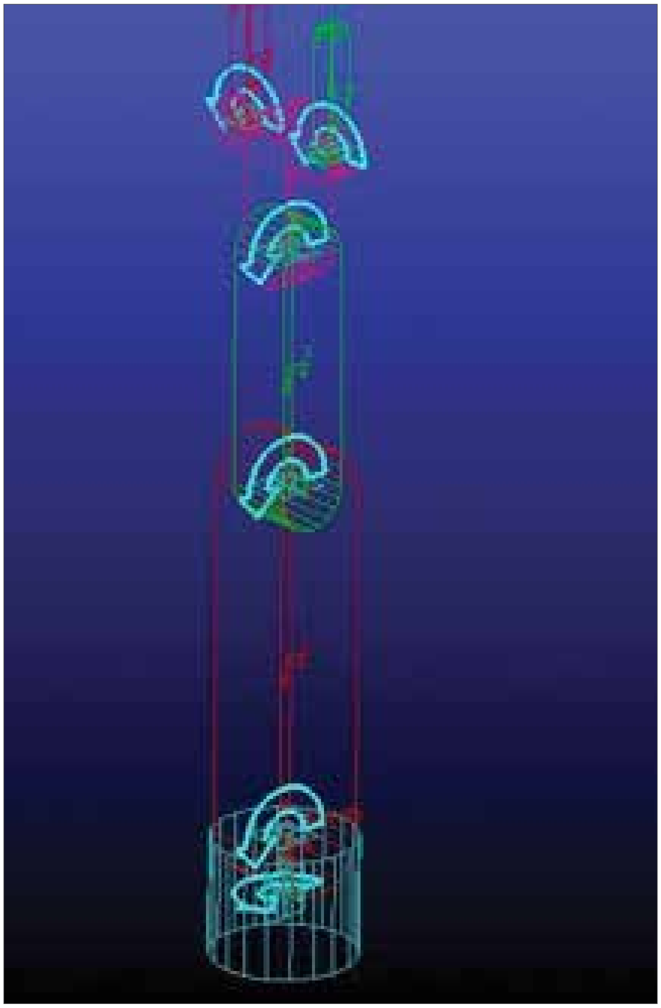




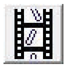
## Step 6. Add Motions to Test the Model

a. Click the **Rotational Joint Motion** tool from the Main Toolbar. Use the default speed of **30.0.**

b. Select the joint between **lower_link** and **base**.
c. Rename the newly created motion **motor_1.**
d. Modify **(Right Click >> motor_1 >> Modify...)** the newly created motion. Add a negative sign to the function line to reverse its direction
e. Add motions to the other 5 revolute joints, using the default speed of 30 for each motion. It is not necessary to change the direction of sign/direction of these motions.
f. Rename each of the motions as shown.
g. Simulate for the default of **5 seconds** and **50 steps**.



## Step 7. Test the Model

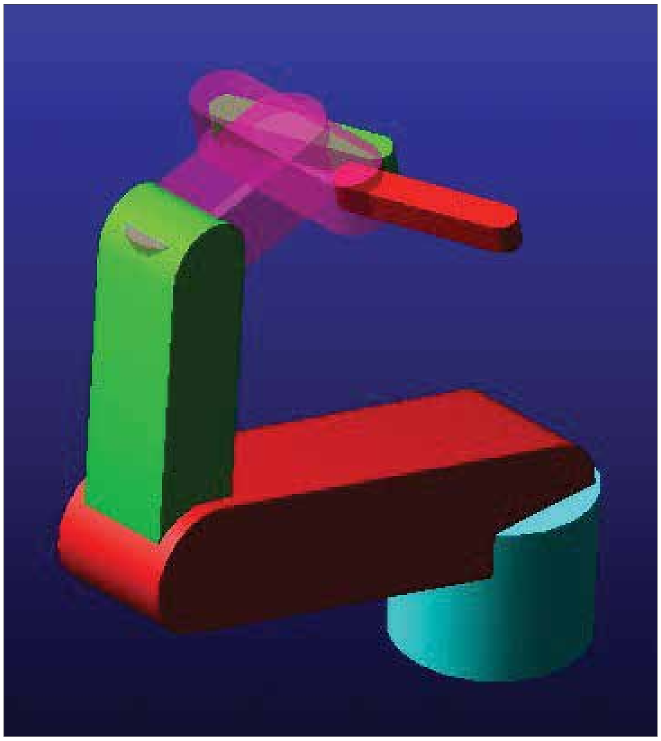
a. Click on the **animation** icon. 

b. Use the slider bar to navigate to **frame 26.**

c. Shown is the **iso (Shift+I) shaded (Shift+S)** view of the model at frame 26 (time 2.5000). Manipulator_base has been made transparent for visualization.

   a. If your model does not behave as shown, attempt to make the necessary changes.

   b. Likely, the issued can be fixed by reviewing the slides (5-8 for position related issues, and 12-13 for joint related issues). If this does not resolve the issue, load robot_arm_shortcut1.bin and continue from there.



## Step 8. Build Objects to Grasp I

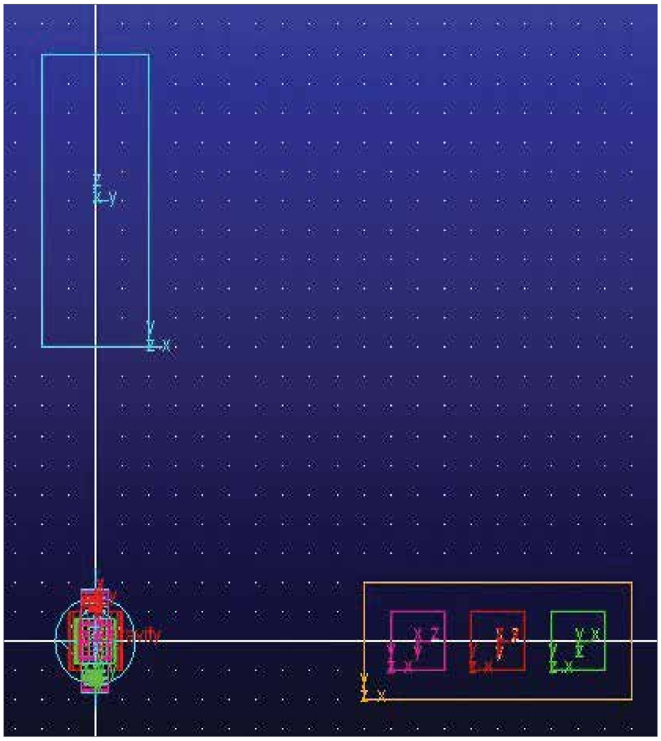
a. Switch working grid to **Global XZ** and spacing to **20 x 20.** Switch to **top view**.

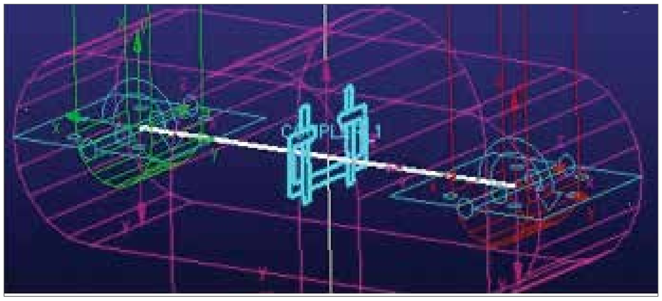
b. Use the **Rigid Body: Box** to build boxes as shown below by selecting the appropriate corner locations.

c. Modify the block geometry of each part, changing the **Z component of Diagonal Corner Coords** to **-5** for the large ‘platforms’ and 40 for each of the cubes.

d. Rename the geometries of the newly created bodies as shown below.

e. Change the mass of each cube to **20g. Modify >> Define Mass By: User Input**






## Step 9. Build Objects to Grasp II

Now we will fix the platforms to ground, and create contact between the boxes and platforms so they do not fall into space.

a. Select **Contact** icon 
b. Make sure **Solid to Solid** is selected in the **Contact Type** drop down menu
c. Right click in the **I Solids** and **J Solids** dialog box and use the Pick to select the geometries in the Main Window.
d. Repeat for the other three cubes.
e. Create **fixed joints** between each **platform** and **ground**.

## Step 10. Couple the Motion of the Grippers

Now we will fix the platforms to ground, and create contact between the boxes and platforms so they do not fall into space.

a. Set the Working Grid back to **Global YZ** and switch to **Right View.**
b. Delete the motions acting on the gripper revolute joints.
c. Select the **Joint Coupler.** 
d. Choose the **gripper_right revolute joint**, then the left, to define a motion coupler.
e. Modify the coupler as shown. This constrains the motion of the joints to be equal in magnitude but opposite in direction.

## Step 11. Add Forces to the Gripper

a. Select **Create a Rotational Spring-damper.**

b. Select **gripper_right** then **manipulator_base** for the bodies to define the spring, then the center of the revolute joint for location.

c. Modify the torsion spring to have a **Stiffness** of **10**, **Damping** of **5** and a **Preload** of **10**. The causes the grippers to spring slightly open by default.

d. Select **Create a Torque.**

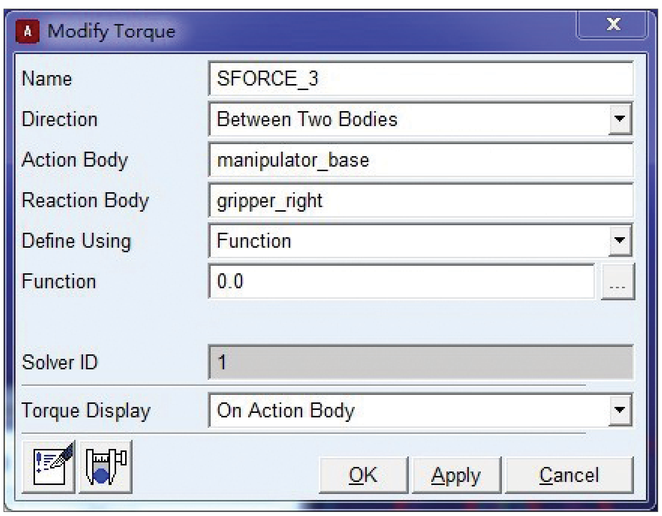
- Note: Although the torque should act between **gripper_right** and **manipulator_base**, use the default **Space Fixed** option (which reacts on ground) to take advantage of the default **Normal to Grid** for direction. The **SFORCE** will later be modified to react on **manipulator_base**.

e. Select **manipulator_left** for the body to define the **SFORCE** and the center of the **gripper_right** revolute joint for location.

f. Change the appearance of the SFORCE to have a color of **blue** and a size of **11** for visibility.

g. Rename the torque **SFORCE_grip_torque**.

h. Modify the SFORCE as shown.




## Step 12. Test the Gripper Forces

a. Set the other motions in the model to be 0 and verify the operation of the manipulator.

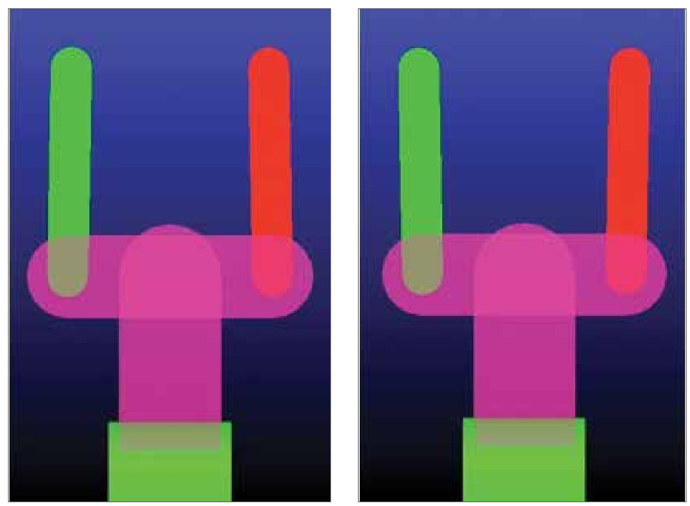
b. Set all 4 of the joint motion (**motor_1, motor_2, etc.**) function definitions to be **0**.

c. Modify **grip_torque's function** to be **20.**

d. Simulate for **3 seconds, 30 steps**, the grippers should now settle in a slightly closed position, as shown.

e. Change **grip_torque's function** to be **0.**

f. If the grippers do not behave as shown, refer to the Modify dialog boxes and images for the torsion spring and grip_torque on the previous slide to check their definitions

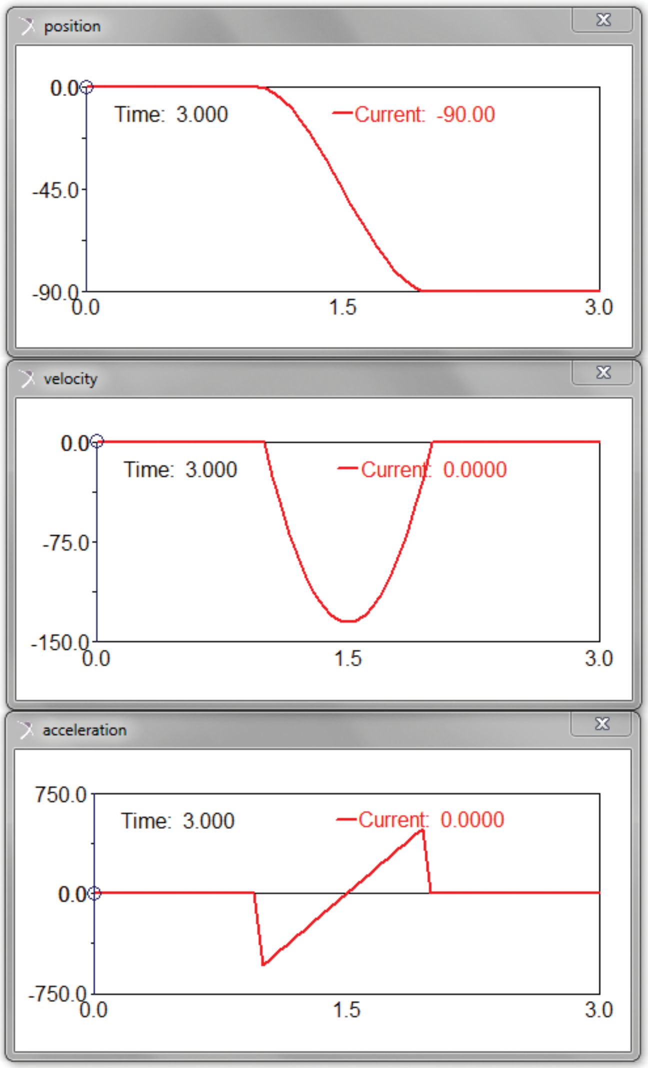


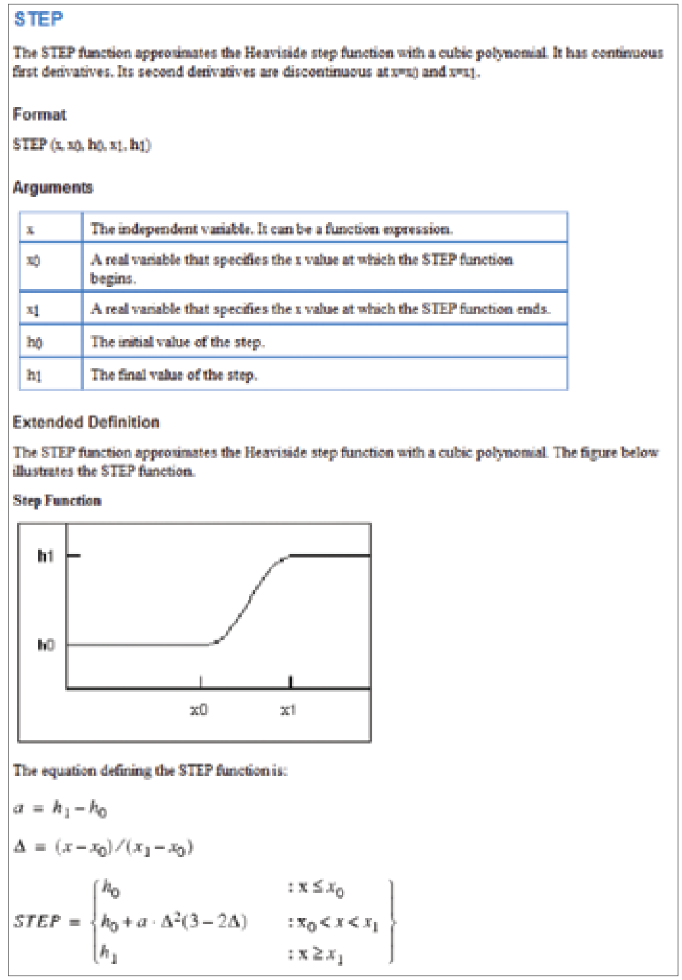
## Step 13. The Step Function Introduction

A step function will be used to define the motion of the robot. The step function dependent variable has an initial valve h0, before the independent variable, x, reaches x0, a final value h1 after the x reaches x1, and a smooth step in between. "Smooth" mean the first derivative is continuous (i.e. no instantaneous change in acceleration). Position, Velocity, and Acceleration of a step defined motion are show below.

Step Function: **-90d*step(time,1,0,2,1)**

As you can see, detailed information on Adams functions can be found in the Adams help documentation



**STEP**

The STEP function approximates the Heaviside step function with a cubic polynomial. It has continuous first derivatives. Its second derivatives are discontinuous at x=x0 and x=x1.

**Format**

STEP (x, x0, h0, x1, h1)

**Arguments**

| | |
|---|---|
| x | The independent variable. It can be a function expression. |
| x0 | A real variable that specifies the x value at which the STEP function begins. |
| x1 | A real variable that specifies the x value at which the STEP function ends. |
| h0 | The initial value of the step. |
| h1 | The final value of the step. |

**Extended Definition**

The STEP function approximates the Heaviside step function with a cubic polynomial. The figure below illustrates the STEP function.

**Step Function**



The equation defining the STEP function is:

$a = h_1 - h_0$

$\Delta = (x - x_0)/(x_1 - x_0)$

$$STEP = \begin{Bmatrix} h_0 & : x \le x_0 \\ h_0 + a \cdot \Delta^2(3 - 2\Delta) & : x_0 < x < x_1 \\ h_1 & : x \ge x_1 \end{Bmatrix}$$

## Step 14. "Stepping" the Robot

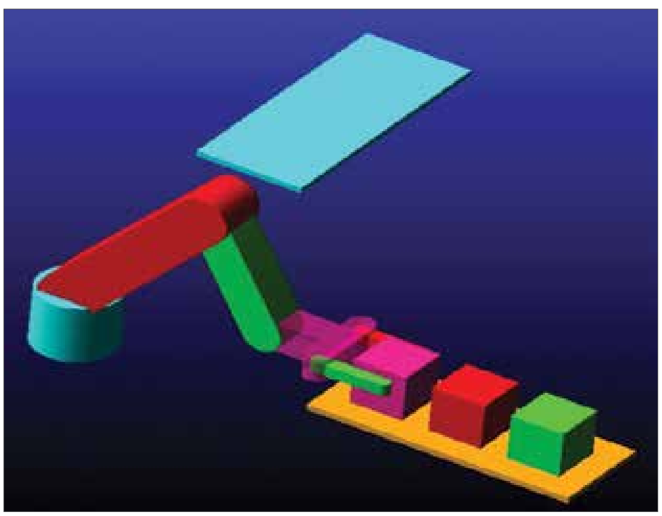
Now, motions will be defined with step functions to bring into position to grip cube_1.

a. Define the following step functions. Be sure delete what is already in the function box and select apply in the motion modify window.

b. Run a simulation for **1 second, 20 steps.**

c. At the end of the simulation, the gripped should be positioned to grip the cube, as shown.

| Motions | Function |
|---|---|
| Motor_1 | step(time,0,0,1,-40d) |
| Motor_2 | step(time,0,0,1,-110d) |
| Motor_3 | step(time,0,0,1,60d) |
| Motor_4 | 0 |



## Step 15. Grip the Cube

Next create a contact and friction force between the grippers and the cube.

a. Create a solid to solid contact between **gripper_left** and **cube_1.**

b. Change the **Static** and **Dynamic Coefficients** as shown.

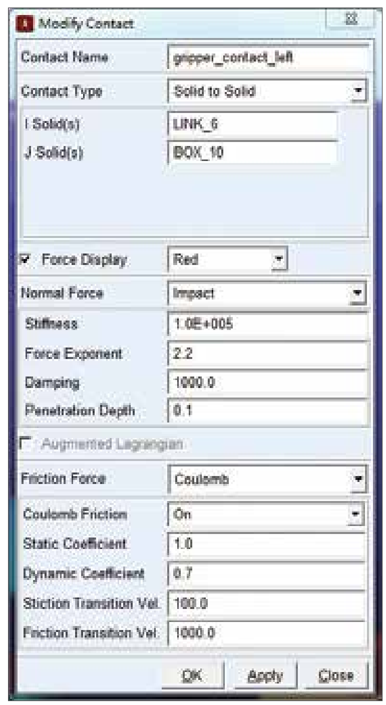
c. Repeat for **gripper_right** and **cube_1.**

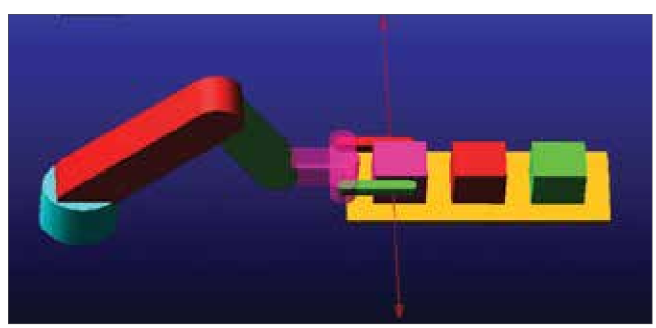
d. Rename each **gripper_contact_[left/right].**

Use a step function to set torque equal to 0 from 0-1sec, allowing the spring the keep the grippers in the slightly open position, then apply 8000 N*m when in position.

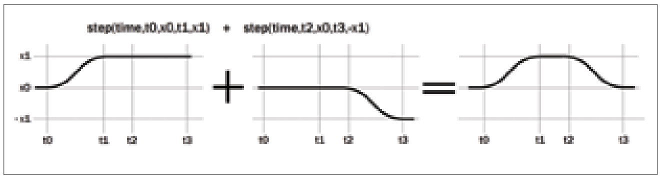
e. Modify **grip_torque's function** to be **-step(time,1,0,1.1,-8000)**

f. Simulate for **1.1 seconds, 55 steps**. Confirm that the gripper makes contact with cube_1 as shown.



## Step 16. Adding Steps

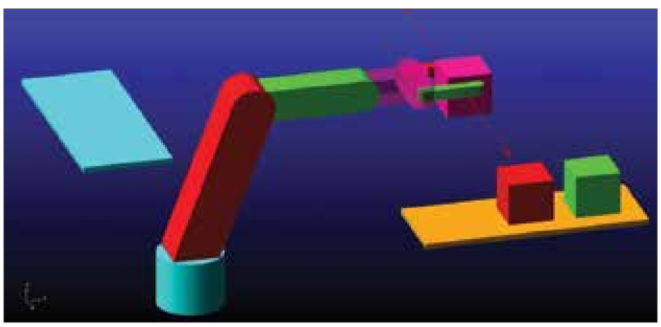



a. Add the following steps to the specified motion functions:

| Motions | Step to Add |
|---|---|
| Motor_1 | step(time,1.1,0,2,20d) |
| Motor_2 | step(time,1.1,0,2,40d) |
| Motor_3 | step(time,1.1,0,2,-60d) |

b. Simulate for **2 seconds, 40 steps.**

c. Verify that your simulation matches with the image.



## Step 17. Finalize Definition of Motions and Torques

The chart below describes the necessary step values required to complete the entire operation. Try to figure out the missing values on your own. Recall that x values are the independent value (time) and the h values are the dependent variables (position/torque) and the format for the step function (with time defined as X) is step(time,x0,h0,x1,h1)

| | Already Done | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Movement Description | Step1 Move into position | | Step2 Grip the cube | | Step 3 Lift | | Step4 Move to platform2 | | Step5 Lower into place | | Step6 Release | | Step7 Raise manipulator | |
| Time Values | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 |
| | 0 | 1 | 1.1 | 2 | 2 | 3 | 2.5 | 3.5 | 3 | 4 | 4 | 4.1 | 4.1 | 5 |
| Motion/ Torque Values | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 |
| motor_1 | 0 | -40d | N/A | N/A | 0 | 20d | N/A | N/A | | | N/A | N/A | | |
| motor_2 | 0 | -110d | N/A | N/A | 0 | 40d | N/A | N/A | | | N/A | N/A | | |
| motor_3 | 0 | 60d | N/A | N/A | 0 | -60d | N/A | N/A | | | N/A | N/A | | |
| motor_4 | N/A | N/A | N/A | N/A | N/A | N/A | | | N/A | N/A | N/A | N/A | N/A | N/A |
| grip_torque | N/A | N/A | 0 | 8000 | N/A | N/A | N/A | N/A | N/A | N/A | | | N/A | N/A |

Add to motion definitions one step at a time and simulating to verify the model behaves as expected.

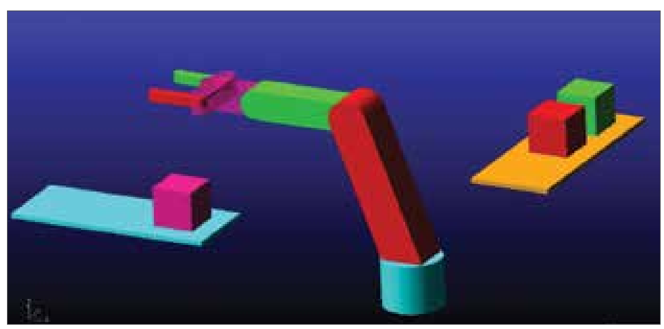
Note that all h0 values are zero and h1 always describes motion relative to the current position.

The necessary motion to place the cube on the platform and release it is essentially the reverse of picking it up. In other words, steps 3, 5, and 7 are very similar.

For example, the final function definition of motor_2 is:
**step(time,0,0,1,-110d) + step(time,1.1,0,2,40d) + step(time,2,0,3,-40d) + step(time,4.1,0,5,40d)**

Step 4 is simple a rotation of the base to bring the cube over platform_2. Note that its x values overlap steps 2 & 5.

Try to simulate the whole operation (0-4 seconds) with your values. If you are having trouble, continue to the next step.



## Step 18. Finalize Definition of Motions and Torques

| | Already Done | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Movement Description | Step1 Move into position | | Step2 Grip the cube | | Step 3 Lift | | Step4 Move to platform2 | | Step5 Lower into place | | Step6 Release | | Step7 Raise manipulator | |
| Time Values | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 | x0 | x1 |
| | 0 | 1 | 1.1 | 2 | 2 | 3 | 2.5 | 3.5 | 3 | 4 | 4 | 4.1 | 4.1 | 5 |
| Motion/ Torque Values | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 | h0 | h1 |
| motor_1 | 0 | -40d | N/A | N/A | 0 | 20d | N/A | N/A | 0 | -20d | N/A | N/A | 0 | 20d |
| motor_2 | 0 | -110d | N/A | N/A | 0 | 40d | N/A | N/A | 0 | -40d | N/A | N/A | 0 | 40d |
| motor_3 | 0 | 60d | N/A | N/A | 0 | -60d | N/A | N/A | 0 | 60d | N/A | N/A | 0 | -60d |
| motor_4 | N/A | N/A | N/A | N/A | N/A | N/A | 0 | 90d | N/A | N/A | N/A | N/A | N/A | N/A |
| grip_torque | N/A | N/A | 0 | 8000 | N/A | N/A | N/A | N/A | N/A | N/A | 0 | -8000 | N/A | N/A |

Final list of motion/torque functions:

| Motions | Function |
|---|---|
| Motor_1 | step(time,0,0,1,-40d) + step(time,1.1,0,2,20d) + step(time,2,0,3,-20d) + step(time,3.1,0,4,20d) |
| Motor_2 | step(time,0,0,1,-110d) + step(time,1.1,0,2,40d) + step(time,2,0,3,-40d) + step(time,3.1,0,4,40d) |
| Motor_3 | step(time,0,0,1,60d) + step(time,1.1,0,2,-60d) + step(time,2,0,3,60d) + step(time,3.1,0,4,-60d) |
| Motor_4 | step(time,1.5,0,2.5,90d) |
| gripper_torque | step(time,1,0,1.1,8000) + step(time,3,0,3.1,-8000) |

## Step 19. Optional Tasks

Torque Demand

a. Switch to **PostProcessor** and examine the results of the simulation.

b. Look at torque demands **(Source:Objects >> motor _x>> Element Torque >> Mag),** and gripper contact forces**(Source:Objects >> gripper_contact_[left/ right]>> Element Torque >> Mag).**

c. Note the sporadic spikes in torque required to maintain the smooth step motion.

d. Switch back to **View** and increase the **contact damping** to **100** and re-simulate.

e. How do the torque demands and contact forces look now?

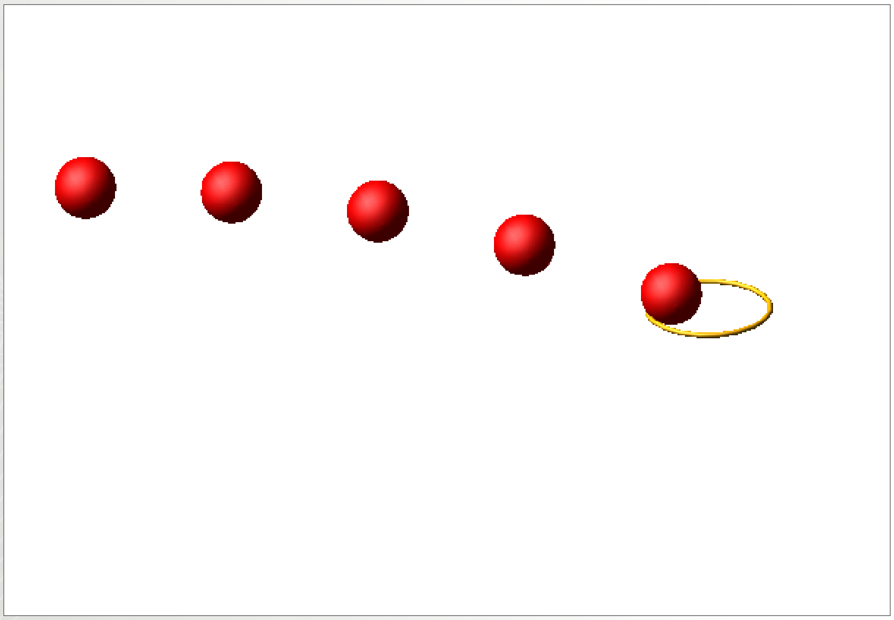
Move the Remaining Cubes

For simplicity, the robot sits the block down in the same position on platform 2 as it was on platform 1.

a. Use sketch paper to derive the necessary angles to sit the block near the far edge of platform 2 , where cube 3 is on platform 1.

b. Try to create the additional steps necessary to move the remaining blocks. Derive the necessary angles by hand or use trial and error to determine the correct values.

# Example 19: Adams Optimization



## Software Version

Adams 2013.2

## Problem Description

Shooting a basketball accurately is no easy task without proper practice. The thousands of iterations that basketball players make is the process of optimizing the move to find the best combination of initial vertical and horizontal velocity. In this example, this process is simulated in Adams/View to find the optimal velocity combination to score a three-point.

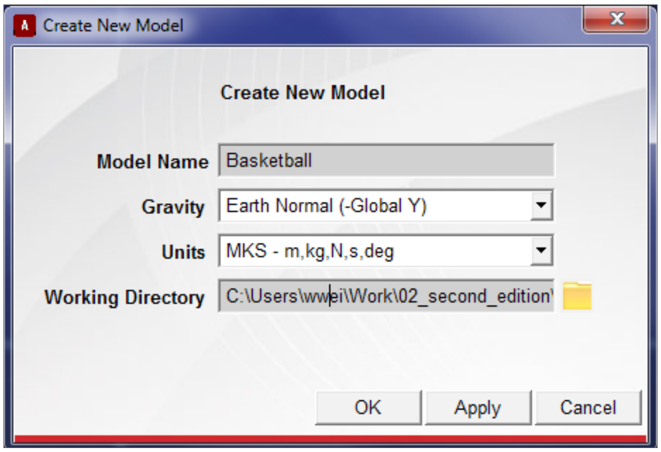
## Step 1. Start Adams/View.

a. Start Adams/View 2013.2.

b. Select **New Model**.

c. Name the model to Basketball and change the Unit to **MKS**. We use MKS here because we are going to model half of the basketball court.

d. Click **OK.**

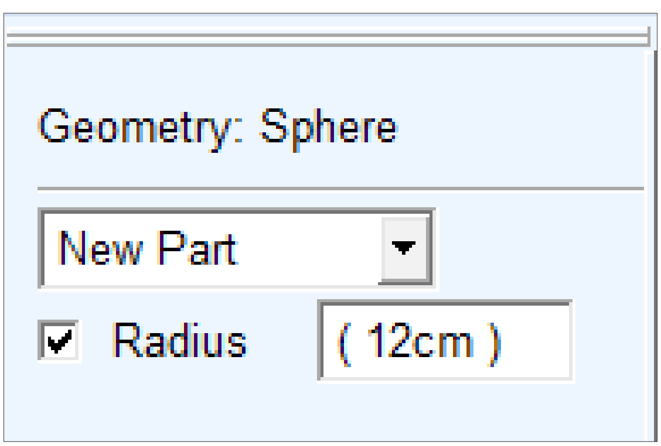



## Step 2. Create the Ball.

a. From **Bodies** ribbon, select **RigidBody: Sphere**.

b. Select **New Part** and **check Radius**.

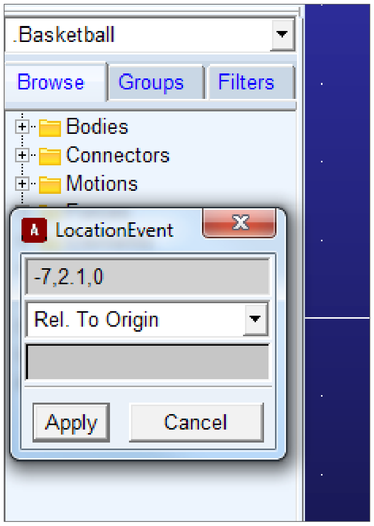
c. Set the **Radius** to **12cm**. The dimensions of the ball, the hoop and their positions are all set according to the standard of NBA.




d. Right click in the window to invoke Location Event.

e. Set the location of the ball at **(-7, 2.1, 0)**.

f. Click **Apply**.




g. Press **F** to fit the view and show the ball.

h. Right click on PART_2 and select Rename. Rename PART_2 to Ball.

## Step 3. Change the Working Grid.

a. Go to **Settings->Working Grid**.

b. Set Orientation to **Global XZ Plane**. This is because in the next step, when we create the hoop using Torus, the torus is created in the plane of Working Grid.

## Step 4. Create the Hoop.

a. From **Bodies** ribbon, select **RigidBody: Torus**.

b. Check both **Minor Radius** and **Major Radius** and set them to **1cm** and **24cm** respectively. Major radius is the radius of the center line of the hoop while minor radius is the radius of the ring itself.

c. Right click in the background to invoke Location Event and set the coordinate to **0, 3.05, 0**.

d. Rename PART_3 to Hoop.

e. Press F to fit the view.

## Step 5. Create Fixed Joint.

a. From **Connectors** ribbon, select **Joint: Fixed**.

b. Set **Construction methods** to **2 Bodies-1 Loc** and **Normal to Grid**.

c. Pick Hoop and then ground.

d. Pick Hoop.MARKER_2 as location. Now we fixed the hoop to the ground.

## Step 6. Create Contact.

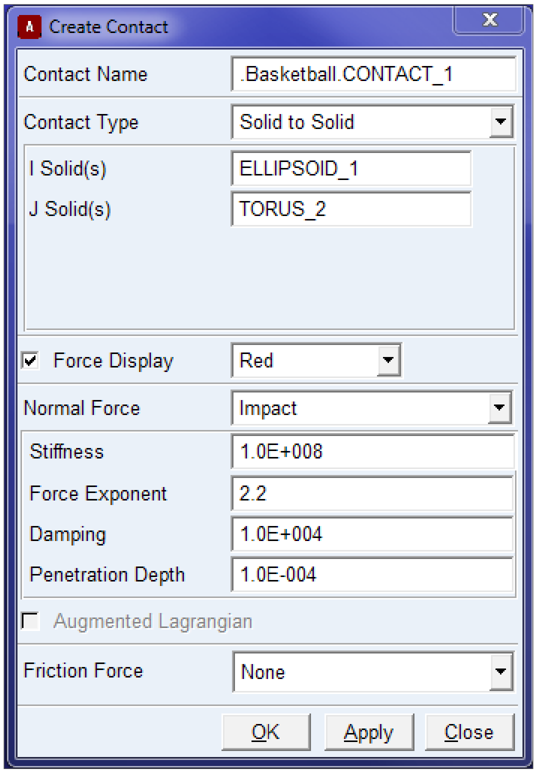
a. Now we create a contact between the ball and the hoop.

b. From **Forces** ribbon, select **Special Forces: Create a Contact**.

c. Right click in the I Solid(s) field and select **Contact Solid->Pick** and pick the ball.

d. Similarly, pick the hoop as J Solid.

e. We will leave the rest of the settings as default in this tutorial. Feel free to change these parameters later to see how these affect the simulation.




## Step 7. Create Design Variables.

a. From **Design Exploration** ribbon, select **Design Variable: Create a Design Variable**.

b. Name the variable **x_vel**.

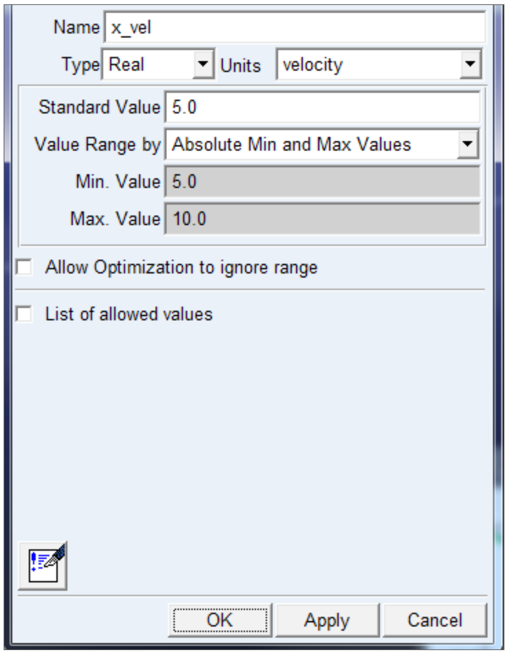
c. Set the Units to **velocity**.

d. Set Standard Value to **5**.

e. Select Value Range by: **Absolute Min and Max Values**.

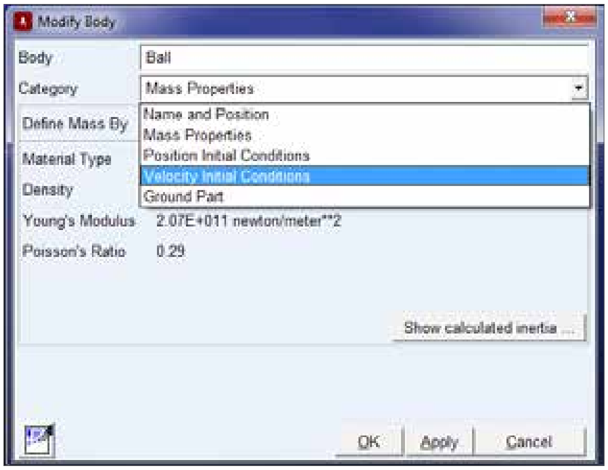
f. Set **Min. Value** and **Max. Value** to **5** and **10** respectively.

g. Click **Apply** and create another variable y_vel using the same settings.

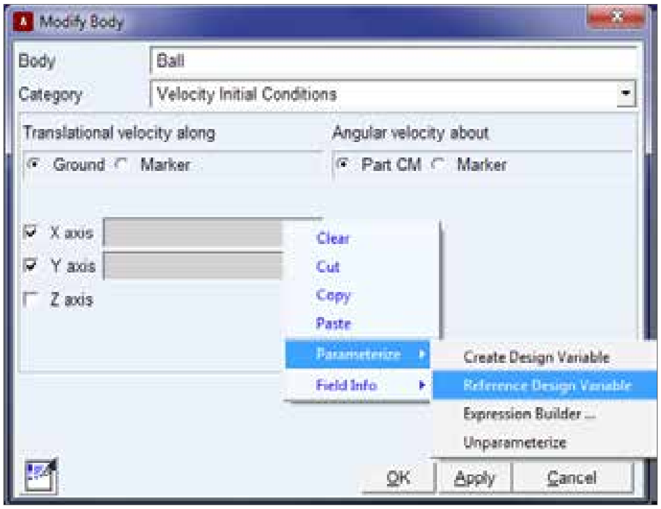



## Step 8. Impose Initial Velocities to the Ball.

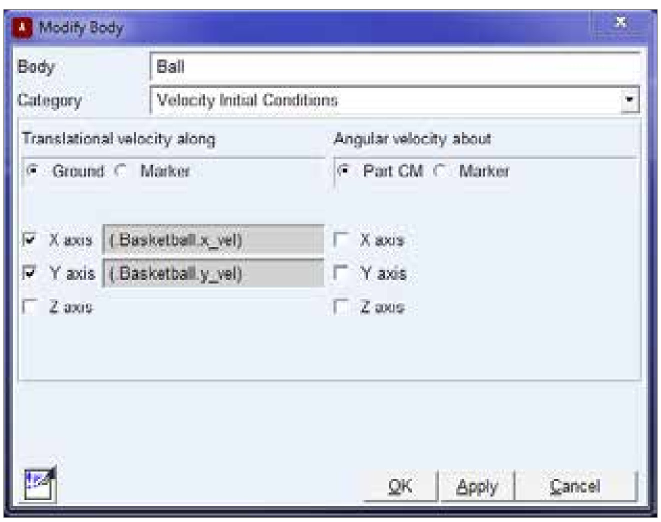
a. From the model tree, right click on part Ball and select Modify.



b. Under Category popup menu, pick **Velocity Initial Conditions**.
c. Check **X axis** and **Y axis**.




d. Right click in the field and select **Parameterize->Reference Design Variable**.
e. Select x_vel and y_vel respectively. Now the initial velocity of the ball will change during each iteration as the design variable x_vel and y_vel follows different combinations.
f. Click **OK**.




## Step 9. Create a Measurement.

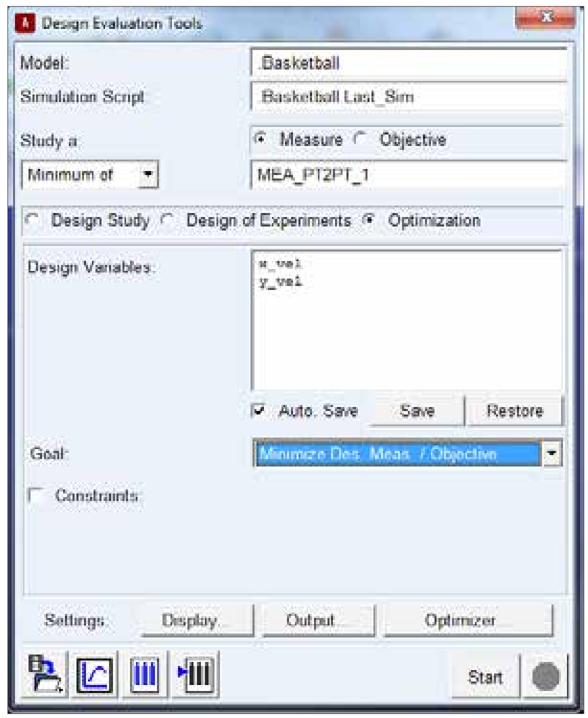
a. From **Design Exploration** ribbon, select **Measures: Create a New Point to Point Measure**.
b. Pick Ball.MARKER_1 and Hoop.MARKER_2. This measures the distance between the center of the ball and the center of the ball. When the distance is zero (within tolerance), we assume that the ball is on target.

## Step 10. Run a Simulation.

a. Run a simulation with 1 second and 50 steps.
b. The simulation script with default run will automatically be used in later iterations by the design evaluation tools.

## Step 11. Set Up Design Evaluation Tools.

a. From **Design Exploration** ribbon, select **Design Evaluation: Design Evaluation Tools**.
b. Pick **Study a Minimum of**.
c. Select the measurement we created in the field.
d. Select **Optimization**.
e. Right click in the field of Design Variables and select x_vel and y_vel.



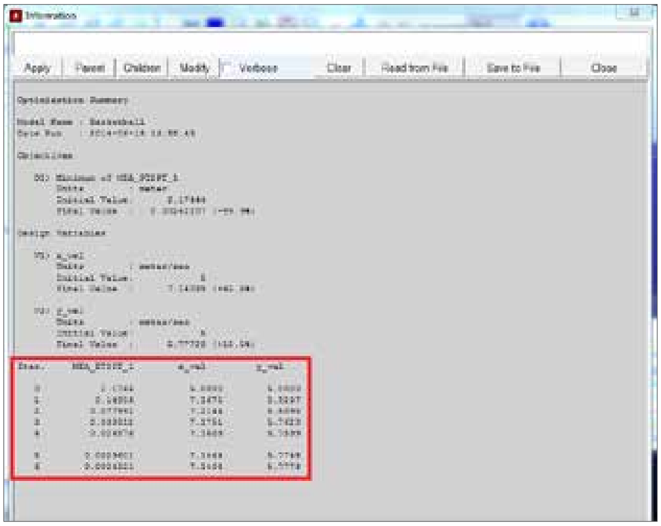

f. Pick the Goal as **Minimize Des. Meas**.
g. Click **Start**.

## Step 12. Review Optimization Results.

a. Click **Create tabular report of results**.

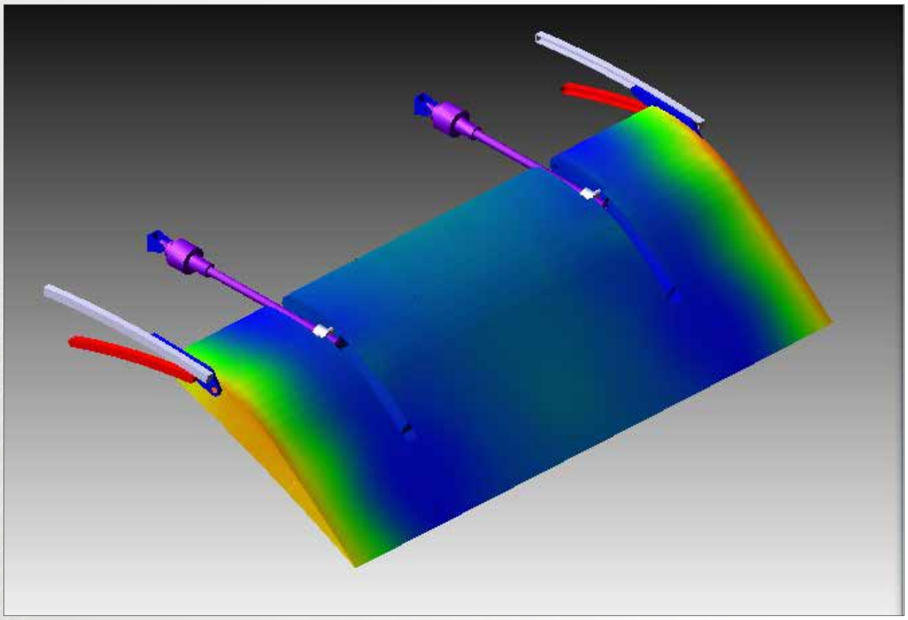
b. Then click **OK**.

c. In the message window, we find that the best combination is a x_vel of 7.1404 m/s and a y_vel of 5.7773 m/s. The distance between two centers is




about 0.0024m.

# Example 20: Airplane Control Surface



## Software Version

Adams 2013.2

## Problem Description

In this example, we will be studying the dynamics of an airplane control surface. The topics covered in this example include importing .mnf files, using .cmd file to generate features and the benefit of including flexible bodies in simulation.

## Step 1. Import Start File

a. Start Adams/View 2013.2.

b. Select **Existing Model** and browse to file **Control_Surface_start.cmd**.

c. Click **OK**. If you can't see the model immediately, press F to fit the model to the view window.

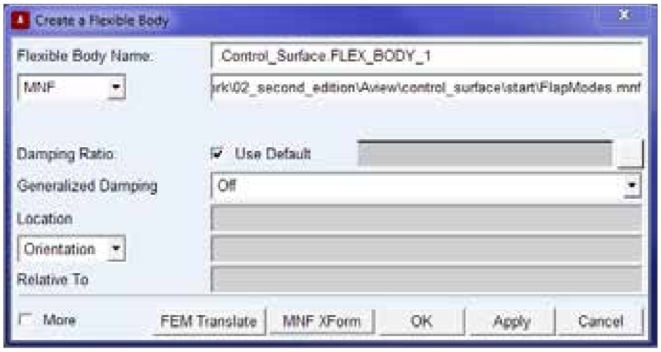
## Step 2. Import .mnf File

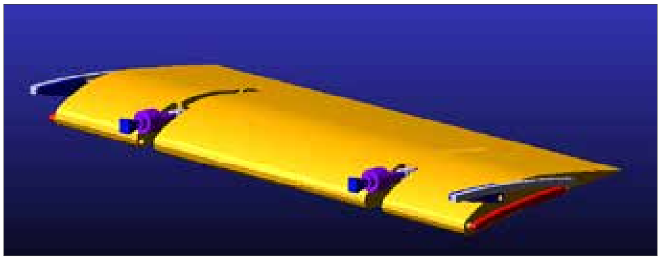
a. At this stage the control surface is not included in the model. There are two ways to include it in the model. The first one is to import the parasolid geometry and then make it a flexible body. The second one is to generate the .mnf file first and then import it into the model. In our example, we use the second method. This is the case when .mnf has been prepared by CAE engineer and then provided to Adams Engineer.

b. From **Bodies** ribbon, select **Flexible Bodies: Adams/Flex: create flexible body via MNF import**.

c. Right click in the file next to file type and browse for **FlapModes.mnf**.

d. Make sure that you **don't change the flexible body name**. Otherwise you won't be able to create joints later by importing the .cmd file we provide to you. Leave the rest of the fields as default and click OK.



## Step 3. Import .cmd File to Create Joints

a. We have been exporting and importing .cmd files many times throughout our tutorial kit. In this example, we will use .cmd files to create joints.

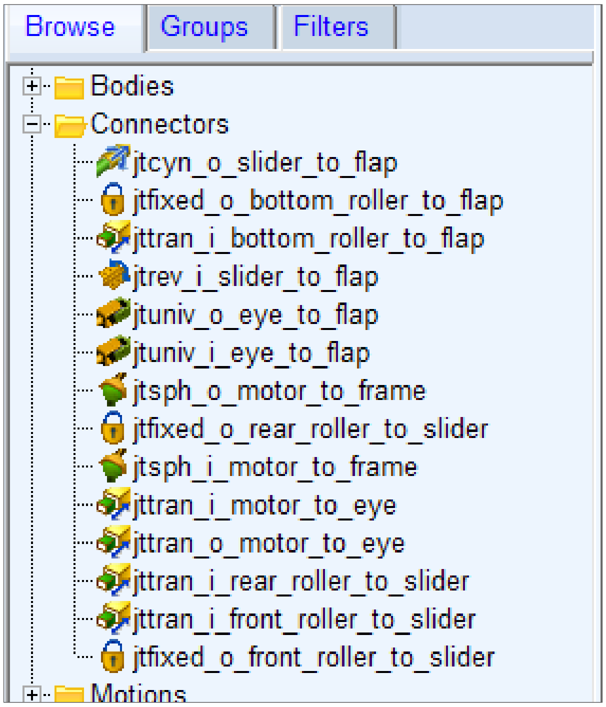
b. From the menu bar, go to **File->Import**.

c. Select **Adams/View Command File** for **File Type**.

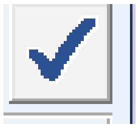
d. Right click in the field next to File To Read and browse for **Create_Joints.cmd**.

e. Click **OK.** Then you will find that several joints have been created under Connectors. What is included in .cmd file is called Adams/View Command Language. Actually, you can right click on the .cmd file and use any text editor to view its content. For more information on Adams/View Command Language, please **refer to Adams Online Help**.

## Step 4. Compare Between Rigid and Flexible Body




a. From Simulation ribbon, select either mode of simulation and verify the model.

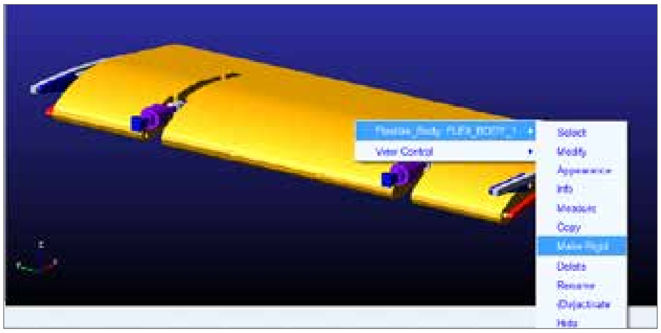


b. The result says that we have 80 degrees of freedom with no redundant contraint equations.

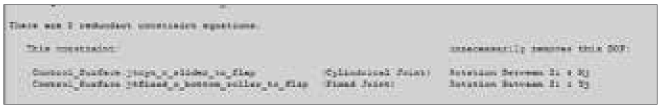
c. Close the information window and make the control surface rigid.

```
VERIFY MODEL: .Control_Surface

  80 Gruebler Count (approximate degrees of freedom)
  12 Moving Parts (not including ground)
   1 Flexible_Bodys
   1 Cylindrical Joints
   1 Revolute Joints
   2 Spherical Joints
   5 Translational Joints
   2 Universal Joints
   3 Fixed Joints
   2 Motions
   6 Point_Curves

  80 Degrees of Freedom for .Control_Surface

There are no redundant constraint equations.


Model verified successfully
```

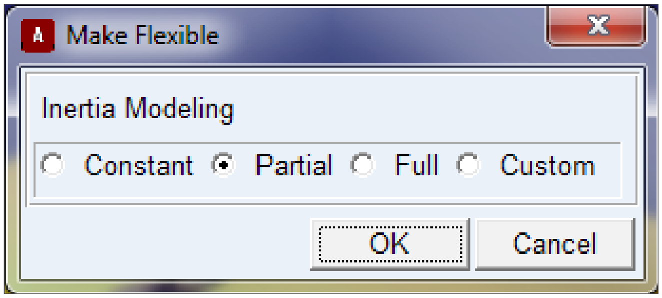
d. Right click on control surface and select **Make Rigid**.




e. Verify the model again. Now the result tells us that there two redundant constraints. Sometimes, Adams will still run even with the presence of redundant constraints because Adams will remove redundant constraints during simulation as long as they are consistent. However, as the simulation runs, chances are that the redundant constraints will become inconsistent, which will cause the simulation to fail. Hence, including a flexible body in the model when necessary will help us to circumvent this situation.

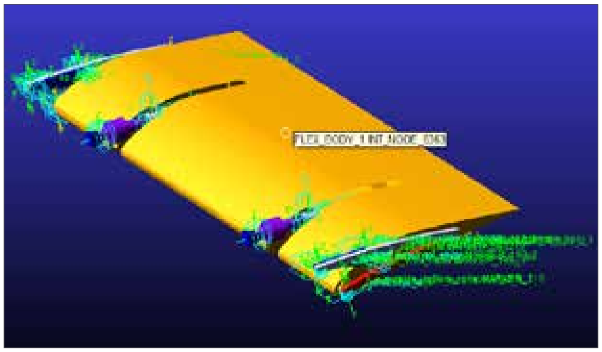


f. Close the information window and make the body flexible again. Pick **partial inertia modeling**.

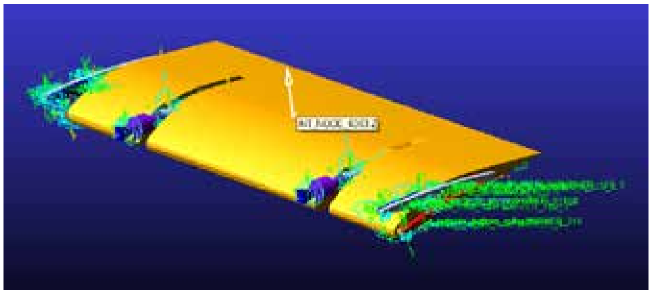



## Step 5. Add Loads to the Control Surface

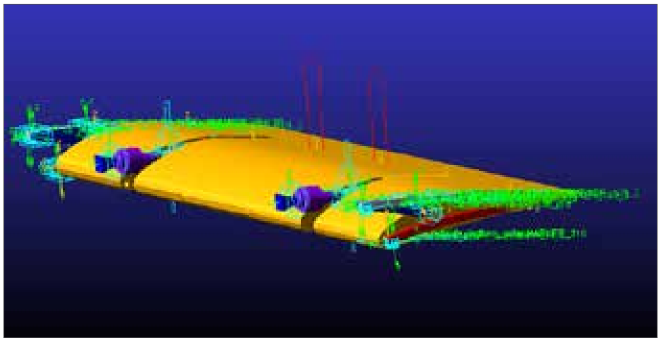
a. To facilitate selection process, press Shift+S to change the render mode to Shaded. Notice that you can now find some nodes with maize coordinate system markers.
b. From **Forces** ribbon, select **Applied Forces: Create a Single Component Force**.
c. Select **Space Fixed** for Run time Direction.
d. Select **Pick Feature** for Construction.
e. Pick Flex_Body_1 as the body.
f. Pick Node 6363 as the point of application. You can right click on the area around its location and pick Node 6363 from the selection list.



g. Pick **Node 6363.Z** as direction. Again, if you are having trouble pick the feature, right click on the region and select from the selection list.



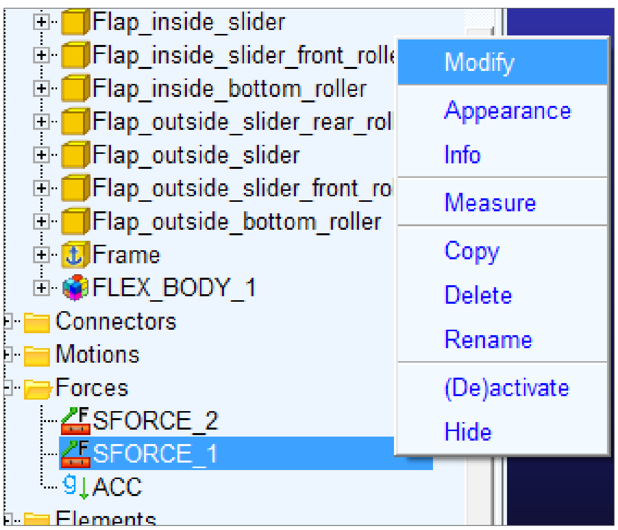
h. Use the same method to create a force at Node 6364 with the same direction.

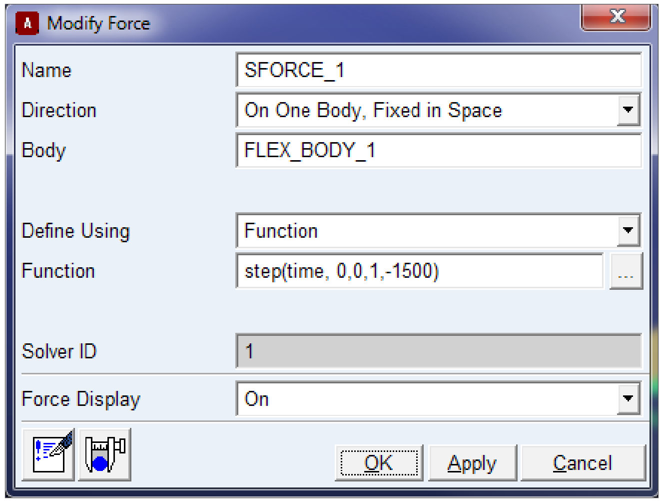


i. Right click on **SFORCE_1** and select **Modify**.

j. Enter **step(time, 0,0,1,-1500)** in the function field.

k. Modify **SFORCE_2** to have the same function. These forces are not aerodynamically accurate. But they are sufficient for illustrative purposes. With data available, you can easily exert forces on flexible bodies in the same manner.

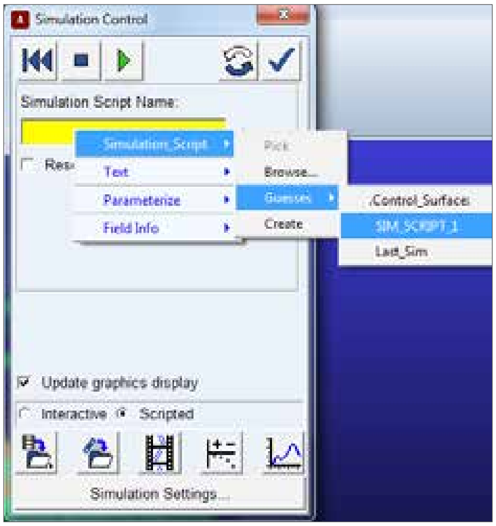



## Step 6. Run a Scripted Simulation

a. From **Simulation** ribbon, select **Simulate: Run a Scripted Simulation**.

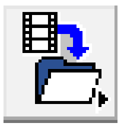
b. Right click in the field and select **SIM_SCRIPT_1**.

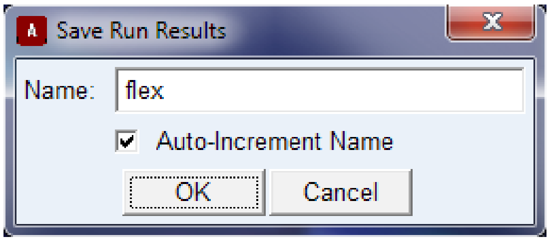



c. Click **Start Simulation**.

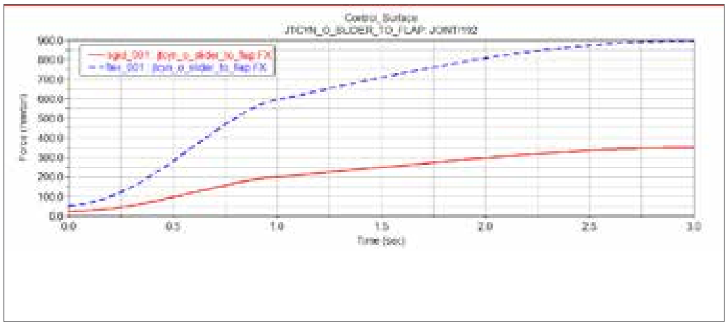
## Step 7. Review Simulation result

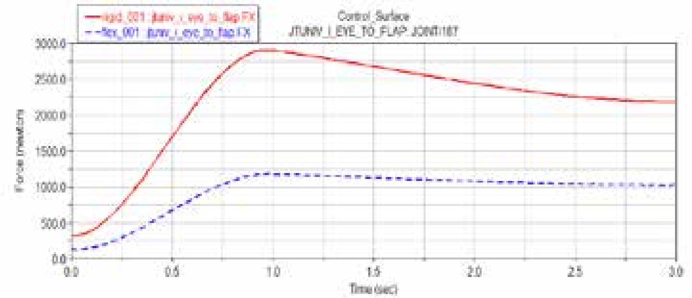
a. From the simulation control panel, save the simulation result as flex.





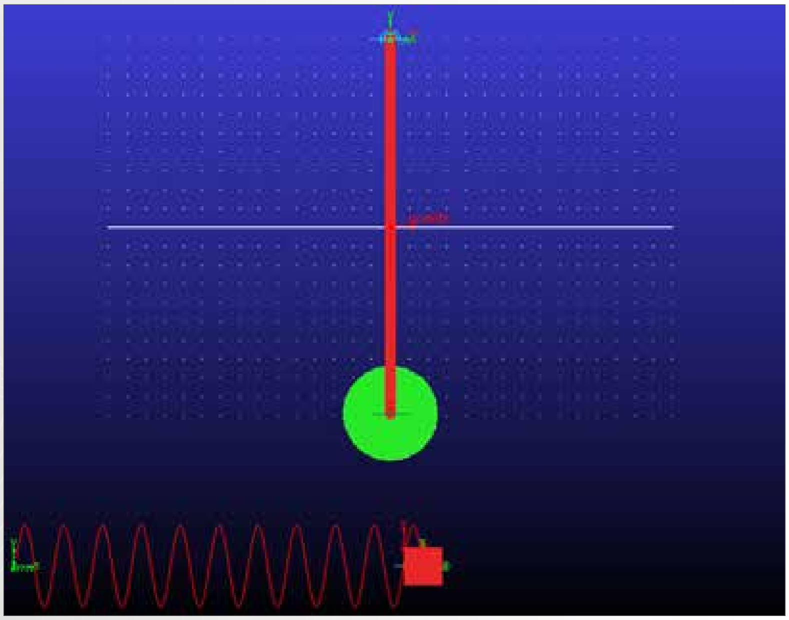
b. Make the body rigid and run the same scripted simulation again. Save the simulation result as rigid.
c. Press **F8** to bring the postprocessing window.
d. Now you can explore the simulation results.




e. You may find that the difference of two modeling methods is distinct in terms of forces at some of the joints. Please explore the result sets and compare the results of two different modeling techinques.

# Example 21: More on Pendulum



## Software Version

Adams 2013.2

## Problem Description

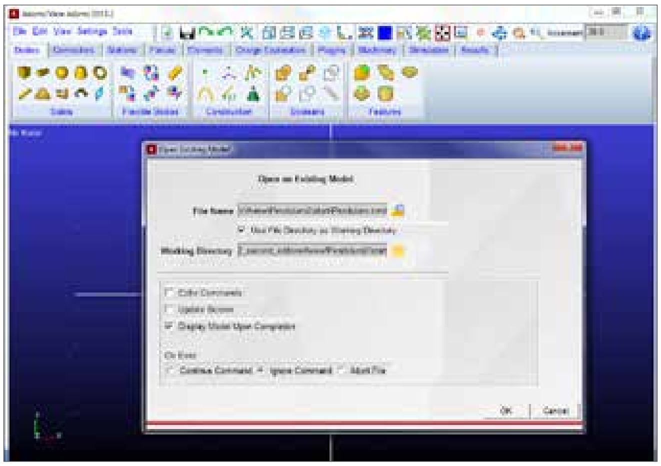
In our previous example of a pendulum, we studied the dynamics of a simple pendulum. In this example, we will further investigate the pendulum but with more complicated models. We will focus on comparing results of pendulums with different set ups. The parameters of the double pendulum referenced in this example is provided by Mechanical Engineering Control Systems Lab at California Polytechnic State University.

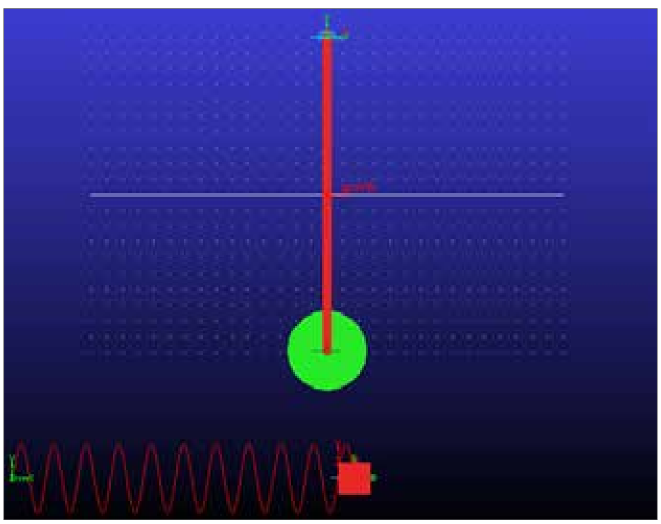
## Step 1. Start Adams/View

a. Start Adams/View 2013.2.

b. Select **Existing Model** and browse for **Pendulum.cmd** in this example's start folder.

c. Click **OK**.

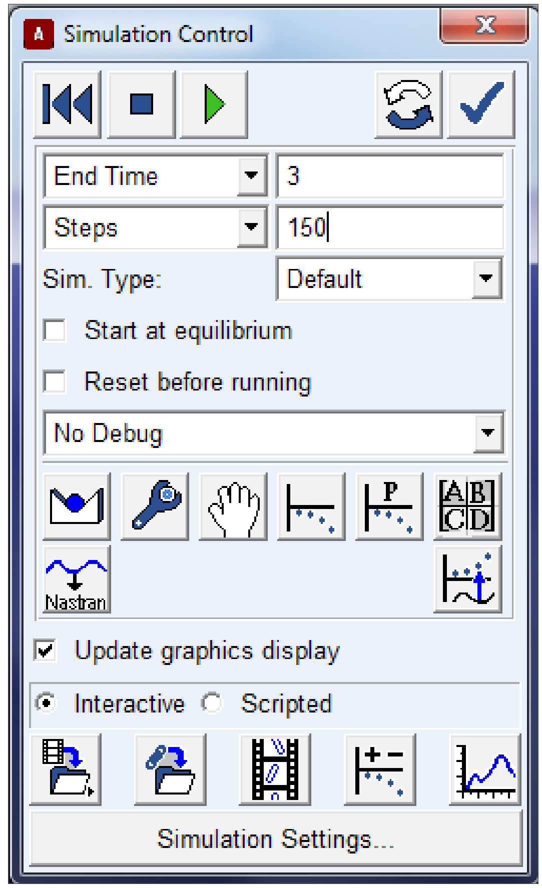


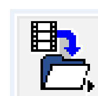


## Step 2. Run a Baseline Simulation

a. The configuration of the pendulum at default is explained now. The center of mass of the rod is defined at the end of the rod. And the rod has no inertia at all. This makes the default configuration a simple pendulum. Beneath the pendulum is a simple mass-spring system. It has the same natural frequency as the pendulum.

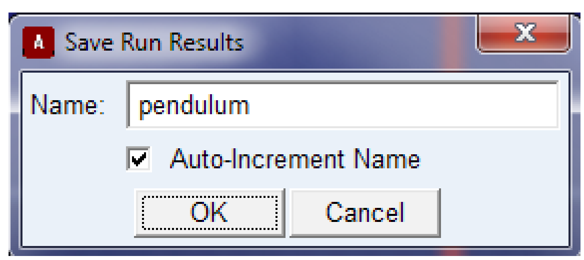
b. Run an interactive simulation with 3 seconds and 150 steps.



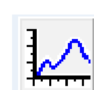



c. Click **Save Run Results**.

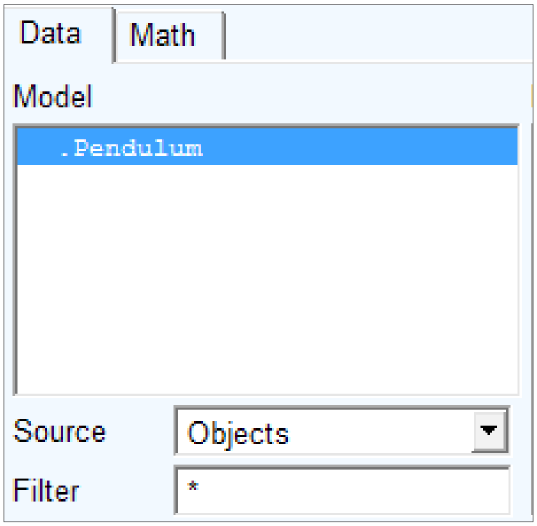
d. This will store the simulation result automatically so that we can plot the last run with previous runs together. Name the runs as pendulum and check Auto-Increment Name. This will add a number after the name for each run to distinguish them from each other.



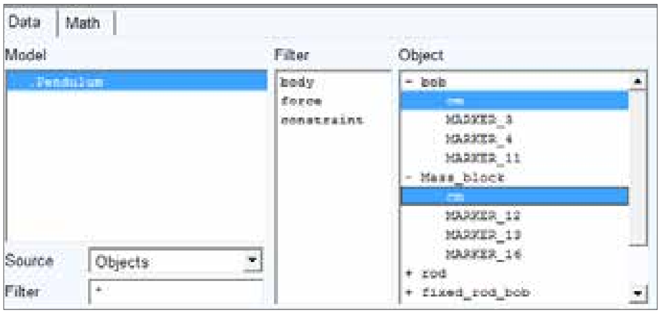

e. Click plotting to enter postprocessing window.



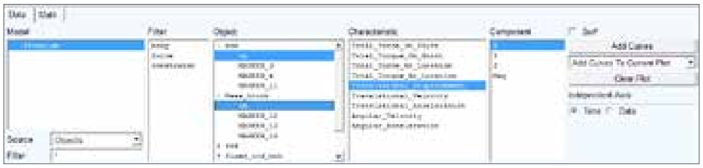
f. In plotting page, make sure that **Objects** is selected as the source.

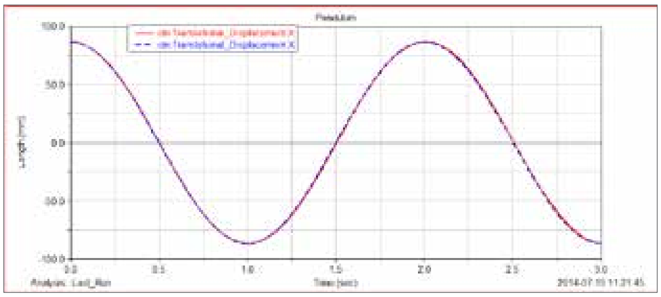



g. Then, hold Ctrl key and highlight both **bob.cm** and **Mass_block.cm**.




h. Then select **Translational_Displacement** and **X** component. Click **Add Curves**.





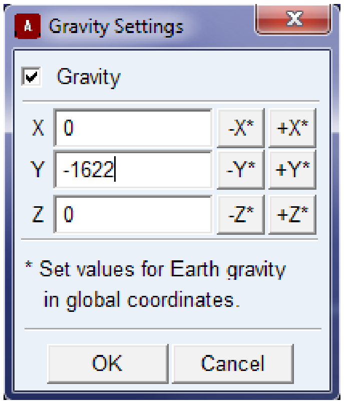
i. It can be found that these two systems have exactly the same amplitude and natural frequency. They act like each other despite the distinct geometric difference.

## Step 3. Move the Pendulum to the Moon

a. What if the pendulum is swinging on the moon? We can simply change the value of the gravity and check the results.

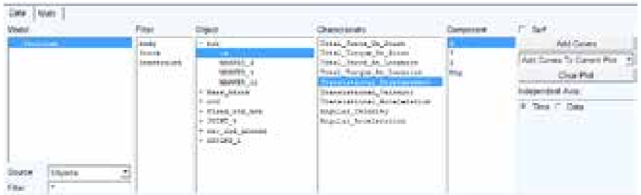
b. From the menu, go to **Settings->Gravity**.

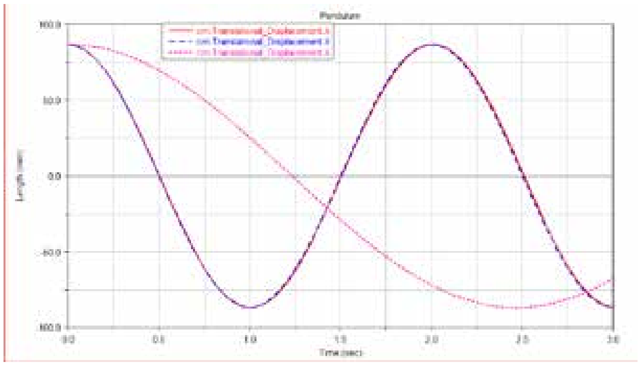
c. Change the value to **-1622** (**MMKS** unit system). Click **OK**.




d. Run the simulation with the same settings (3 seconds, 150 steps) again and save the run results as well.

e. Now, switch to postprocessing window again and plot the **X component of Translational_Displacement of bob.cm**. The plot shows that the period of the pendulum is longer as it is inversely proportional to the square root of the gravity.



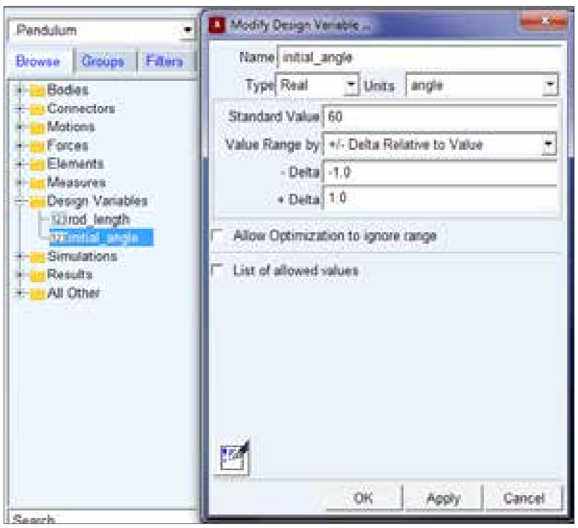


f. Go to **Settings->Gravit**y again and **click on –Y** to change the gravity back to default value.

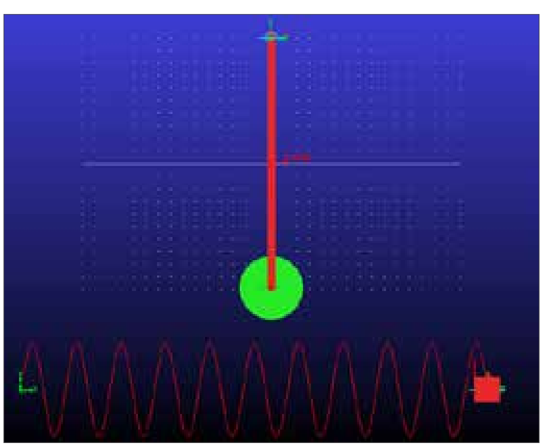
## Step 4. Change the Initial Position to a Larger Angle

a. In this step, the initial angle of the pendulum is increased from 5 degrees to 60 degrees. The aim is to show that small angle approximation will fail at relatively large deflection.

b. From the model tree, double click Design Variable **initial_angle**. Then change the value from **5** to **60**. Click **OK**.



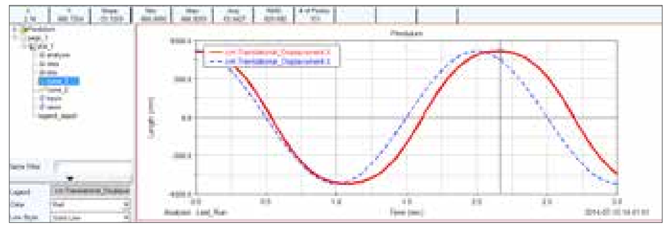

c. You should notice that the mass block as well as the spring has extended to a new initial position corresponding to the pendulum. No graphical update occurs to the pendulum because the modification will only show up when the simulation starts. The model is parameterized so that once we change the initial_angle variable, the other parameters change accordingly.

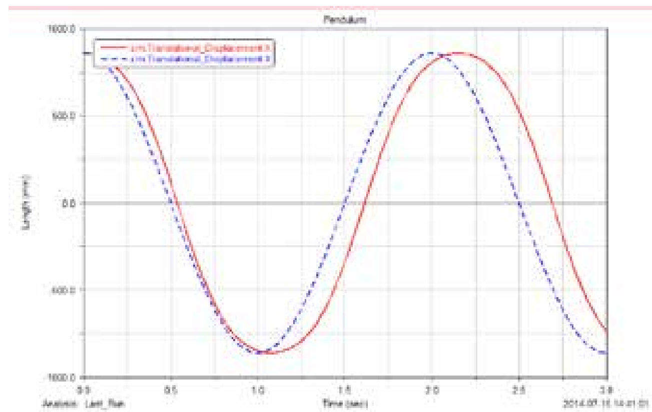


## Step 5. Run the Simulation Again

a. Again, run a 3 seconds-150 steps simulation and save the result.

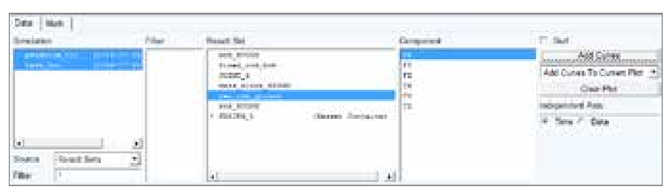
b. Return to postprocessing window and plot the translational displacement of bob.cm and mass_block.cm. Notice that the period has changed from 2 seconds to 2.16 seconds.

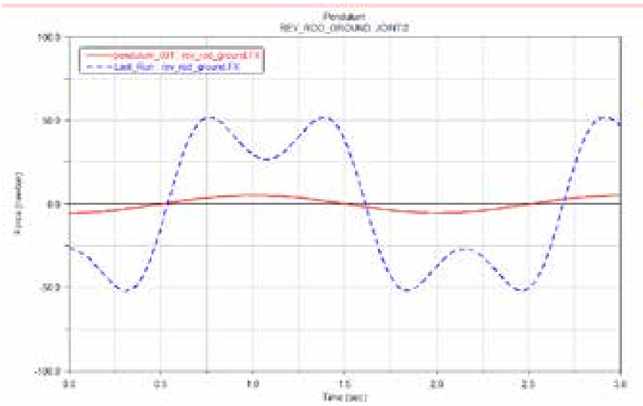





c. The change in period may not be that significant. However, if we check the force on the revolute joint at the end of the rod, we can find a huge difference.

d. Change Source from Objects to **Result Sets** and select the first and the last simulation, select **rev_rod_ground** and pick **FX** component and then click **Add Curves**.






e. The curve in magenta is the curve of the large deflection. It is not even a sinusoidal curve. This reminds us that the equation

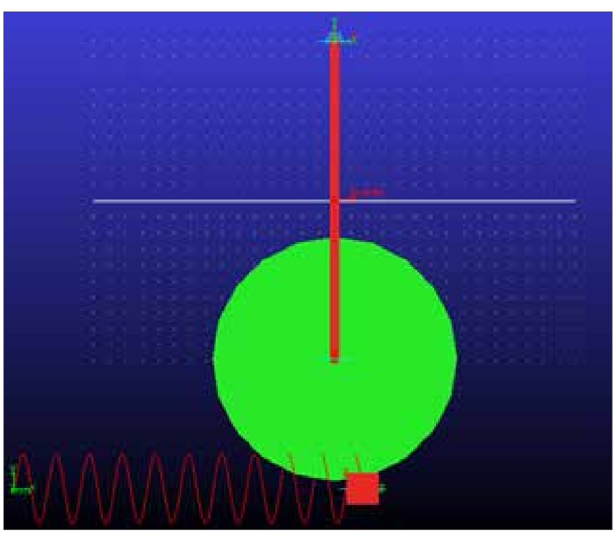
$$T \approx 2\pi\sqrt{\frac{L}{g}}$$

applies only to pendulums with small angle deflection.

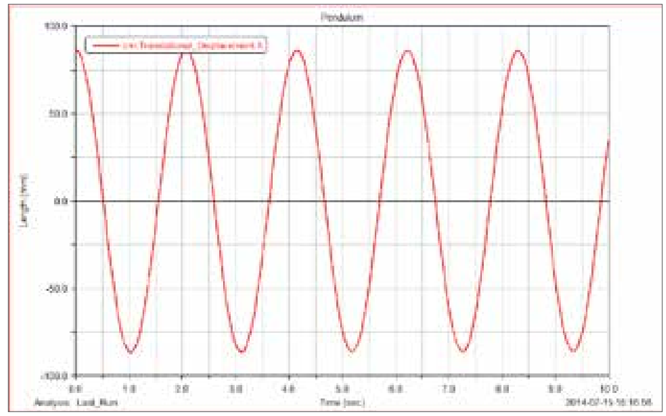
## Step 6. Add a Revolute Joint between Rod and Bob

a. In this, step, we will investigate the effect of the inertia of the bob on the oscillation of the pendulum.

b. Double click **CYLINDER_2** from the model tree and change the radius to **15in**. Click **OK**.



c. Run the simulation and plot the x component translational displacement of bob.cm as we have done in previous steps.

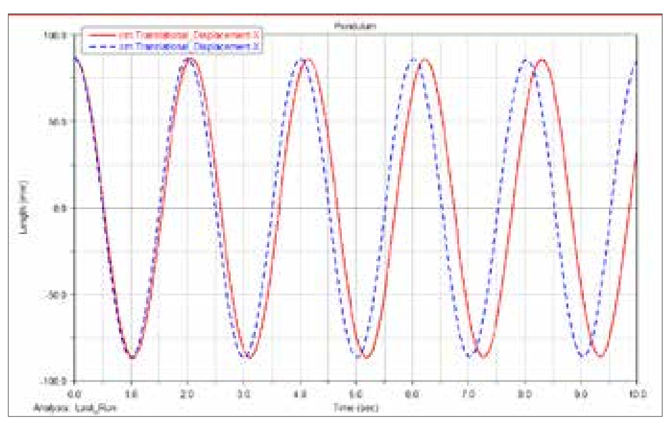



d. From the animation you can already find that as the radius of the bob increases, the period also increases. Simple pendulum neglects the effect of the inertia of the bob.

e. Now, **deactivate joint fixed_rod-bob** and **activate joint rev_rod_bob**.


f. Run the simulation again. In postprocessing window, in the same plot, add the curve of x component of translational displacement of bob.cm.

g. We find that the pendulums gradually oscillate out of phase. The one with the revolute joint (blue dashed curve) oscillates faster.

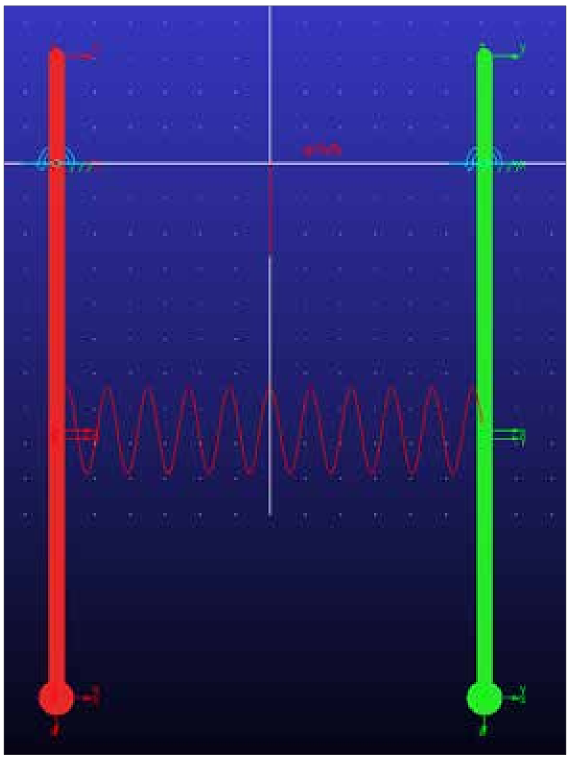



h. The reason is that as the bob is set free, it is not rotating about its center of mass. That portion of energy is converted to the translational motion of the pendulum thus making the pendulum to finish one swing quicker.

## Step 7. Double Pendulum

a. In this step, a double pendulum connected by a spring is modeled This is a system with two DOFs. We will run simulations with different initial conditions to find out the two normal modes of the system.

b. Go to **File->New Database**. Select **Existing Model** and pick **Double_Pendulum.cmd** from the exercise

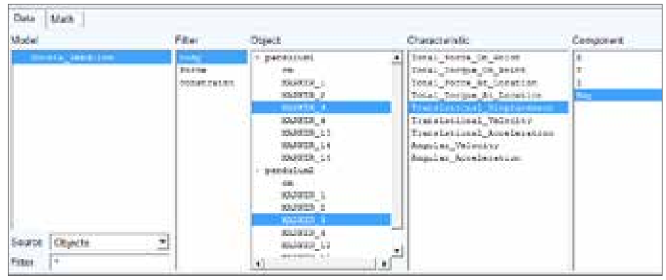


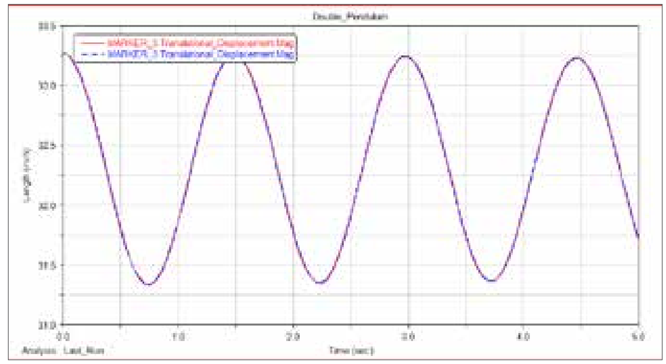
start directory.

c. Double click on each of the revolute joints and then click **Initial Conditions** to view the I.C. at each Joint.

d. From **Simulation** Ribbon, run an **interactive simulation** with 5 seconds and 100 steps.

e. Then press F8 to enter postprocecssing window. Hold Ctrl key and highlight both **pendulum1. MARKER_3** and **pendulum2_MARKER_3**. Under Characteristic, select **Translational Displacement.** Under Component, select **Mag**. Then click **Add Curves**. It can be found that the period of the system is 1.5 seconds. The two pendulums have the same magnitude of oscillation but in the opposite direction of motion.

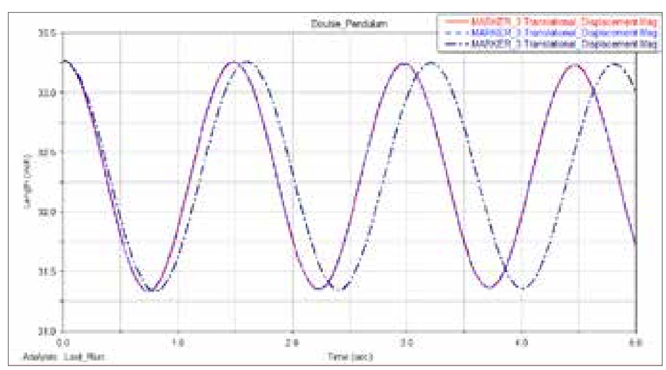





f. Double click on one of the revolute joints and click **Initial Condition**. Change the initial rotational displacement to the opposite sign so that initially, the pendulums will start with the same amount of deflection in the same direction.

g. Run the simulation again and press F8 to enter postprocessing window.

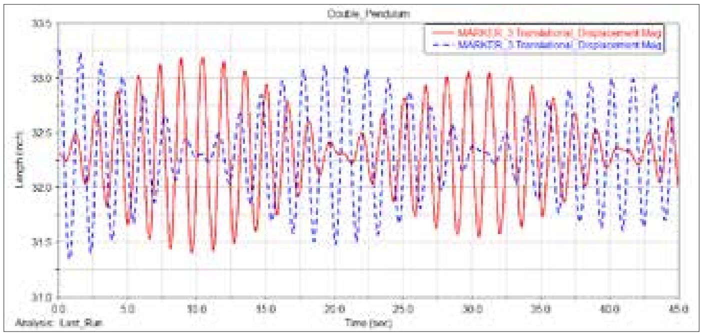
h. Plot Translational Displacement of pendulum2. MARKER_3 on the same page with the previous curves. This time, it can be found that the period of the system now is 1.6 seconds. The two pendulums still have the same magnitude of oscillation but they have the same phase angle.



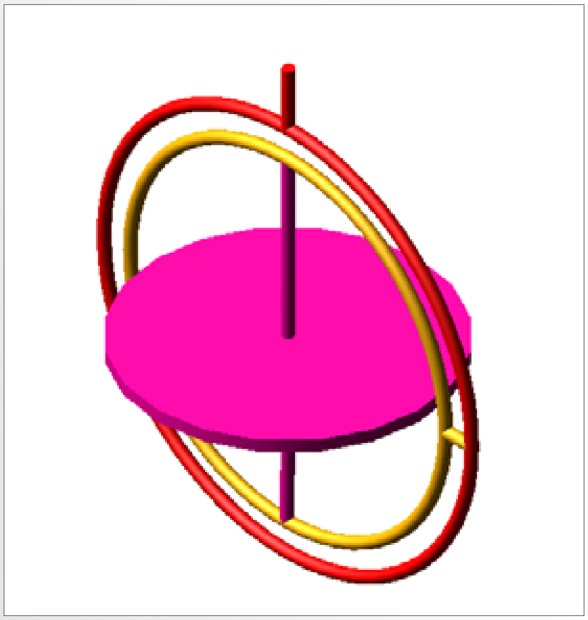

i. Another interesting simulation would be setting the initial displacement of pendulum1 to zero and maintain the initial angle of pendulum2. You may change the

Initial Conditions like instructed in Step 7f. Then, run a simulation with 45 seconds and 900 steps.

j. Now, plot the magnitude of Translational_Displacement of MARKER_3 of both pendulums again. The curves are not simply sinusoidal anymore. But we can still find out a periodic oscillation.



# Example 22: Gyroscope



## Software Version

Adams 2013.2

## Problem Description

A gyroscope is a device for measuring or maintaining orientation. A gyroscope usually comprises an outer cage (frame), an inner cage (gimbal) and an inertia wheel (rotor). The rotor is spinning at high speed so that external disturbances cannot change its orientation too much due to its large angular momentum. In this example, we will show you how to build a gyroscope model in Adams/View and properly measure the angle as well as export the data. Outer cage, inner cage and inertia wheel are connected to each other via revolute joints while the assembly is connected to the ground via a spherical joint to assume any orientation.

## Step 1. Create a New Database

a. Open Adams/View 2013.2.

b. Select **New Model**.

c. Name the new model **Gyro**.

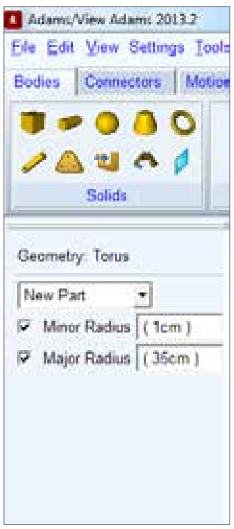
d. Select the directory of your choice.

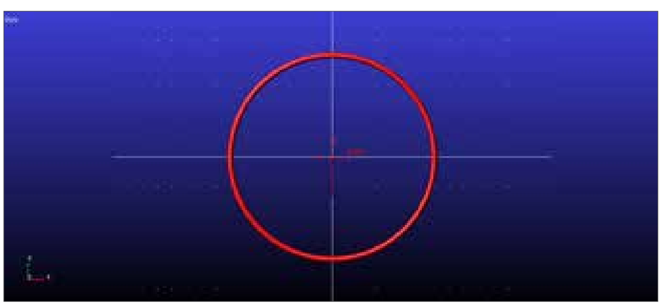
## Step 2. Create the Outer Cage

a. From **Bodies** ribbon, select **RigidBody: Torus**.

b. Select **New Part** and check **Minor radius** and **Major radius**. Set them to **1cm** and **35cm** respectively.

c. Click on the **Origin** and the outer cage will be created.



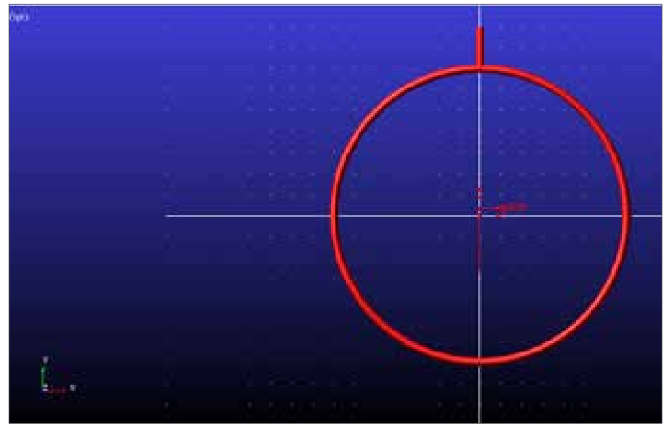
d. From **Bodies** ribbon, select **RigidBody: Cylinder**.

e. Select **Add to Part** and leave length unchecked. Check **radius** and set it to **1cm**.

f. Select the torus just created and right click at the background to invoke the Location Event.

g. Enter **0, 350, 0**, and click apply then enter **0, 450, 0**. These steps will create a cylinder under PART_2 and it starts at 0, 350, 0 and ends at 0, 450, 0.

h. Rename the part to **OuterCage**.



## Step 3. Create the Inner Cage

a. From **Bodies** ribbon, select **RigidBody: Torus**.

b. Select **New Part** and check **Minor radius** and **Major radius**. Set them to **1cm** and **30cm** respectively.

c. Click on the Origin and the inner cage will be created.

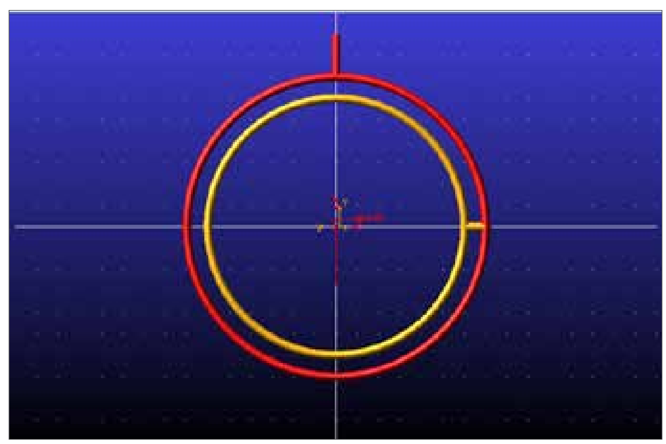
d. From **Bodies** ribbon, select **RigidBody: Cylinder**.

e. Select **Add to Part** and leave length unchecked. Check **radius** and set it to **1cm**.

f. Select the torus just created and right click at the background to invoke the Location Event.

g. Enter **300, 0, 0**, and click apply then enter **350, 0, 0**.

h. Rename PART_3 to **InnerCage**.



## Step 4. Create the Inertia Wheel

a. From **Bodies** ribbon, select **RigidBody: Cylinder**.

b. Select New Part and leave length unchecked. Check **radius** and set it to **1cm**.

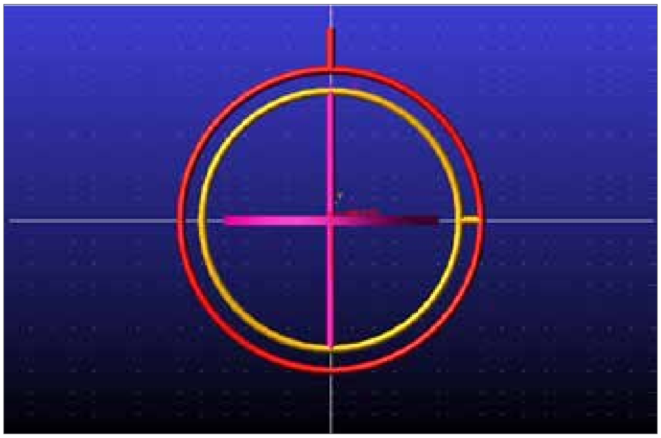
c. Right click in the background and enter **0, 300, 0** in the Location Event. Click apply and enter **0, -300, 0**.

d. From **Bodies** ribbon, select **RigidBody: Cylinder**.

e. Select **Add to Part** and leave length unchecked. Check **radius** and set it to **25cm**.

f. Select the cylinder just created (PART_4) right click in the background and enter **0, -10, 0** in the Location Event. Click apply and enter **0, 10, 0**.

g. Rename PART_4 to **InertiaWheel**.



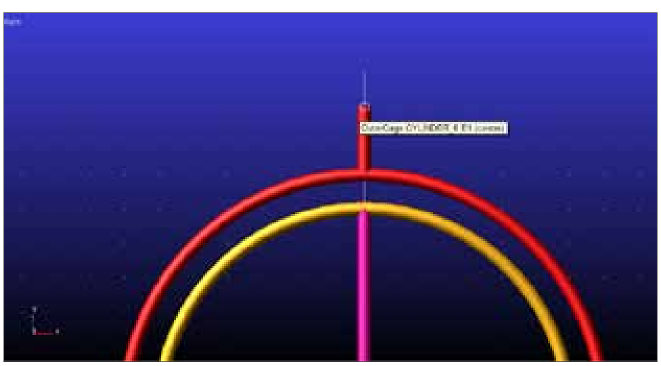
## Step 5. Create Spherical Joint

a. From **Connectors** ribbon, select **Joint: Spherical**.

b. Select **2 Bodies-1 Loc** and **Normal to Grid**.

c. Pick OuterCage first and then ground.

d. Select **OuterCage.CYLINDER_6.E1** (center) as the location. Note that the actual name of the point may vary but the idea is to create a spherical joint at the end of the cylinder on outer cage. If the point is not automatically snapped on, right click around it and select it from the selection list.



## Step 6. Create Revolute Joints

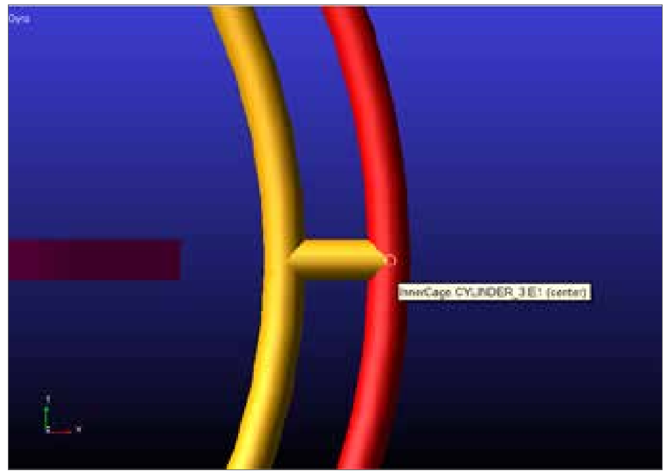
a. Change the working grid orientation to **Global YZ Plane**.

b. From the **Connectors** ribbon, select **Joint: Revolute**.

c. Select **2 Bodies-1 Loc** and **Normal to Grid**.

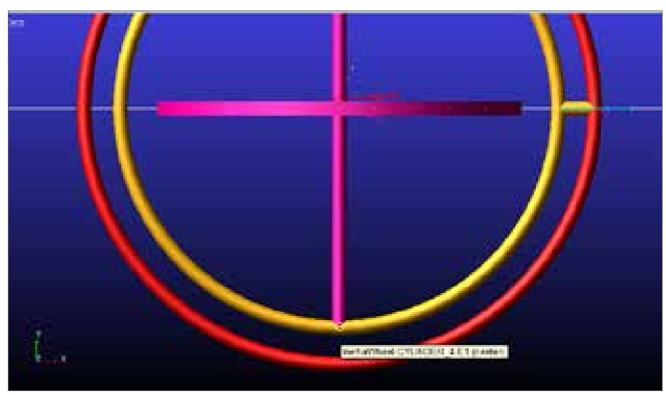
d. Pick OuterCage first and then InnerCage.

e. Select the end of the cylinder of Innercage.



f. Change the orientation of the working grid to **Global XZ Plane**.

g. Create a revolute joint between InnerCage and InertiaWheel at the connecting cylinder.



## Step 7. Verify the Model

a. From **Simulation** ribbon, run a simulation with 1 second and 50 steps. Check Start at equilibrium. The gyro should sit still.

## Step 8. Create a Disturbance

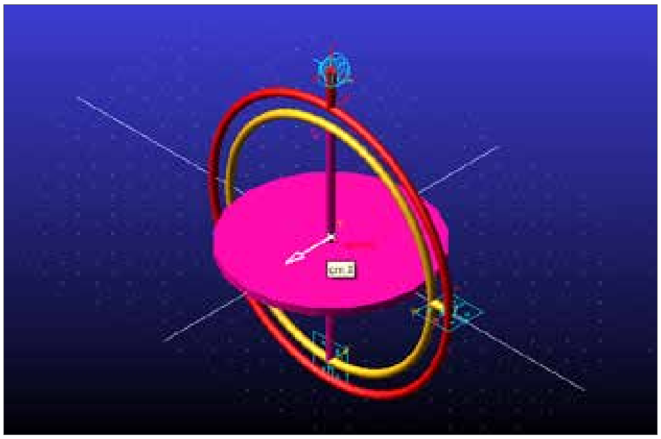
a. From **Forces** ribbon, select **Applied Forces: Single Component Force**.

b. Select **Body Moving** for Run-time Direction.

c. Select **Pick Feature** under Construction.

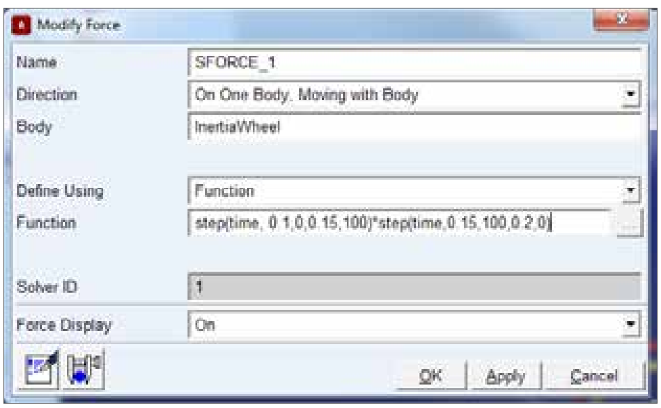
d. Select **InertiaWheel**, then right click on the center of **InertiaWheel** and pick **InertiaWheel.cm** from the list.

e. Select **cm.X** for the directon. You may need to switch to Iso view (shift+I) to help you make the selections.



f. When the force has been created, double click it under the model tree.

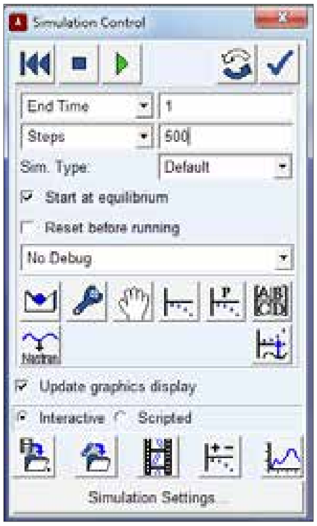
g. Under **Function**, enter **step(time, 0.1,0,0.15,100)*step(time,0.15,100,0.2,0)**. This function simulates an impulse disturbance applied upon inertia wheel that has a width of 0.1 second and an amplitude of 100 N.



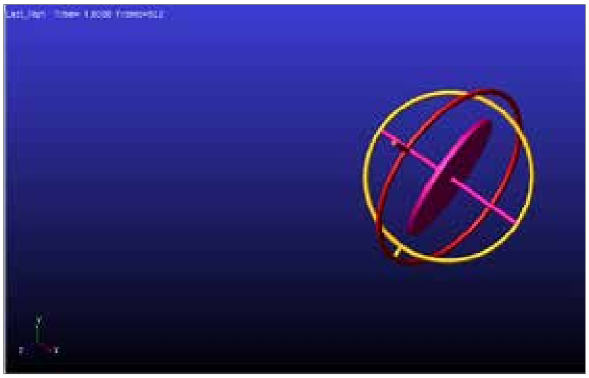

## Step 9. Run a Baseline Simulation

a. Run an interactive simulation of 1 seconds and 500 steps.

b. Check **Start at Equilibrium**.




c. It can be found that the gyro, without the inertia wheel spinning, moves rather drastically, especially towards the end of simulation.



## Step 9. Add Initial Condition to Inertia Wheel

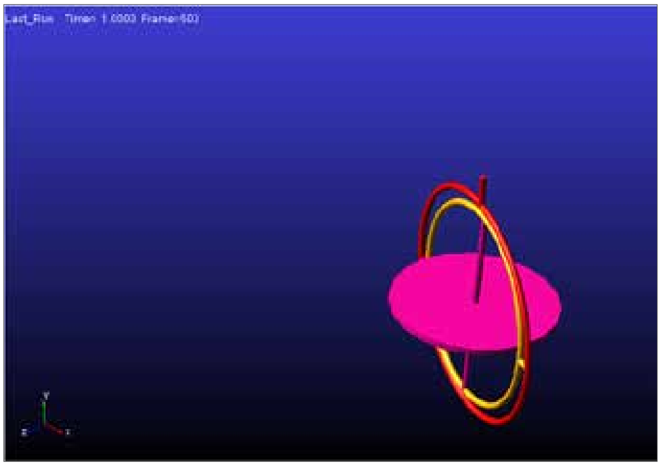
a. From the model tree, double click on **JOINT_3**.

b. Click on **Initial Conditions**.

c. Check Rot.Velo. and enter **36000**.
d. Click **OK**.

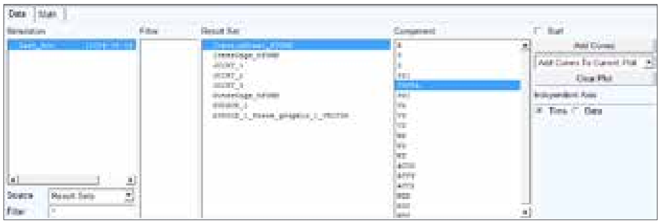
## Step 10. Run Simulation Again

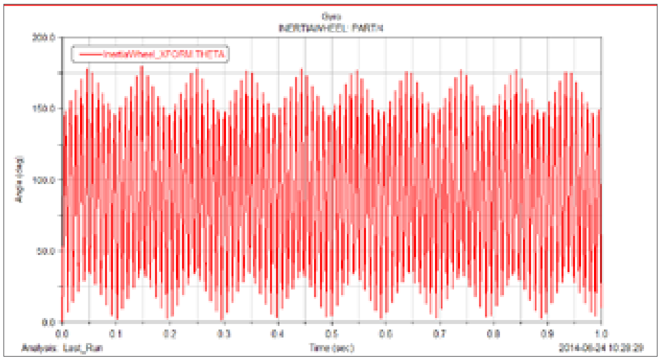
a. Run a simulation of 10 seconds and 500 steps.
b. Check **Start at Equilibrium**.
c. This time, although the cages were still moving due to the disturbance, the inertia wheel didn't behave as in the previous case. Due to the inertia of the spinning wheel, the gyro barely rotates.



## Step 11. Check Simulation Results

a. Press F8 to enter postprocessing window.
b. Under **Simulation list**, select **Last Run**.
c. Under **Source**, pick **Result Sets**.
d. Under **Result Set list**, hightlight **InertiaWheel_ XFORM**.
e. Highlight **THETA** under **Component**.
f. Click **Add Curves**.





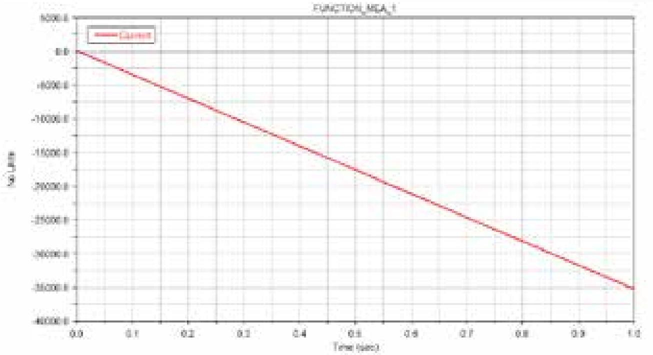
g. The plot of theta, angular data of the inertia wheel rotating around y axis is rapidly changing and discontinuous. In real-world application, this will cause difficulties for control algorithms.

## Step 12. Build an Angle Measurement

a. To avoid the problem mentioned above, we create a new measurement using function AY and check the results again.
b. From **Design Exploration** ribbon, select **Measures: Create a new Function Meaure**.
c. In the function builder, enter **AY(InertiaWheel. cm)*rtod**. Here, function AY measures the angle of center of mass of inertia wheel against global Y axis. Function rtod converts the result from radian to degree.

## Step 13. Run Simulation Again

a. Run the simulation again and check the results.

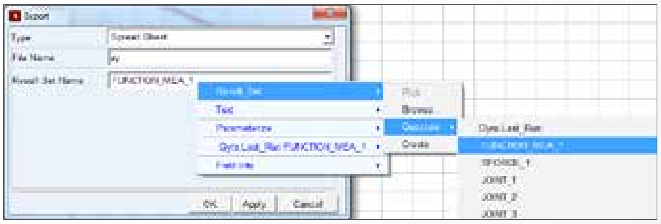


b. The result is now a continuous line.
c. To export the numerical data, in post processing

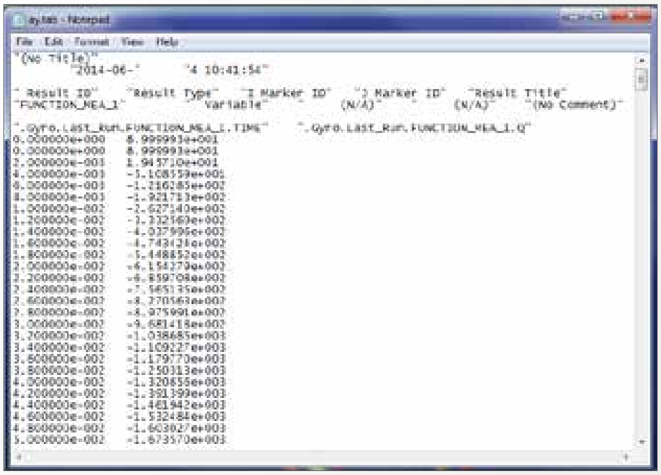
window, go to **File->Export->Spreadsheet**.

d. Enter file name **ay**.

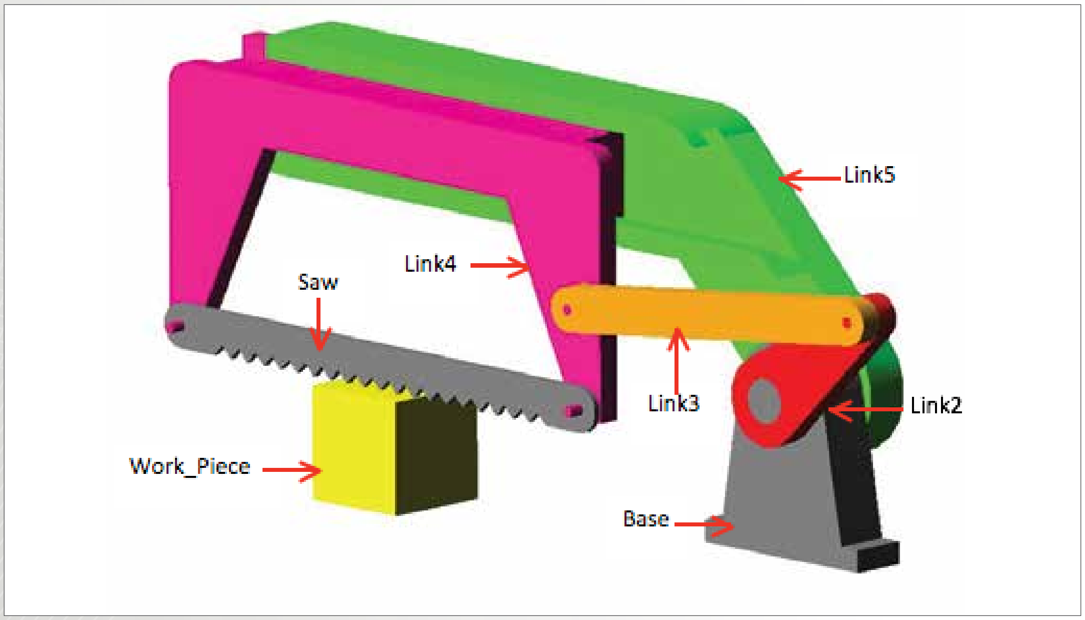
e. Right click in Result Set Name and browse for **FUNCTION_MEA_1**. The click **OK**.




f. Now, in your working directory, there should be a new file **ay.tab**. Open it with any type of text editor, and you can postprocessing the data for your own application.



# Example 23: Power Hacksaw Mechanism




## Workshop Objectives

Use **Adams/View** to

- Simulate the power hacksaw mechanism
- Create translational joint and revolution joints
- Apply motion to a revolution joint
- Define a contact between a solid and a solid

Use **Adams/PostProcessor** to

- Plot the horizontal stroke of the saw blade as a function of the angle of link 2.

## Software Version

Adams 2013.2

# Files Required

- **hackSaw.x_t**
- Located in the directory **exercise_dir/ Example 23**

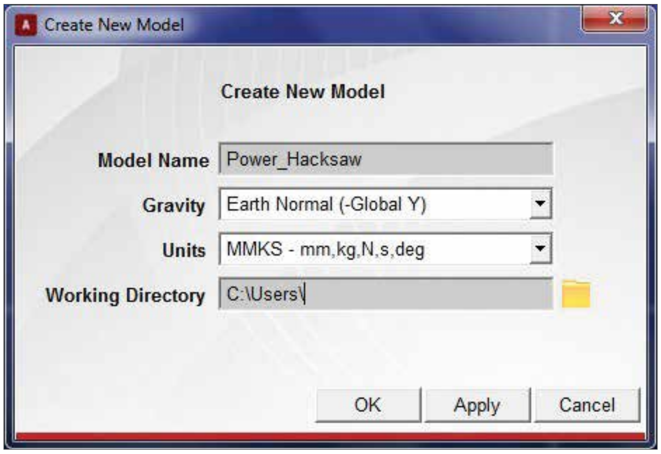
# Problem Description

- The model represents the power hacksaw, which is an offset crank slider mechanism.
- Link 2 is being rotated at a given velocity.
- Link 5 pivots at O5 and its weight forces the saw blade against the work piece while the linkage moves Link 4 back and forth on Link 5 to cut the work piece.

Adapted from Robert L. Norton (2012). Design of Machinery (Fifth Editon)

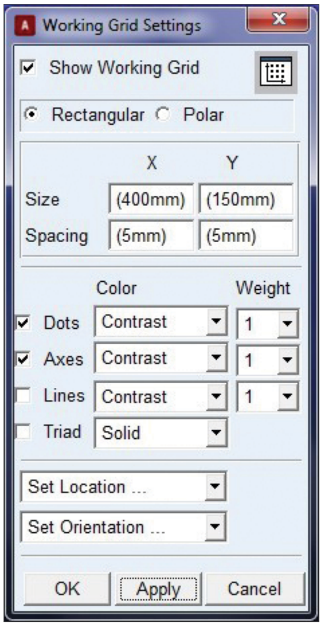
### Step 1. Start Adams/View and Create a Database

a. Start Adams/View.
b. From the Welcome dialog box, select **New Model**.
c. Replace the contents of the **Model Name** text box with **Power_Hacksaw**
d. Select **OK**.



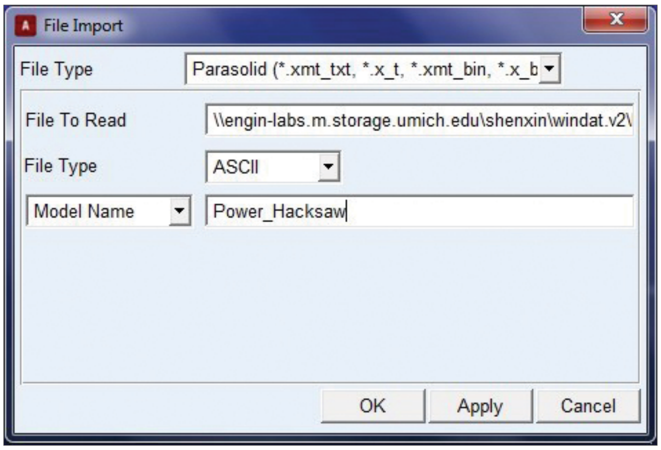

### Step 2. Set Up Work Environment

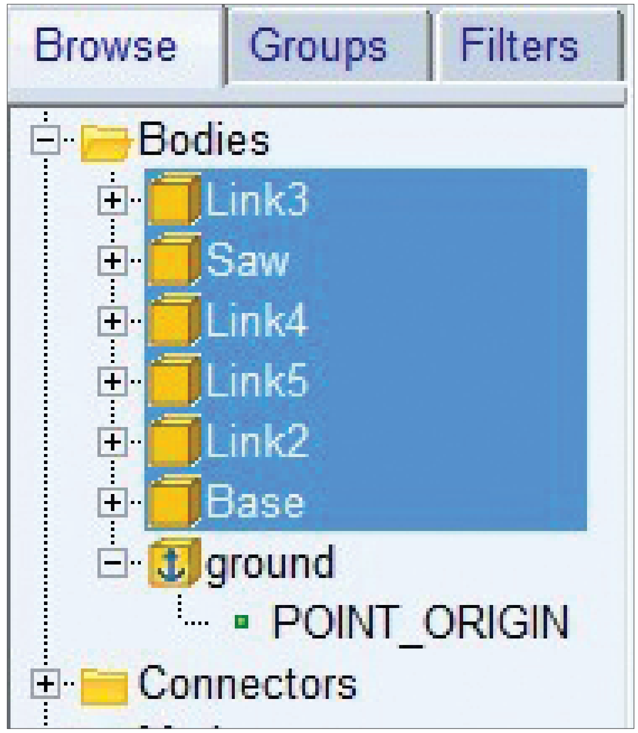
a. From the **Setting** menu, select **Working Grid**.
b. Set the grid size along X to 400 mm and along Y to 150 mm, and the grid spacing for X and Y to 5mm.
c. Select **OK**.
d. Press F4 on the keyboard to display the coordinates.



## Step 3. Import Part

a. From the Main Menu, select **File**, then click **Import...**
b. Replace the contents of **File Type** with **Parasolid**
c. Right-click the blank beside **File to Red** and select **Browse**.
d. Locate saved file **hackSaw.x_t** and click **OK**.
e. Select the default Model Name and type **Power Hacksaw** into the blank.
f. Click **OK**.




## Step 4. Move the Import Parts

a. From the Main Toolbox, select the ribbon Bodies.
b. Select **Geometry: Point**.



c. Click the origin point **(0.0, 0.0, 0.0)** in the working space.
d. Rename the new point as **POINT_ORIGIN**
e. Select all the parts under the **Bodies** tree.
f. From the Main Toolbox, right-click **Position: Reposition objects**.

g. Select **Position-Move**.

h. In the **Position:Move** dialog, check **Selected** and select **From To** method.
i. Select the point **Base.cm**, and then select **POINT_ ORIGIN**.

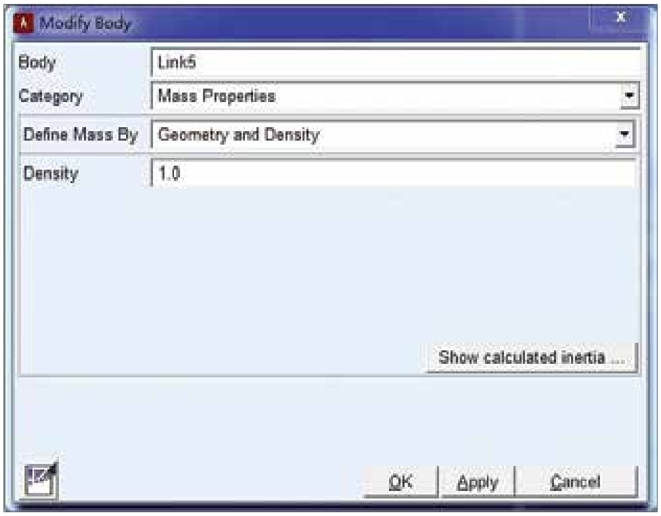
### Step 5. Change Mass of Link5

a. From the **Model Brower**, right-click **Link5** below the **Bodies** tree.

b. Select **Modify**.

c. Set **Density** to **1.0**.

d. Click **OK**.




### Step 6. Create Work Piece

a. From the Main Toolbox, select **Bodies**, and then select **RigidBody:Box**.

b. Use the default construction method **New Part**.

c. Check **Length, Height and Depth**, and then enter **50.0 mm**

d. Right -click at location **(-265, -55, 0)** in the working area.

e. Right-click the part and point to **Part: PART**, and then select **Rename**.

f. Enter **.Power Hacksaw.Work_Piece** in the **New Name** content.

### Step 7. Color the Parts

a. From **Model Browser**, left-click the plus sign beside **Base** displayed under the **Bodies** tree

b. Right-click **SOLID1**, and then select **Appearance**.

c. In the **Edit Appearance** dialog, enter **Gray** beside **Color**.

d. Click **OK**.

e. Repeat the above steps to change the color of the other parts.

| Part Name | Color Name |
|---|---|
| Link2 | Red |
| Link3 | Maize |
| Link4 | Magenta |
| Link5 | Green |
| Saw | Gray |
| Work Piece | Yellow |
| Base | Gray |

### Step 8. Connect the Parts Using Revolute Joints

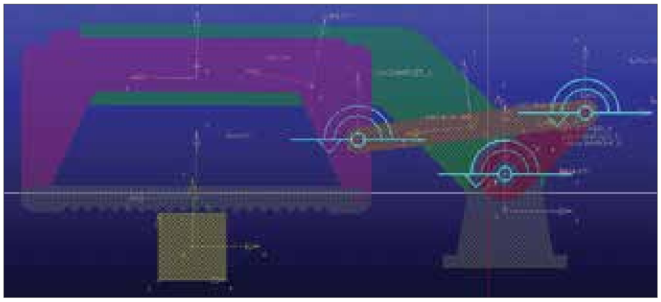
a. From the Main Toolbox, select **Connectors**, and then select **Create a revolute joint**.

b. To select the parts to attach, click the part **Base** and **Link 2**

c. Click the point in the table to set the joint's location.

d. Repeat the above steps to create three more revolute joints.

| 1st Body | 2nd Body | Joint Location |
|---|---|---|
| Link 2 | Link 3 | Link2.SOLID4.E16(center) |
| Link 3 | Link 4 | Link4.SOLID3.E56(center) |
| Base | Link 2 | base.SOLID1.E28(center) |
| Base | Link 5 | base.SOLID1.E28(center) |

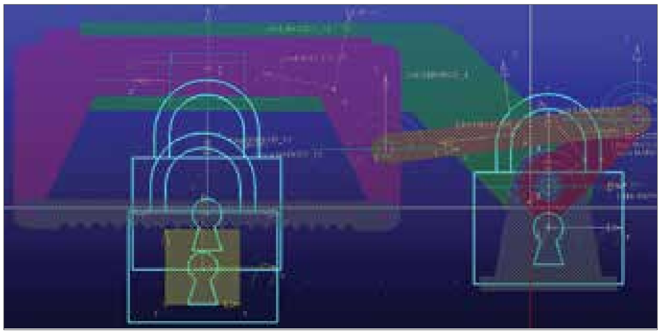


## Step 9. Create Fixed Joints

a. From the Main Toolbox, select **Connectors**, and then select **Create a fixed joint**.

b. Click the part **Saw**, the part **Link 4** and the CG of **Saw**.

c. Repeat the above steps to create fixed joints between **Base, Work Piece** and ground.



## Step 10. Connect Link 4 and Link 5 Using a Translational Joint

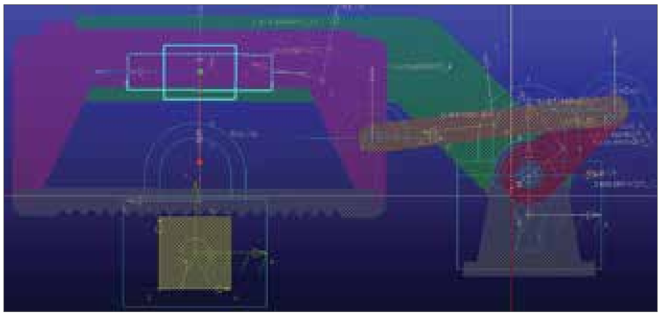
a. From the Main Toolbox, select **Connectors**, and then select **Create a Translational joint.**

b. Click the part **Link 4,** the part **Link 5** and the CG of **Link 4.**

c. Right-click the CG of Link 4 and select **Link4.cm.X**

d. Select any vector in **X-direction**

e.



## Step 11. Create Contact between Saw and Work Piece

a. From the Main Toolbox, select the ribbon Forces, and then select **Create a Contact.**

b. Right-click the text box of **I Solid(s),** point to **Contact_ Solid,** and then select **Pick**. Select the part **Saw**.

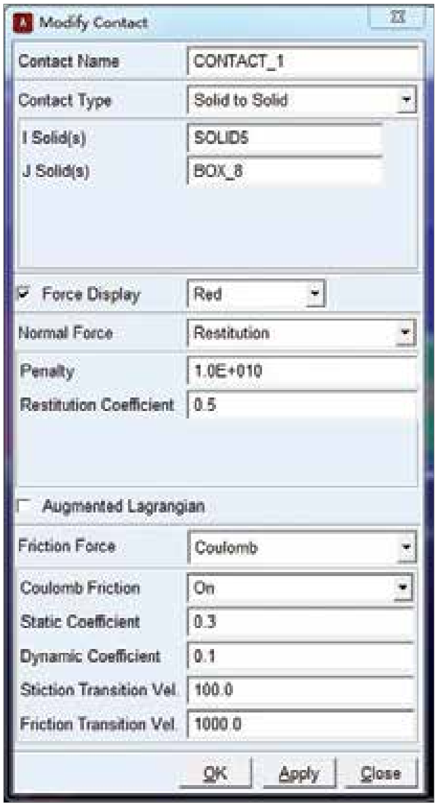
c. Select **Work_Piece** as **J Solid(s).**

d. Change the **Normal Force** to **Restitution**.

e. Set **Penalty** to **1.0E+010.** Set **Restitution Coefficient** to **0.5.**

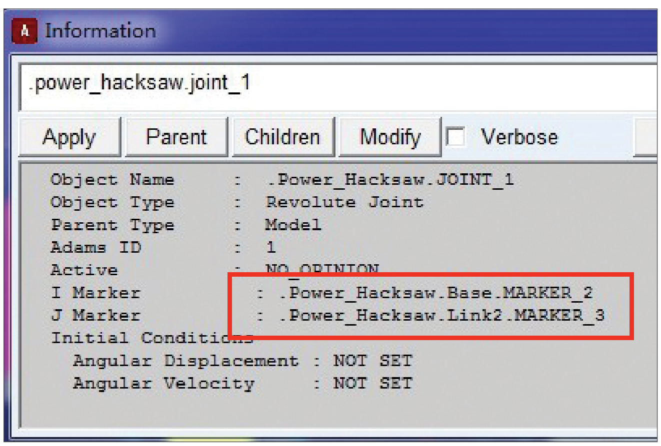
f. Select **Coulomb** as **Friction Force.**

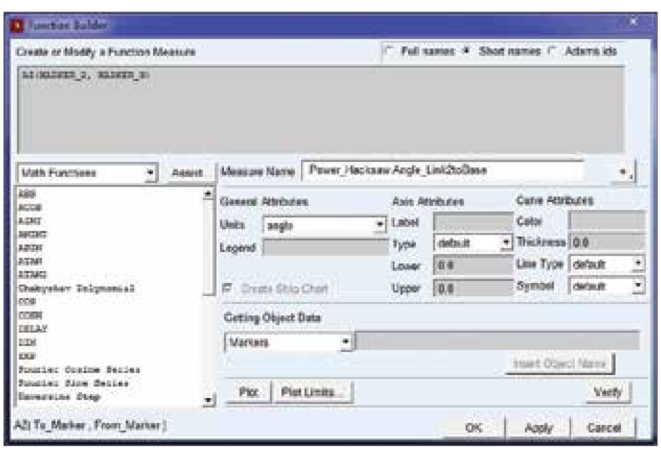
g. Click **OK**.



## Step 12. Create an Angle Measure

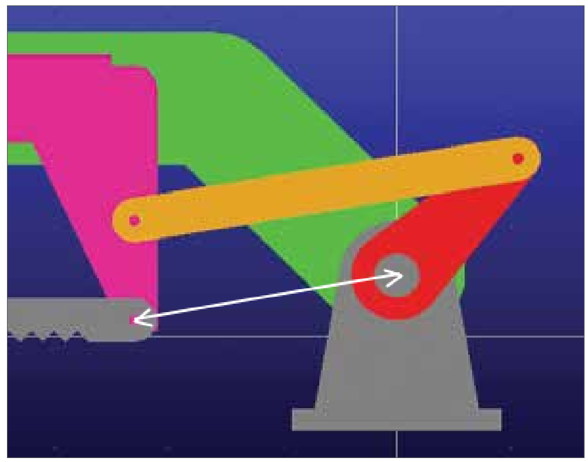
a. Under the **Connectors** tree in the **Model Browser**, right-click the revolute joint between **Link2** and **Base**.
b. Select **Info** to see the names of **I Marker** and **J Marker.**
c. Close the **Information** dialog.
d. From the Main Toolbox, select the **Design Exploration.**
e. Select **Create a new Function Measure.**
f. In the **Function Builder** dialog, enter **Angle_Link2toBase** as **Measure Name**.
g. Select **angle** as **Units**.
h. Select **Displacement**, select **Angle about Z**, and then click **Assist**…
i. In the **Angle about Z** dialog, right-click the contents of **To Marker** and **From Marker**, point to **Marker**, and then select **Browse**…
j. In the **Database Navigator**, select the markers in **Step b**.
k. Click **OK**.




## Step 13. Create a Horizontal Distance Measure

a. Select **Point** to **Point Measure**
b. Select **Displacement** as **Characteristic** and **Global X** as **Component**
c. Select the left hole center of the saw and then select **Base.POINT1**
d. Rename the measurement as **stroke**.



## Step 14. Create Motion on a Revolution Joint

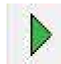
a. From the Main Toolbox, select the ribbon **Motions**, and then select **Rotational Joint Motion**.
b. Enter **30** in **Rot. Speed**
c. Select the revolution joint between the **Link2** and the ground.
d. From the model browser, expand **Motions**.
e. Right-click **MOTION_1** and select **Modify**.
f. In **Function (time)**, enter **55d + 30d*time**. (55d is the initial angel to keep Link2 horizontal at the beginning of the simulation.)
g. Click **OK**.

## Step 15. Simulate the Motion of Your Model

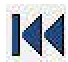
a. Click the ribbon **Simulation**, and then select the **Run an Interactive Simulation tool.**

b. Set up a simulation with an **End Time** of **15** seconds and **Step Size** of **0.1.**

c. Select the **Simulation Start** tool.

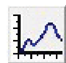


d. To return to the initial model configuration, select the **Reset** tool.



## Step 16. Use Adams/PostProcessor

a. In the Simulation Control panel, click **Plotting**.

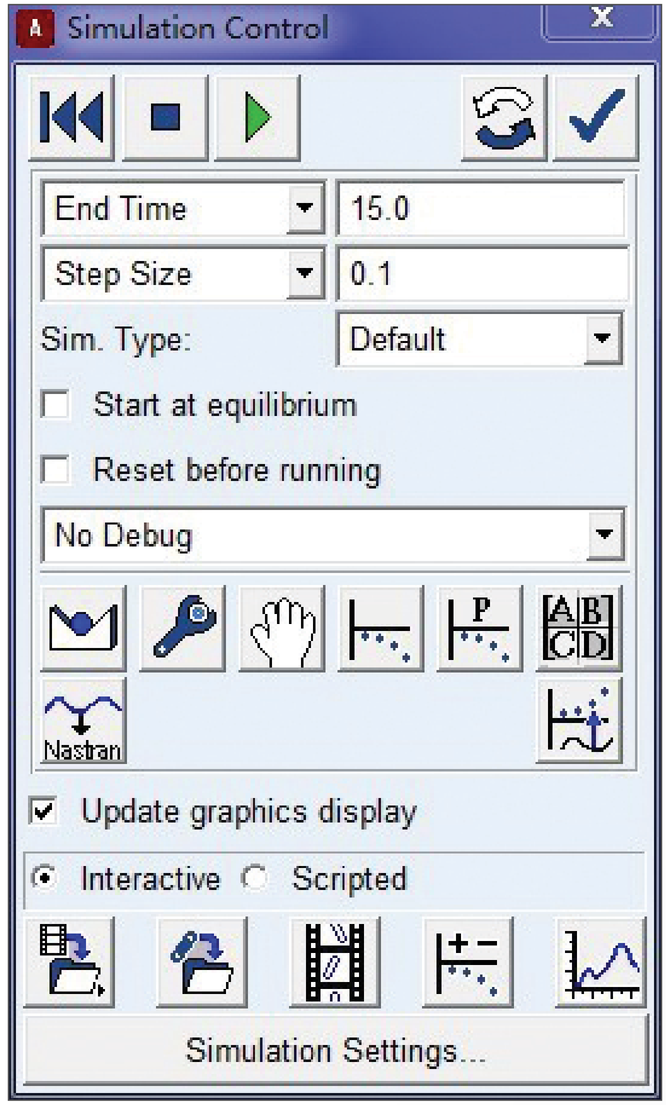


b. In the **Adams/PostProcessor** windows, select **Data** as **Independent Axis.**

c. Select **Angle_Link2toBase** in the **Independent Axis Browser**, and then click **OK**.

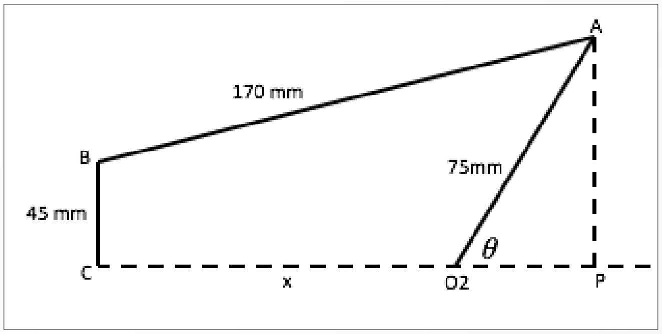
d. Select **Measure** as **Source**, and then select **stroke**

e. Click **Add Curve**.



## Step 17. Compare Results

### Theoretical Solution



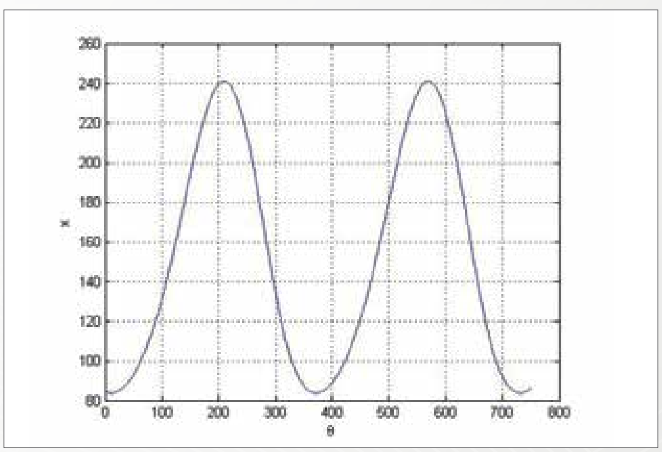

$$AP = 75\sin(\theta)$$

$$(AP - BC)^2 + (CO_2 + PO_2) = AB^2$$

$$\Rightarrow (75\sin(\theta) - 45)^2 + (x + 75\cos(\theta))^2 = 170^2$$

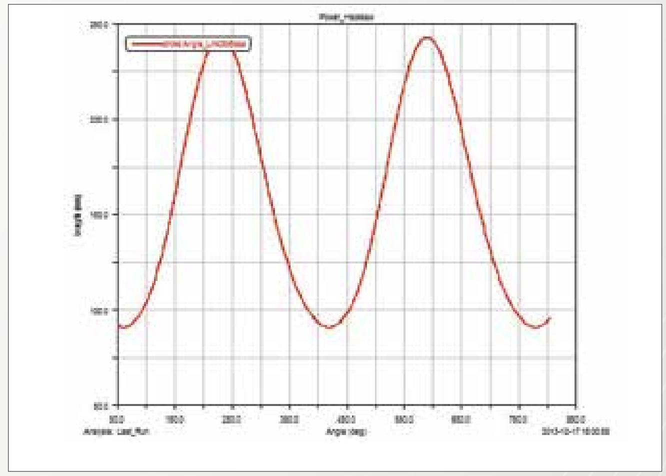
$$\Rightarrow x^2 + 150\cos(\theta) + (-21250 - 6750\sin(\theta)) = 0$$

$$x = 75\left(-\cos(\theta) + \sqrt{\cos^2(\theta) + \frac{6}{5}\sin(\theta) + \frac{34}{9}}\right)$$



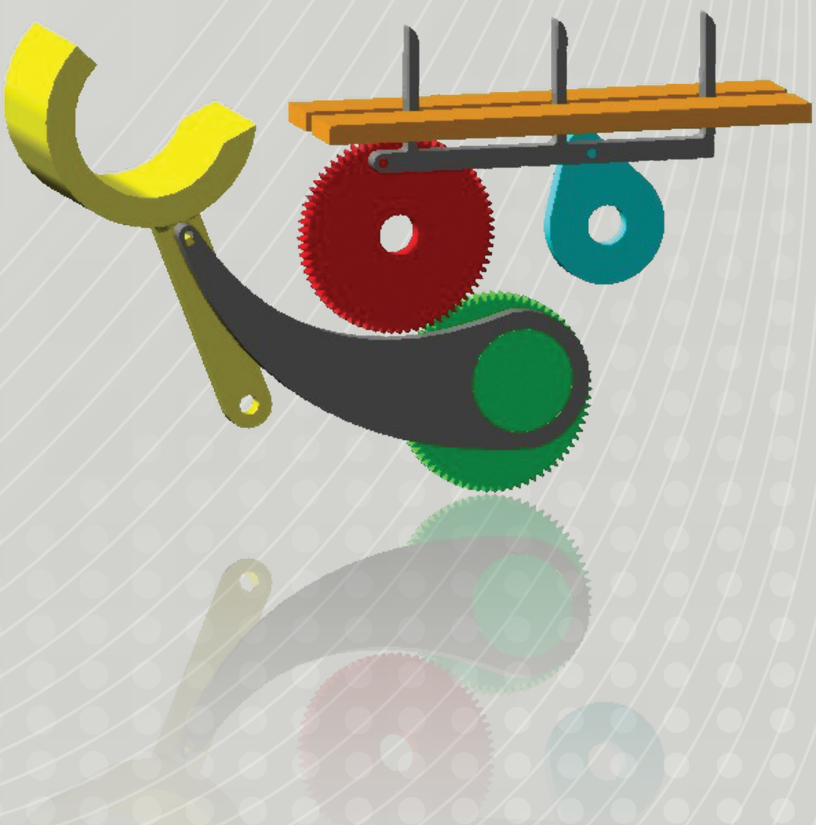

### Adams Solution

The Adams solution is exactly the same as the theoretical solution.

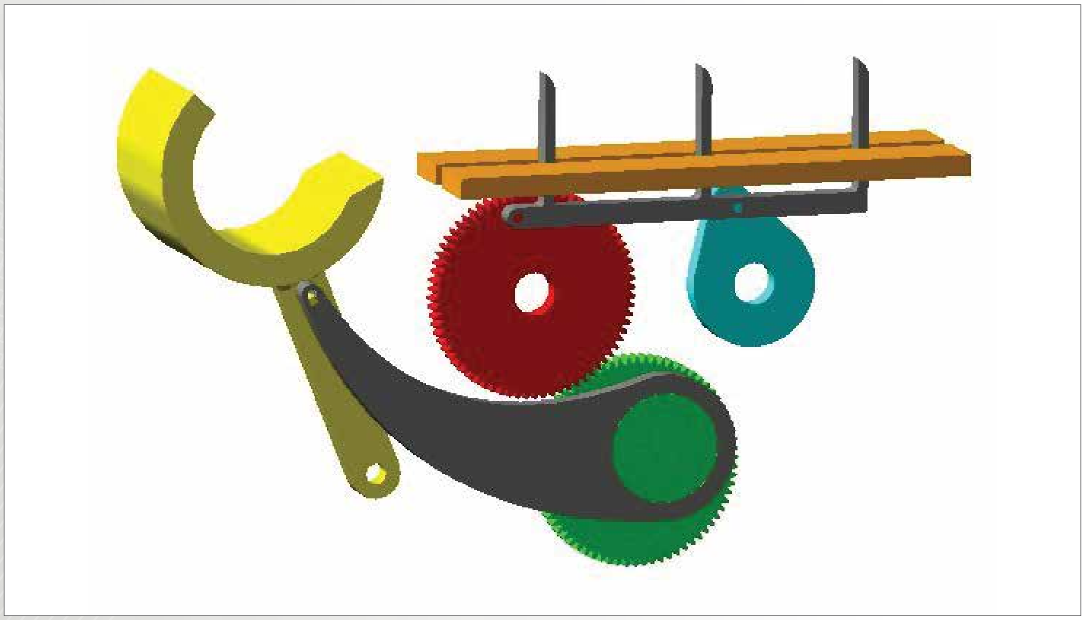


# Section III: Textbook Problems

*In this section, you'll learn how to solve some of the textbook problems using Adams. All these problems are created in reference to the textbook Design of Machinery (Fifth Edition) by Robert L. Norton (2012). All the mechanisms that we chose have been widely used in automotive and manufacturing industry. We hope you can solve the other similar textbook problems using Adams after you finish this section.*



# Example 24: Walking Beam Indexer



## Workshop Objectives

Use **Adams/view** to

- Simulate the walking beam indexer with a pick-and-place mechanism
- Import existing .x_t file
- Create a gear pair
- Duplicate part
- Create an angle measurement

Use **Adams/Postprocessor** to

- Calculate the horizontal stroke of the walking beam and the angular displacement of the placing arm.

# Software Version

- Adams 2013.2
- Adams/Machinery Plugin with gear module is required

# Files Required

- **crank.x_t**
- **walkingBeam.x_t**
- **placeArm.x_t**
- **link.x_t**
- Located in the directory **exercise_dir/Example 24**

# Problem Description

- The model represents the walking beam indexer with a pick-and-place mechanism.
- The crank is being rotated at a given velocity.
- For one revolution of the crank, the walking beam pushes products forward one step.
- The articles are caught by the place arm.

Adapted from Robert L. Norton (2012). Design of Machinery (Fifth Editon)

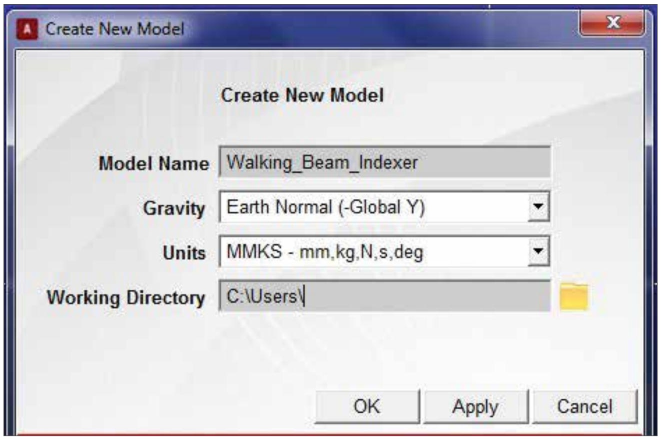
### Step 1. Start Adams/View and Create a Database

a. Start **Adams/View.**

b. From the Welcome dialog box, select **New Model.**

c. Replace the contents of the **Model Name** text box with **Walking_Beam _Indexer.**

d. Select **OK**.




### Step 2. Set Up Work Environment

a. From the **Setting** menu, select **Working Grid**.

b. Set the grid size along X and Y to **250 mm,** and the grid spacing for X and Y to **5mm**.

c. Select **OK**.

## Step 3. Create Design Points

a. From the Main Toolbox, select the ribbon **Bodies**, and then select the **Construction Geometry: Point.**



b. Use the default setting for points, which are **Add to Ground** and **Don't Attach**.

c. Place the design point at **X = 55** and **Y = 80.**

d. Right-click the design point, Point to **Point: POINT_1**, and then select **Rename**.

e. Replace **POINT_1** with **ground.O2**.

f. Right-click the design point, Point to **Point: O2**, and then select **Modify**.

g. Change X coordinate to 57 and Y coordinate to 82.

h. Select **OK**.

i. Repeat the above steps to create the following design points in the following table.

| | X Location | Y Location | Z Location |
|---|---|---|---|
| O2 | 57 | 82 | -10 |
| O4 | -51 | 82 | 0 |
| O6 | -128 | 0 | -10 |
| B | 10.517 | 7.641 | 0 |
| F | -100 | 136 | 10 |
| D | -62.695 | 120.252 | 0 |
| C | -164.089 | 89.871 | 0 |

## Step 4. Import Parts

a. From the Main Menu, select **File**, then click **Import**…

b. Replace the contents of **File Type** with **Parasolid**

c. Right-click the blank beside **File to Red** and select **Browse**.

d. Locate the save file **crank.x_t**.

e. Click **OK**.

f. Replace **Model Name** with **Part Name** and type **Crank** into the blank beside the **Part Name.**

g. Right-click the blank beside **Location** and select **Pick Location.**

h. Select the point **ground.O2** in the working area.

i. Replace the contents of **Orientation** with **0.0, 0.0, 73.0.**

j. Click **OK**.

k. Repeat the above steps to import the other three parts in.

## Step 5. Create Gears

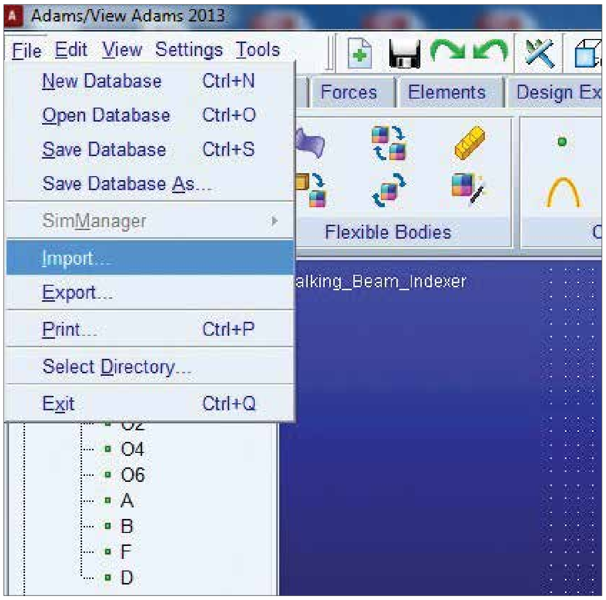
a. From the Main Toolbox, select ribbon **Machinery**, and then select **Create gear pair.**

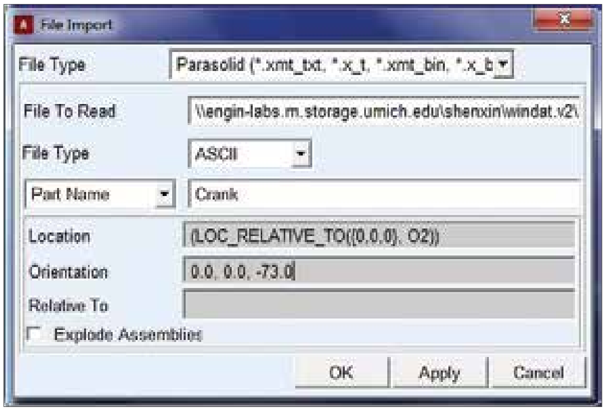
b. Choose **Spur** in **Gear Type**, and then click **Next**.

c. Choose **Detailed** in **Method**, and then click **Next**.

d. Set the parameters in the **Geometry** dialog

| Part Name | Location Point | Orientation |
|---|---|---|
| Crank | ground.O2 | 0, 0, -73 |
| Walking_Beam | ground. D | 0, 0, 0 |
| Link | ground. B | 0, 0, -25 |
| Place_Arm | ground. O6 | 0, 0, -68 |



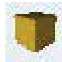
according to the following figure.

e. Click **Next** with the default setting of **Material** and **Connection**

f. Click **Finish**.

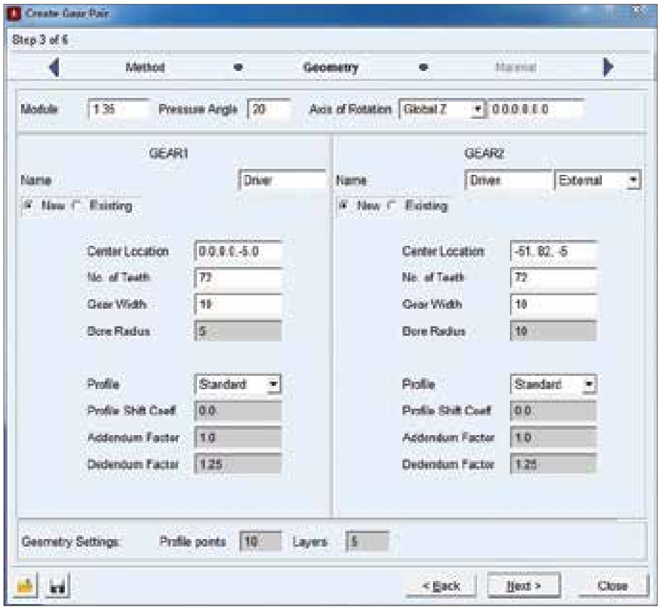
## Step 6. Create Platen

a. From the Main Toolbox, select ribbon **Bodies**, and then select **RigidBody:Box**



b. Use the default construction method **New Parts**.

c. In the **Geometry: Box** dialog, check **Length** and enter




250.0 mm. Check **Height** and enter 10.0 mm. Check **Depth** and enter 30.0 mm.

d. Click the point **ground.F.**

e. Right-click the part and point to **Part: PART_10**, and then select **Rename**.

f. Enter **.Walking Beam Indexer.Platen_Left** in the **New Name** content.

g. From the Main Toolbox, right-click **Position: Reposition objects**

h. Select **Position –Move.**

i. In the **Position: Move** dialog, check **Selected** and **Copy**. Choose **Vector** and enter -40.0 mm below **Distance**

j. Select the part **Platen_Left.**

k. Right-click the CG of the part **Platen_Left.**

l. Select **Platen_Left.cm.Y** and click **OK**.

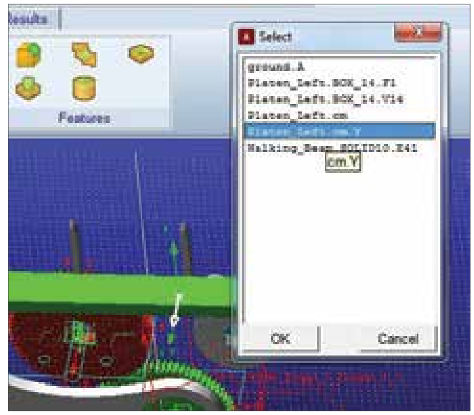
m. Click **Unparamerized**.

n. Rename the new part as **Platen_Right.**

## Step 7. Create Eccentric Cam

a. From the Main Toolbox, select ribbon **Bodies**, and then select **Construction Geometry: Arc/Circle.**

b. In the Geometry: Circle dialog, check **Radius** and




enter 25 mm, then check **Circle**.

c. Left-click **ground.B** in the working area.

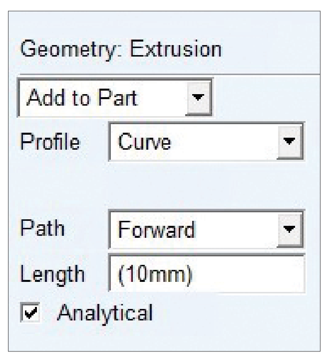
d. From the Main Toolbox, select ribbon **Bodies**, and then select **Rigidbody: Extrusion.**

e. Select the items in **Geometry: Extrusion** as shown in **Figure 7.**

f. Click the gear **Driver_1** in the working area, and then select **PART_9.CIRCLE_18.**

g. From the Model Browser, left-click the plus beside **PART_9.**

h. Right-click **CIRCLE_18**, and then select **Hide**.



## Step 8. Color the Parts

a. From **Model Browser**, left-click the plus beside **Platen_Right** displayed under the **Bodies** tree
b. Right-click **SOLID1** under the **Crank** and select **Appearance**.
c. In the **Edit Appearance** dialog, enter **Cyan** beside **Color**.
d. Click **OK**.
e. Repeat the above steps to change the color of the other parts.

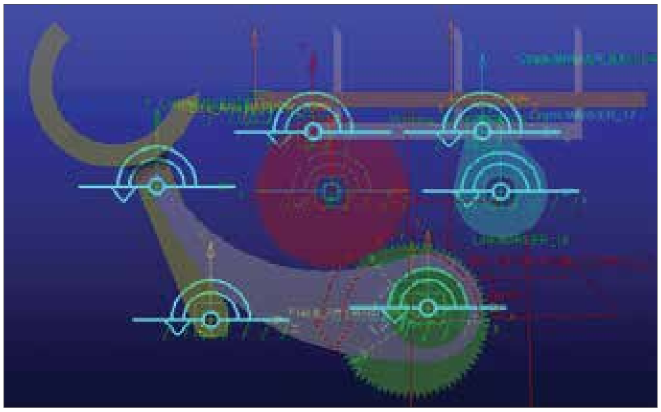
| Part Name | Color Name |
|---|---|
| Platen_Right | Maize |
| Platen_Left | Maize |
| Crank | Cyan |
| Walking_Beam | White |
| Place_Arm | Yellow |
| Link | White |
| Driven_1 | Red |
| Driver_1 | Green |

## Step 9. Connect the Parts Using Revolute Joints

a. From the Main Toolbox, select ribbon **Connectors**, and then select **Create a revolute joint.**

b. To select the parts to attach, click the part **Place_Arm** and ground (the background)
c. Click the point **ground.O6** to set the joint's location.
d. Repeat the above steps to create three more revolute joints.

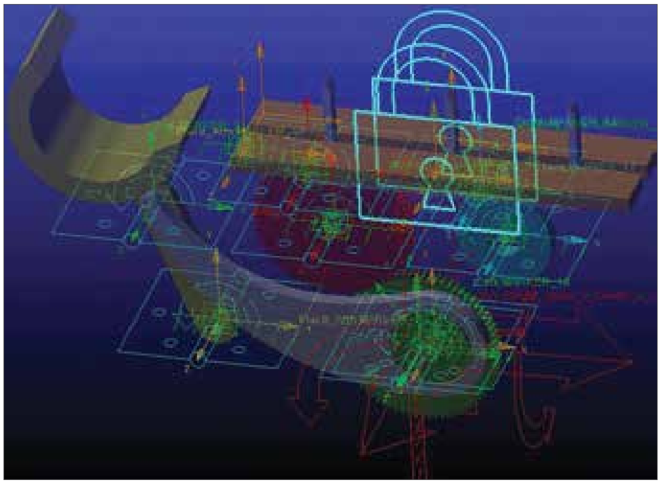
| 1st Body | 2nd Body | Joint Location |
|---|---|---|
| Place_Arm | ground | O6 |
| Place_Arm | Link | C |
| Link | Driver_1 | B |
| Walking_Beam | Driven_1 | D |
| Walking_Beam | Crank | Walking_Beam.SOLID15.E64 |
| Crank | ground | O2 |



## Step 10. Fix the Platen to the Ground

a. From the Main Toolbox, select ribbon **Connectors**, and then select **Create a fixed joint.**

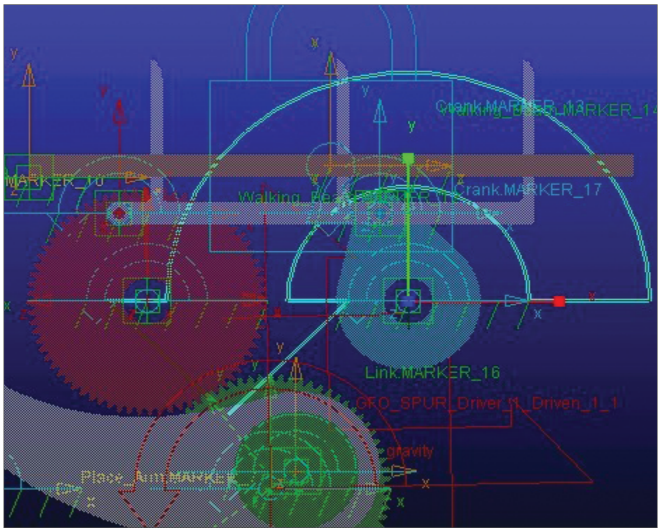
b. Click the part **Platen_Left**, ground and the CG of Platen.
c. Repeat the above steps to create a fixed joint between **Platen_Right** and ground.



## Step 11. Create Motion on a Revolution Joint

a. From the Main Toolbox, select the ribbon **Motions**, and then select **Rotational Joint Motion.**

b. Select the revolution joint between the **Crank** and the ground.




## Step 12. Create an Angle Measure

a. Under the **Connectors** tree in the **Model Browser,** right-click the revolute joint between **Place_Arm** and the ground.

b. Select **Info** to see the names of **I Marker** and **J Marker.**

c. Close the **Information** dialog.

d. From the Main Toolbox, select the **Design Exploration**.

e. Select **Create a new Function Measure**.

f. In the **Function Builder** dialog, enter **Angle_PlaceArmtoGround** as **Measure Name**.

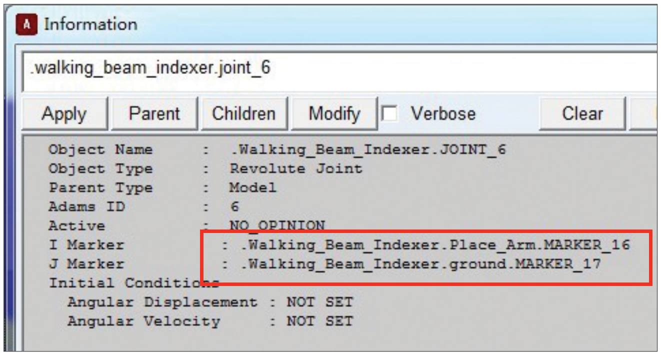
g. Select **angle** as **Units**.

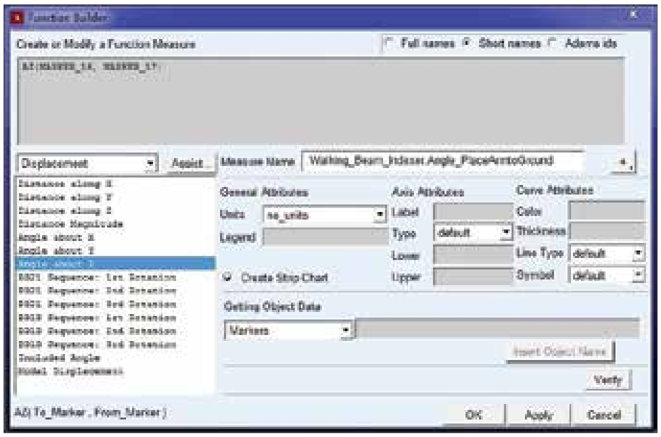
h. Select **Displacement**, select **Angle about Z**, and then click **Assist**…

i. In the **Angle about Z** dialog, right-click the contents of **To Marker** and **From Marker,** point to **Marker**, and then select **Browse**…

j. In the **Database Navigator**, select the markers in **Step b.**

k. Click **OK**.



## Step 13. Create a Function Measure

a. Select **Construction Geometry: Marker**

b. Create a **MARKER_42** at **(-100.0, 146.0, 0.0)**

c. From the Main Toolbox, select the **Design Exploration.**

d. Select **Create a new Function Measure.**

e. In the **Function Builder** dialog, enter **Stroke_Walking_Beam** as **Measure Name.**

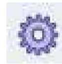
f. Select **length** as **Units**.

g. Enter the following into the **Create or Modify a Function Measure** **(DY(MARKER_41, MARKER_42, MARKER_42)/ ABS(DY(MARKER_41, MARKER_42, MARKER_42)) + 1) / 2 * DX(MARKER_41, MARKER_42, MARKER_42)**

h. Click **OK**.

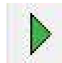
**Note:** you can design another better way to measure the horizontal stroke of the walking beam for the portion of their motion where its tips are above the top of the platen.

## Step 14. Simulate the Motion of Your Model

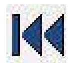


a. Click the ribbon **Simulation**, and then select the **Run an Interactive Simulation tool.**

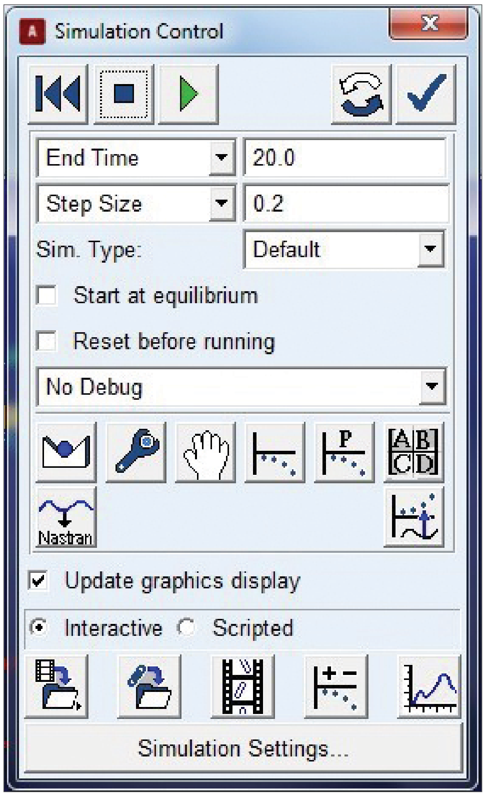
b. Set up a simulation with an **End Time** of **20** and **Step Size** of **0.2.**

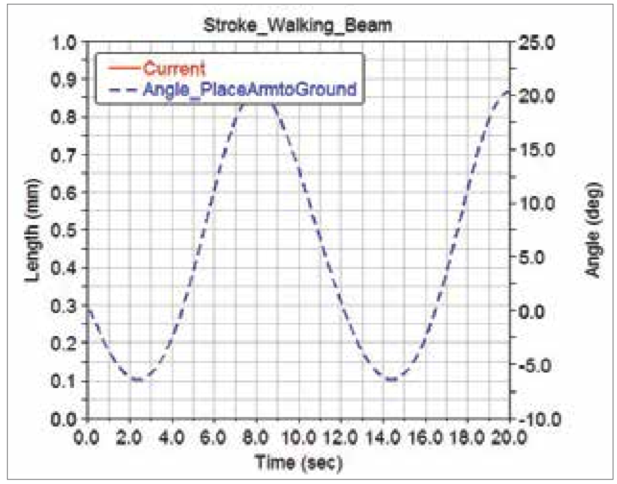


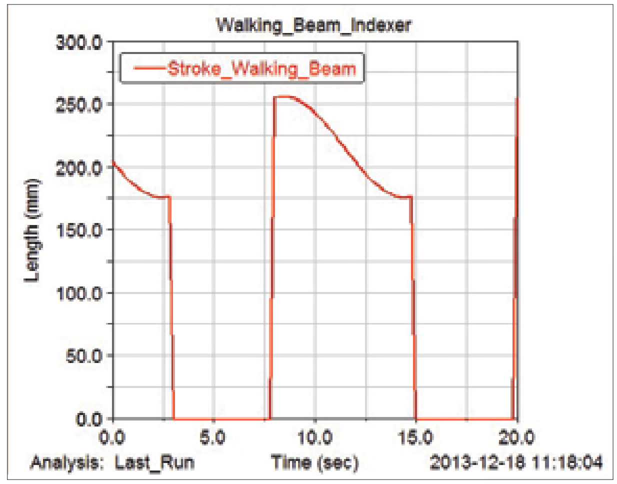
c. Select the **Simulation Start** tool.



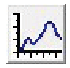
d. To return to the initial model configuration, select the **Reset** tool.




## Step 15. Use Adams/PostProcessor



a. In the Simulation Control panel, click **Plotting**.

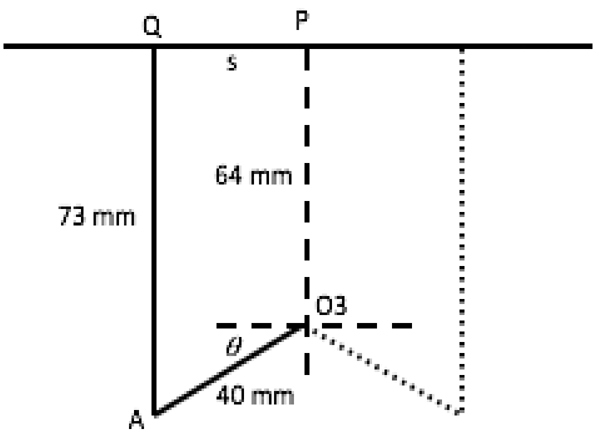
b. Select **Objects** as **Measure**.

c. Select **Angle_PlaceArmtoGround** or **Stroke_Walking_Beam.**

d. Click **Add Curve**.

## Step 16. Compare Results

Theoretical Result




$$AO_3^2 = (AQ - O_3P)^2 + QP^2$$

$$\Rightarrow 40^2 = (73 - 64)^2 + s^2$$

$$\Rightarrow s = 38.97$$

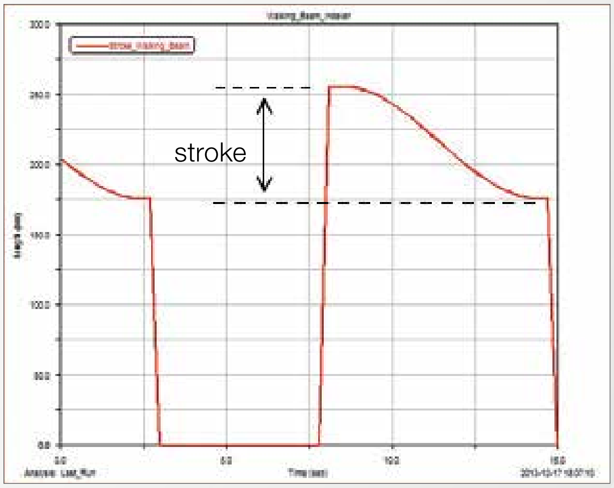
$$stroke: 2s = 77.94mm$$

$$\alpha = 180^\circ - 2\arcsin\left(\frac{AQ - PO_3}{AO_3}\right) = 154^\circ$$

The portion of one revolution of Link 2 is

$$1 - \frac{154}{360} = 57.2\%$$

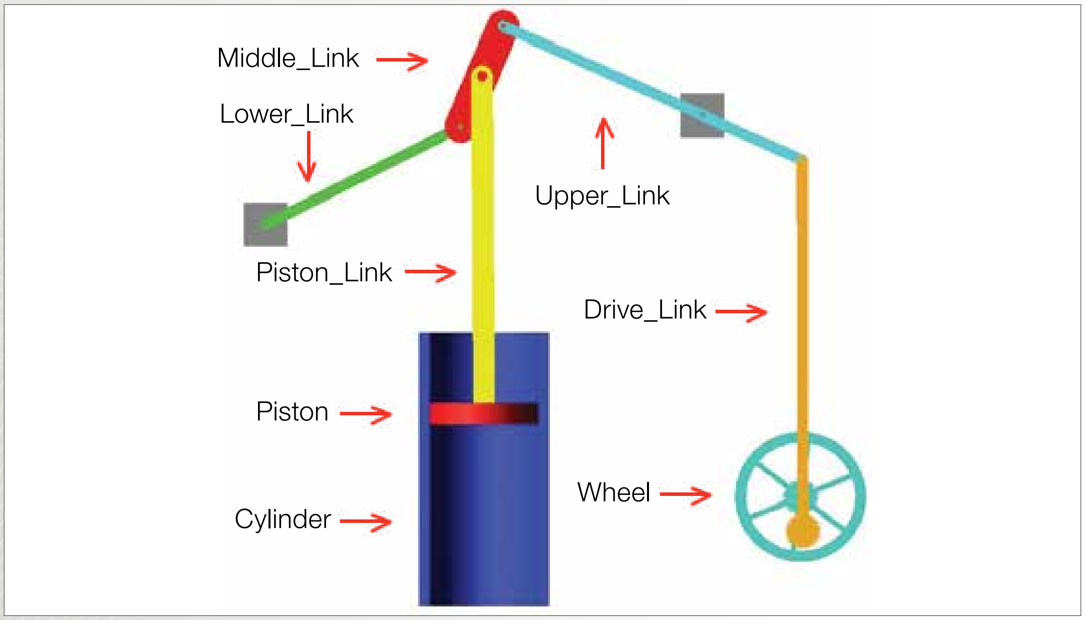
Adams Solution




Stroke = 254.9749 -177.1884 = 77.78 mm

The 1mm difference between the Adams solution and exact solution is caused by the thickness of the platen.

# Example 25: Watt's Linkage in a Steam Engine




## Workshop Objectives

Use **Adams/view** to

- Simulate the Watt's linkage in a steam Engine.
- Import .x_t file
- Create revolute joints and a translational joint
- Create a gear pair

## Software Version

- Adams 2013.2
- Adams/Machinery Plugin with gear module is required

## Files Required

- **wheel.x_t**
- Located in the directory **exercise_dir/Example 25**

# Problem Description

- The model represents the Watt's linkage used in a steam engine.
- The piston is being constrained to move along a straight line.
- The piston pushes three-bar linkage system to rotate the wheel.

Adapted from Robert L. Norton (2012). Design of Machinery (Fifth Edition)

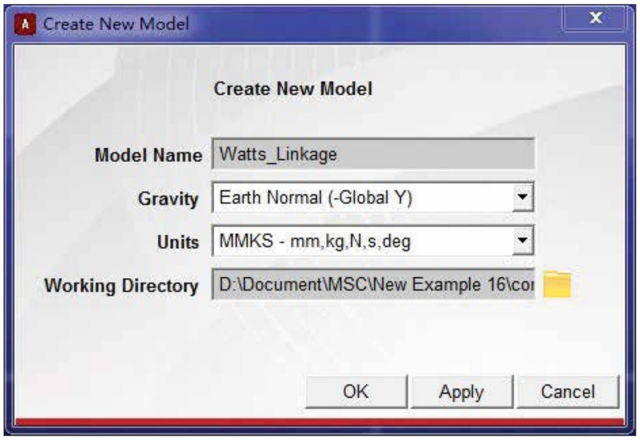
### Step 1. Start Adams/View and Create a Database

a. Start Adams/View.

b. From the Welcome dialog box, select **New Model.**

c. Replace the contents of the **Model Name** text box with **Watts_Linkage.**

d. Select **OK**.

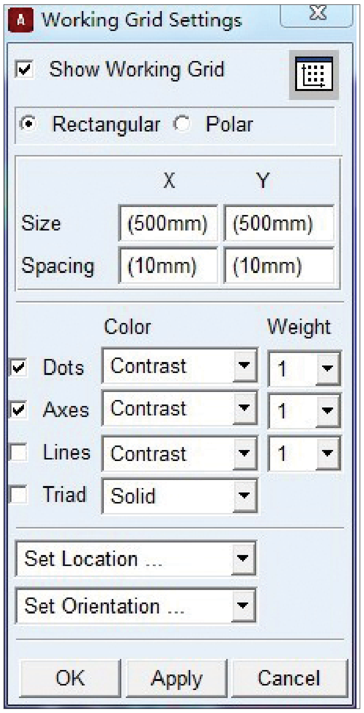



### Step 2. Set Up Work Environment

a. From the **Setting** menu, select **Working Grid**.

b. Set the grid size along X and Y to **500 mm**, and the grid spacing for X and Y to **10mm**.

c. Select **OK**.



## Step 3. Create Design Points



a. From the Main Toolbox, select the ribbon **Bodies**, and then select the **Construction Geometry: Point.**

b. Use the default setting for points, which are **Add to Ground** and **Don't Attach**. Place the design point at X = 0 and Y = 350.

c. Right-click the design point, Point to **Point: POINT_1**, and then select **Rename**. Replace **POINT_1** with **ground.A.**

d. Repeat the above steps to create the design points.

| | X location | Y location | Z location |
|---|---|---|---|
| O2 | -200 | 350 | 0 |
| O4 | 200 | 450 | 0 |
| O8 | 290 | 100 | -17.5 |
| A | 0 | 350 | 0 |
| B | 0 | 450 | 0 |
| C | 300 | 450 | 0 |
| D | 0 | 100 | 0 |
| E | 320 | 110 | 0 |
| P | 0 | 400 | 0 |

## Step 4. Import Part

a. From the Main Menu, select **File**, then click **Import**…

b. Replace the contents of **File Type** with **Parasolid**.

c. Right-click the blank beside **File to Read** and select **Browse**.

d. Located the saved file **wheel.x_t**

e. Click **OK**.

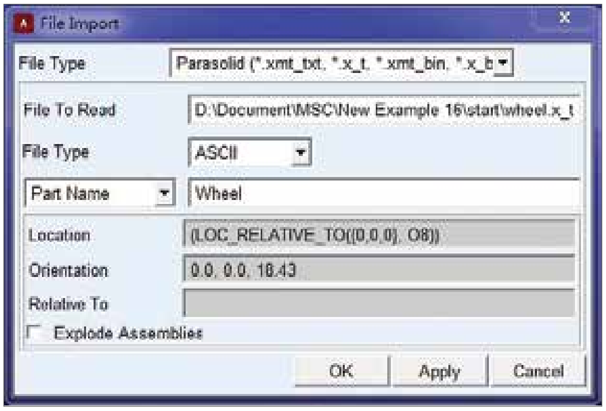
f. Replace **Model Name** with **Part Name** and type **Wheel** into the blank beside the **Part Name.**

g. Right-click the blank beside **Location** and select **Pick Location.**

h. Select the point **ground.O8** in the working area.

i. Replace the contents of **Orientation** with **(0.0, 0.0, 18.43).**

j. Click **OK**.




## Step 5. Create Cylinder

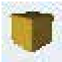
a. From the Main Toolbox, select ribbon **Bodies**, and then select **RigidBody:Cylinder.**

b. Use the default construction method **New Parts**.

c. In the **Geometry: Cylinder** dialog, check **Length** and enter **250.0 mm**. Check **Radius** and enter **60.0 mm**.

d. Click the point **(0, 0, 0)**, move upwards and click.

e. Rename the new part as **Cylinder**.



f. Select **RigidBody:Box**.

g. Select **Add to Part** as method in **Geometry: Box dialog.**

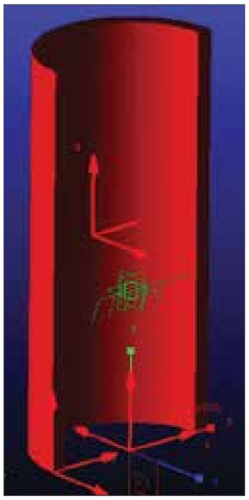
h. Check **Length** and enter **120 mm**. Check **Height** and enter **250 mm**. Check **Depth** and enter **60mm.**

i. Left-click the part **Cylinder** in the working area, and then left-click **(-60, 0, 0).**

j. Select **Booleans: Cut out a solid with another.**

k. Select **Cylinder**, and then select **Box**.

l. Repeat the above procedure to cut **Cylinder** use a smaller cylinder with the radius of **50mm**.



## Step 6. Create a Piston

a. From the Main Toolbox, select ribbon **Bodies**, and then select **RigidBody:Cylinder.**

b. Use the default construction method **New Parts.**

c. In the **Geometry: Cylinder** dialog, check **Length** and enter 20.0 mm. Check **Radius** and enter 50.0 mm.

d. Click the point **D** in the working area, move downwards and click

e. Right-click the part and point to **Part: PART_4**, and then select **Rename**.

f. Enter **Piston** in the **New Name** content.

g. Right-click **Piston** in the model browser and select **Modify**.

h. Enter **(7.801E+004(kg/meter**3))** for **Density**.

## Step 7. Create Ground Supports

a. Select **RigidBody:Box.**

b. Select **New Part** as method in **Geometry: Box dialog**.

c. Check **Length** and enter **40mm.** Check **Height** and enter **40 mm**. Check **Depth** and enter **40mm.**

d. Left-click **(-220, 330, 0).**

e. From the Main Toolbox, right-click Position: **Reposition objects**

f. Select **Add a boss.**

g. Enter **2mm** to **Radius** and **5mm** to **Height**, and then select the center of the **Box** in the working area.

h. Rename the new part as **Ground_Support1**

i. Repeat the above steps to create **Ground_Support2** at **(180, 430, 0).**

## Step 8. Create Linkages

a. Select **RigidBody:Link.**

b. Use the default construction method **New Parts.**

c. Check **Width** and enter **20mm.** Check **Depth** and enter **5mm**.

d. **DO NOT** check **Length**.

e. Left-click the point **D** and point **P.**

f. Rename the link as **Piston_Link.**

g. Repeat the above steps to create five more linkages.

h. Select **Add a hole** or **Add a boss** to add a hole of boss at the end of these linkages.

Remarks: Use **set the view to isometric** to choose point (290,100,0).

| Name | Width | Depth | Point 1 | Point 2 |
|---|---|---|---|---|
| Piston_Link | 20 mm | 5 mm | D | P |
| Lower_Link | 10 mm | 5 mm | O2 | A |
| Middle_Link | 30 mm | 5 mm | A | B |
| Upper_Link | 10 mm | 5 mm | B | C |
| Drive_Link | 10 mm | 5 mm | C | E |
| Gear_Link | 10mm | 5mm | (290,100,0) | E |

## Step 9. Move Parts into Difference Layers

a. From the Main Toolbox, right-click **Position: Reposition objects.**

b. Select **Position-Move.**

c. In the **Position:Move** dialog, check **Selected** and select **Vector** method.

d. Select the part, and then select any vector into or out of the working area.

e. Choose **Unparameterize** in the warning message.

| Part Name | Distance | Direction |
|---|---|---|
| Lower Link | 10 mm | into |
| Middle Link | 5 mm | into |
| Drive Link | 5 mm | out |
| Gear Link | 5mm | in |
| Ground_Support1 | 52.5 mm | into |
| Ground_Support2 | 42.5 | into |

## Step 10. Create Gears

a. From the Main Toolbox, select ribbon **Machinery**, and then select **Create gear pair**.

b. Choose **Spur** in **Gear Type,** and then click **Next**.

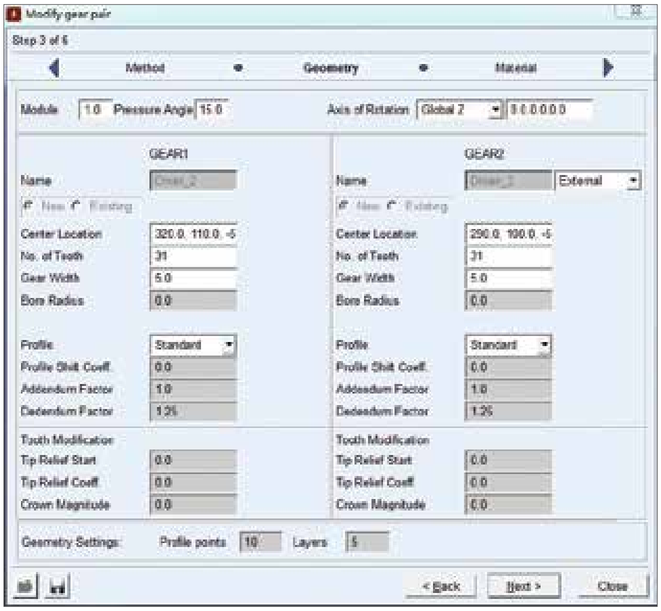
c. Choose **3D Contact** in **Method**, and then click **Next**.

d. Set the parameters in the **Geometry** dialog according to Figure.

e. Click **Next** with the default setting of **Material**.

f. Create **Gear Connection** according to Figure

g. Click **Next**, and then Click **Finish**.






## Step 11. Color the Parts

a. From **Model Browser,** right-click **Ground_Support1** displayed under the **Bodies** tree. Select **Appearance**.

b. In the **Edit Appearance** dialog, enter **Gray** beside **Color**

c. Repeat the above steps to change the color of the other parts.

| Part Name | Color Name |
|---|---|
| Wheel | Aquamarine |
| Drive_Link | Maize |
| Groud_Support1 | Gray |
| Ground_Support2 | Gray |
| Middle_Link | Red |
| Piston_Link | Yellow |
| Upper_Link | Cyan |
| Lower_Link | Green |
| Piston | Red |
| Cylinder | Blue |
| Driven_1 | Aquamarine |
| Driver_1 | Maize |

## Step 12. Connect the Parts Using Revolute Joints

a. From the Main Toolbox, select ribbon **Connectors**, and then select **Create a revolute joint.**

b. To select the parts to attach, click the part **Lower_Link** and ground (the background)

c. Click the point **ground.O2** to set the joint's location.

d. Repeat the above steps to create three more revolute joints.

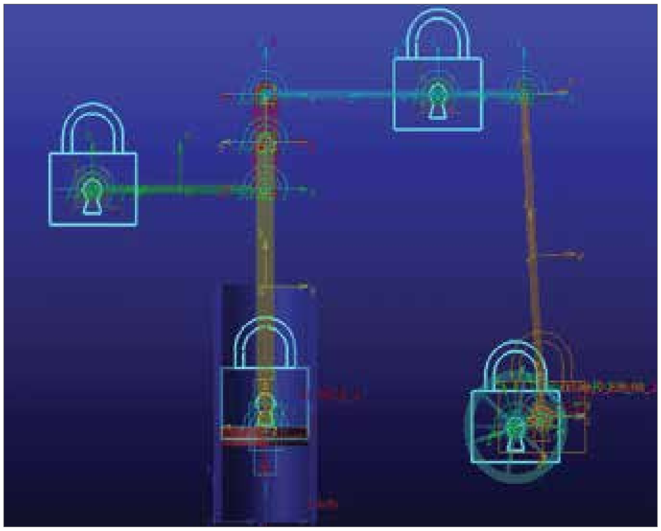
| 1st Body | 2nd Body | Joint Location |
|---|---|---|
| Lower_Link | ground | O2 |
| Middle_Link | Lower_Link | A |
| Middle_Link | Piston_Link | P |
| Middle_Link | Upper_Link | B |
| Piston_Link | Piston | D |
| Upper_Link | ground | O4 |
| Drive_Link | Upper_Link | C |
| Gear_Link | Ground | O8 |
| Gear_Link | Driver_1 | E |

## Step 13. Fix the Ground_ Support1 to the Ground

a. From the Main Toolbox, select ribbon **Connectors**, and then select **Create a fixed joint.**

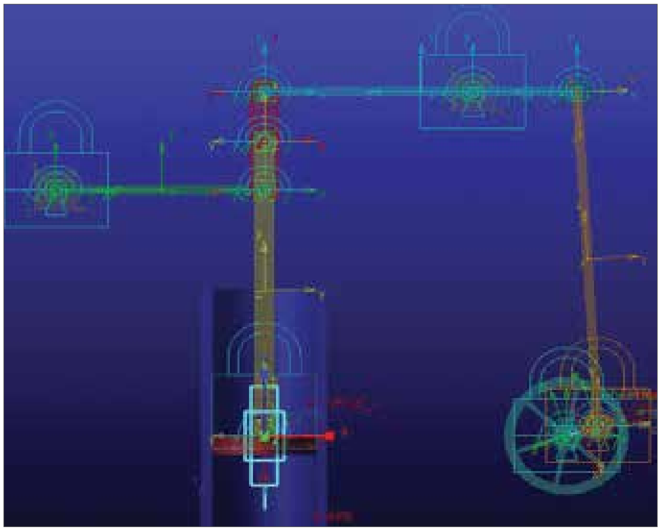
b. Click the part **Ground_Support1,** ground and the CG of **Ground_Support1.**

c. Repeat the above steps to create two fixed joint between: **Ground_support2** and the ground, Cylinder and the ground, Driven_1 and Wheel.



## Step 14. Create a Translational Joint

a. From the Main Toolbox, select **Connectors**, and then select **Create a Translational joint.** 
b. Click the part **Pistion**, the part **Cylinder** and the CG of **Piston**.
c. Right-click the CG of Link 4 and select **Piston.cm.X**
d. Select any vector in X-direction
e. Right-click the translational joint in the **model browser,** and then select **Modify**
f. Click **Initial Conditions**, and enter **86** for **Trans. Displ.**



## Step 15. Review All the Constraints

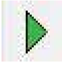
| | | | |
|---|---|---|---|
| 1 | Fixed joint | Ground_ Support1 and ground | O2 |
| 2 | Fixed joint | Ground_ Support2 and ground | O4 |
| 3 | Fixed joint | Cylinder and ground | Cylinder.cm |
| 4 | Fixed joint | Wheel and driven_1. gear_part | O8 |
| 5 | Revolute joint | Lower link and ground | O2 |
| 6 | Revolute joint | Lower link and middle link | A |
| 7 | Revolute joint | Piston link and middle link | P |
| 8 | Revolute joint | Upper link and middle link | B |
| 9 | Revolute joint | Upper link and drive link | C |
| 10 | Revolute joint | Upper link and Ground_ support2 | O4 |
| 11 | Revolute joint | Piston link and Piston | D |
| 12 | Revolute joint | Gear Link and ground | O8 |
| 13 | Revolute joint | Gear Link and driver_1. gear_part | E |
| 14 | Translational joint | Piston and cylinder with initial Conditions, 86 for Trans. Displ. | Along the vertical DIR. |

## Step 16. Create a Force

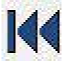
a. From the Main Toolbox, select the ribbon **Forces**, and then select **Create a Force.**
b. Select **Piston**, and then select point **D**.
c. Move upwards and create a force in y direction.
d. Expand the **Forces** tree in the **Model Browser.**
e. Right-click **SFORCE_1**, and then select **Modify**.
f. Enter **120*(STEP(time, 0.25, 0.0, 0.26, 1) + STEP(time, 0.27, 0.0, 0.28, -1) +STEP(time, 0.55, 0.0, 0.56, 1) + STEP(time, 0.57, 0.0, 0.58, -1) +STEP(time, 0.81, 0.0, 0.82, 1) + STEP(time, 0.83, 0.0, 0.84, -1))** into **Function(time).**
g. Click **OK**.

## Step 17. Simulate the Motion of Your Model

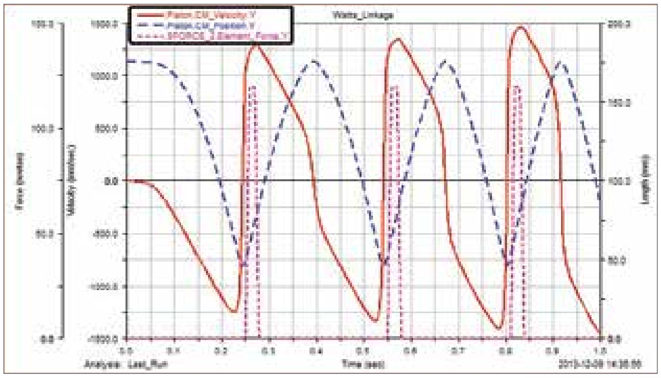
a. Click the ribbon **Simulation**, and then select the **Run an Interactive Simulation** tool.
b. Set up a simulation with an **End Time** of **1** and **Step Size** of **0.01.**



c. Select the **Simulation Start** tool.

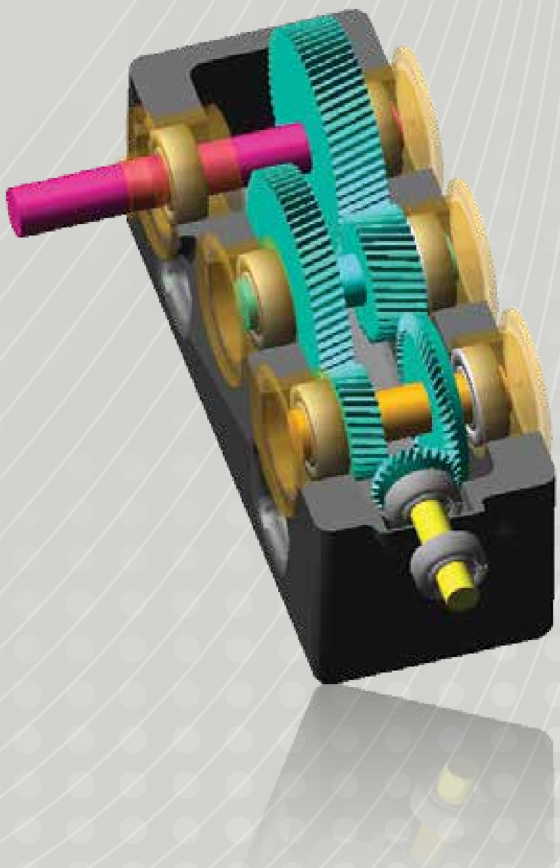


d. To return to the initial model configuration, select the **Reset** tool.
e. Click Plotting to start PostProcessor.
f. Select Objects as Source.
g. Plot the y component of **CM_Position** of **Piston**, **CM_Velocity** of **Piston** and **SPFORCE** in the graph.

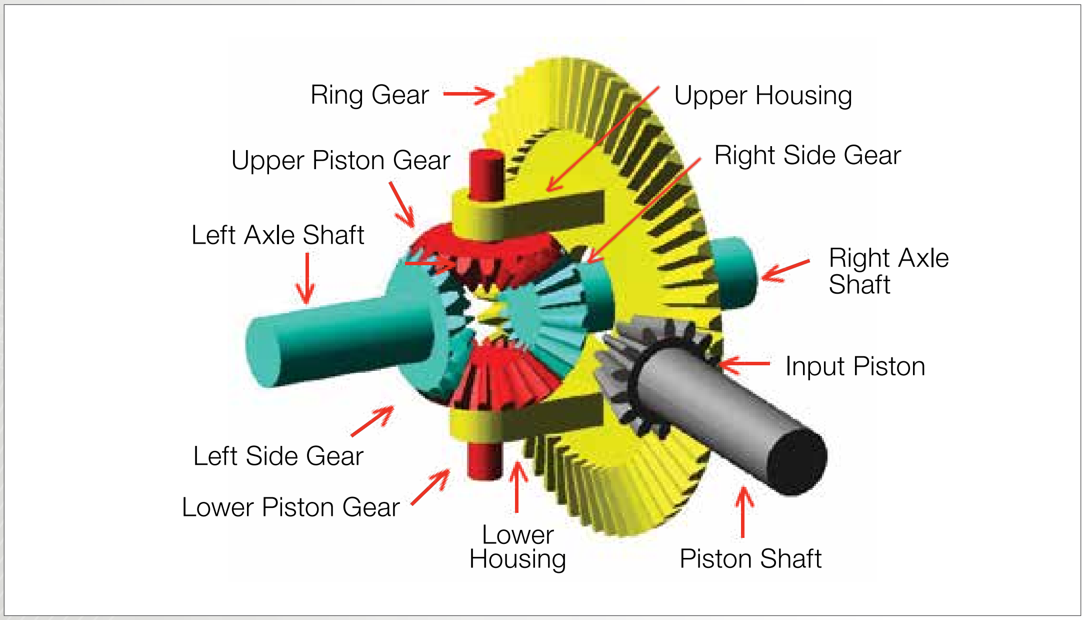


# Section IV. Adams/Machinery Applications

*In this section, you will learn how to use a powerful simulation tool, Adams/Machinery, to help you solve real world problems more easily, especially for those models with Gears, Bearings, Belts, Chains, Cables or Electric Motors in their drive systems.*



# Example 26: Open Differential




## Workshop Objectives

Use **Adams/view** to

- Simulate open differential when a vehicle makes moves straightly and then turns.
- Create Gear Pairs with existing gears and non-existing gears.
- Apply force and motion to the revolute joint.
- Learn the function of Booleans.

Use **Adams/Postprocessor** to

- Calculate the torque ration between the pinion shaft and the axle shafts

## Software Version

- Adams 2013.2
- Adams/Machinery Plugin with gear module is required

## Problem Description

- The model represents how a differential works.
- The left and right side gears have teeth on their side and they are attached directly to the end of the left and right axel shafts.
- The left and right axel shafts can turn freely on bearing in the ends
- The ring gear is attached to the input pinion which takes power from the pinion shaft which comes from the transmission.
- When the pion shaft turns and the ring rear and housings turns.

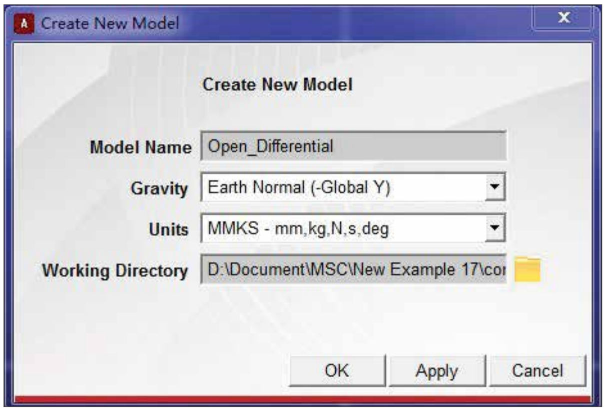
### Step 1. Start Adams/View and Create a Database

a. Start Adams/View.

b. From the Welcome dialog box, select **New Model.**

c. Replace the contents of the **Model Name** text box with **Open_Differential.**

d. Select **OK**.



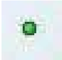

### Step 2. Set Up Work Environment

a. From the **Setting** menu, select **Working Grid**.

b. Set the grid size along X and Y to **150 mm**, and the grid spacing for X and Y to **5mm**.

c. Select **OK**.

## Step 3. Create Design Points

a. From the Main Toolbox, select the ribbon **Bodies**, and then select the **Construction Geometry: Point**. 

b. Use the default setting for points, which are **Add to Ground** and **Don't Attach**.

c. Place the design point at **X = -45** and **Y = 0.**

d. Right-click the design point, Point to **Point: POINT_1**, and then select **Rename**.

e. Replace **POINT_1** with **ground.A.**

f. Select **OK**.

g. Repeat the above steps to create the following design points.

| | X location | Y location | Z location |
|---|---|---|---|
| A | -45 | 0 | 0 |
| B | 45 | 0 | 0 |
| C | 0 | 43 | 0 |
| D | 0 | -43 | 0 |
| E | 40 | 0 | 137 |
| F | 0 | 50 | 0 |
| G | 0 | -50 | 0 |
| H | 40 | 0 | 200 |

## Step 4. Create Five Pairs of Gears

a. From the Main Toolbox, select ribbon **Machinery**, and then select **Create gear pair**. 

b. Choose **Bevel** in **Gear Type**, and then click **Next**.

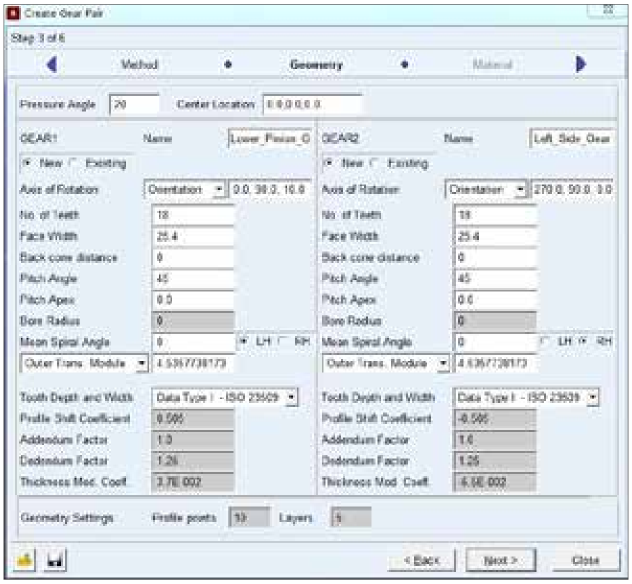
c. Choose **3D Contact** in **Method**, and then click **Next**.

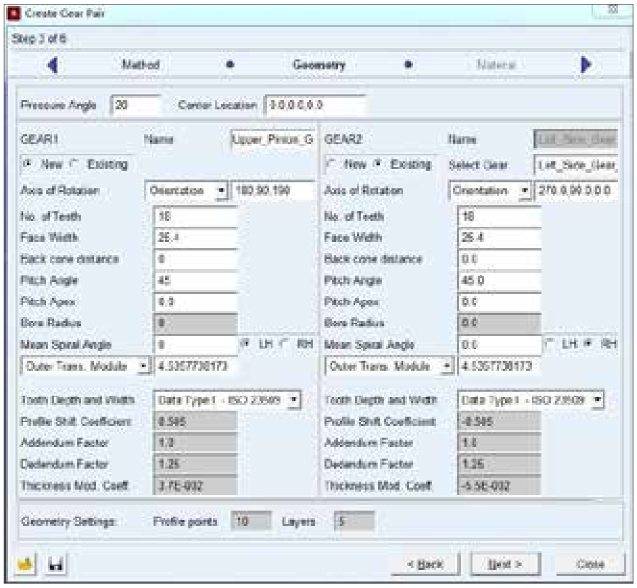
d. Set the parameters in the **Geometry** dialog according to the following figures. Note that some gears are created using **Existing** option.

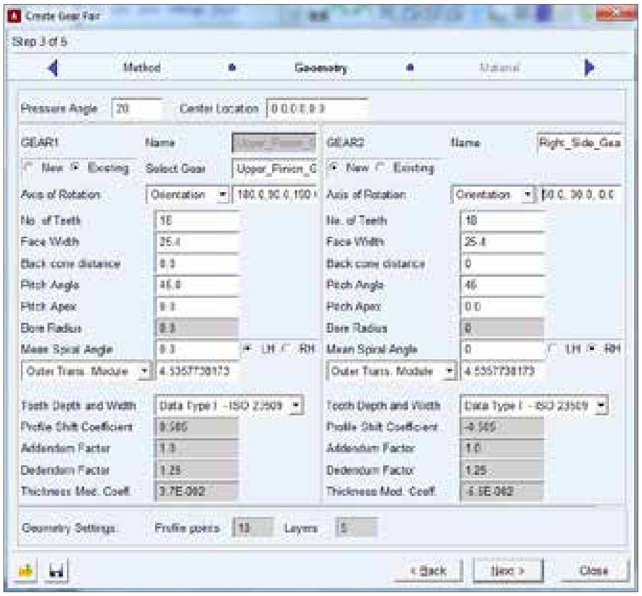
e. Click **Next** with the default setting of **Material**

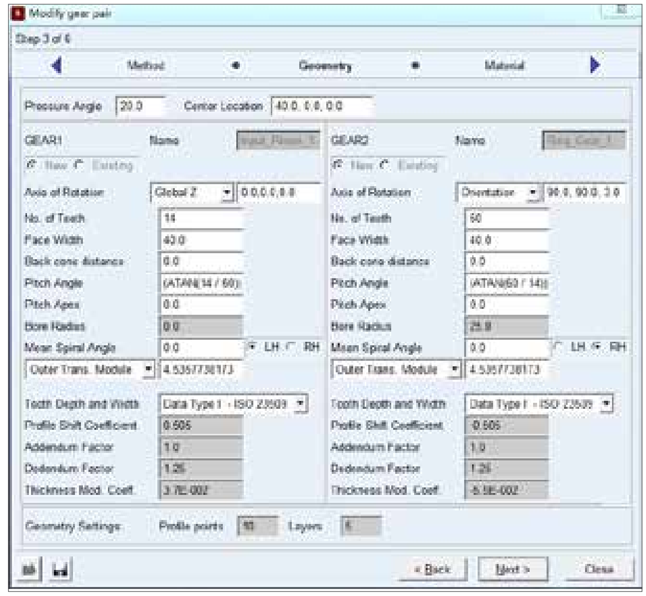
f. In the **Connection page**, select **none** in Type. Only select **Rotational Joint** and **ground** for **Ring_ Gear.**

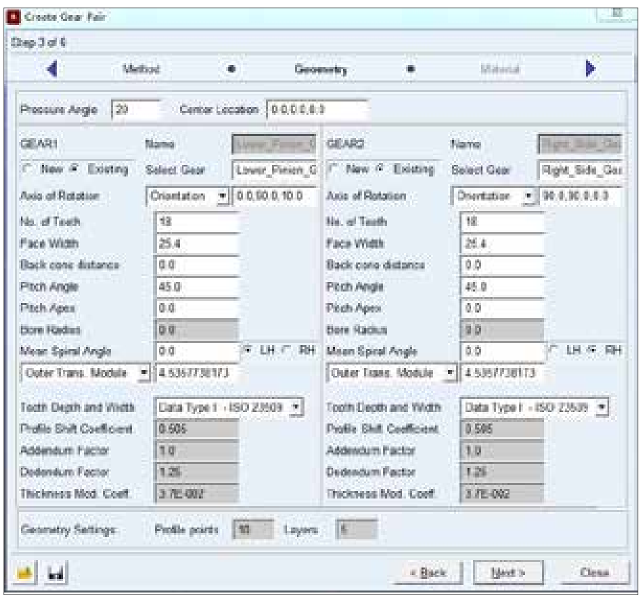
g. Click **Finish**.

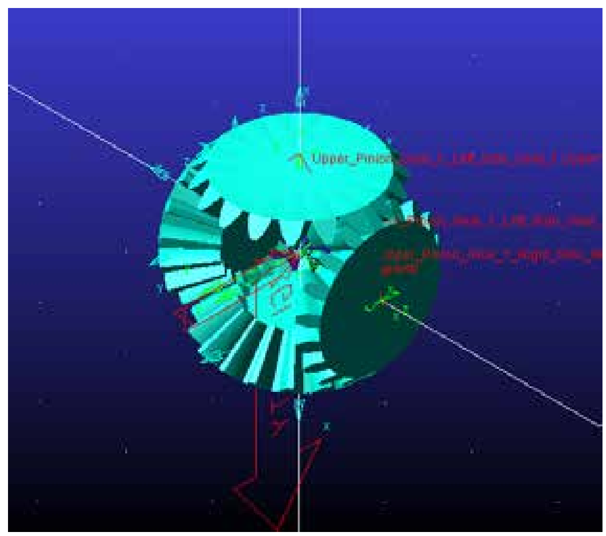










## Step 5. Create Axle Shafts

a. From the Main Toolbox, select ribbon **Bodies**,  and then select **RigidBody:Cylinder.**

b. Use the construction method **Add to Part.**

c. In the **Geometry: Cylinder** dialog, check **Length** and enter **100.0 mm.** Check **Radius** and enter **20.0 mm.**

d. Select **Left_Side_Gear**. Click the point **A,** move left and click.

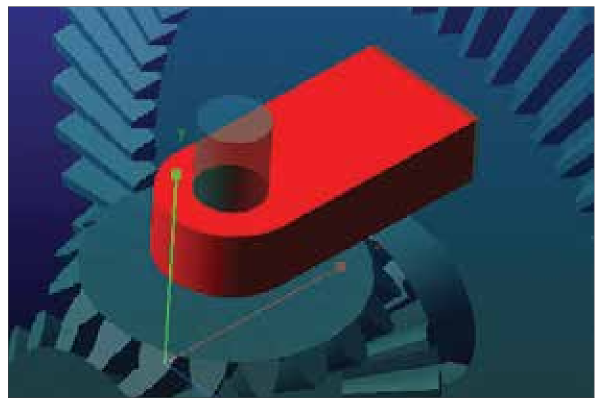
e. Repeat the above steps to create the shaft for **Right_ Side_Gear.**

## Step 6. Create Shafts for Pinion Gears

a. From the Main Toolbox, select ribbon **Bodies**, and then select **RigidBody:Cylinder.**
b. Use the construction method **Add to Part**.
c. In the **Geometry: Cylinder** dialog, check **Length** and enter **50.0 mm**. Check **Radius** and enter **10.0 mm.**
d. Click **Upper_Pinion_Gears.**
e. Click the point **C**, move upwards and click.
f. Repeat the above steps to create the shaft for **Lower_Pinion_Gear.**

## Step 7. Create Housings

a. Select **RigidBody:Box.**
b. Select **Add to Part** as method in **Geometry: Box dialog.**
c. Check **Length** and enter **70 mm**. Check **Height** and enter **20 mm**. Check **Depth** and enter **40mm**. Click the point **F.**
d. Rename the new part as **Upper_Housing**.
e. From the Main Toolbox, right-click **Position: Reposition objects.**
f. Select **Position-Move**.
g. In the **Position:Move** dialog, check **Selected** and select **Vector** method. Enter **20mm** in Distance.
h. Select **Upper_Housing**, and then select any vector pointing into the working area.
i. From the Main Toolbox, select ribbon **Bodies**, and then select **RigidBody:Cylinder.**
j. Select **Add to Part** as method in **Geometry: Box dialog.**
k. In the **Geometry: Cylinder** dialog, check **Length** and enter **20.0 mm**. Check **Radius** and enter **20.0 mm**.
l. Select **Upper_Housing**. Click the point **F**, move upwards and click.
m. Select **Booleans: Cut out a solid with another**.
n. Select **Upper_Housing.BOX27**, and then select **Upper_Housing.Cylinder28.**
o. From the Main Toolbox, select ribbon **Bodies**, and then select **RigidBody:Cylinder.**
p. Select **Add to Part** as method in **Geometry: Box dialog.**
q. In the **Geometry: Cylinder** dialog, check **Length** and enter **20.0 mm.** Check **Radius** and enter **10.0 mm**.
r. Select **Upper_Housing**. Click the point **C**, move upwards and click.
s. Select **Booleans: Cut out a solid with another.**
t. Select **Upper_Housing.CGS29**, and then select **Upper_Housing.Cylinder30.**
u. Select **Position-Move**.
v. In the **Position:Move** dialog, check **Selected** and check **Copy**.
w. Select Vector method and enter **120mm** in **Distance**.
x. Select **Upper_Housing**, and then select any vector which is vertical downwards.
y. Rename the new part as **Lower_Housing**



## Step 8. Create Pinion Shaft

a. Select **Setting** -> **Working Grid...**
b. Select **Global YZ** in **Set Orientation**
c. Click **OK**.
d. From the Main Toolbox, select ribbon **Bodies**, and then select **RigidBody:Cylinder**.
e. Use the construction method **Add to Part.**
f. In the **Geometry: Cylinder** dialog, **DO NOT** check **Length**. Check **Radius** and enter **10.0 mm**.
g. Click **Input_Pinion_Gear.**
h. Click the point E, and then click **point H.**

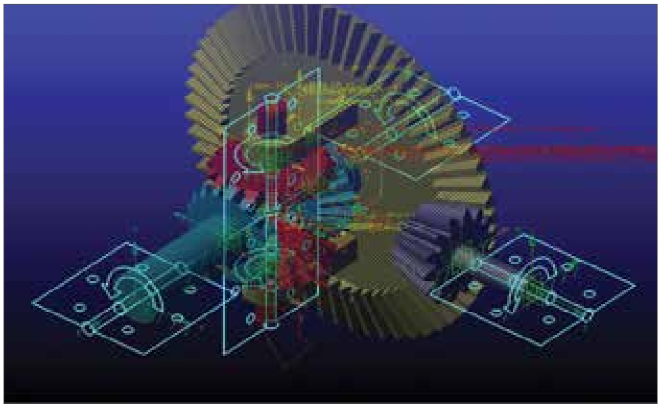
## Step 9. Color the Parts

a. From **Model Browser**, left-click the plus beside **Platen_Right** displayed under the **Bodies** tree
b. Right-click **Right_Axle_Shaft** and select **Appearance**.
c. In the **Edit Appearance** dialog, enter **Aquamarine** beside **Color**.
d. Click **OK**.
e. Repeat the above steps to change the color of the other parts.

| Part Name | Color Name |
|---|---|
| Right_Axle_Shaft | Aquamarine |
| Right_Side_Gear | |
| Upper_Pinion_Gear | Red |
| Lower_Pinion_Gear | |
| Input_Pinion | Gray |
| Ring_Gear | Yellow |
| Upper_Housing | |
| Lower_Housing | |

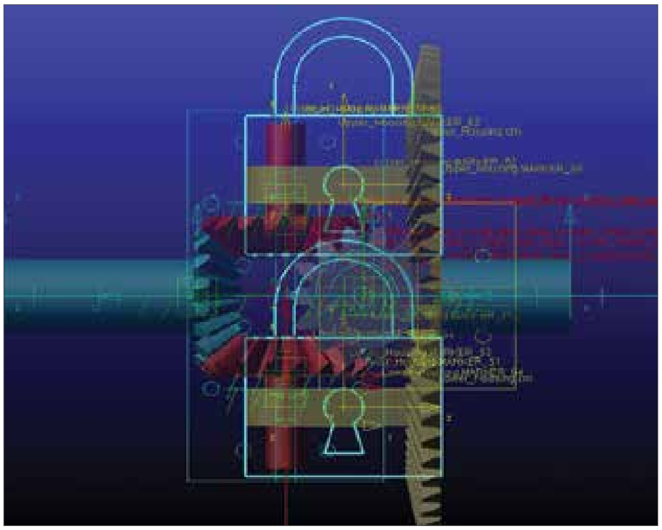
## Step 10. Create Revolute Joints

a. From the Main Toolbox, select ribbon **Connectors**, and then select **Create a revolute joint.** 
b. Click the part **Left_Pinoin_Gear** and the ground.
c. Click the center point at the end of the shaft of **Left_ Pinion_Gear.**
d. Repeat the above steps to create the revolute joint between **Right_Pinoin_Gear** and the ground.
e. Change the grid orientation to **Global XZ.**
f. Create revolute joints between
   a. **Lower_Housing** and **Lower_Pinion_Gear** at point **F**
   b. **Upper_Housing** and **Upper_Pinion_Gear** at point **G**
g. Change the grid orientation to **Global XY**
h. Create revolute joints between
   a. **Input_Pinion _Gear** and the ground at point **H**



## Step 11. Create Fixed Joints

a. From the Main Toolbox, select ribbon **Connectors**, and then select **Create a fixed joint.** 
b. Click the part **Upper_Housing, Ring_Gear** and point **Upper_Housing.cm.**
c. Repeat the above steps to create another fixed joint between **Lower_Housing and Ring_Gear.**



## Step 12. Create Motion on Revolution Joints

a. From the Main Toolbox, select the ribbon **Motion**, and then select **Rotational Joint Motion**.

b. Select the revolution joint between the **Right_Shaft** and the ground.

c. Rename it as **Motion_Right**.

d. Right-click **Motion_Right** in the **model browser,** and select **Modify**.

e. Enter **10 + STEP(time, 1, 0, 2, 5.0)** into **Function**

f. Repeat the above steps to create **Motion_Left.**

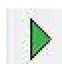
g. Enter **10 - STEP(time, 1.0, 0.0, 2.0, 5.0)** into **Function** for **Motion_Left.**

## Step 13. Simulate the Motion of Your Model

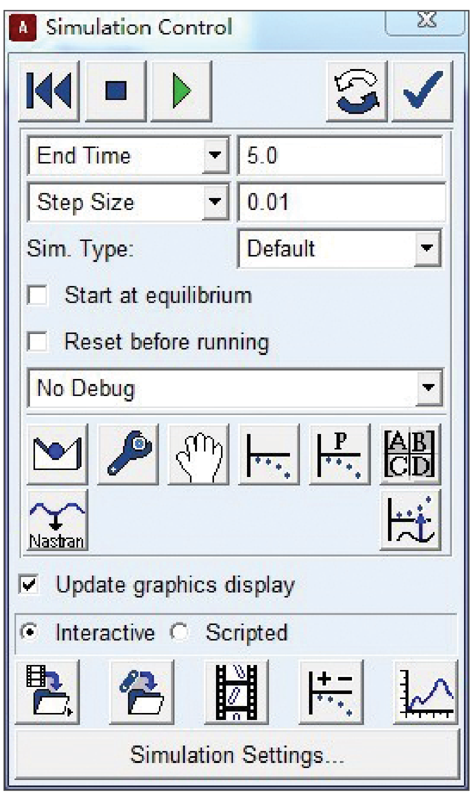
a. Click the ribbon **Simulation**, and then select the **Run an Interactive Simulation** tool.

b. Set up a simulation with an **End Time** of 5 seconds and **Step Size** of **0.01.**

c. Select the **Simulation Start** tool.



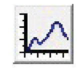
d. To return to the initial model configuration, select the **Reset** tool.




**Remarks:** It may take several minutes to run the simulation.

## Step 14. Use Adams/PostProcessor

a. In the Simulation Control panel, click **Plotting**.

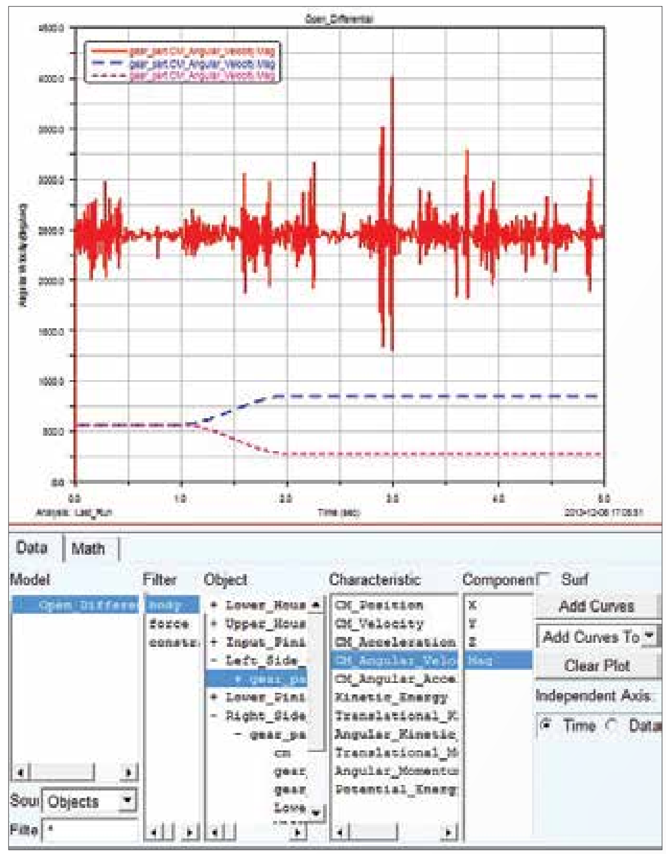


b. Select **Source** as **Objects**.

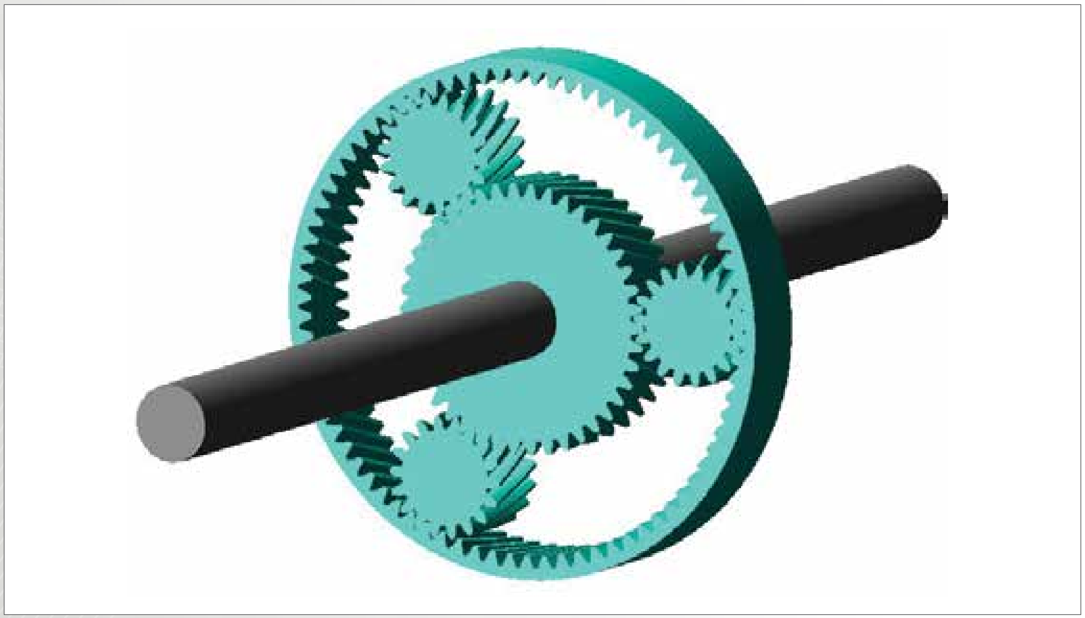
c. Select **body** as **Filter**.

d. Select **CM_Angular_Velocity** of **Left_Side_Gear, Right_Side_Gear** and **Input_Pinion_Gear..**

e. Click **Add Curve**.



# Example 27: Planetary Gear Sets Modification



## Workshop Objectives

- Modify the current planetary model and compare results in post-processor
- Get familiar with wizard interface

## Software Version

Adams 2013.2

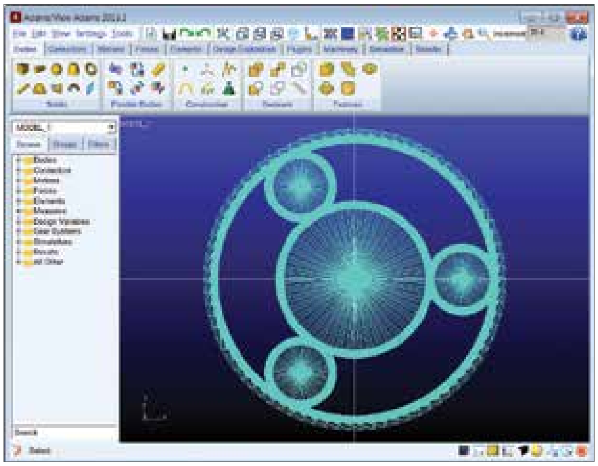
## Files Required

- Planetary_start.cmd
- Located in the directory **exercise_dir/Example 27**

## Step 1. Launch Adams and Import start file

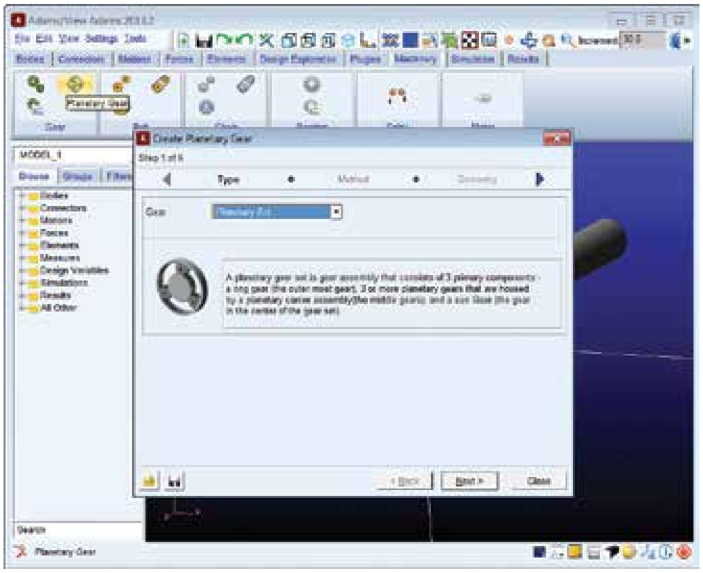
To get started: import the initial model:

a. Launch **Adams/View**
b. Select **Existing Model**
c. Browse for **Planetary_start.cmd** and hit **OK**

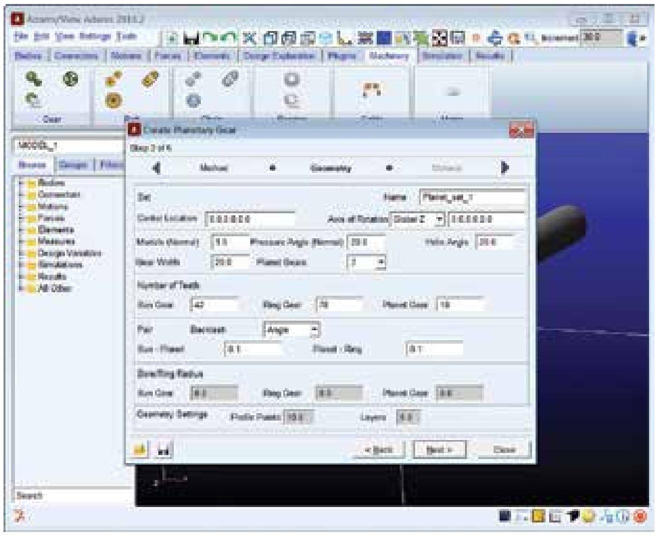


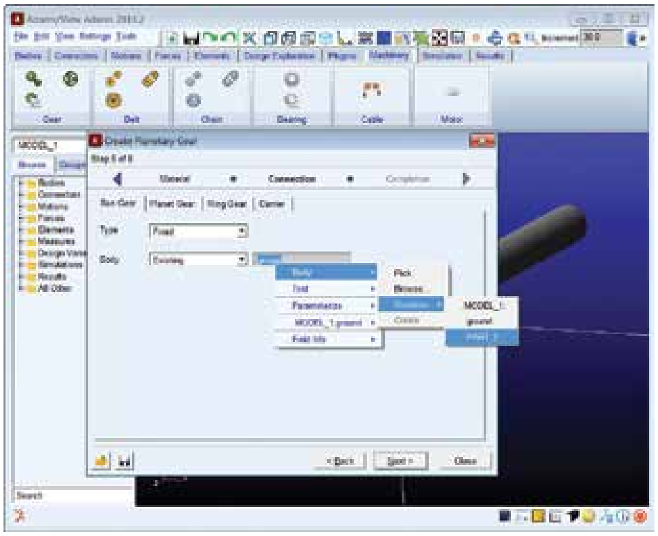
## Step 2. Create the Planetary Gear

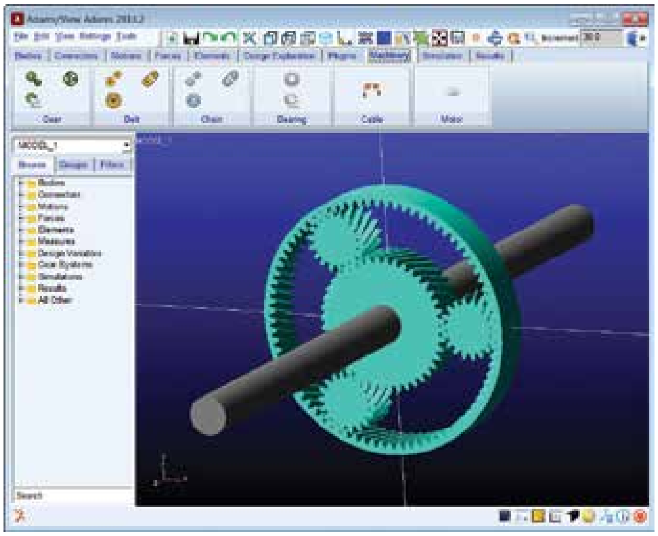
a. Click Planetary Gear under "Machinery" Tab.
b. Click Next.
c. Choose "Simplified", then Next.
d. On Geometry page, choose the default setting, click Next.
e. On Material page, choose the default setting, click Next.



f. On Connection page, fix the Sun gear to the existing "PART_2".
g. Click Next. Then click Finish.







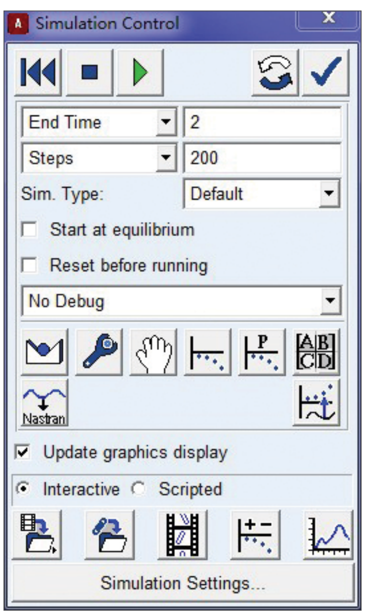
## Step 3. Run the simulation

a. Click on the run icon from the Simulation tab

b. Set “End Time” and “Steps” as shown.

c. Hit the **Run** button to run the simulation.

d. Click on **Save** the results to save the last simulation as P1

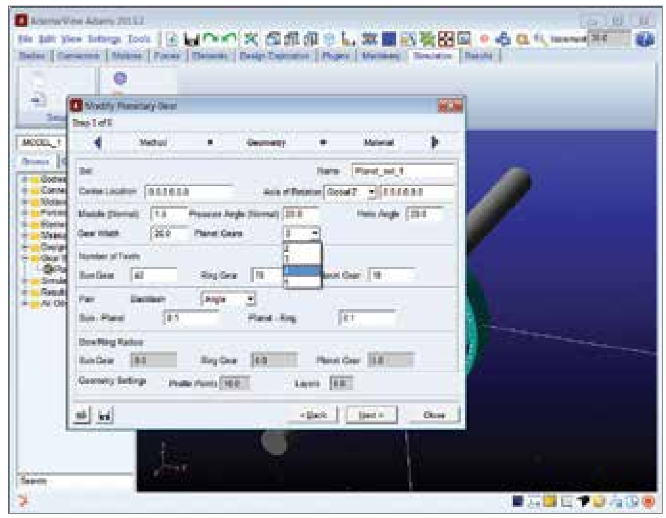



## Step 3. Change the number of planet gears

a. Right click **Planet_set_1** in **Gear systems** in **model browser**, and choose **Modify**.

b. Click **Next** until you get to the **Geometry** setup window.

c. Change the number of planet gears from **3** to **4**, and then continue to click **Next** until the Finish button appears. Then click **Finish** to complete the wizard.



## Step 4. Change the number of teeth

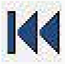
Next, change the number of teeth for **Planetary gear set 1**. To do this:

a. Modify **Planet_set_1** again and run through the wizard, but change the **Number of Teeth** on the **Geometry** step.

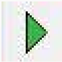
b. Follow the rule:
**Sum(Sun-teeth + 2*planet-teeth) should equal Ring-teeth**
**and**
**Sum(Sun-teeth + Ring-teeth) should be EVENLY divisible by number of planets**

c. For example:
**Sun Gear: 25; Ring Gear: 71; Planet Gear: 23**

## Step 5. Run the simulation again

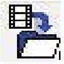


a. Click on the **Rewind** button

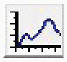


b. Run another simulation with **2 seconds** and **200 steps**

c. Save simulation results as **P2**



## Step 6. Open postprocessor



a. Go to the **post processor**

b. Change Source to **Result Sets**

## Step 7. Compare results

a. Choose **P2** in **Simulation**

b. Click the result **REQ_SIMPLIFIED_Planet_1_to_ring** and **Total Force** (which means the contact force between Ring gear and one of the planet gears in Gear set 1).

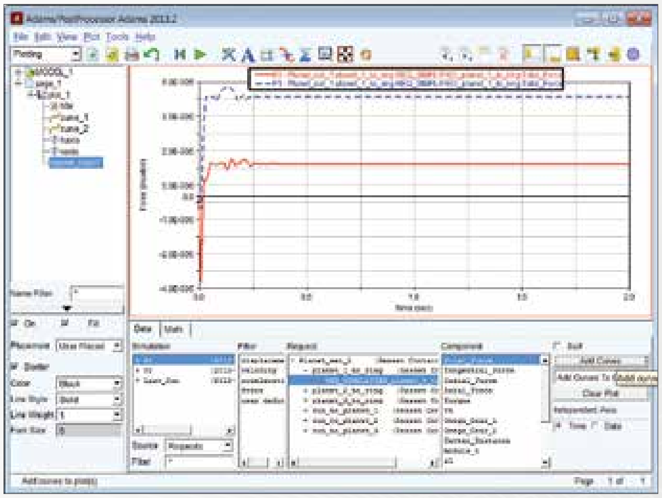
c. Click **Add curve**

d. Choose **P1** in **Simulation**

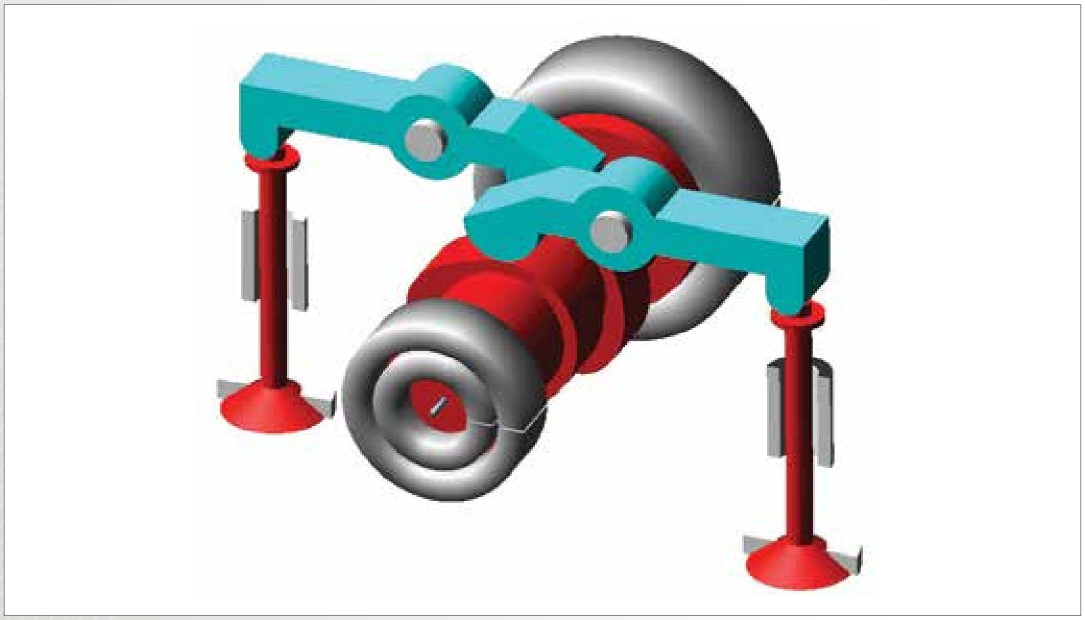
e. Click the result **REQ_SIMPLIFIED_Planet_1_to_ring** and **Total Force**

f. Click **Add curve**

In P1, there are 3 planetary gears; In P2, there are 4 planetary gears. That's why the contact force on each planetary gear is decreased from P1 to P2.



# Example 28: Bearing System Workshop



## Workshop Objectives

- Investigate the system dynamics with Ideal Joint and with Bearing.
- Compare the effect of Ideal Joint and Bearing on system dynamics.

## Software Version

Adams 2013.2

## Files Required

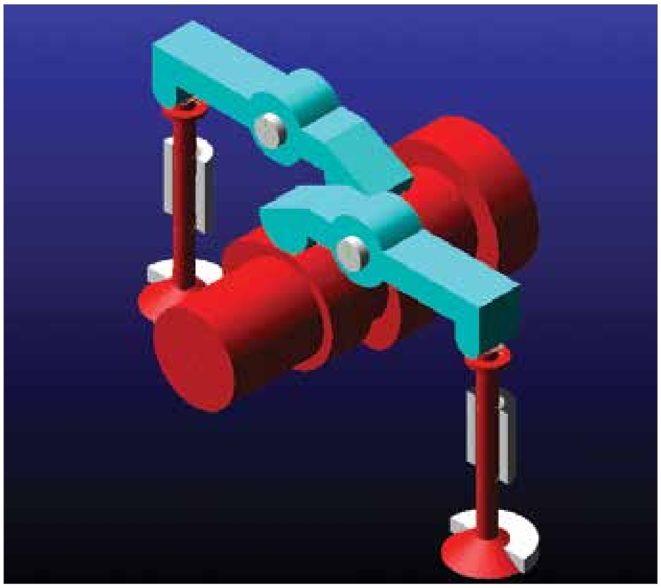
- **CamBearings_start.cmd**
- Located in the directory **exercise_dir/Example 28**

## Step 1. Import Start File

To get started: import the starting model:

a. Open **Adams/View** from the directory **exercise_dir/Example 24**

b. Import the file **CamBearings_start.cmd**.



## Step 2. Simulate Baseline System

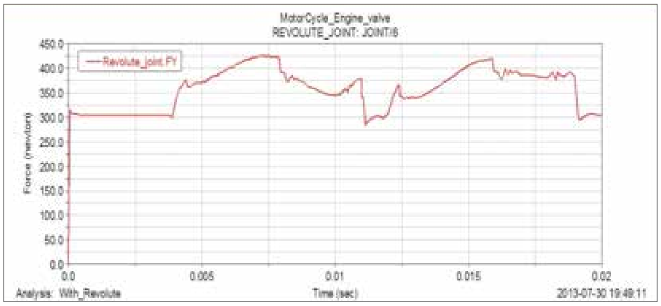
Run a simulation to get familiar with the system operation. To do this:

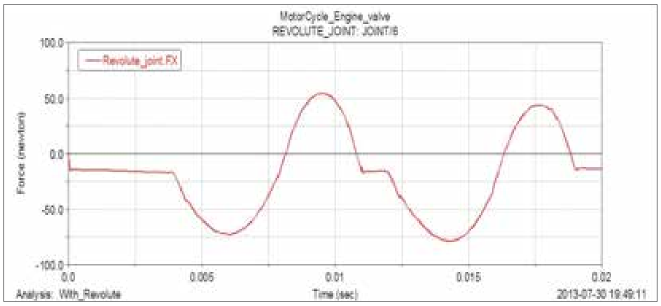
a. Run a simulation for **(1/50)s, 100** output steps.

b. Animate the system a few times and inspect the behavior.

c. Save the results as **With_Revolute.**

d. In **Adams/Post processor** plot the result of **FX** and **FY** for Revolute Joint.



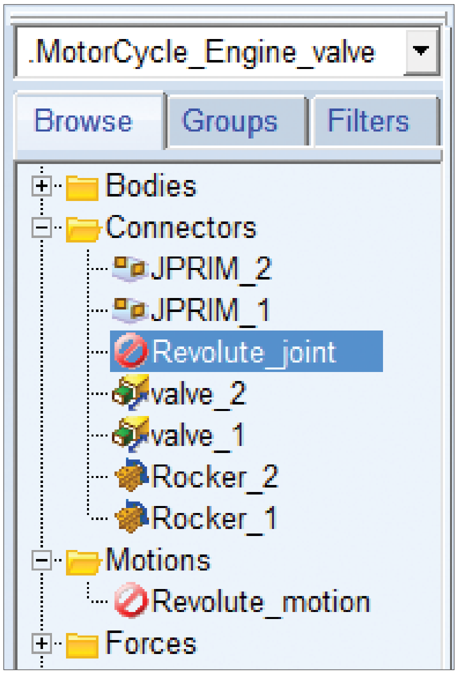

## Step 3. Create First Bearing

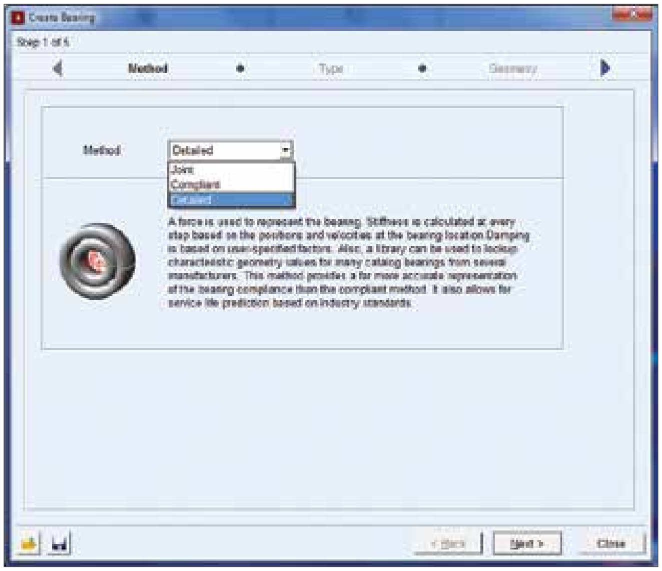
Next, create the first Bearing in the model. Currently there is a revolute joint and motion between the camshaft and ground. We will replace this system with bearing system. To do this:

a. Deactivate **Revolute_motion, Revolute_joint.**

b. Select the **Create Bearing icon.**

c. In the Bearing Method select **Detailed**





d. Advance to the **Type** tab. Select the Type as **Needle Roller Bearing with/without internal ring**.

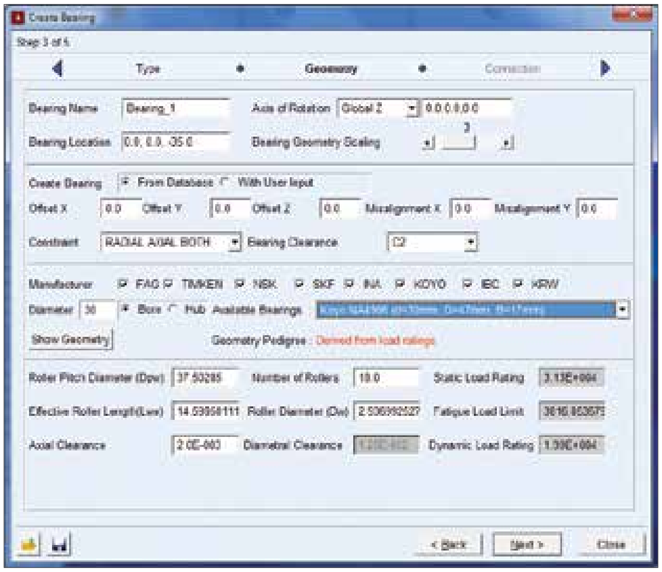
e. Advance to the **Geometry** tab and input all the parameters, as shown:

   a. Select Bearing location as **MARKER_71**.

   b. Select Create Bearing **From Database**.

   c. Enter Bearing **Diameter** as **30 mm.**

   d. Select Bearing Type **Koyo NA4906.**



f. Advance to the **Connection** tab and input all the parameters as shown:

   a. Select **Shaft** as **Main_Shaft** and **Housing** as ground.

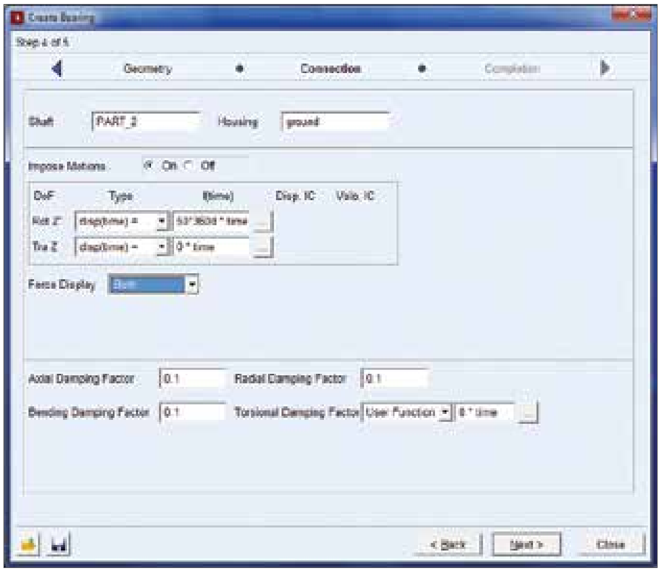
   b. Select Impose Motion **On**.

   c. Input rotation about z-axis as: **disp(time) = 50*360d*time**

   d. Translation about z-axis is fixed

   e. Leave rest as default

g. Advance to the **Completion** tab and Finish the



operation to build the first Bearing.

## Step 4. Create Second Bearing

Next, create the second Bearing in the model. To do this:

a. Select the **Create Bearing** icon.

b. Select **Method** and **Type** as same as first Bearing.

c. Advance to the **Geometry** tab and input all the parameters as shown:

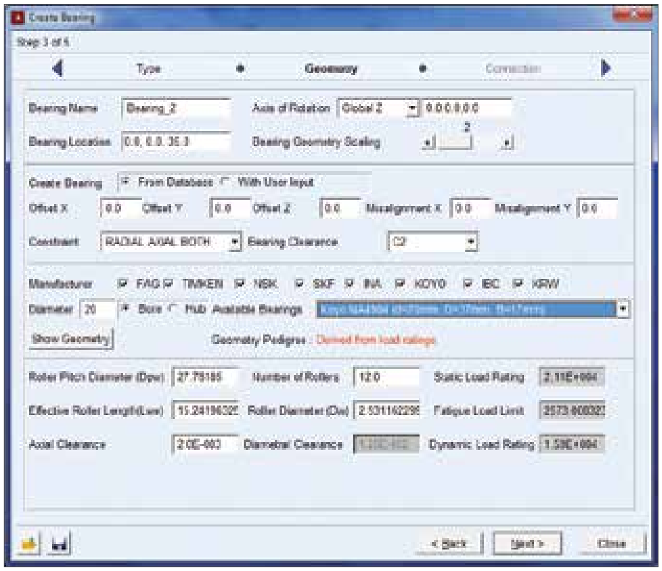
   a. Select Bearing location as **MARKER_1**.

   b. Enter Bearing **Diameter** as **20 mm**.

   c. Select Bearing Type **Koyo NA4904**.

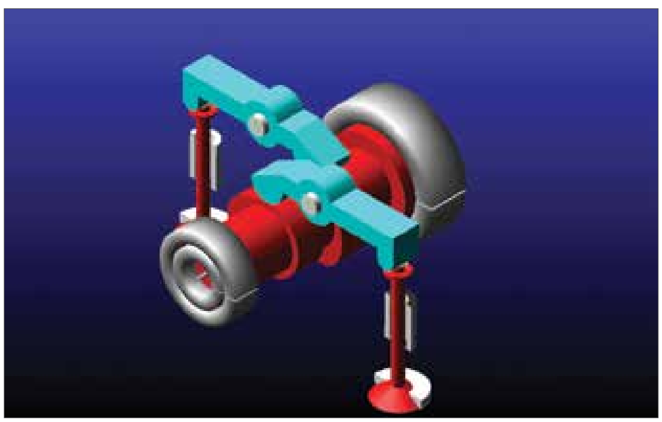
d. Advance to the **Connection** tab and Select **Shaft** as **Main_Shaft** and **Housing** as ground. Leave rest as default.

e. Advance to the **Completion** tab and Finish the operation to build the Second Bearing.



## Step 5. Check Completed Model

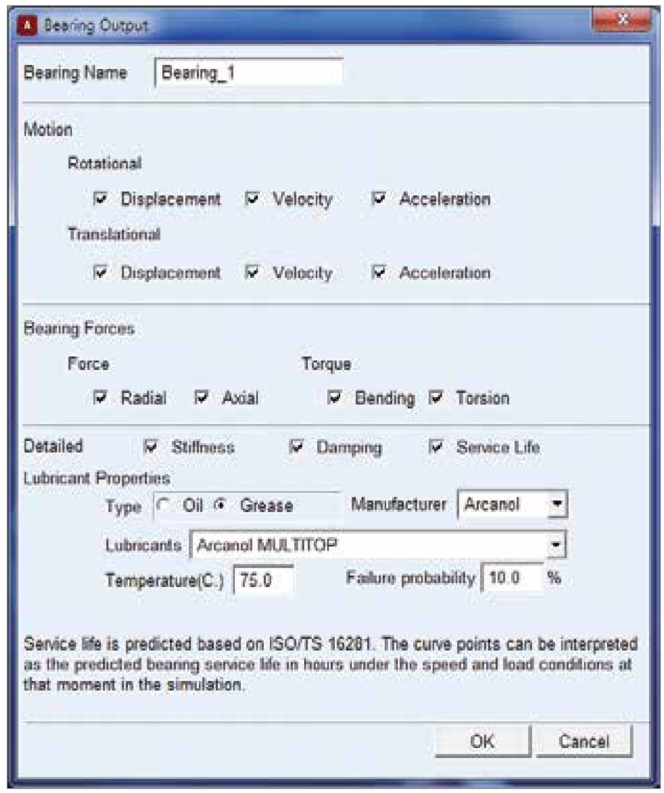
The model with both the Bearings should look like as follows:



## Step 6. Create Bearing REQUESTs

Adams/Machinery provides common bearing properties in the form of REQUEST elements. To create a REQUEST for the bearing characteristics:

a. From the main toolbar select the **Machinery** tab and then the **Bearing Output** button.
b. Select the first bearing: **Bearing_1.**
c. Select all of the output characteristics. Select lubrication properties as shown in figure.
d. Hit OK to create the REQUEST.
e. Repeat the process for the second Bearing in the model.



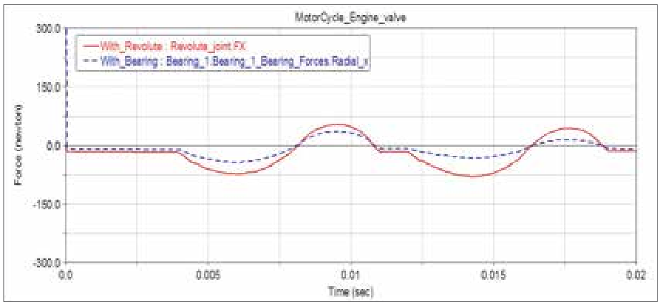
## Step 7. Simulate and Investigate

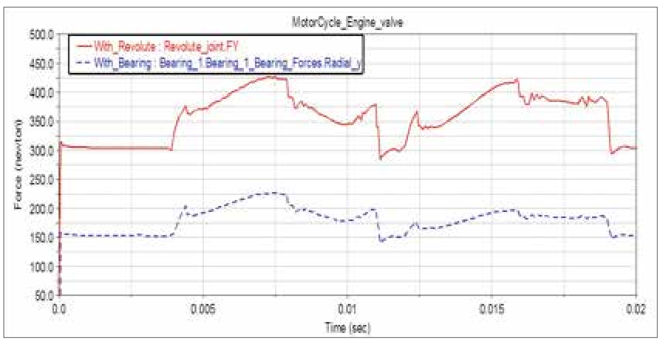
**Run a simulation for the bearing system:**

a. Run a dynamic simulation for **(1/50)s, 100** output steps.

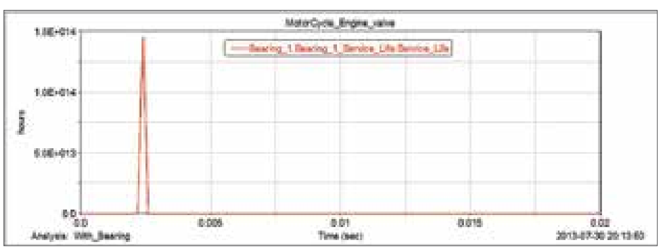
b. Save this analysis to the database as **With_Bearing**.

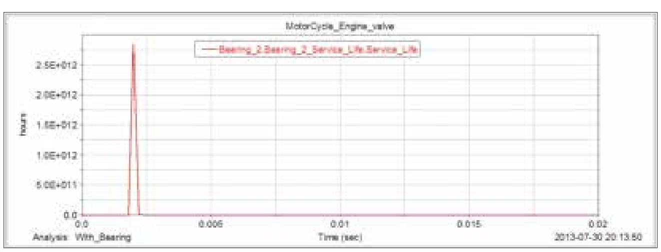
c. Switch to **Adams/PostProcessor** and compare the result of FX and FY of Revolute joint with that of Bearing Forces **Radial_X** and **Radial_Y** of first bearing.




d. Plot Service life for both the bearings.




## Step 8. Further Investigation

Investigate the system performance in other ways.

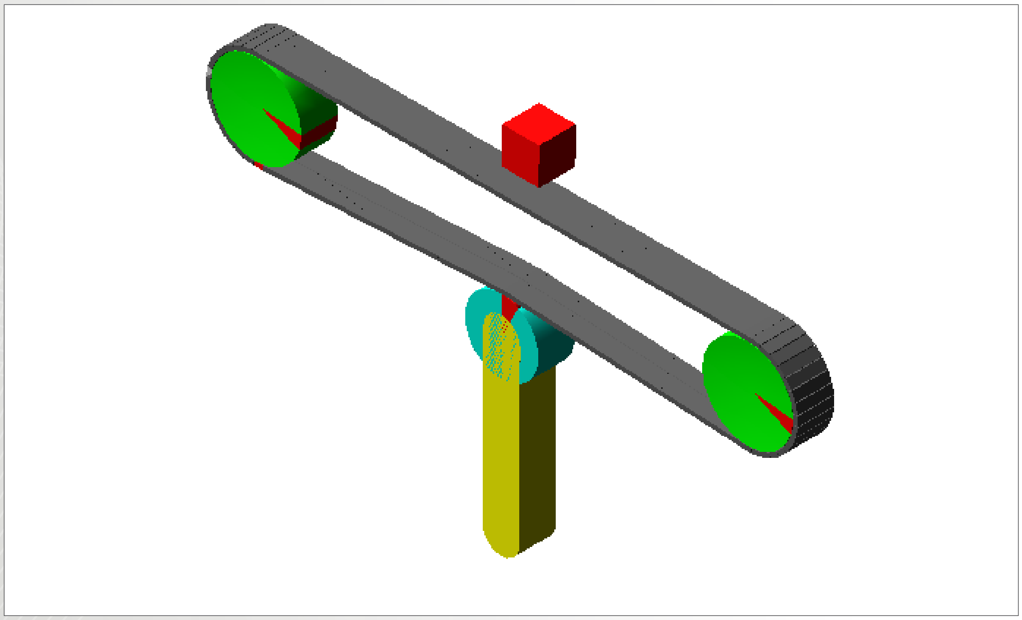
a. Check the Displacement along x and y axis.

b. Change Bearing 2 to bearing type **Koyo NA4904R** and compare the results (Koyo NA4904R has 9 rollers as compared to Koyo NA4904 which has 12 rollers).

c. Change the damping factors and check its effect on the results.

d. Try changing lubricant properties from bearing output and compare the results.

# Example 29: Simple Belt Example



## Software Version

Adams 2013.2

## Problem Description

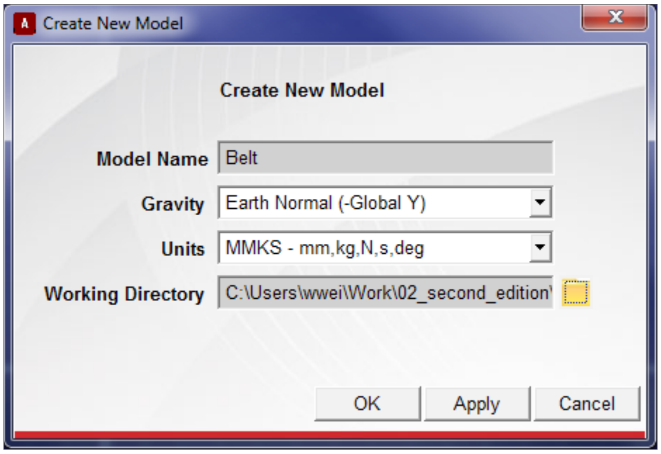
Build a simple belt system with two pulleys and one tensioner. The pulleys are rotating about axes parallel to global z axis. The center distance is 400mm. The pulley is actuated to rotate at constant speed. A box is hanging above the belt system and is dropped at simulation time=0.5s. Adams/View Command Language is used to create multiple contacts and Simulation Script is used to deactivate the fixed joint between the box and the ground.

## Step 1. Open Adams/View and Create a New Model

a. Open Adams/View 2013.2 and select **New Model**.

b. Name the model **Belt** and select a Working Directory.

c. Click **OK.**




## Step 2. Build the Pulleys

a. To build a belt system, we would create the pulleys and tensioners first.

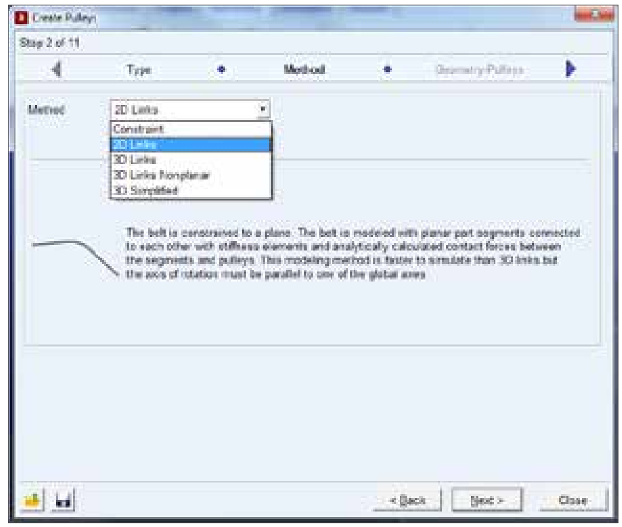
b. From **Machinery** ribbon, select **Belt: Create Pulley**.

c. Change the **Pulley Set Type** to **Smooth**. This configuration creates a simple flat belt system. By the end of this example, feel free to choose other types of pulley sets.

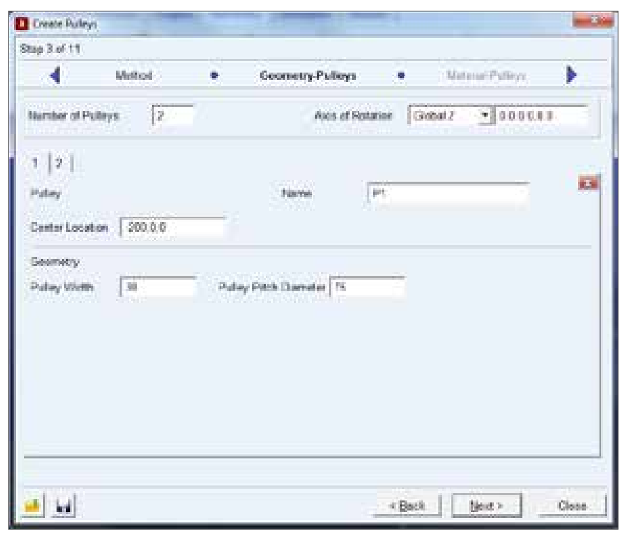

d. Click **Next**.

e. Select **Method: 2D Links**. This method is only used for the cases where the axis of rotation of pulleys are parallel to the global axes. Since this is exactly what we are modeling, we can use this method instead of more complicated ones. Browse through each method to learn the differences among each other.

f. Click Next.




g. Name the first pulley **P1**. Enter **(-200, 0, 0)** for **Center Location**.




h. Click on Tab 2 and name the second pulley **P2** and enter **(200, 0, 0)** for Center Location. Here we are making the center distance of the pulleys to be 400 mm. This is not a large distance. The reason we use such a small distance is that in this way, with less elements modelled, it is faster to run the simulation. This is sufficient for illustrative purposes.

i. Click **Next**.

j. Leave the material properties of pulleys as default. Click **Next**.

k. Leave the connection page as default and click **Next**. In this tutorial, we are actuating the pulley using the built in module of Adams/Machinery. We can also use other bodies to actuate the pulley. For example, we can choose Type: Fixed to fix the pulley to an existing shaft. Later, we can impose a motion/torque on the shaft to actuate the pulley. Here, by selecting Type: Rotational, Adams/Machinery will automatically create a revolute joint between the pulley and the body specified.

l. Now we are asked to select the output of the pulley. Check all the parameters and click **Next**.

m. Now we have completed the specification of the pulleys. In the next step, we will navigate to the pages where tensioners are specified.

## Step 3. Build the Tensioner

a. Under Geometry-Tensioners page, change the **Number of Tensioner with Deviation Pulley** to **1**.

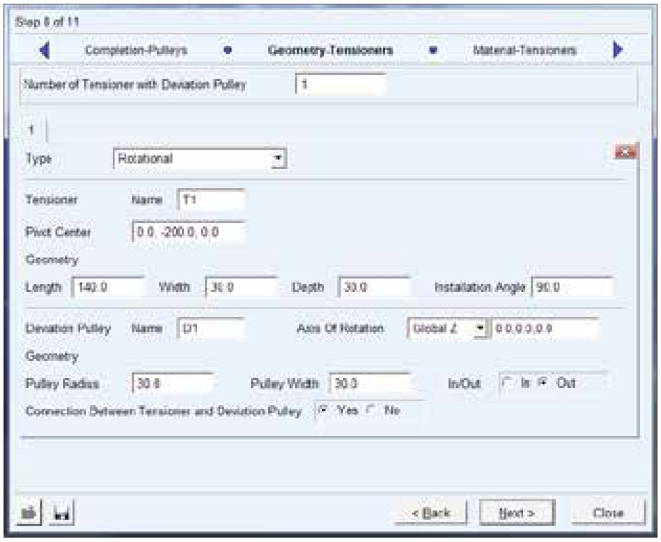
b. Pick **Rotational Type** and Name the Tensioner to **T1**.

c. Enter **(0, -200, 0)** for Pivot Center. This will locate the tensioner beneath the belt system.

d. Enter **140, 30, 30** for Length, Width and Depth respectively. Set the **Installation Angle** to **90**. This means that the deviation pulley is installed 90 degrees counter-clock wise about z axis, which is desired in this case.

e. Name the Deviation Pulley **D1** and enter **30** for **Pulley Radius**.

f. Under In/Out, pick **Out**. This is important because this determines which side the belt will wrap around the deviation pulley. Click **Next.**




g. Leave the material properties as default and click **Next**.

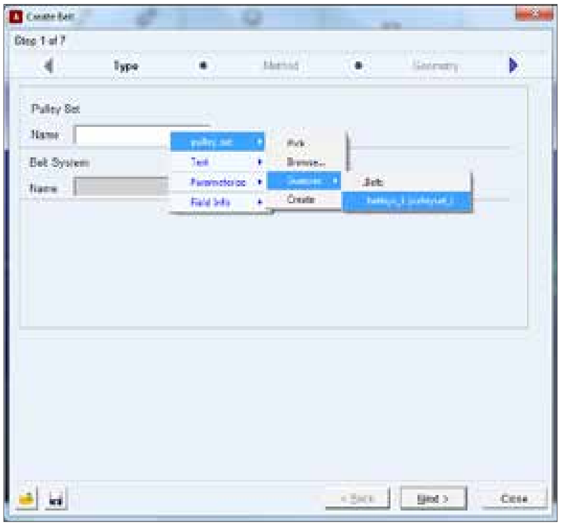
h. Under Connection page, we connect the tensioner to ground.

i. Enter **300, 0.1, 50** for Stiffness, Damping and Preload. These values are for illustrative purposes. Click **Next** and then **Finish**.

## Step 4. Build the Belt

a. From **Machinery** ribbon, select **Belt: Create Belt**.

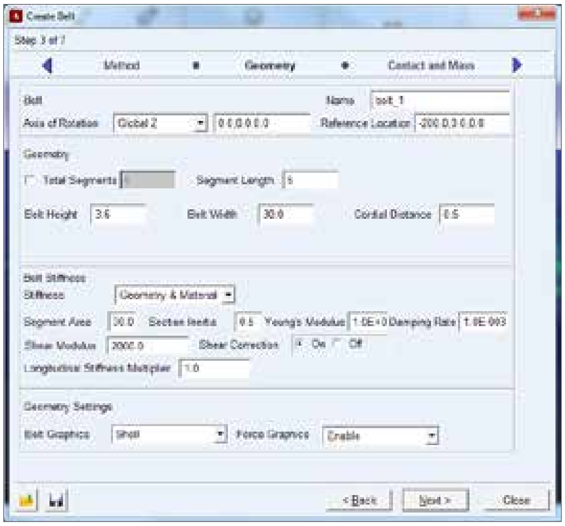
b. Right click in the name field and go to **pulley_set->Guesses->beltsys_1.pulleyset_1**.




c. Click **Next**.

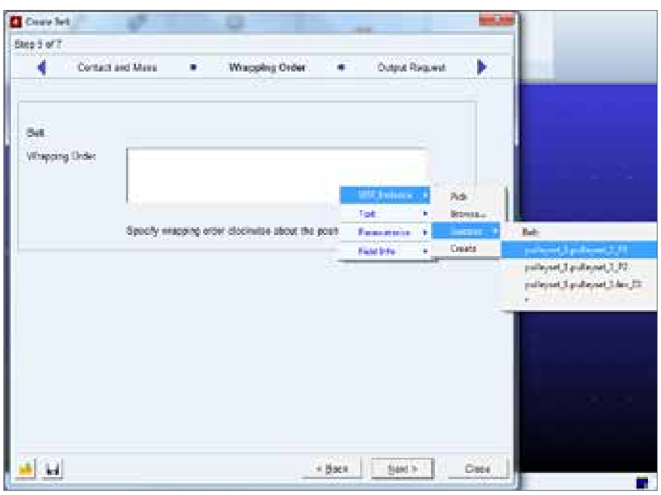
d. Select **2D Links** Method, or make sure the method is consistent with your pulley modeling method. Click Next.

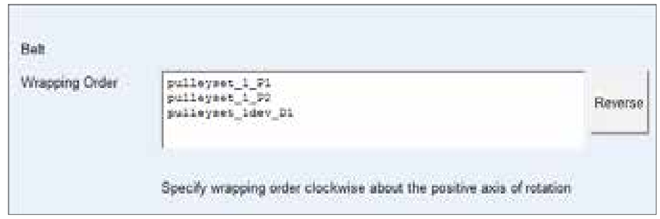
e. There are a number of parameters we can set to determine the geometry of the belt. Press F1 to check what each of the parameters does. Change the **Segment Length** to **7.5** and then click **Next**.



f. This page is to specify the contact and mass property of belts. This is beyond the scope of this tutorial. If you want to learn more about this, press F1 to view the Help documents. Click **Next**.

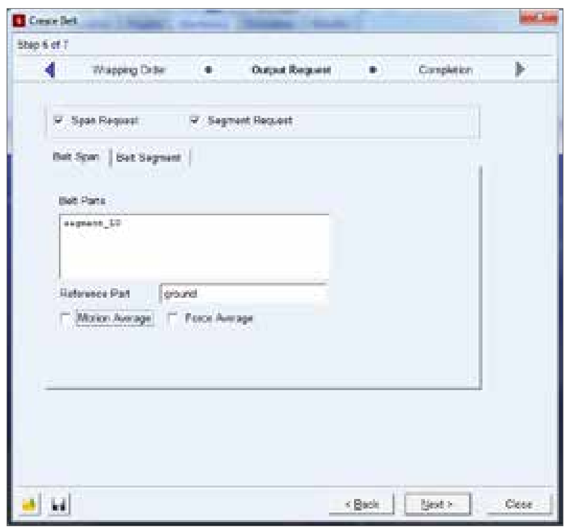
g. Now we have to specify the Wrapping Order. Right click in the field and go to **UDE_Instance->Guesses->Pulleyset_1.pulleyset_1_P1**. Choose **P2** and **D1**. Make sure the wrapping is in the order of P1, P2 and D1. The wrapping order is **clockwise** about the positive axis of rotation. Click **Next**.

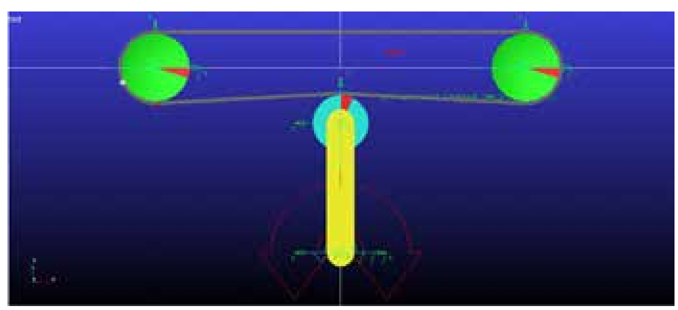





h. A message will tell you the number of segments and the tension. Write the number of segments down because it will later be used when we are creating the contacts. It is 138 in this example. Click **Yes**. You might see a warning. This is merely telling us that we should choose C++ solver. We will set this when we are about to run the simulation.

i. Check Span Request and Segment Request. Right click in the field to select any of the segments. What is important is to get to know that we can get useful outputs from Adams/Machinery. Click **Next** and then **Finish**.

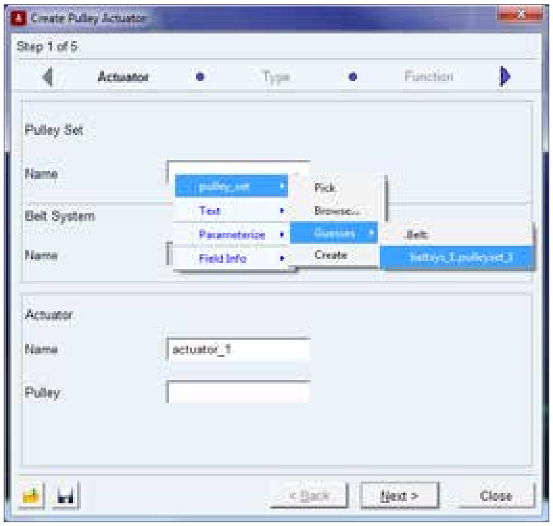




## Step 5. Create Actuator

a. From **Machinery** ribbon, select **Belt: Belt Actuation Input**.

b. Right click in the Name field and select the pulley set we have created.




c. For the **actuator pulley**, select **P1**. Click **Next**.


d. Pick **Motion** for **Type**. Click **Next**.

e. Select **Function: User Defined** and enter **step(time, 0,0,0.25,40)**. This defines a step function in Adams. Step function in Adams has been explained in Example 18. Click **Next**.

f. Check all the requests. Click **Next** and then **Finish**.

## Step 6. Set Solver Settings

a. From the menu bar, select **Settings->Solver->Executable**.

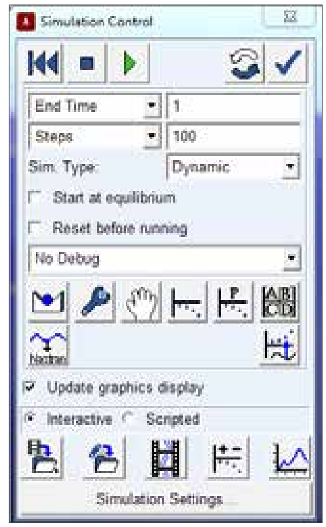
b. Under **Solver**, select **C++**.

c. Change category from Executable to Dynamics and select Integrator: HHT.

## Step 7. Run a Simulation

a. Run a simulation with 1 second and 100 steps. Set **Sim. Type** to **Dynamic**. Make sure that the model works well.

b. The simulation make take time depending on the configuration of your machine.



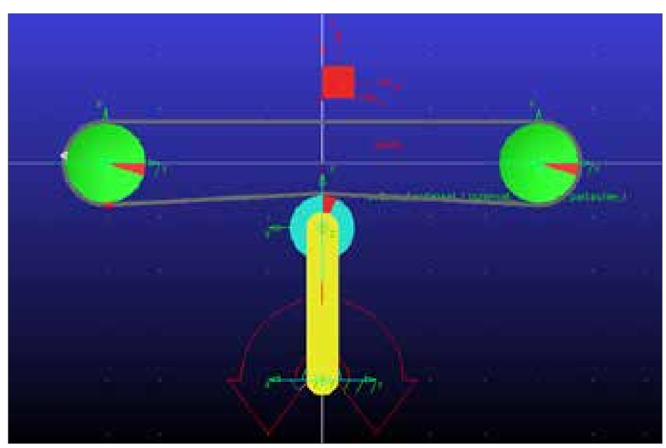

## Step 8. Create a Box

a. From **Bodies** ribbon, select **RigidBody: Box**.

b. Pick **New Part** and check all three dimensions. Enter **30** in all the fields. This will create a cube that is equal in depth as the belt.

c. Right click in the background and invoke the Location Event.

d. Enter **0, 60,-15** as the corner of the cube.

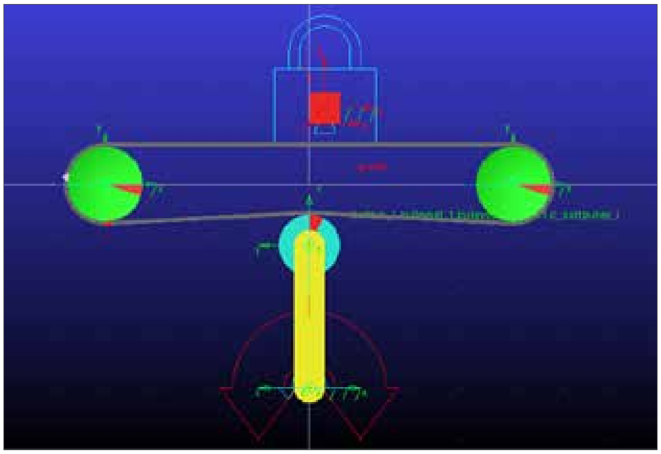


## Step 9. Create a Fixed Joint

a. Now a fixed joint will be created between the box and the ground. This joint will later be deactivated during simulation using script.

b. From **Connectors** ribbon, select **Joint: Fixed**.

c. Pick the box first and then the ground. Pick PART_#.cm as the location. You can right click around the cube to invoke the select list. # is the number of the part in your model.



## Step 10. Create Contacts using Adams/View Command Language

a. In order to model the interaction between the dropping box and the belt, contacts should be added between the box and the belt. Since the belt is composed of 138 segments, we have to add 138 contact pairs. This is tedious to accomplish if we create them separately. However, Adams/View Command Language enables us to include this process in a for loop.

b. Right click on contact.cmd located in the start file folder and edit it with any text editor of your choice.

c. When the file opens, you should see the following lines.

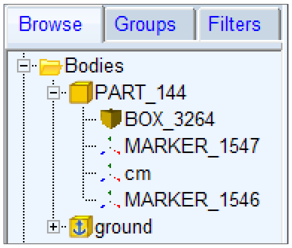
```
FOR variable_name=tempreal start_value=1 end_value=138
    contact create &
    contact_name = (eval(".Belt.CONTACT_"//RTOI(tempreal)))  &
    adams_id = (eval( unique_id("CONTACT") ))&
      &
    i_geometry_name = BOX_3264  &
    j_geometry_name = (eval("segment_"//RTOI(tempreal)//".block"))  &
      &
      &
      &
    stiffness = 1.0E+005  &
    damping = 10.0  &
    exponent = 2.2  &
    dmax = 0.1
END
```

d. Make several changes to the template.

e. In **line 1**, replace the nubmer after “end_value =”with the number of segments you write down in step 4h.

f. In **line 3**, replace Belt with your model name in case you name your model in your own way.

g. In **line 6**, replace BOX_3264 with BOX_# where # is the number of the box. You can browse your model tree to determine this number.



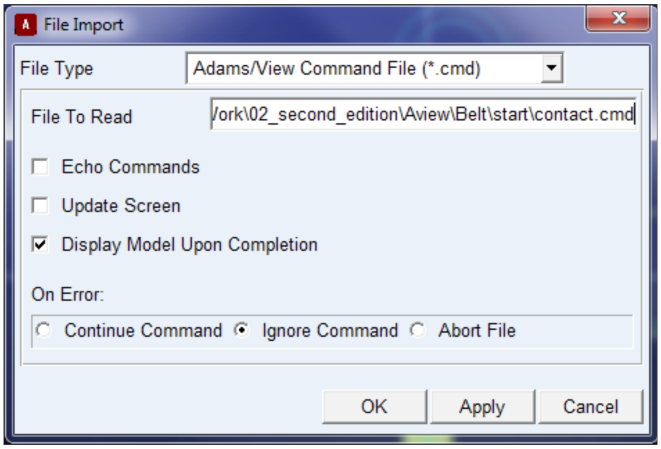

h. Save the changes to **contact.cmd**. For more information about Adams/View Command Language, please refer to Adams Online Help Document.

## Step 11. Import the .cmd

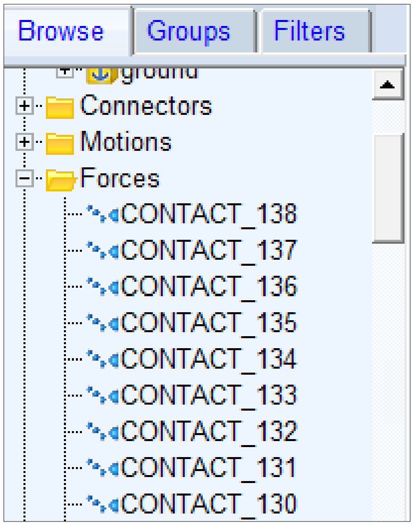
a. From the menu bar, select **File->Import**.

b. Under **File Type**, select **Adams/View Command File (*.cmd)**.

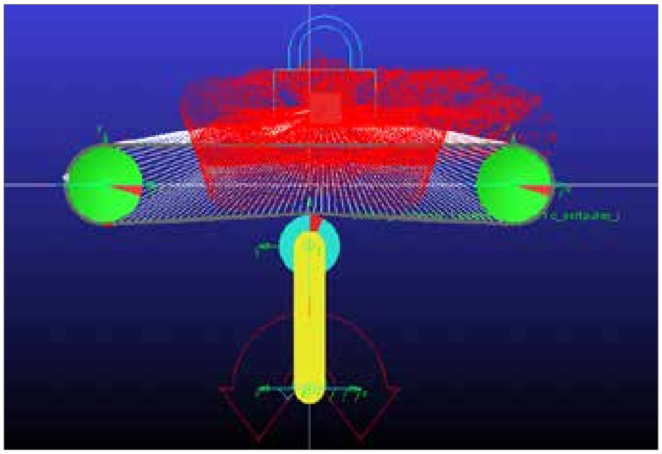
c. Right click in the File to Read field and browse to **contact.cmd**. Click **OK**.




d. Expand the Forces in model tree and you will find that 138 contacts has been added.




e. Now the view window is messy because of the contacts added.

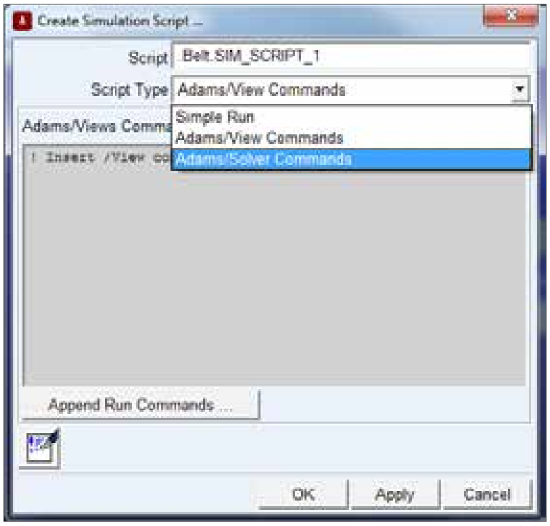


f. Press V to hide the icons.

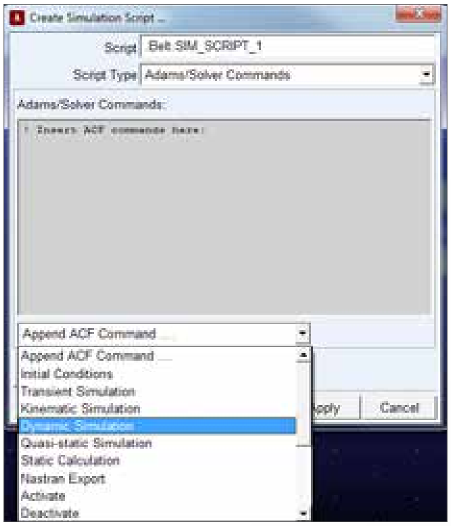
## Step 12. Create a New Simulation Script

a. From **Simulation** ribbon, select **Setup: Create a new Simulation Script**.

b. Choose **Script Type** as **Adams/Solver Commands**.

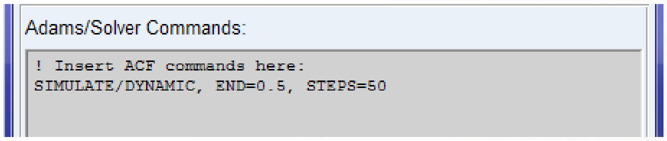



c. Under the menu of **Append ACF Command**, choose **Dynamic Simulation**.

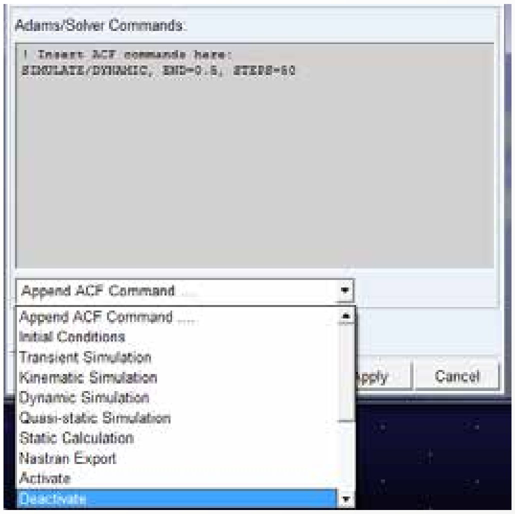



d. Enter **50** and **0.5** for Number of Steps and End Time. Click **OK**.

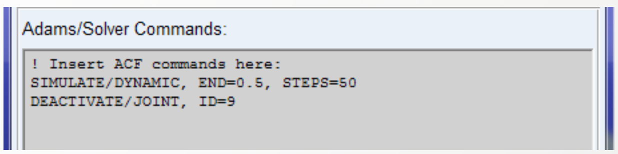
e. Now a new line has been appended to the script.




f. From the drop down menu, select **Deactivate**.



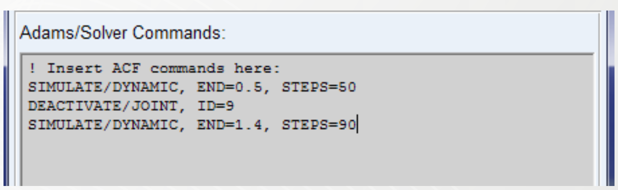

g. Right click and browse for the fixed joint in the Joint Name field. Click **OK**. Note that the ID number of the joint may vary.




h. Now another line has been added.

i. Add another line of dynamic simulation which ends at t=1.4s with 90 steps.

j. The final Adams/Solver Commands would look like this. The simulation will start with a dynamic simulation at t=0. At t=0.5s, the fixed joint is deactivated and the cube drops. A dynamic simulation continues and ends at t=1.4s. Click **OK**.




## Step 13. Run a Scripted Simulation

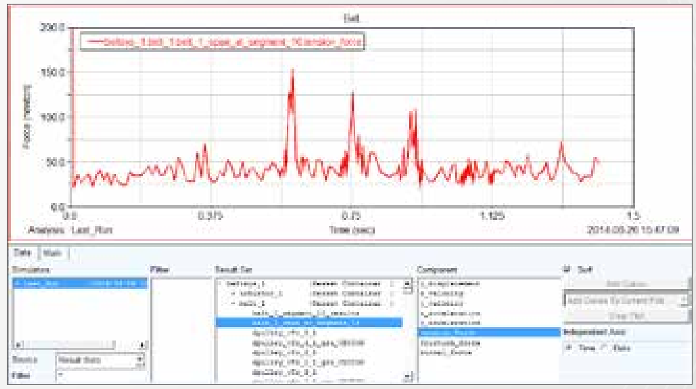
a. From **Simulation** ribbon, select **Simulate: Run a Scripted Simulation**.

b. Right click in the Script Name field and select the script just created.

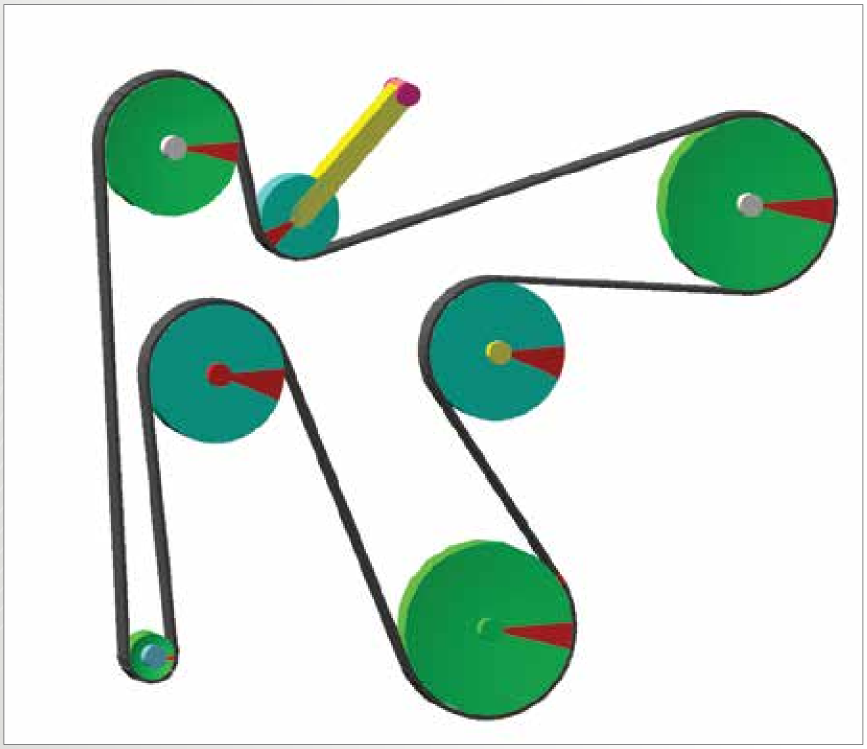
c. Start Simulation.

d. The simulation would take some time depending on your machine. After simulation, press F8 to enter postprocessing window and you may check the result output of the belt now.

e. Select **Result Sets** as **Source**, expand **beltsys_1** and then **belt_1**. Pick **belt_1_span_at segment_10**. Select **Tension_force** and then click **Add Curves**.



# Example 30: Serpentine Belt System



## Workshop Objective

Use **Adams/Machinery**

- Create, simulate and animate the serpentine belt system.

## Software Version

- Adams 2013.2
- Adams/Machinery Plugin with belt module is required

## Files Required

- serpentine_belt_start.cmd
- Located in the directory **exercise_dir/Example 30**

# Problem Description

- The model represents a serpentine belt system.
- The crank shaft is being rotated at a given velocity.
- The rotation of crank shaft pulley is transferred to the other pulleys through the belt.

# Tips before you start

- Deactivate Solver Compatibility Checking:

a. When reading/writing model files Adams/View will, by default, check modeling elements for compatibility with the C++ Solver.

b. This checking can be time consuming for models with many parts.

c. Turn this check off via:

   a. **Tools -> Command Navigator** and then
   b. **defaults ->** solver and then
   c. **Compatibility Checking = Off**

- Turn off Model Verify:

a. **Adams/Solver** checks for redundant constraints, massless parts, etc before simulations. This can be time consuming for large models.

b. Turn off Model Verify via:

   a. **Settings -> Solver -> Executable**
   b. Set **Verify First = No**

### Step 1. Import File

a. Start **Adams/View.**

b. From the Welcome dialog box, select **Existing Model.**

c. Click the file folder icon, and the Select Directory dialog box appears.

d. Find and select the directory **Exercise_dir/belt_module**

e. Click **OK**.

f. Click on the file folder icon of the **File Name**, select the file

g. **serpentine_belt_start.cmd** and click **Open**.

h. Click **OK** on the **Open Existing Model** dialog box

### Step 2. Build the Pulley Set

a. Click on **Create Pulley** icon under Belt ribbon.

b. Set **Belt System Nam**e as **beltsys_1.**

c. Set **Pulley Set Name** as **pulleyset_1.**

d. Select **Smooth** as a **Type**.

e. Click on **Next**

f. Select **2D Links** as a Method

g. Click on **Next**

### Step 3. Geometry & Material Properties

a. Enter **4** in the field **Number of Pulleys** and hit **Enter**.

b. Select **Axis of Rotation** as **Global Z**

c. Click on **tab "1**" and enter Name **crank_shaft_p** for Pulley1

d. Enter **0,0,0** in field **Center Location**.

e. Enter **30** in the field **Pulley Width**

f. Enter **150** in the field **Pulley pitch diameter**

g. Click on **tab "2", "3"** and **"4"** to enter values for Pulley2, Pully3 and Pully4.

| Pulley# | Name | Center Location | Width | pitch diameter |
|---|---|---|---|---|
| 1 | crank_shaft_p | 0,0,0 | 30 | 150 |
| 2 | alternator_shaft_p | -300,0,0 | 30 | 40 |
| 3 | ac_shaft_p | -260,440,0 | 30 | 120 |
| 4 | power_strg_shaft_p | 250,350,0 | 30 | 150 |

h. Click on **Next** button

i. Specify Pulley Material

j. Keep default material for Pulleys 1, 2, 3 and 4

k. Click on **Next** button

## Step 4. Pulley Connection

a. Click on **Pulley1** tab to define connection for **crank_shaft_p.**

b. Select **Type** as **Fixed** to fix the pulley to **crank_shaft**

c. Select **Existing** for **Body** to enter existing part name to which pulley will be connected using selected Type as **Fixed**.

d. Right click -> **Body -> Guesses ->** and select part name **crank_shaft** (or Browse for existing part)

e. Click on **tab "Pulley2", ""Pulley3"** and **"Pulley4"** to enter values for Pulley2, Pulley3 and Pulley4.

| Pulley | Type | Body |
|---|---|---|
| Pulley1 | Fixed | Existing -> crank_shaft |
| Pulley 2 | Fixed | Existing -> alternator_shaft |
| Pulley 3 | Fixed | - Existing -> ac_shaft |
| Pulley 4 | Fixed | Existing -> power_strg_shaft |

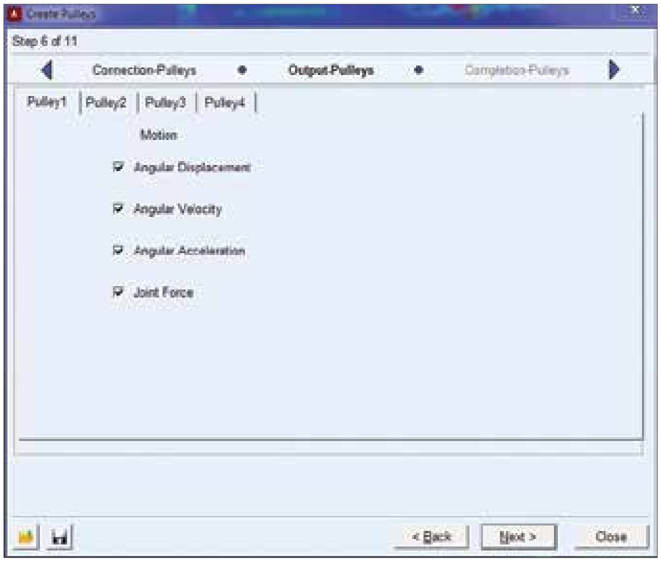
## Step 5. Specify Pulley Outputs

a. Select default outputs for Pulleys 1, 2, 3 and 4 for post processing using **Adams/PostProcessor**

b. Click on **Next** button.

c. Completion-Pulleys page concludes the pulley specification.

d. Click on **Next** button.




## Step 6. Geometry-Tensioners

a. Enter 3 in the field Number of Tensioner with Deviation Pulley and hit Enter.

b. Select Type as Fixed for tab "1"

c. Enter Name as dev1 for Deviation Pulley1

d. Select Axis of Rotation as Global Z

e. Enter 20,240,0 in the field Center Location.

f. Enter 60 in the field Pulley radius

g. Select **Out** for In/Out

| | Type | Deviation Pulley Name | Axis of Rotation | Center Location | Pulley Radius | Pulley Width |
|---|---|---|---|---|---|---|
| Tab 1 | Fixed | dev1 | Global Z | 20,240,0 | 60 | 30 |
| Tab 2 | Fixed | dev2 | Global Z | -230,240,0 | 60 | 30 |
| Tab 3 | Rotational | dev3 | Global Z | 20,240,0 | 35 | 30 |

h. Repeat the above steps to setup Tab 2 & 3.

Additional setup in Tab 3:
**Tensioner3 Name:** Ten3

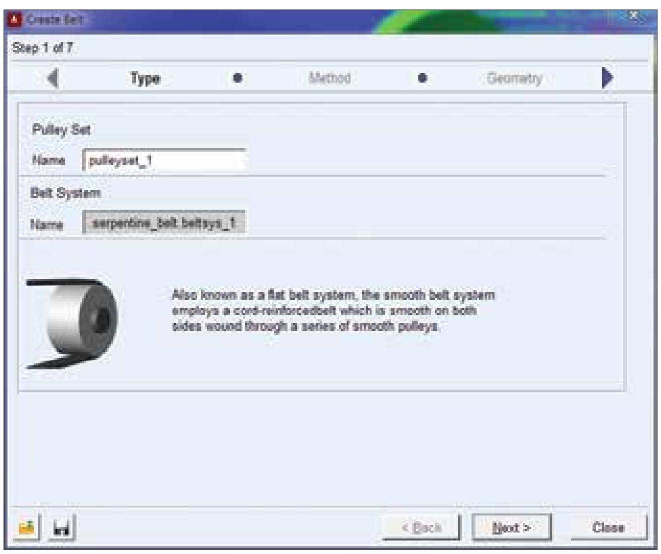
**Pivot Center:** -50.0, 470.0, 0.0
**Length:** 140
**Width**: 20
**Depth:** 30
**Installation Angle**: 225
**Connection Between Tensioner and Deviation Pulley:** Yes

## Step 7. Material & Connection Tensioners

a. Keep default material for Deviation Pulleys 1, 2 and 3 and Tensioner
b. Click on **Next**
c. Click on **Tab "3"** to define connection for tensioner3.
d. Select **Yes** for Tensioner connector
e. Select **Existing** for **Body** to enter existing part name.
f. Right click -> **Body -> Guesses ->** and select part name **tensioner_shaft** (or Browse for existing part)
g. Enter **100** for **Stiffness, 1** for **Damping** and **100** for **Preload**
h. Click on **Next**.
i. Click on **Finish**

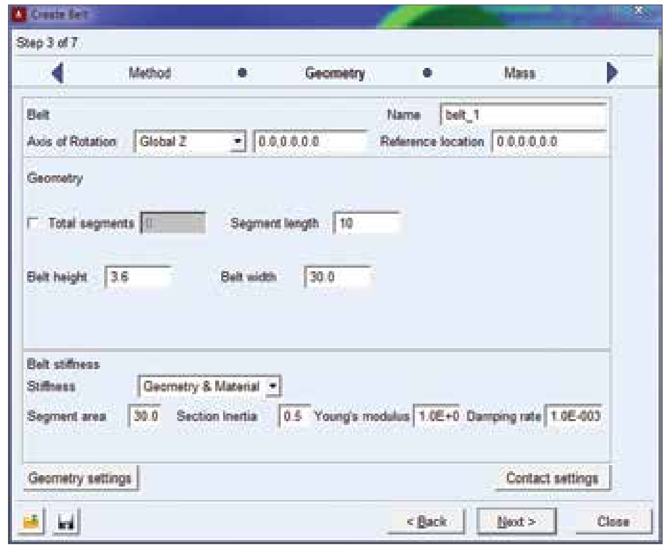
## Step 8. Create Belt

a. Click on **Create Belt** icon under Belt ribbon.
b. Select the name of the existing **pulleyset** to be used. Right click -> **Name -> Guesses ->** and select **pulleyset** name (or Browse for existing pulleyset).
c. Select **Next** to advance to the next Step.
d. On the **Method** Step specify **2D Links** for the **belt type.**
e. Click on **Next**.



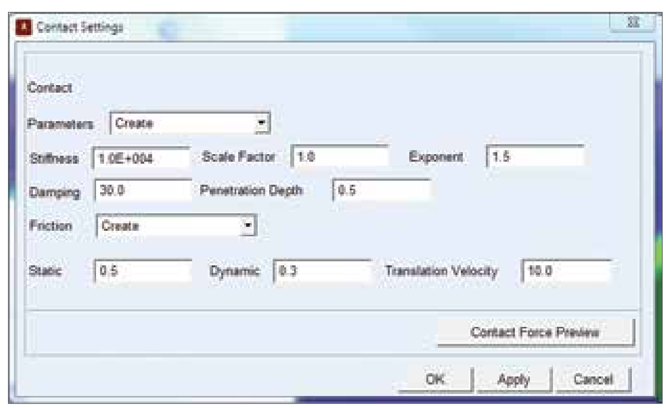

## Step 9. Belt Geometry

a. Select **Axis of Rotation** as **Global Z**
b. Enter **10** in the field for **Segment Length.** Note that the user can also choose a larger segment length to reduce the computation time and then gradually refine the model if the model has been verified.
c. Enter **3.6** for **Belt height** and **30.0** for **Belt width**
d. Enter **Belt Stiffness** values as shown.




e. Select the Contact Settings button and use the settings shown on the following page.
f. Click on **Next** button.

## Step 10. Belt Contact Settings

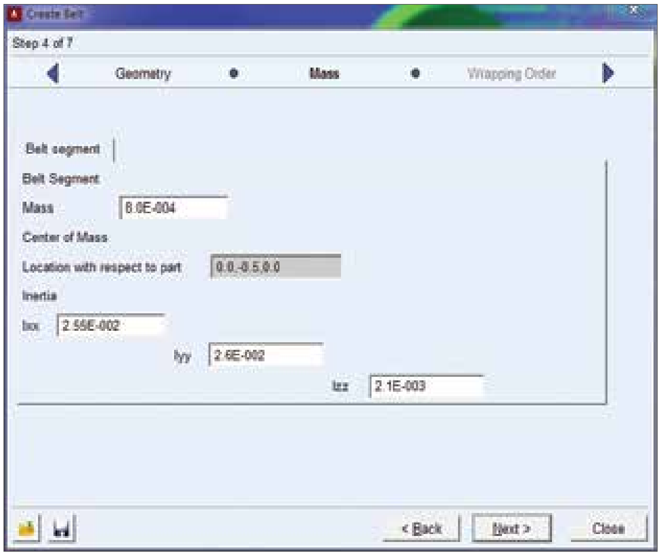



In the **Contact Settings** screen use the values shown here:

a. Specify the properties as shown.
b. Select OK to return to the wizard tab.

## Step 11. Belt Mass

a. Specify the properties as shown.
b. Click on **Next** button.




## Step 12. Belt Wrapping Order

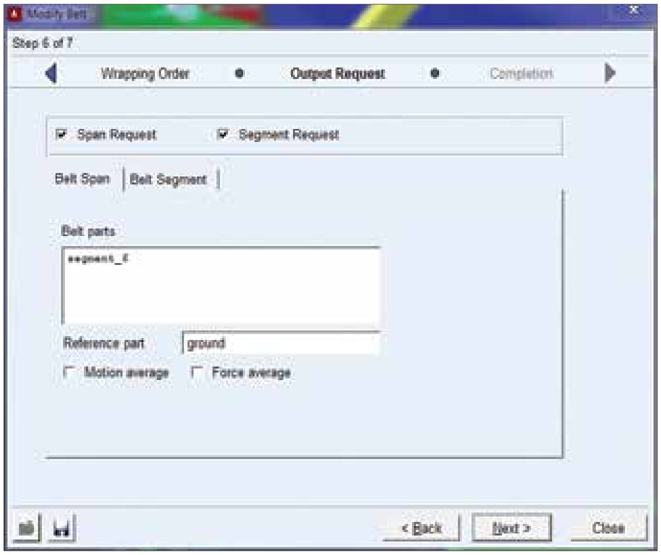
a. Read the Notes below and then use the following for the belt wrapping order:
.serpentine_belt.beltsys_1.pulleyset_1**.pulleyset_1_crank_shaft_p**
.serpentine_belt.beltsys_1.pulleyset_1**.pulleyset_1dev_dev2**
.serpentine_belt.beltsys_1.pulleyset_1**.pulleyset_1_alternator_shaft_p**
.serpentine_belt.beltsys_1.pulleyset_1**.pulleyset_1_ac_shaft_p**
.serpentine_belt.beltsys_1.pulleyset_1**.pulleyset_1dev_dev3**
.serpentine_belt.beltsys_1.pulleyset_1**.pulleyset_1_power_strg_shaft_p**
.serpentine_belt.beltsys_1.pulleyset_1**.pulleyset_1dev_dev1**

b. **Important Notes on Belt Wrapping:**

- Belts must be wrapped in a **clockwise** fashion with respect to the belt axis of rotation.
- Right-click and 'pick' functionality does not currently work for pulleys – you must use the 'Guesses' and 'Browse' functionality.
- Note that pulley names are shown in the modeling window to help when using the 'Guesses' functionality.
- Click on **Next** button

## Step 13. Belt Output

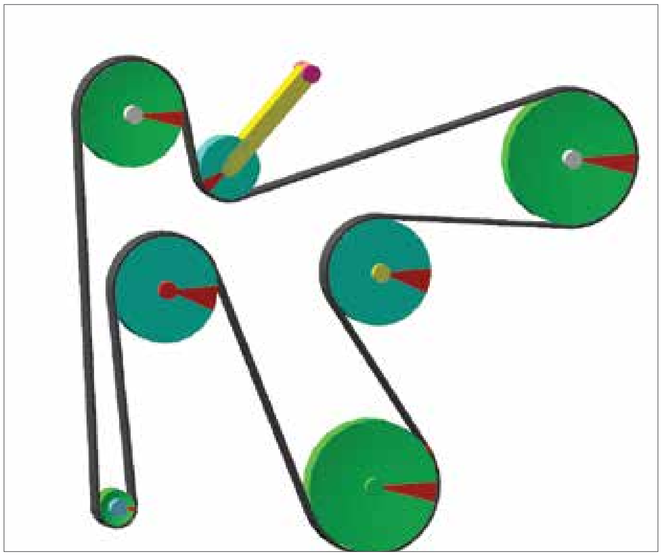
a. Select both the **Span Request** and **Segment Request** checkboxes.
b. Right click in the field and select parts, then select Browse/Guess. Select an arbitrary belt part from the list for both the belt span and segment outputs.
c. Follow the similar procedure in b and select ground as Reference part.
d. Click on **Next** button.



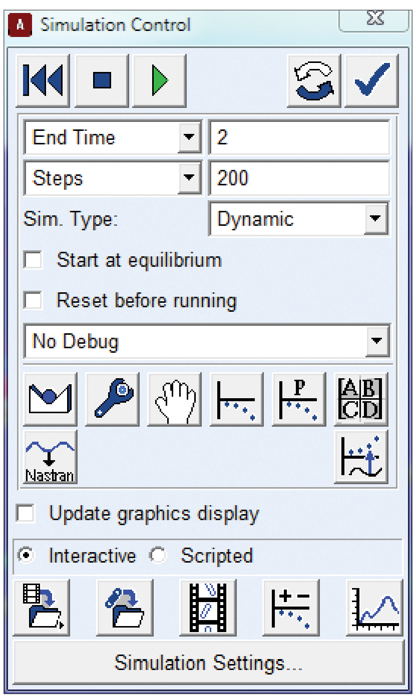
## Step 14. Belt Completion

The system with a properly wrapped belt should look like the following:

## Step 15. Simulation Setup

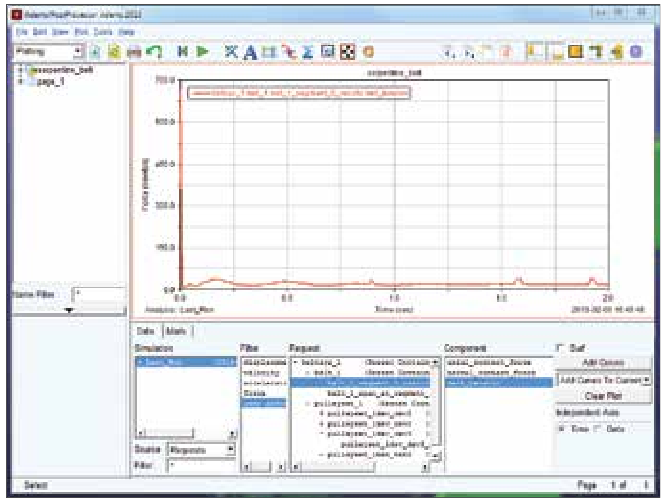


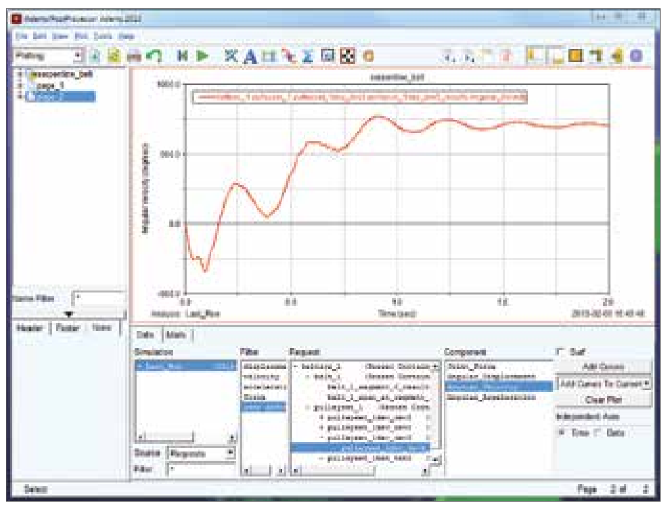
Ensure that appropriate settings are used for Adams/ Solver while handling belt (many parts) simulations. Do the following:




a. Open the Solver Dynamics settings via the menu picks:
   a. Settings -> Solver -> Dynamics
b. For Dynamics use these parameters:
   a. Integrator = HHT
   b. ERROR = 1e-5
c. In this dialog box change the **Category** to be **Executable** and make the changes:
   a. Choice = C++
   b. Verify First = No

The changes above instruct the Solver to use the HHT integrator which is best for belt/chain models.





## Step 16. Simulate & Animate

a. Ensure that there is a MOTION on the crank shaft joint, JOINT_1. The MOTION should be of type Velocity and use the following function expression to gradually spin the crank shaft input up:
**step5(time, 0, 0, 1, 180d)**

b. Run a dynamic simulation for **2** seconds, **200** output steps.

c. Animate the results.

d. Note that the simulation may take a long time depending on the configuration of your machine.

## Step 17. Investigate System

Create further animations and plots to illustrate things such as:

a. How does the belt tension (found in the belt segment REQUEST) vary over time?

b. Does slippage occur in the belt tensioner? (View the angular velocity of the tensioner deviation pulley through time).

## Workshop Questions

1. When specifying a belt wrapping order, what orientation must be used?..........

2. Pulleys can be connected to the model using three different specifications; list them: ..........

3. Which integrator is suggested for belt/chain systems (systems with many inter-connected parts that move only slightly with respect to one another)?..........

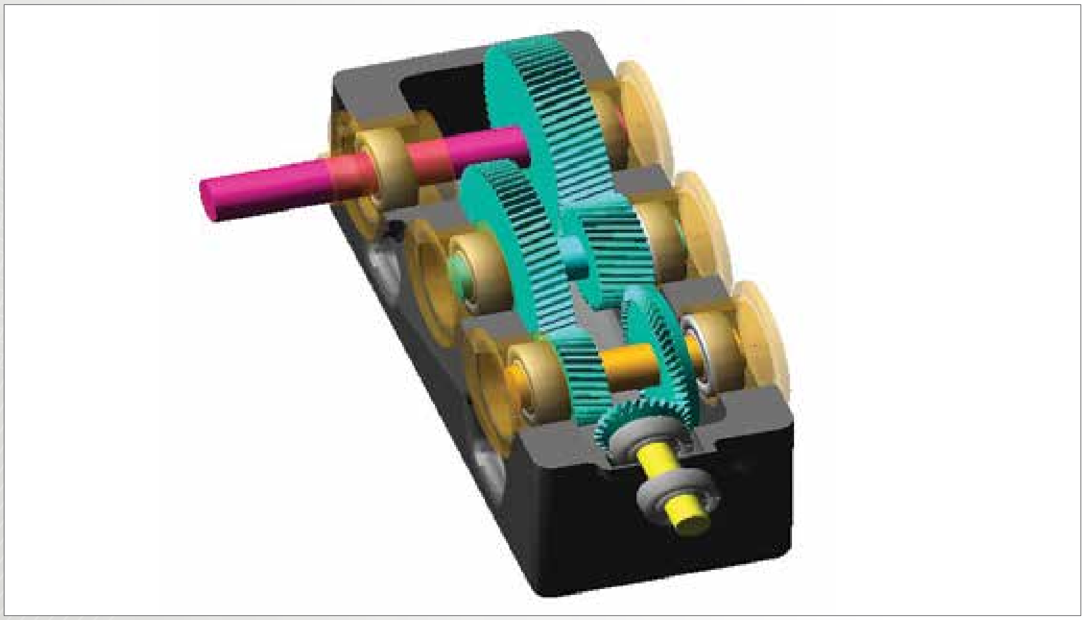
Answers:

1. When wrapping the belt the pulleys must be specified in a clockwise sequence with respect to the pulley axis of rotation.

2. Pulleys can be attached to the model using rotational joints, fixed joints or bushings (compliant).

3. The HHT integrator is generally best for belt/chain-type systems. Run your simulation with the default ERROR control (1e-5) for HHT and then re-run with tighter ERROR control to see if results change. If results change with tighter ERROR then run a solution convergence study on the ERROR parameter to identify a proper setting.

# Example 31: Gear Train



## Workshop Objectives

- Create both the helical gear pairs and the bevel gear pair
- Get familiar with the parameters to set up the gear pair
- Learn to convert a rigid part to a flexible part
- Use dummy parts to connect a flexible part with bearings or gears

## Software Version

- Adams 2013.2
- Adams/Machinery Plugin with gear and bearing module is required

## Files Required

- Withbacklash.cmd
- Directory: **exercise_dir/Example 31**

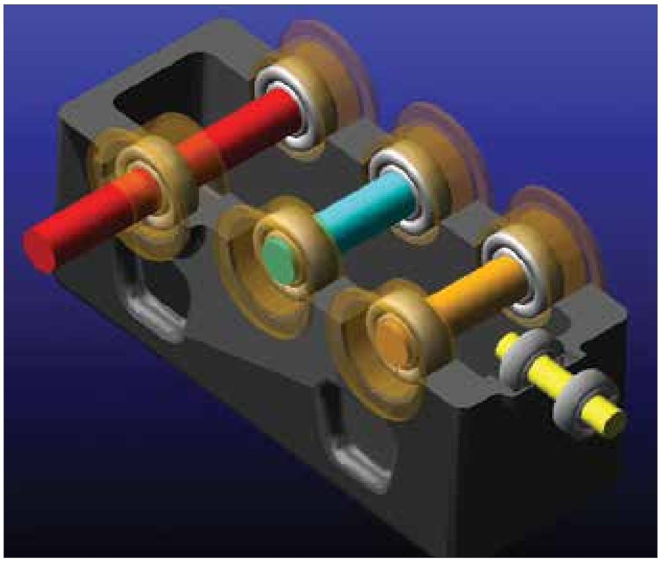
## Step 1. Launch Adams and Import start file

To get started: import the initial model:

a. Launch **Adams/View**

b. Select **Existing Model**

c. Browse for **Withbacklash.cmd** and hit **OK**



## Step 2. Add the gears pairs

a. Under the **Machinery** ribbon, select **Create gear pair.**

b. Select **Helical** as **Gear Type,** and click **Next**.

c. Select **3D Contact** as **Method**, and click **Next**.

d. Set up the **Geometry** page as shown in Figure. Center location of Gear 2 is (0.0, 194.55609294, 0,0).

e. Use the default settings of **Material**, and click **Next**.

f. For **GEAR1**, select **Fixed**. Right click the content of **Body** and select **Body->Pick**

g. Click **PART_6** in the working area, and click **Gear2**.

h. For **GEAR2**, select **Fixed**. Right click the content of **Body** and select **Body->Pick**

i. Click **PART_5** in the working area, and click **Next**.

j. Click Finish.

k. Repeat the above steps to create the other two pairs of gears.

| | Gear type | Method | Connection | | | |
|---|---|---|---|---|---|---|
| | | | Gear 1 | | Gear 2 | |
| | | | Type | Body | Type | Body |
| Gear pair 1 | Helical | 3D Contact | Fixed | PART_6 | Fixed | PART_5 |
| Gear pair 2 | Helical | Simplified | Fixed | PART_5 | Fixed | PART_3 |
| Gear pair 3 | Bevel | Simplified | Fixed | PART_3 | Fixed | PART_2 |

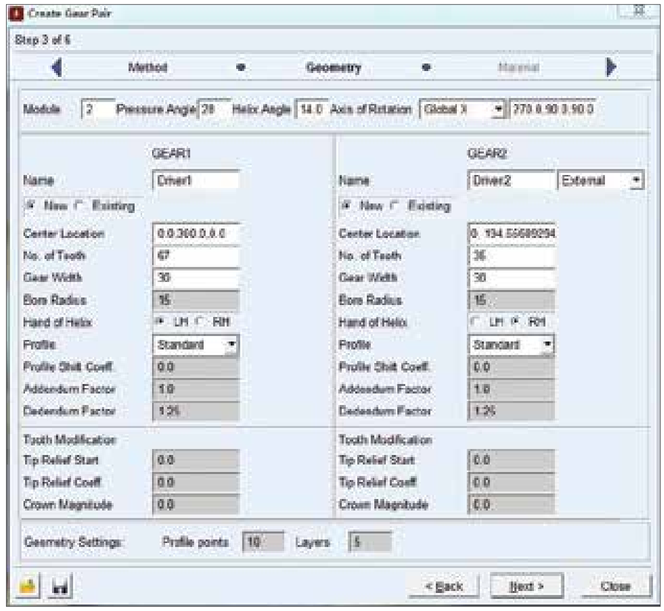



Figure: Geometry Setting of Gear Pair 1

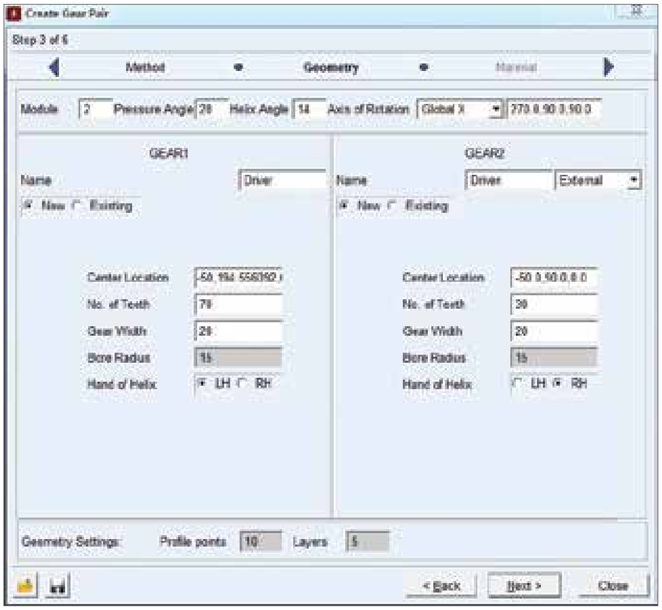



Figure: Geometry Setting of Gear Pair 2

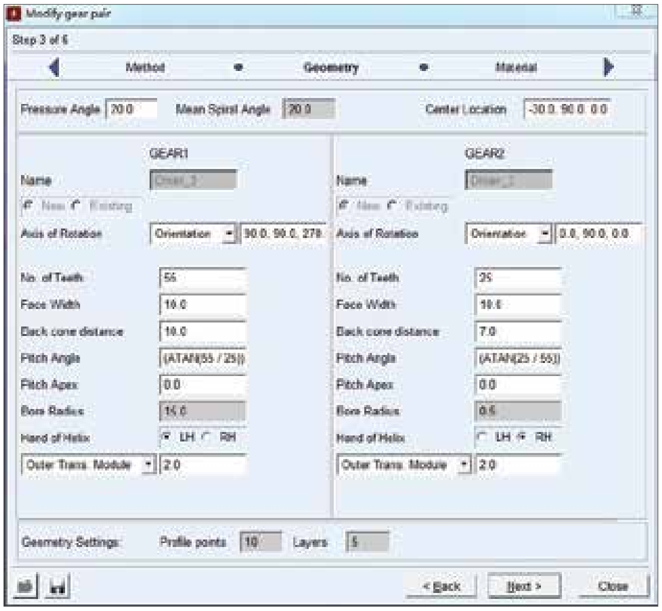



Figure: Geometry Setting of Gear Pair 3

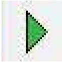
## Step 3. Run the simulation

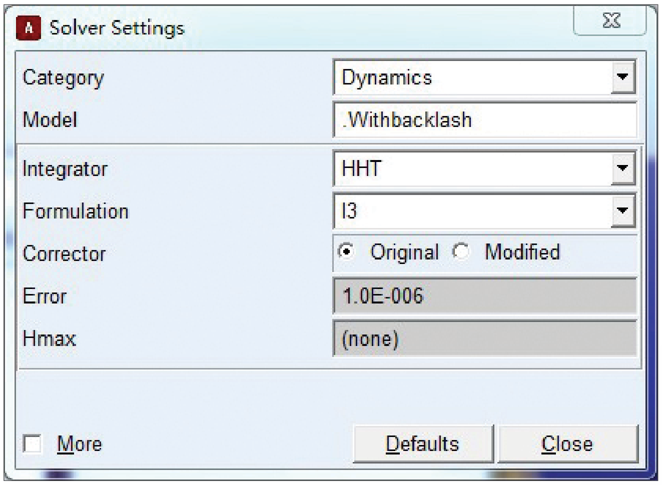
a. Click on **Run an Interactive Simulation.**

b. Click **Simulation Settings…** at the bottom of **Simulation Control panel.**

c. Select **Dynamics** as **Category**, and select **HHT** as **Integrator**.

d. Run another simulation with **0.2 seconds** and **200 steps**






## Step 4. Connect a gear to a dummy part

a. Right-click **Driver_1_Driven_1**, and select **Modify**.

b. Click **Next** to the **Connection** page.

c. Under the **GEAR1**, replace content of the **Body** with **Gshaft** part.

## Step 5. Convert PART_6 to a flexible body.

a. Right-click **PART_6** in the working area.

b. Select **Part: PART_6 -> Make Flexible**

c. Click on **Create New.**

d. Enter **8** in **Number of Modes.**

e. Check **Stress Analysis** and **Advanced Setting.**

f. Select **Size** in **Element Specification**.

g. Enter **3.0mm** for **Element Size** and **2.0mm** for **Minimum Size.**

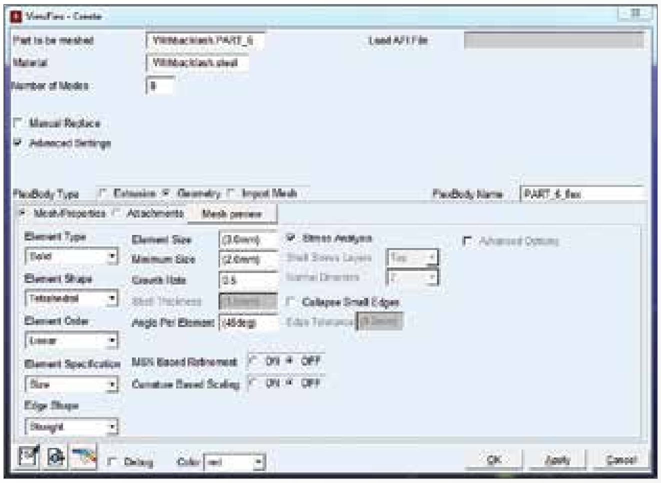
h. Click **Attachments**, and then click **Find Attachments.**

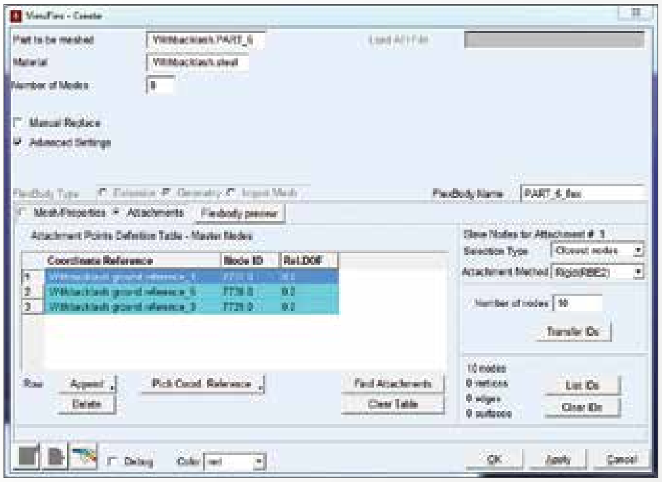
i. Select the **tab 1.**

j. Select **Closest nodes** in **Selection Type** and enter **10** for **Number of nodes.**

k. Apply the same settings for **tab 2** and **tab 3**.

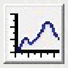
l. Click **OK**



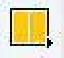


## Step 6. Show results in Adams/Postprocessor

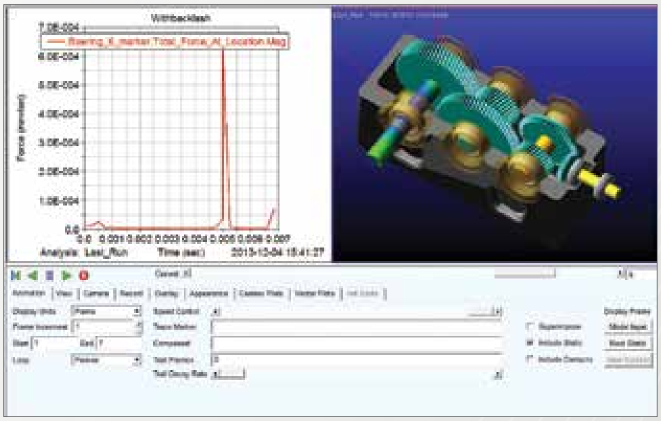
a. Run the simulation again.



b. To start Adams/Postprocessor, Click Plotting in the **Simulation Control** panel.



c. Arrange the animation and plots in the same window by click **Page Layout**



# Example 32: Windshield Wiper Mechanism

## Software Version

Adams 2013.2

## Problem Description

Build the windshield wiper mechanism including electric motor and worm gear. Find the velocities and accelerations of the wipers as they travel through the wiping cycle.

## Step 1. Start Adams/View.

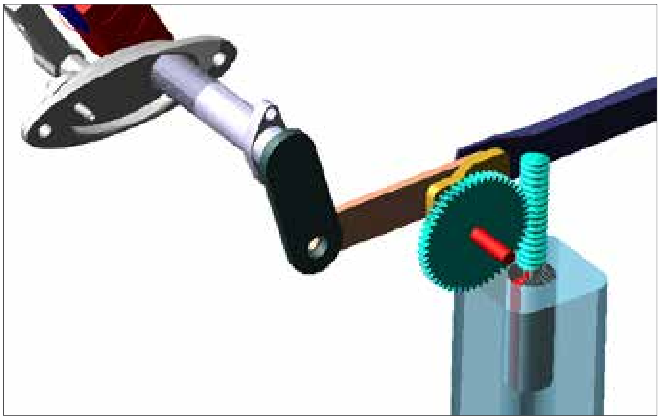
a. Start Adams/View 2013.2.

b. Select Existing Model and browse for Windshield_Wiper_start.cmd in the exercise start folder.

c. Click OK.

## Step 2. Observe the model

a. In our completed model, the system is driving by an electric motor. A worm gear pair transmits the motor torque to the wipers through two four bar linkage systems sharing one common link (mot_link). The spur gear is fixed on the red motor shaft which is rigidly linked to mot_link. A worm gear fixed at the rotor of the motor drives the spur gear. There is a green sphere called Dummy_shaft which is used to locate the position of the motor.



b. We will build the electric motor and worm gear in the following steps.

## Step 3. Create the Motor

a. From the Machinery Ribbon, select Create Motor.

b. Select Analytical in Method, then click Next.

c. Select DC in Motor Type, then click Next.

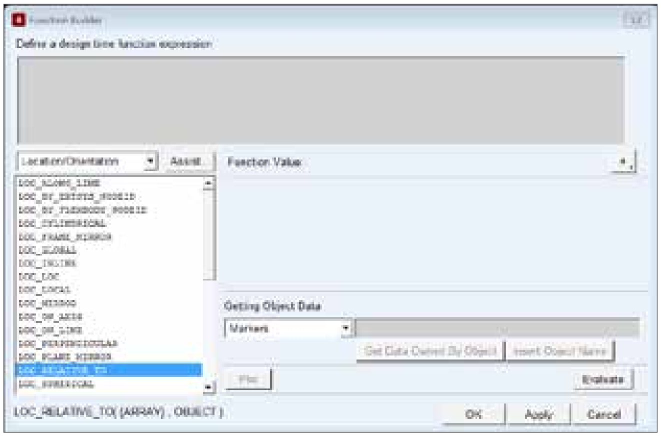
d. To set the motor location, right click in the blank field and select Parameterize->Expression Builder. From the drop down menu on the left, choose Location/Orientation. Double click LOC_RELATIVE_TO.

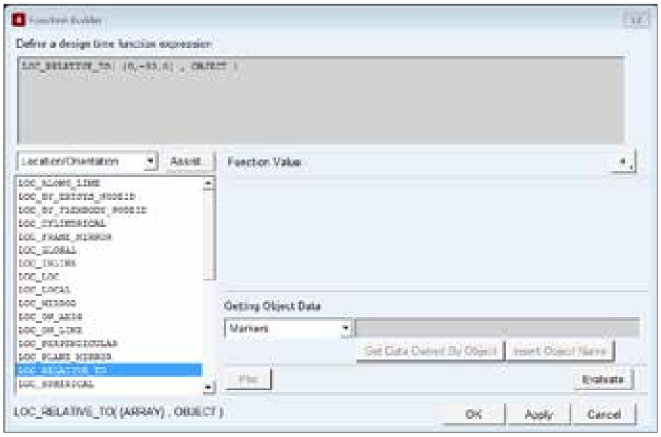
e. Change the Array to 0, -30, 0.

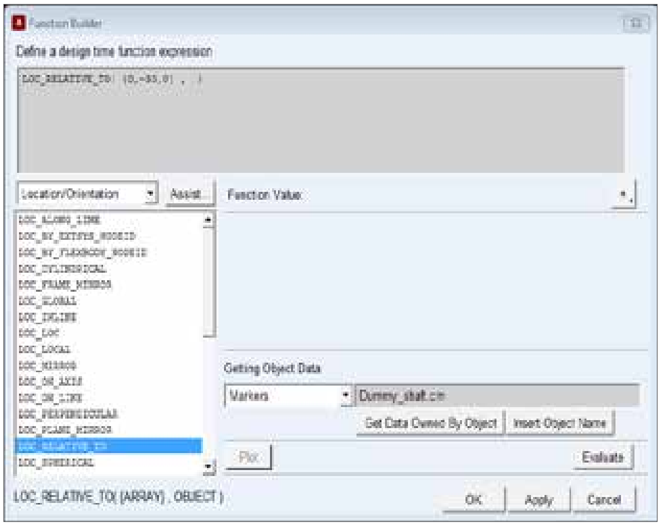
f. Delete OBJECT and leave the cursor as where it is.

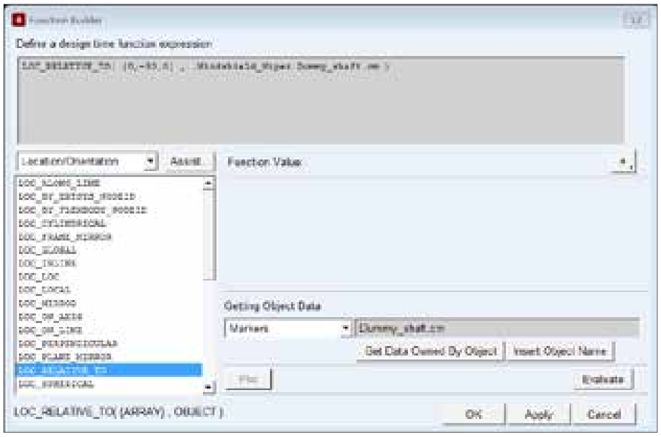
g. Right click in the field of Getting Object Data and click Browse. Select Dummy_shaft.cm.

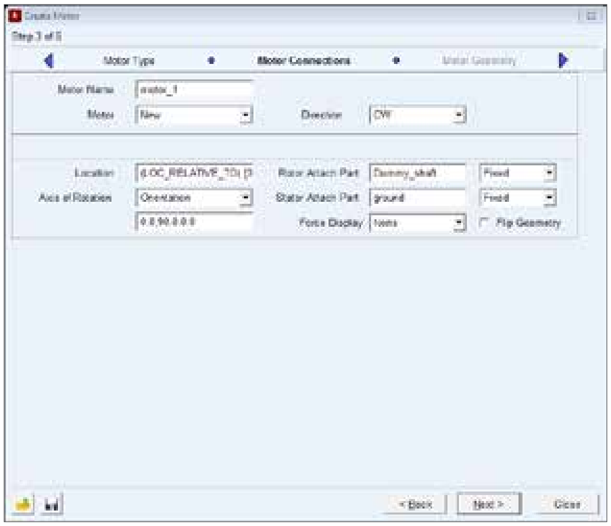
h. At the place where “OBJECT” was, click “Insert Object Name”. Then click OK.



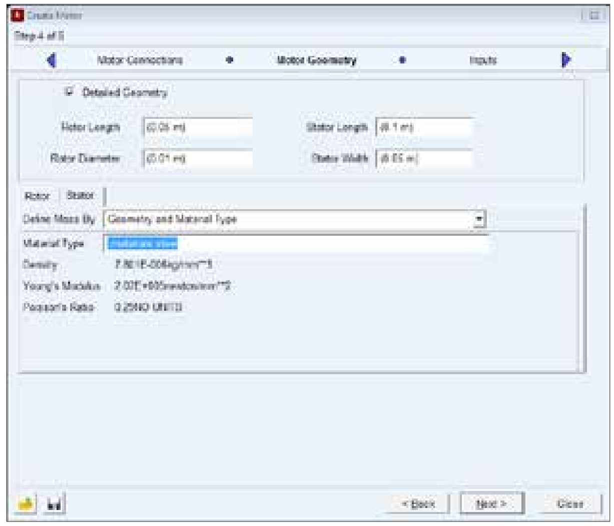



i. Choose CW as Direction. Right click next to Rotor Attach Part and go to Body->Guesses->Dummy_shaft. Select Fixed in the drop down menu on the right. Similarly, fix stator to the ground.

j. Choose Orientation as Axis of Rotation and enter 0, 90, 0. Click Next.

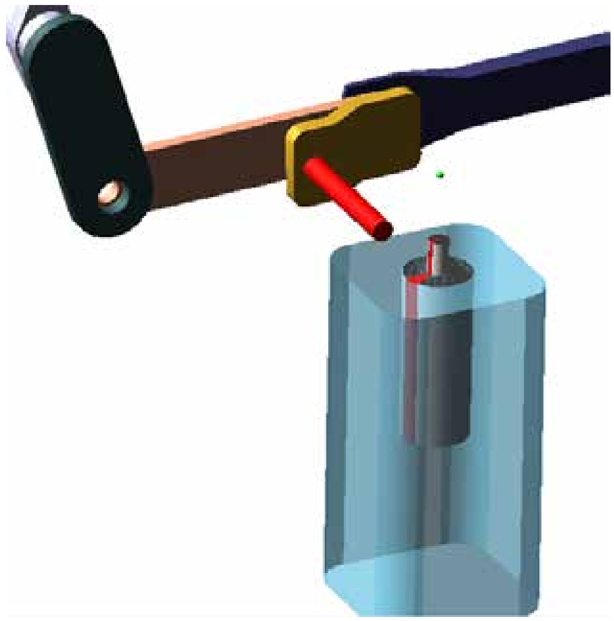
k. Check Detailed Geometry. Set the Motor Geometry as shown in the figure below. Choose Geometry and Material Type to define mass for both Rotor and Stator.




l. Set the Input as shown in the table below, choose Shunt and then click next.

| | |
|---|---|
| No. of Conductors | 16 |
| Flux Per Pole | 0.1 |
| Source Voltage | 24 |
| No. of Paths | 20 |
| Armature Resistance | 0.35 |
| No. of Poles | 12 |

m. Choose Scale Factor for Multiply and set scale factor as 1, then click Finish. Ignore the warning.



## Step 4. Create a Worm Gear Pair

a. From Machinery Ribbon select Create Gear Pair.

b. In Gear Type, select Worm, then click Next.

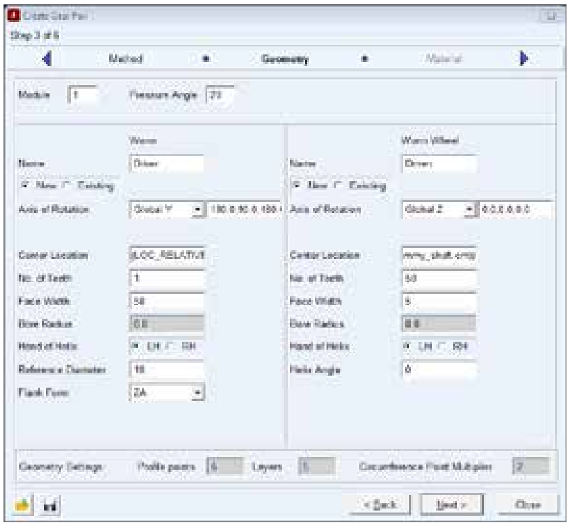
c. Set Method as Simplified, then click Next.

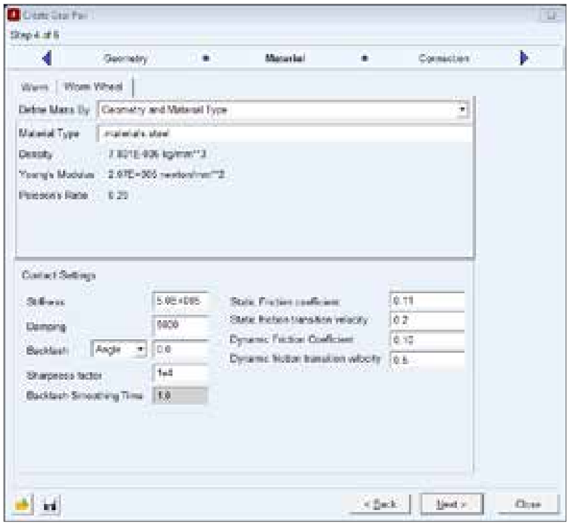
d. Enter **1** in **Module**.

e. Set the Geometry as shown in the table. To set the Center of Location of Worm, right click in the blank field and select Reference_Frame and then select Browse. In the Database Navigator, choose Dummy_shaft.cm. To set the Center of Location of Worm Wheel, use the method illustrated in Step 11 to locate it (30,0,0) relative to Dummy_shaft.cm. Set 0 for Helix Angle in this step as the actual value will be calculated by Adams automatically and it will be changed later. Leave the rest as default and then click Next.

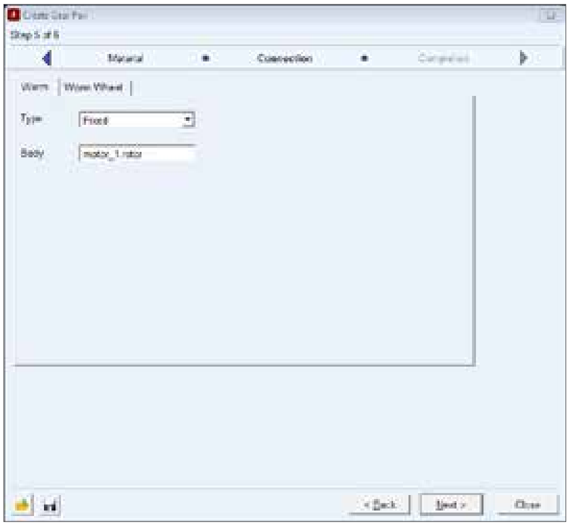
| | Worm | Worm Wheel |
|---|---|---|
| Axis of Rotation | Global Y | Global Z |
| No of Teeth | 1 | 50 |
| Face Width | 50 | 5 |
| Reference Diameter | 10 | |
| Flank Form | ZA | |

f. Choose Geometry and Material Type to define mass for both worm and worm wheel. Then click Next.

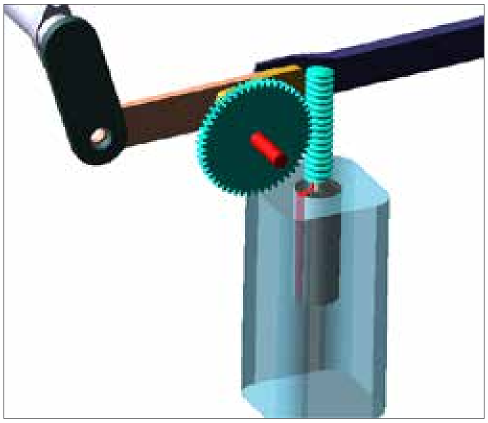




g. Fix the worm gear to the rotor of motor and worm wheel to “Shaft”. Click Next then Finish.



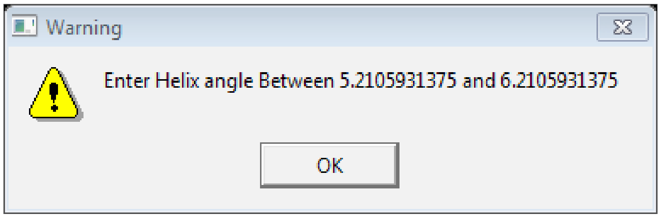
h. It may take some time for Adams to create the gear pair. The final model should look like the configuration below.



## Step 5. Modify Helix Angle

a. From the model tree, double click Driver_1_Driven_1.

b. Navigate to Geometry page. A warning message is shown.




c. Click Ok, then set Helix Angle as 5.7. Click Next and leave everything unchanged. Click Finish on the last page.

## Step 6. Simulate the Model

a. From Simulation Ribbon, select Run an Interactive Simulation and run a simulation of 3 seconds with 300 steps. The mechanism should run smoothly. Make sure that MOTION_1 is deactivated.

## Step 7. Postprocessing the Result

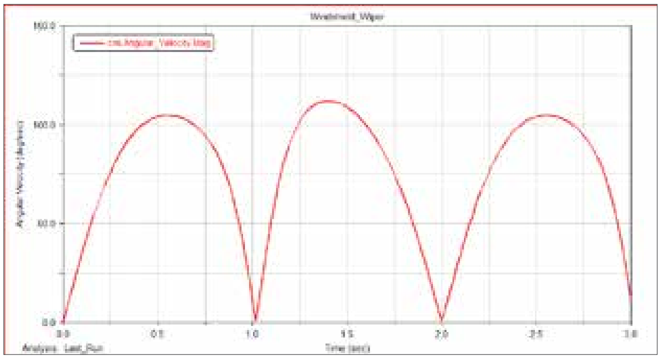
a. Press F8 to enter postprocessing window.

b. Choose Objects as Source. Click body in Filter.

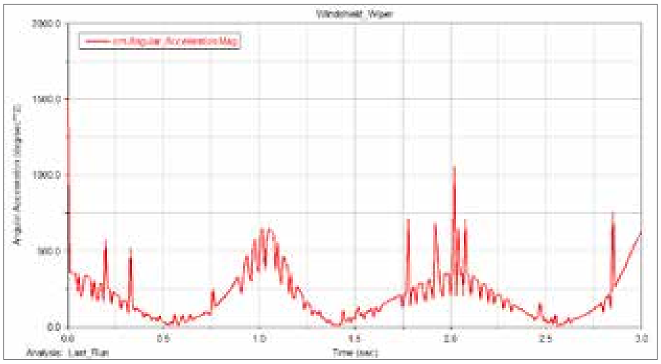
c. Browse for Wiper_left in Object and double click it to expand. Highlight cm.

d. Pick Angular_velocity and then select Mag as Component.

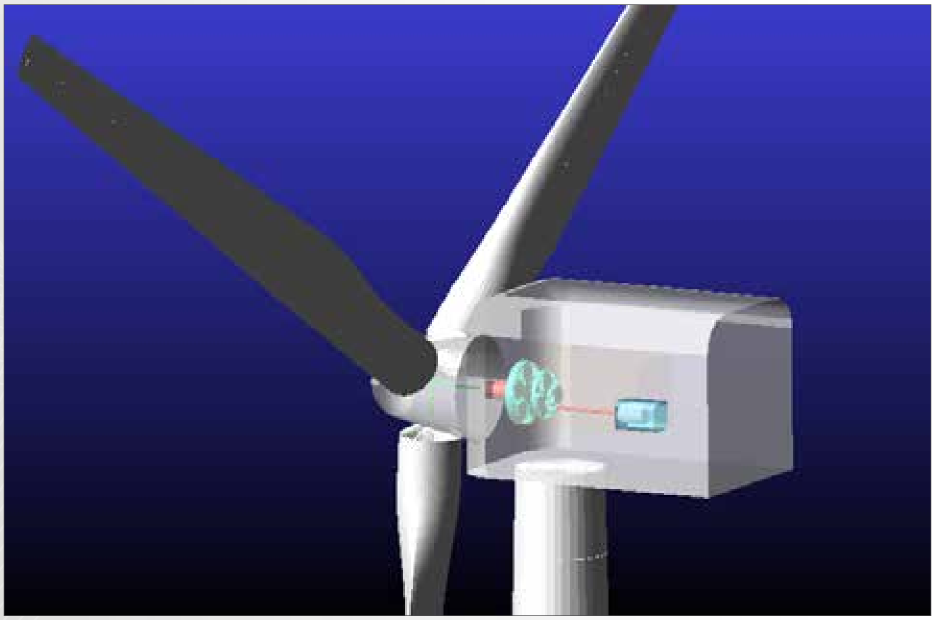
e. Click Add Curves.




f. Similarly, in a new page, select Angular_Acceleration, Mag and click Add Curves.



# Example 33: Windturbine



## Software Version

Adams 2013.2

## Problem Description

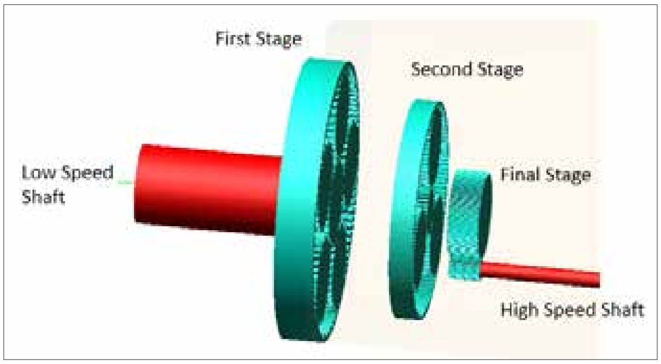
Use Adams/Machinery to model the two-stage planetary gear train in wind turbine's gear box. The output shaft (high speed shaft) is connected to a generator, which is modelled using motor module in Adams/Machinery. The blades and the tower are modelled as flexible bodies. Torque and power generated by the generator can be studied in postprocessing.

## Step 1. Topology of the Gear Train Explained

a. The topology of the gear set is explained first. A planetary gear set requires two inputs to determine one output. In the first stage planetary gear set, the gear carrier (no graphical representation) is connected to the low speed shaft which is rotating at the same angular velocity as the hub. This is our first velocity input. The second velocity input of planetary gear set is usually a zero input (fixed). In stage one, the ring gear is fixed in the gear box which provides another velocity input. The output of this planetary gear set is the velocity of the sun gear.

b. At stage two, the carrier is rigidly connected to the sun gear of stage one (also no graphical representation). The ring gear is fixed to gear box. These two comprises the required input. The predictable output is the rotation of the sun gear.

c. At the final stage, there is a spur gear pair. Low speed gear is connected to the sun gear of stage two. High speed gear drives the high speed shaft which is powering the generator.




## Step 2. Start Adams/View

a. Start Adams/View 2013.2.

b. Choose **Existing Model** and select **windturbine_ start.cmd** under exercise directory.

c. Click **OK**.

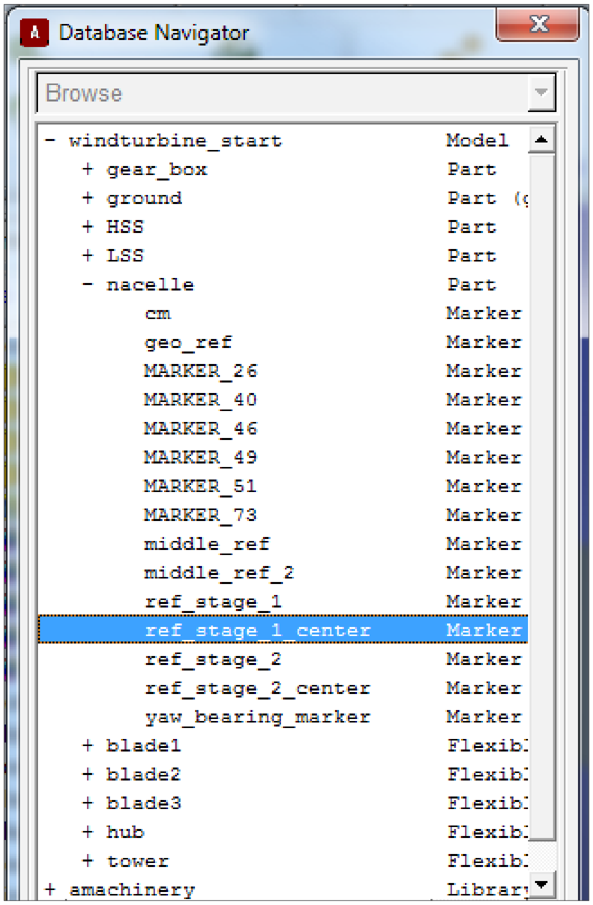
## Step 3. Build First Stage Planetary Gear Set: Geometry

a. From **Machinery** ribbon, select **Gear: Planetary Gear**.

b. Click **Next**.

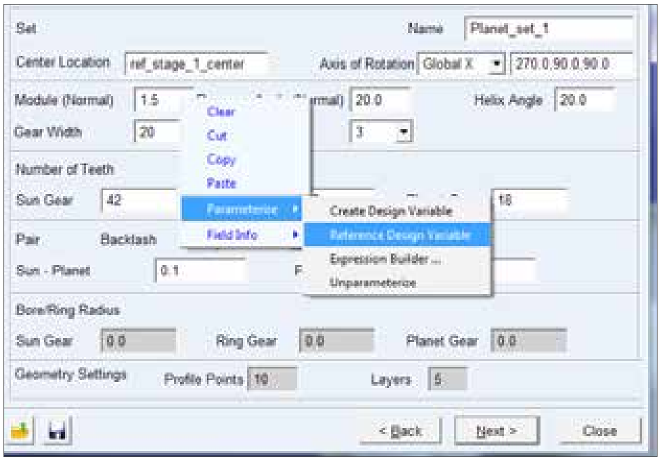
c. Select **Simplified Method**.

d. Right click in the **Center Location** field, direct to **Reference_Frame->Browse**. In **Database Navigator**, find **ref_stage_1_center** and double click.




e. Select **Global X** as **Axis of Rotation**.

f. Right click at Module and go to **Reference Design Variable**.



g. Select **module_stage_1** from the list. Similarly, select **width_stage_1** for **gear width**.

h. Select **5** planet gears and **0** Helix Angle.


i. The number of teeth for sun gear, ring gear and planet gears are **35**, **105**, and **35**. Leave the rest of the field as default. Click **Next**.


## Step 4. Build First Stage Planetary Gear Set: Material

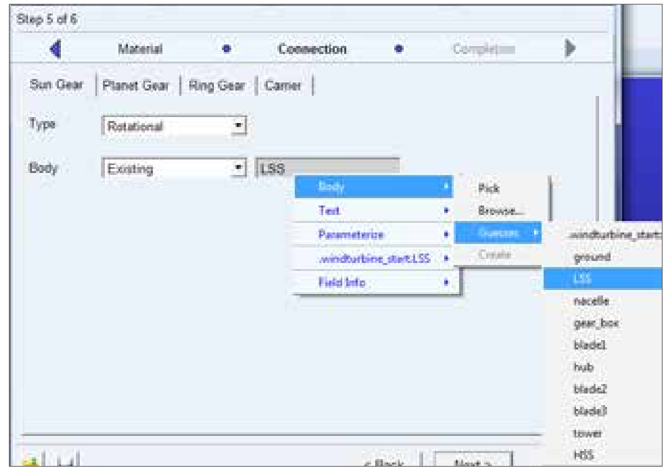
a. Select **Geometry and Material Type** for all the parts.

b. Leave the contact and friction settings as default and click **Next**.

## Step 5. Build First Stage Planetary Gear Set: Connection

a. In this step, connections are built according to the topology described in Step 1.

b. For **sun gear**, select **Rotational** and right click in the field to select **LSS**.

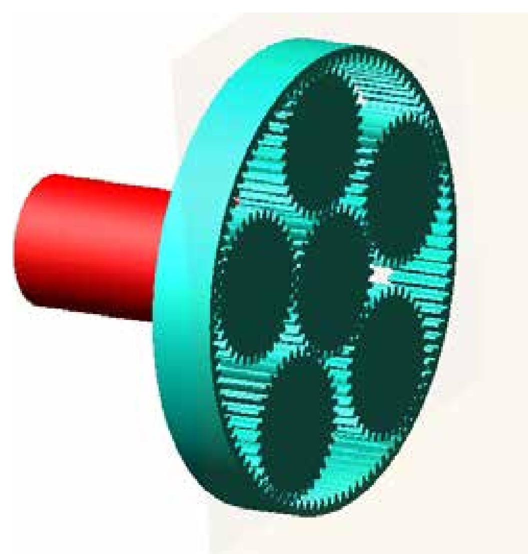



c. For **planet gear**, select **Rotational**.

d. For **ring gear**, select **Fixed** and pick **gear_box** as well.

e. For **carrier**, select **Fixed** and pick **LSS** as the existing body.


f. Click **Next** and then **Finish**. It may take while for Adams to build the gear set depending on the configuration of your machine. Be patient. When the gears have been built, the dialog box will close automatically and the gears will show up like this.



## Step 6. Build Second Stage Planetary Gear Set: Geometry and Material

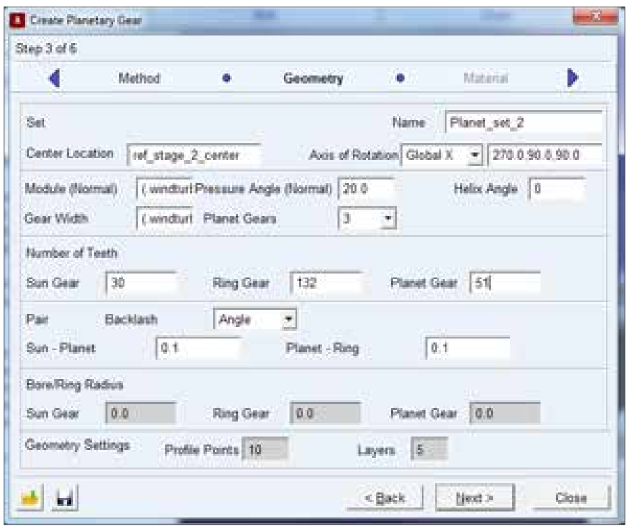
a. Follow the similar procedures as we build the first stage gear sets and navigate to **Geometry** page.

b. Follow the same steps to define center location, module and gear width. This time, change all references from “_stage_1” to “_stage_2”. Select appropriate design variables for module and gear width.

c. Pick **Global X** as axis of rotation.

d. Enter **30**, **132** and **51** for number of teeth of sun gear, ring gear and planet gear.

e. Set number of planet gears to **three** and Helix Angel to **0**. Leave the rest settings as default and click **Next**.




f. Pick **Geometry and Material Type** for mass definition for all parts. Click **Next**.

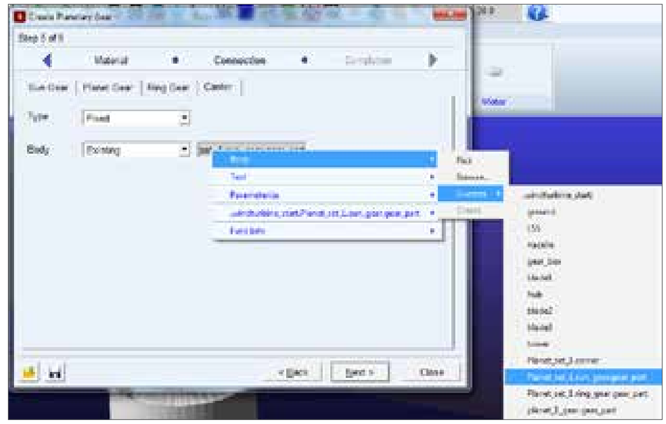
## Step 7. Build Second Stage Planetary Gear Set: Connection

a. Make the sun gear rotate against gear_box, following similar steps in Step 5b.

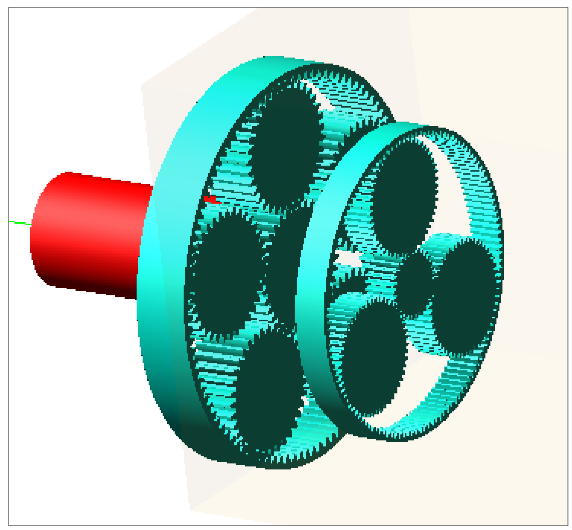
b. Select **Rotational** for planet gear.

**c.** **Fix** ring gear to gear_box.

**d.** **Fix** the carrier to the sun gear of first stage gear set.



e. Click **Next** and then **Finish**. Wait patiently until the gear set is created.



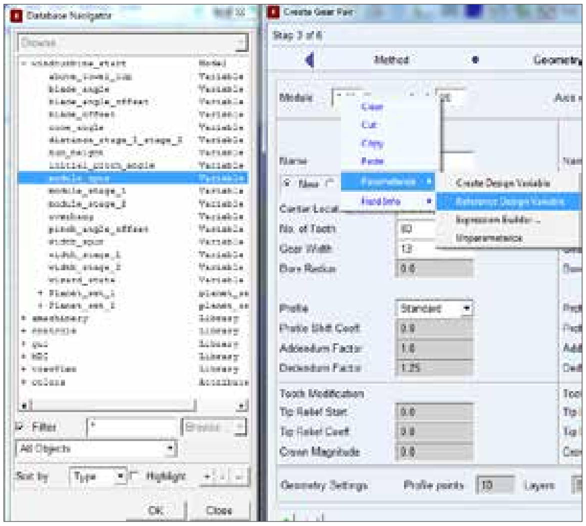
## Step 8. Build Final Stage Spur Gear Set: Geometry

a. From **Machinery** ribbon, select **Gear: Create gear pair**.

b. Select **Spur** as Gear Type. Click **Next**.

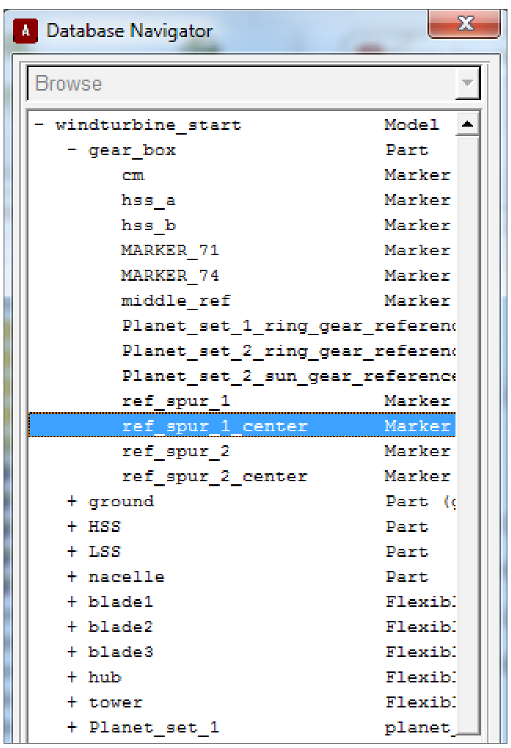
c. Select **Simplified** as Method. Click **Next**.

d. Right click next to **Module** and select **Parameterize->Reference Design Variable**. Pick **module_spur** from the list



e. Pick **Global X** as Axis of Rotation.

f. Right click at **Center Location** under Gear 1 column and select **Reference_Frame->Browse**. Select **ref_spur_1_center** from the Database Navigator.



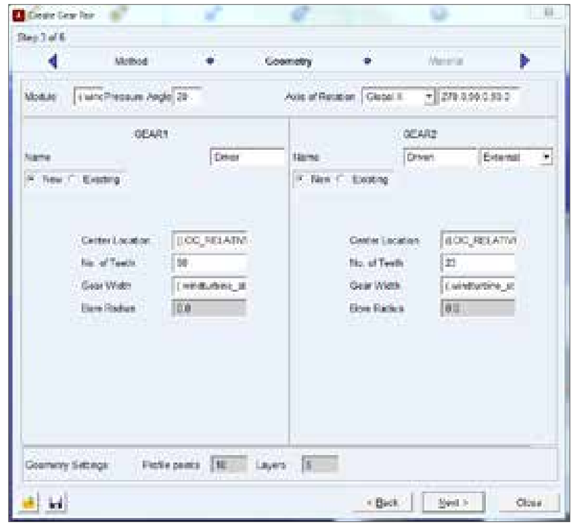

g. Enter **90** for No. of Teeth.

h. Right click at **Gear Width** and go to **Parameterize->Reference Design Variables**. Pick **width_spur**.

i. Move to Gear 2 column. Replace the reference marker in Step 8h to **ref_spur_2_center**.

j. Enter **23** for No. of Teeth and use the same **Gear Width**.

k. Leave the rest as default and click **Next**.



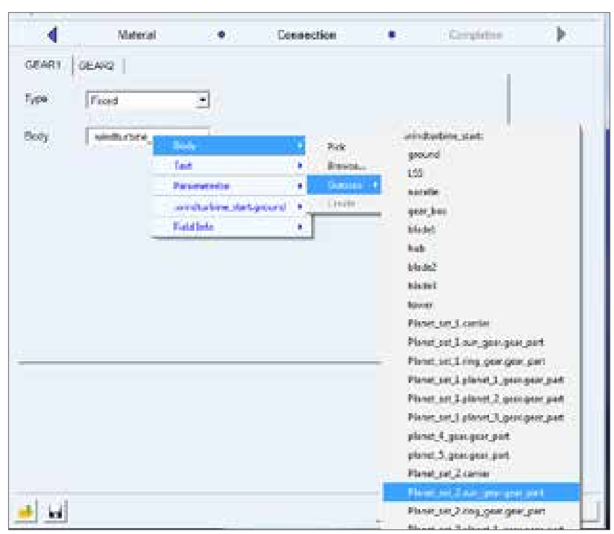
## Step 9. Build Final Stage Spur Gear Set: Material

a. Define mass of gear 1 and gear 2 by **Geometry and Material Type**.

b. Click **Next**.

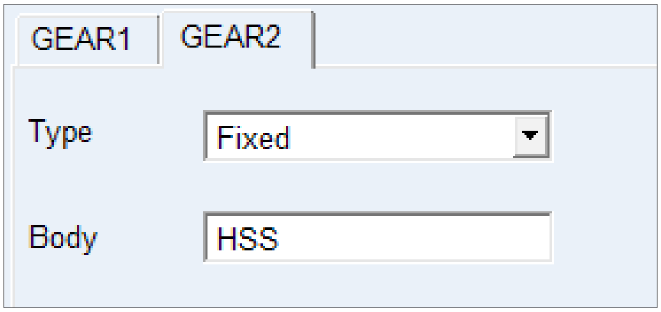
## Step 10. Build Final Stage Spur Gear Set: Connection

a. Apparently, the driver gear should be fixed to the sun gear of second stage planetary gear set while the driven gear should be fixed to high speed shaft.



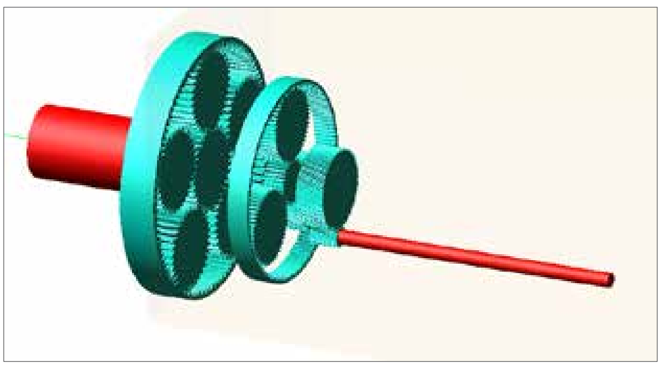
b. Right click in Body and pick the sun gear.

c. Click Gear 2 tab and right click in Body and browse for **HSS**.




d. Click **Next** and then **Finish**. Wait for Adams to build the gear pair.

e. The completed gear train should look like the one in the figure.



## Step 11. Build the Generator

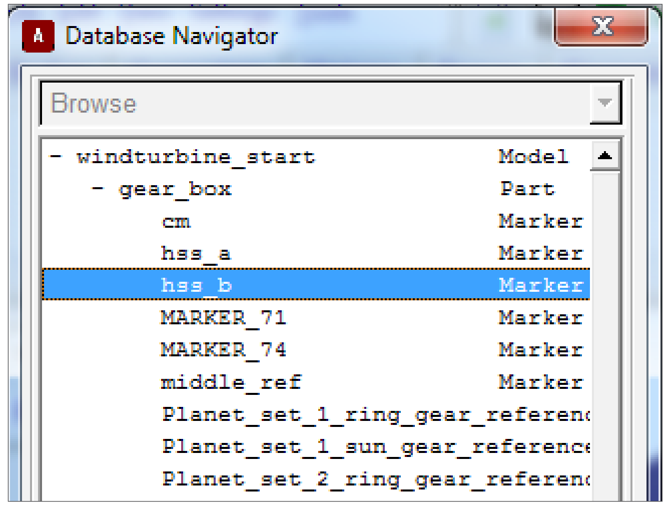
a. There is no built-in generator module in Adams/Machinery but with a few modifications, we can build a generator using the motor module.

b. From **Machinery** ribbon, select **Motor: Create Motor**.

c. Under Method, choose **Curve Based**.

d. Click **Next** and navigate to Motor Connections.

e. Choose **CW** (clockwise) as Direction. Right click at Location and then go to **Reference_Frame->Browse**. In Database Navigator, expand **gear_box** and pick **hss_b**.




f. Pick **Global X** as Axis of Rotation.

g. Fix rotor and stator to hss (high speed shaft) and gear_box repectively. Click Next.

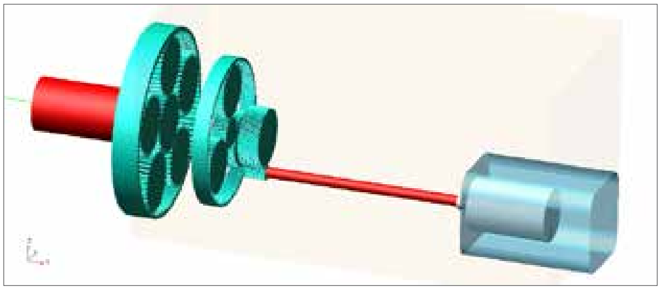
h. Set rotor **length** and **diameter** to **1.5** and **0.5**. Make sure that your current Unit System is **MKS**. Set the stator **length** and **width** to **2.5** and **1.6**.

i. The mass properties of rotor and stator are shown in the following table. Enter the values and click **Next**.

| | Rotor | Stator |
|---|---|---|
| Mass (kg) | 3100 | 50000 |
| Ixx (kg*m^2) | 100 | 400 |
| Iyy (kg*m^2) | 50 | 150 |
| Izz (kg*m^2) | 50 | 150 |

j. For Inputs, select **Enter Spline File**. Right click in the Spline File Name field and browse for **torque_curve.csv** under start folder. Click **Next** and then **Finish**.

k. The complete gear train and generator will look like this.

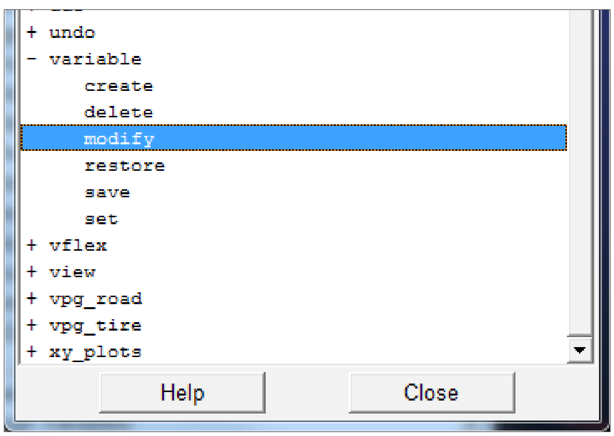


## Step 12. Modify the Generator

a. Now the motor in the model will just work like a motor. Furthermore, the default unit system of angular velocity used by the motor is not necessarily consistent with that of our curve.

b. The first thing we do is cut the power supply to the motor.

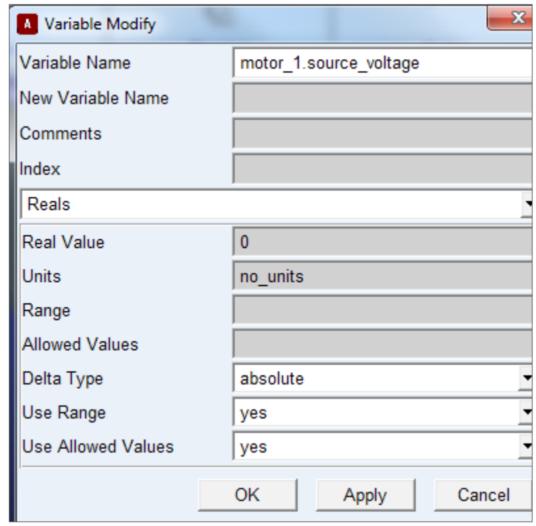
c. From the menu, select **Tools->Command Navigator**.



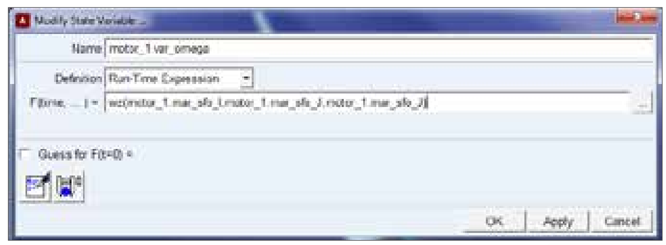

d. Navigate to **variable->modify** and double click.

e. Right click in **Variable Name** and go to **Variable->Browse**. Expand **windturbine_start**, then **motor_1**. Select **source_voltage**.

f. The value is set to 12 by default. Enter **0** in our case. Click OK and close the Command Navigator.

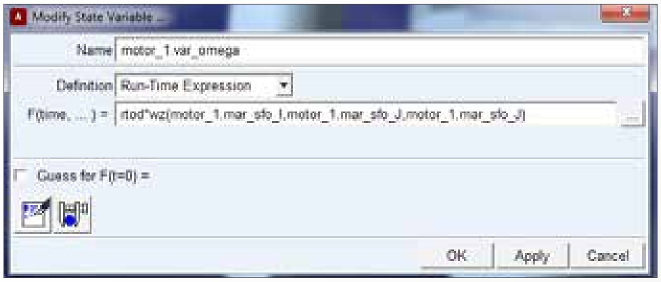


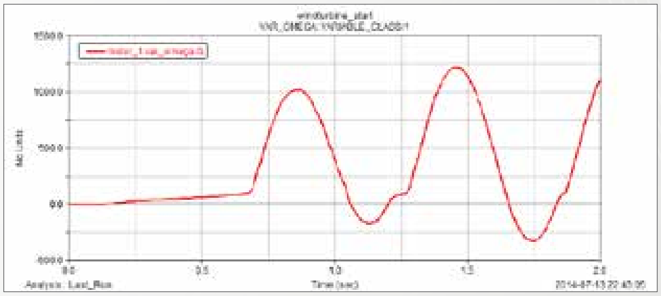

g. From the model tree, expand Motors folder and double click **var_omega** and **var_spline_torque**.

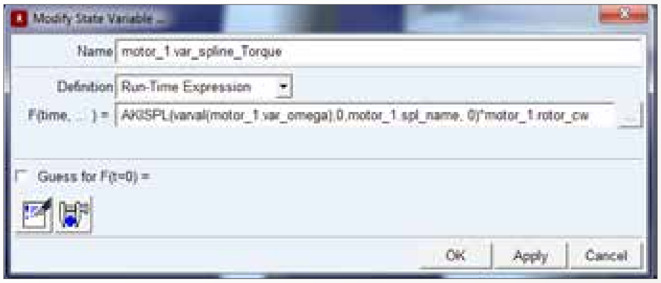




h. Our torque curve builds up the relationship between generator speed and generator torque. Function AKISPL returns the value of the torque at certain speed based on spline fitting. In our curve, the unit of angular velocity is deg/sec, while in the default case, the value var_omega is in rad/sec. 60/(2*pi) converts rad/sec to rpm. So in our case, we rewrite the function of var_omega to rtod*wz(motor_1.mar_sfo_I,motor_1.mar_sfo_J,motor_1.mar_sfo_J). “rtod” is a function that converts the parameter from radian to degree. We also delete “60/(2*pi)” modified in var_spline_Torque function.



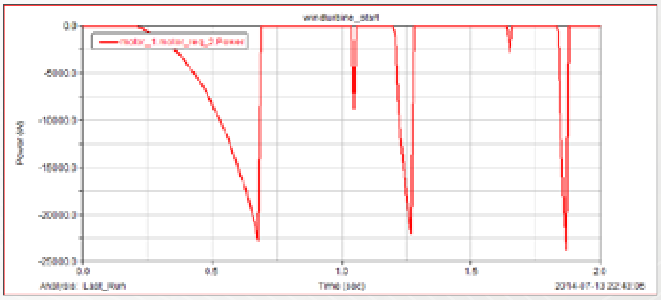




## Step 13. Run Dynamic Simulation

a. Concentrated forces have already been added to the tip of the blades.

b. Run an interactive dynamic simulation with 2 seconds and 200 steps.

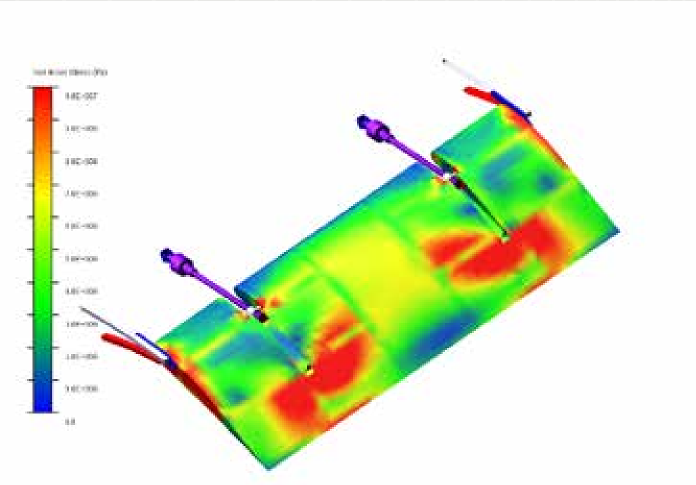
c. We can find that the power output of the generator exhibits a spike trend. This is because then the motor speed exceeds certain speed, we suppress the power output of the generator to protect the generator and grid from overloading. Normally, there is also a brake system to control the speed of the shaft which is not modelled in our example.



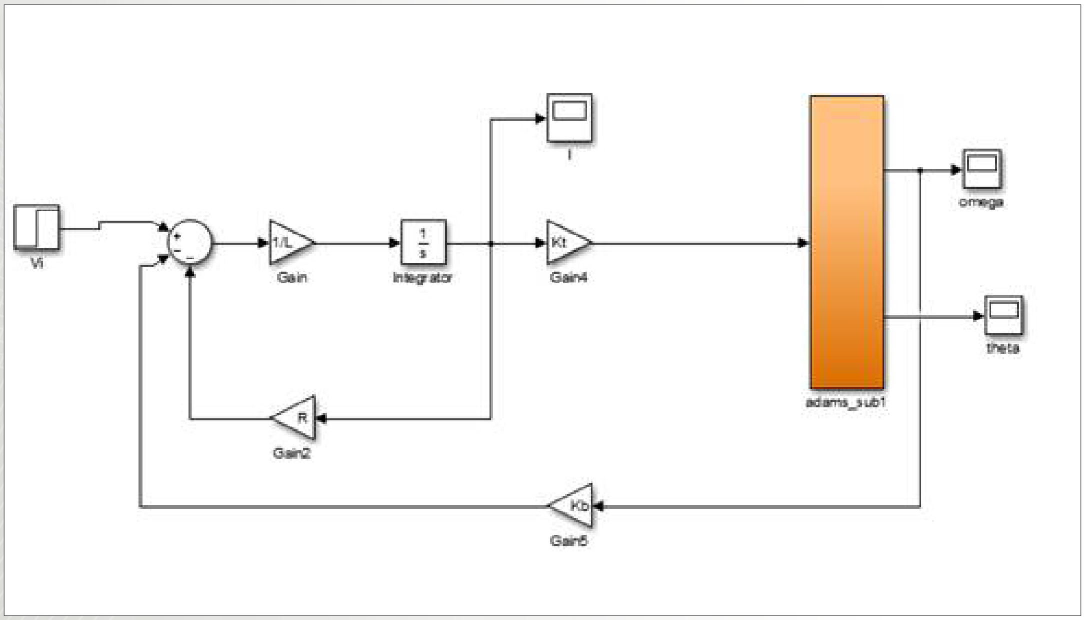
# Section V: Adams/MATLAB Co-simulation

*This section introduces you the benefits of Adams/MATLAB co-simulation. There are many systems require control systems to function properly. MATLAB is popular among controls engineers while the insight of the dynamic systems Adams will provide is unrivaled. The capability to run co-simulation between these two enables engineers to develop better products. In this section level, you will learn:*

- *How to create inputs and outputs in Adams*
- *How to export Adams model for co-simulation*
- *How to setup MATALB/Simulink for co-simulation*
- *How to postprocess the co-simulation result*



# Example 34: DC Motor




## Software Version

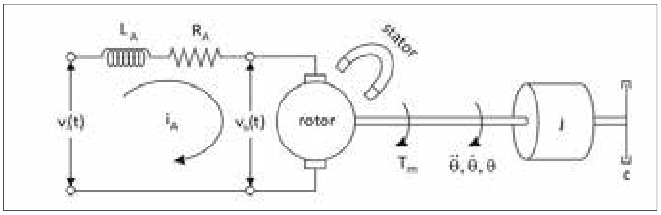
Adams 2013.2

## Problem Description

- Build a simple DC motor connected to an output shaft in both MATLAB/Simulink and Adams/View
- Compare the results of two different simulation methods
- Note: copy all the files in the start file to working directory before you start.
- This example is created in collaboration with Prof. Frank Owen at California Polytechnic State University.

## Step 1: Build the Model in Simulink

a. The schematic of a permanent magnet servo motor is shown below.




b. Apply Kirchoff's Voltage Law to the circuit, where vi is

$$v_i - v_L - v_R - v_b = 0$$

the input voltage, vL and vR are voltage across the inductor and resistor separately. vb is the back EMF proportional to the motor's speed, i.e.,

$$v_b = K_b \cdot \dot{\theta}$$

c. Now, substitute this and write the expression for voltage drop in terms of armature current,

$$v_i - L_A \frac{di_A}{dx} - R_A \cdot i_A - K_b \cdot \dot{\theta} = 0$$

d. The torque output by the motor is proportional to the armature current,

$$T_m = K_t \cdot i_A$$

e. The dynamics of the shaft is.

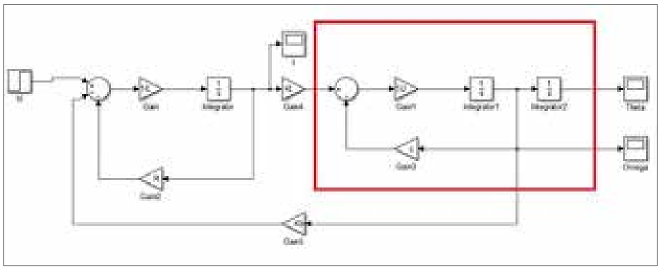
$$T_m - c \cdot \dot{\theta} = J \cdot \ddot{\theta}$$

f. Now we have all the necessary equations to describe the system. Put the higher order derivatives on the left hand side.

$$\frac{di_A}{dx} = \frac{1}{L_A}(v_i - R_A \cdot i_A - K_b \cdot \dot{\theta})$$

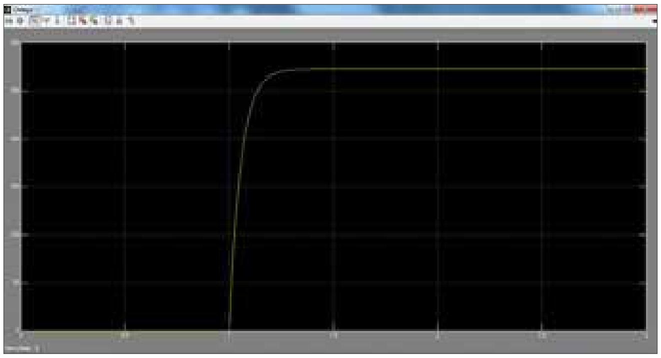
$$\ddot{\theta} = \frac{1}{J}(T_m - c \cdot \dot{\theta})$$

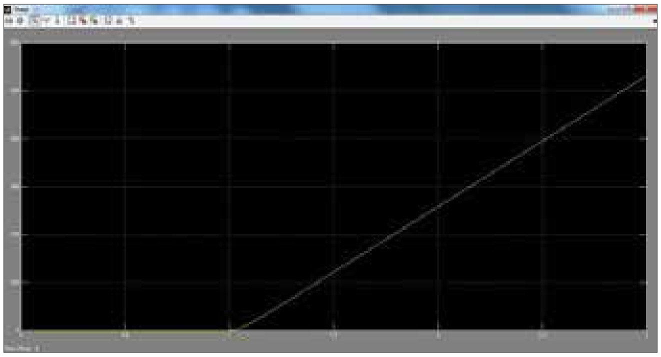
g. Now, build the model in Simulink. Open motor_full. slxThe part highlighted by red rectangle is the mechanical part which will later be replaced by an Adams subsystem.

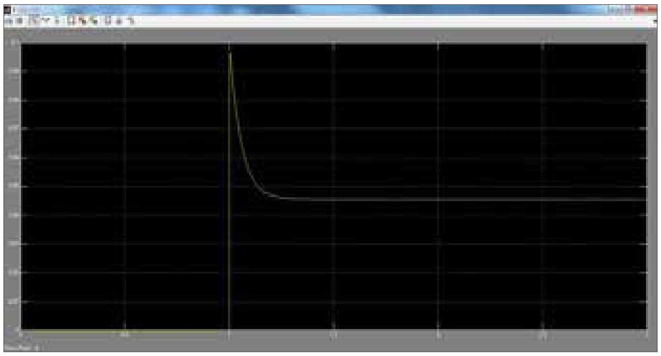


## Step 2: Simulate the Model in Simulink

a. Run the file Motor_initialize.m to set necessary parameters for simulation.







b. Simulate the model for 3 seconds.

c. The scopes should show the following curves.

## Step 3: Create New Model in Adams/View

a. Open Adams/View and select New Model.

b. Set Model Name as motor_shaft.

c. Click OK.

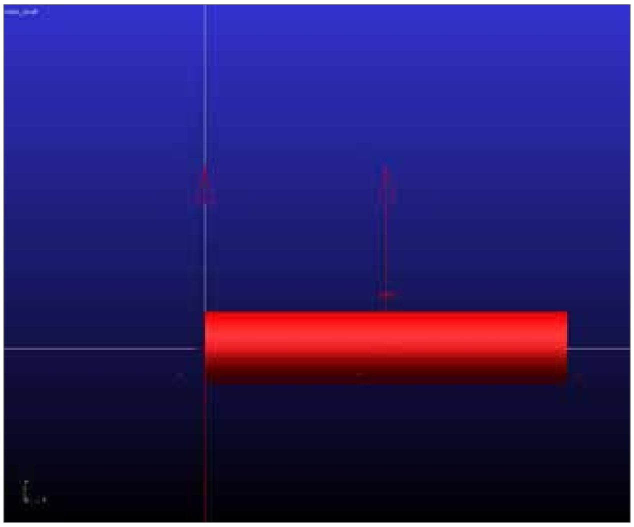
d. Change the model unit from millimeter to meter.

## Step 4: Create the Shaft

a. From the Bodies ribbon, select Rigid Body: Cylinder.

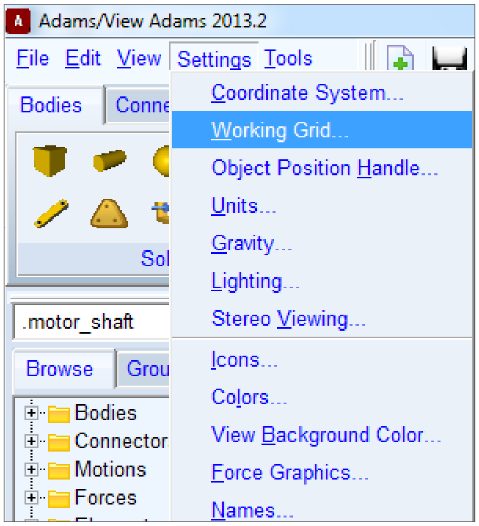
b. Select New Part and check Length and Radius. Set them to 10cm and 1cm respectively.

c. Pick the origin as the first end and click anywhere on the x axis.



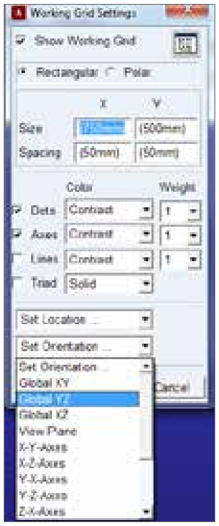
d. The shaft should be created as shown below.

e. Rename the part as shaft.




## Step 5: Add Constraints

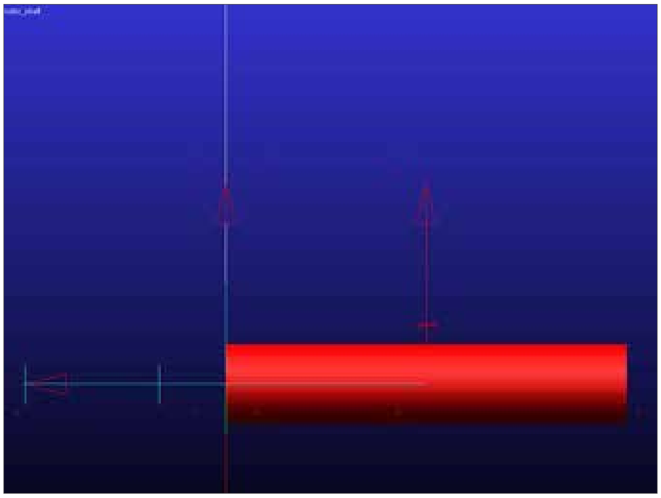
a. From the tool bar, select Settings->Working Grid




b. Change the orientation of the grid to YZ plane. This would help us to utilize the Normal to Grid feature when creating the revolute joint.

c. From Connectors ribbon, select Revolute Joint.

d. Leave the settings as default (2 Bod-1 Loc, Normal to Grid) and pick shaft first and then ground. Pick shaft.



MARKER_1 as the location.

e. The front view (shift + F) should be similar to the figure.

## Step 6: Add Rotational Damping

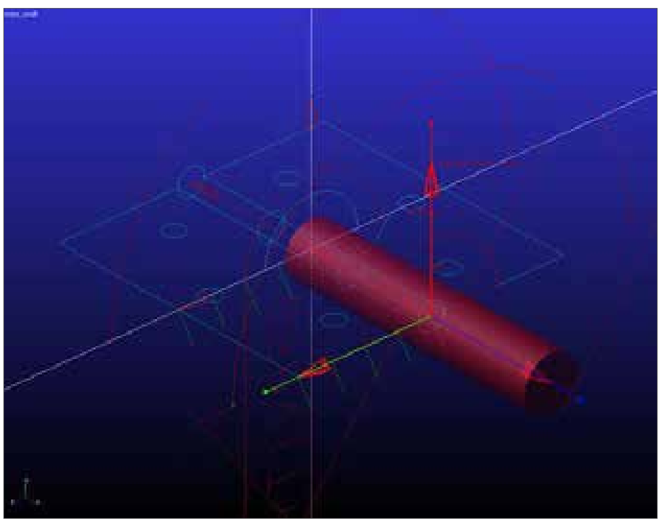
a. From the Forces ribbon, select Rotational Spring-Damper.

b. Set the construction method to 2 Bod-1 Loc and Normal to Grid.
c. Do not check KT as we do not need a spring here.
d. Check CT and set it to 0.0001 (N*m*s/rad).
e. Pick shaft first then ground and choose shaft.cm as the location

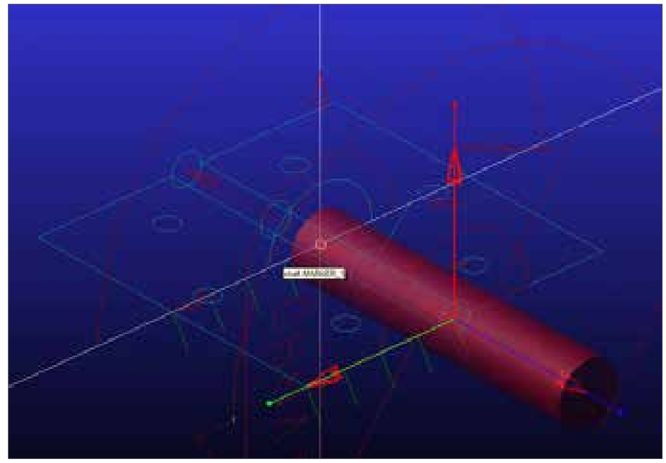
## Step 7: Add Motor Torque

a. From the Forces ribbon, select Toque.
b. Set the Run-time Direction as Space Fixed.
c. Set Construction to Normal to Grid.
d. Pick shaft as the body and pick shaft.MARKER_1 as the action point.

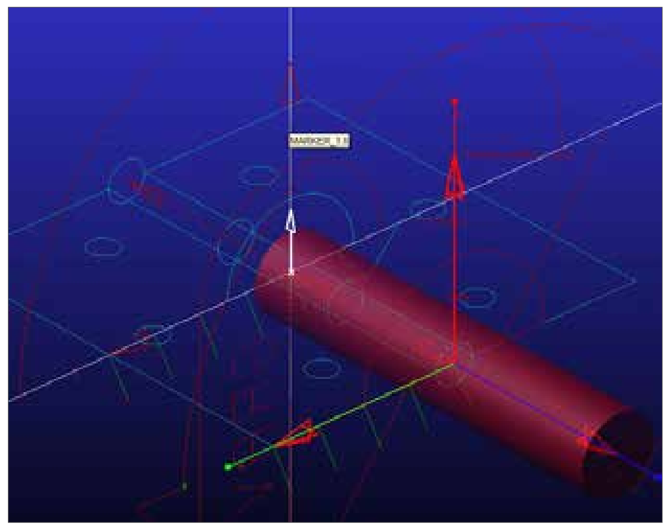
## Step 8: Create Measurement Reference Angle



a. First check the orientation of the shaft.cm. Notice how X, Y, Z axes are oriented. This is important for creating correct measurement.
b. From the Bodies ribbon, select Construction Geometry: Marker.
c. Select Add to Ground and for Orientation, select X-Axis, Y-Axis.
d. Select shaft.Marker_1 as the origin.



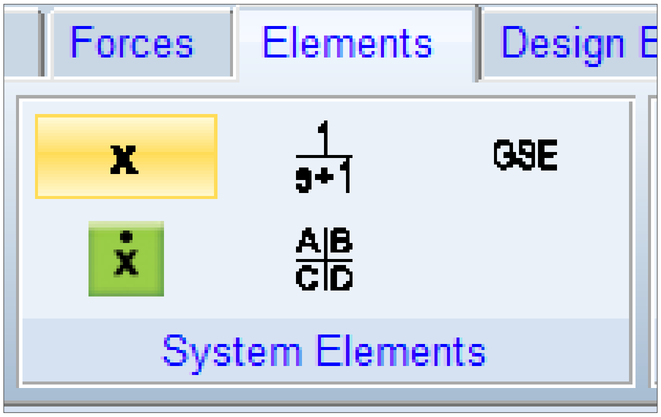
e. Move your mouse around and align it with Marker_1.X.



f. Similarly, select Marker_1.Y as Y-axis.
g. A new marker should be added to ground. In the model tree, expand Bodies->ground and rename the new marker as ang_ref.

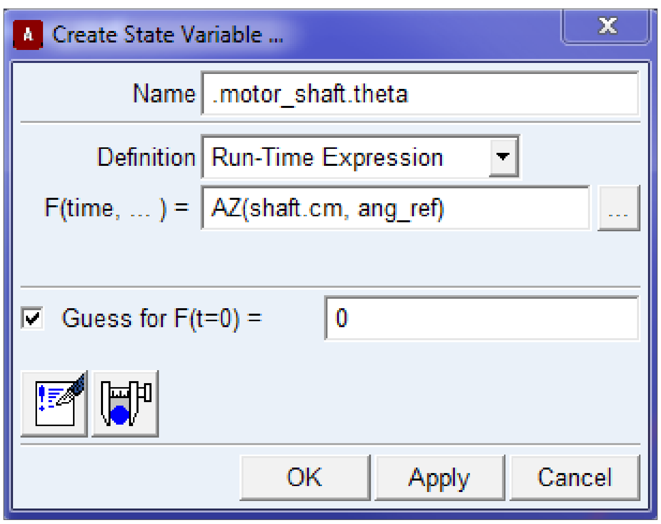
## Step 9: Create State Variables

a. State variables act as the parameters that Simulink and Adams exchange with each other. They work either as input or output to the Adams model. In our model, the input is the motor torque and the output is the angular displacement, angular velocity or any other type of physical quantities that are the result of the motor torque input.
b. From Elements ribbon, select System Elements: Create a State Variable.




c. Set the variable name as motor_torque. Leave the rest settings as default. The value will be passed by Simulink once the simulation starts.

d. Create a new variable named theta. This time, type AZ(shaft.cm, ang_ref) in the function dialog box.

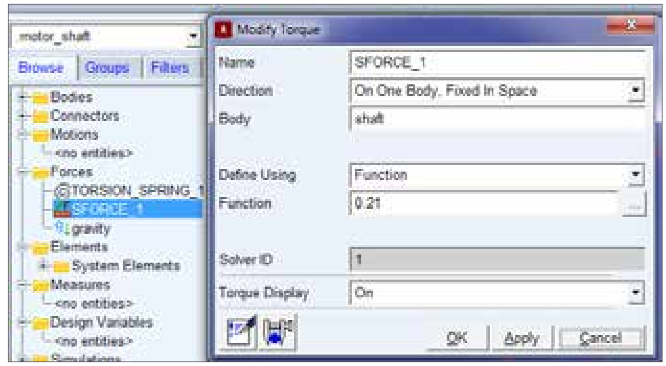



e. Create a new variable named omega. Type WZ(shaft.cm, ang_ref, ang_ref) in the function dialog box. Note that detailed information about functions AZ and WZ can be found in Adams Help Document.

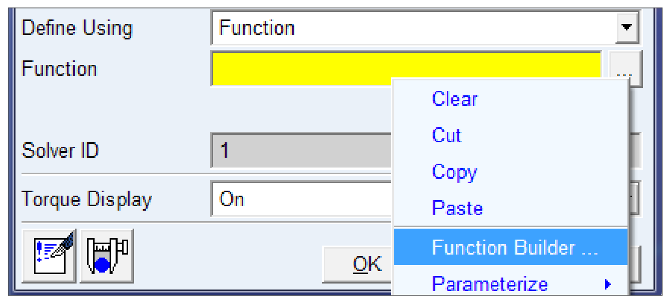
f. The latter two will be the output of the model.

## Step 10: Modify Torque and Damping

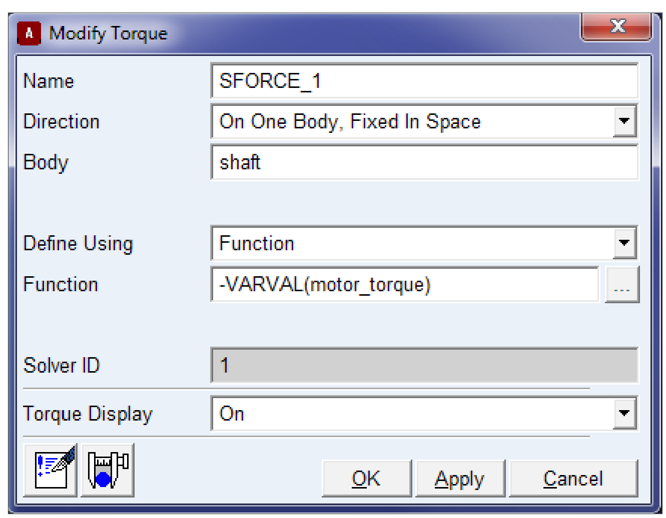
a. Right click on SFORCE_1 and select Modify.




b. Right click in the Function box and select Function Builder.



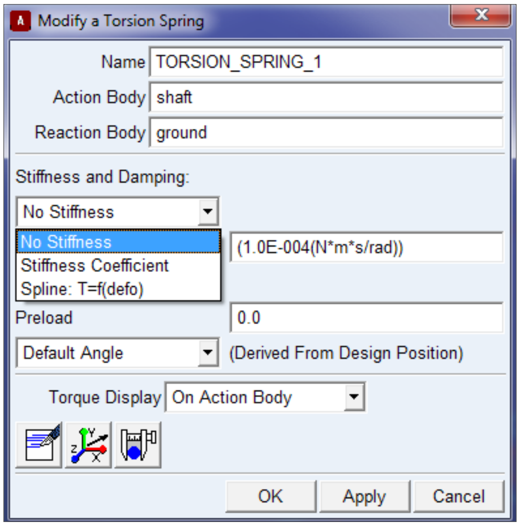

c. Type -VARVAL(motor_torque). This expression builds a function that is equal to the value of the adams variable motor_torque during the runtime. Since motor_torque is later defined as an input controlled by Simulink, we are controlling the shaft with a torque that is input from Simulink model. Note that there is a minus sign here. It is not necessarily to add a minus sign at this stage. If it is found out in later simulation that the out data has opposite sign as expect, a minus sign can be added



then..

d. Click OK.

e. Right click on TORSION_SPRING_1 and then select Modify.

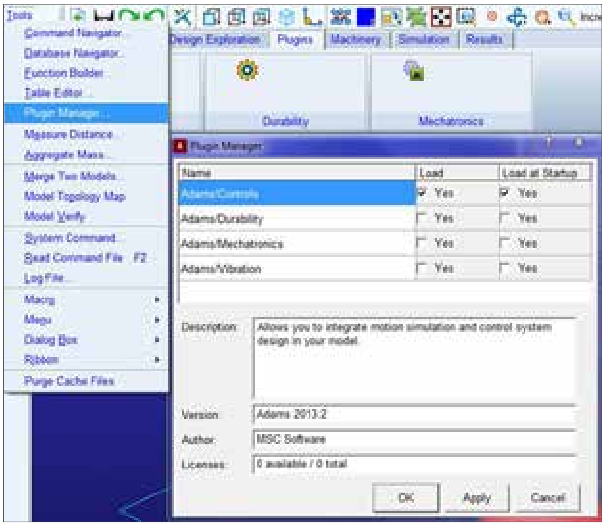



f. Under Stiffness and Damping, choose No Stiffness.

g. Click Ok.

## Step 11: Load Adams/Controls Plugin

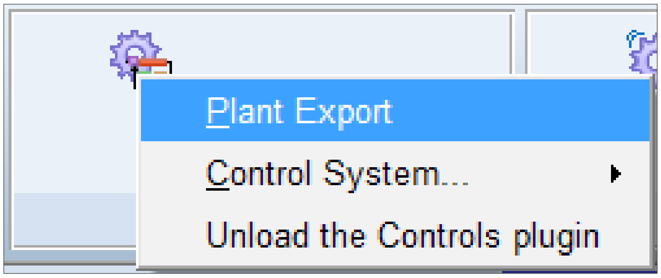
a. From the tool bar, select Tools->Plugin Manager.



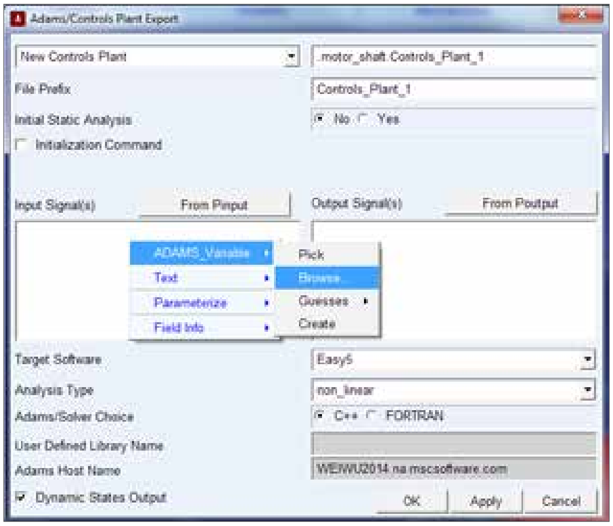

b. Check Adams/Controls plugin.

## Step 12: Export Plant

a. From the Plugins ribbon, select Controls: Plant Export.

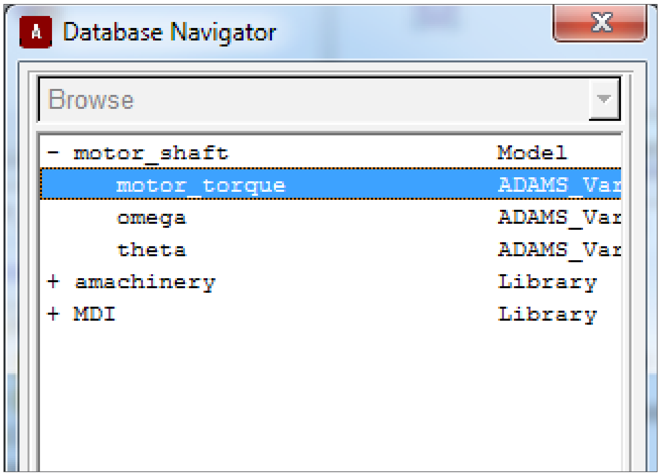



b. Right click in the field of Input Signals, select Adams_




Variable->Browse.

c. From the Database Navigator, double click motor_

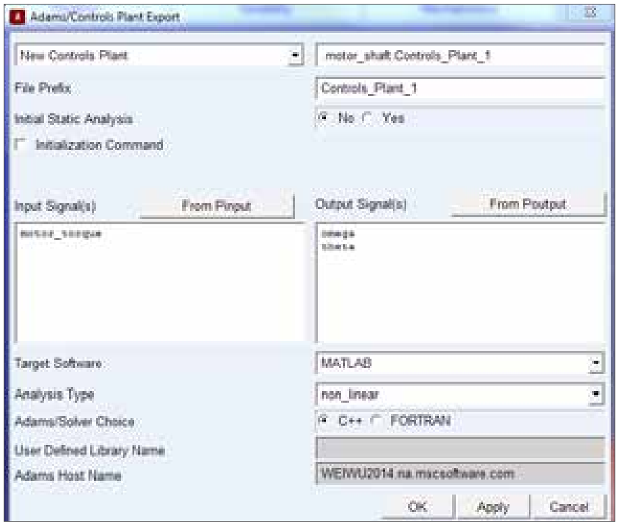



torque.

d. Similarly, choose omega and theta for Output Signals.

e. Choose MATLAB as Target Software.

f. Leave the rest fields as default.




g. Click OK.

h. Now in the working directory, four new files can be found. They are Controls_Plant_1.adm, Controls_Plant_1.m, aviewAS.cmd, Controls_Plant_1.cmd.

## Step 13: Build Co-simulation Model

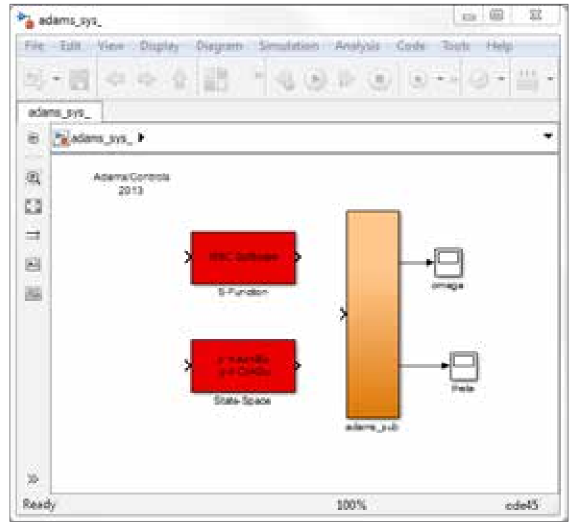
a. From the exercise file directory, copy Motor_initialize.m and post_processing_plot.m to the working directory.

b. Start MATLAB and run Motor_initialize.m to initialize motor parameters.

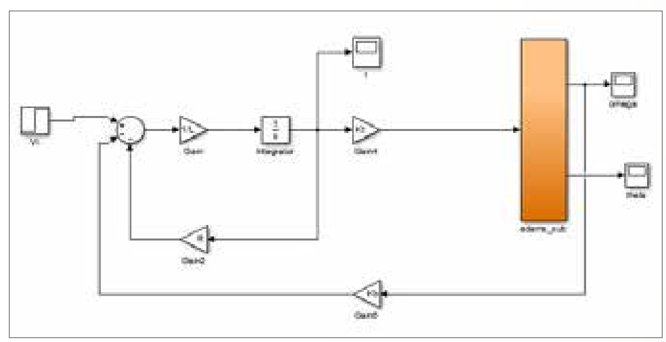
c. Run Controls_Plant_1.m. You will see the following lines in the command window.

```
ans =

16-Jun-2014 10:30:36


%%% INFO : ADAMS plant actuators names :
1 motor_torque
%%% INFO : ADAMS plant sensors   names :
1 omega
2 theta
```

d. Type adams_sys in the command window and then press Entrer. This will bring a Simulink model containing an Adams subsystem. This may take a while.




e. Now, open motor_full.slx and replace the mechanical part with Adams subsystem copied from adam_sys. Don't forget to copy and replace the scopes as well otherwise the post_processing_plot.m in later steps won't work.



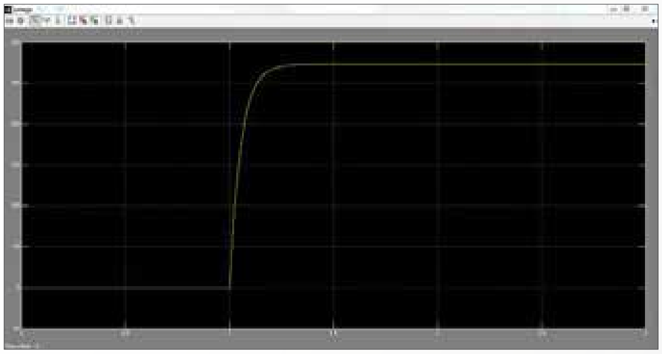

f. Double click on the scope omega. Click on the gear icon on the upper left corner to enter parameter setting of scope.

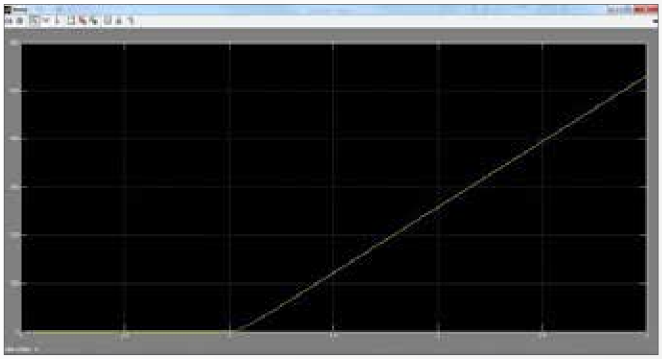
g. Click on History tab. Check Save data to workspace and name Variable name as omega. Repeat this step with theta scope with Variable name theta.

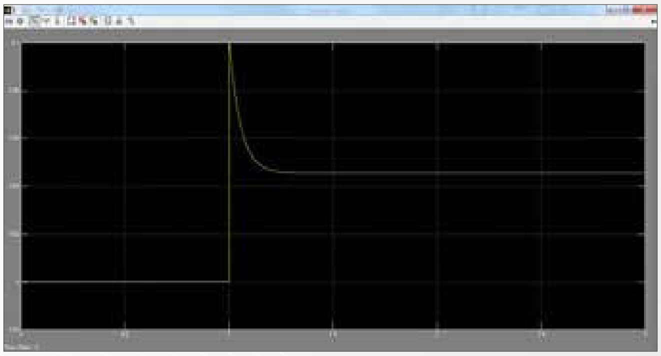
h. Save the model as motor_cosim.slx.

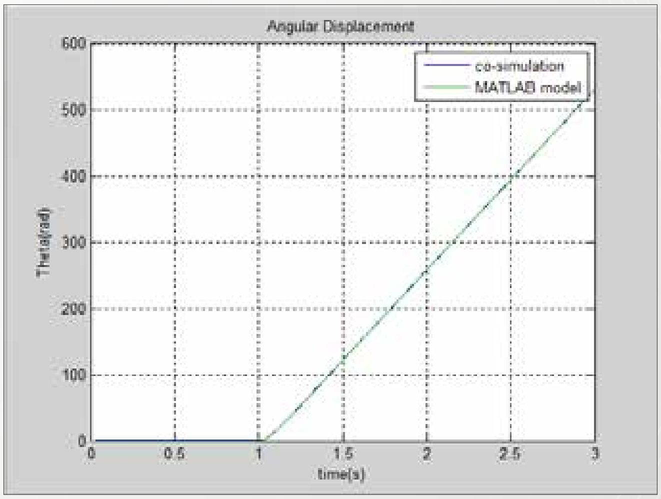
i. Run the simulation for 3 seconds.

j. The scopes show the following results.







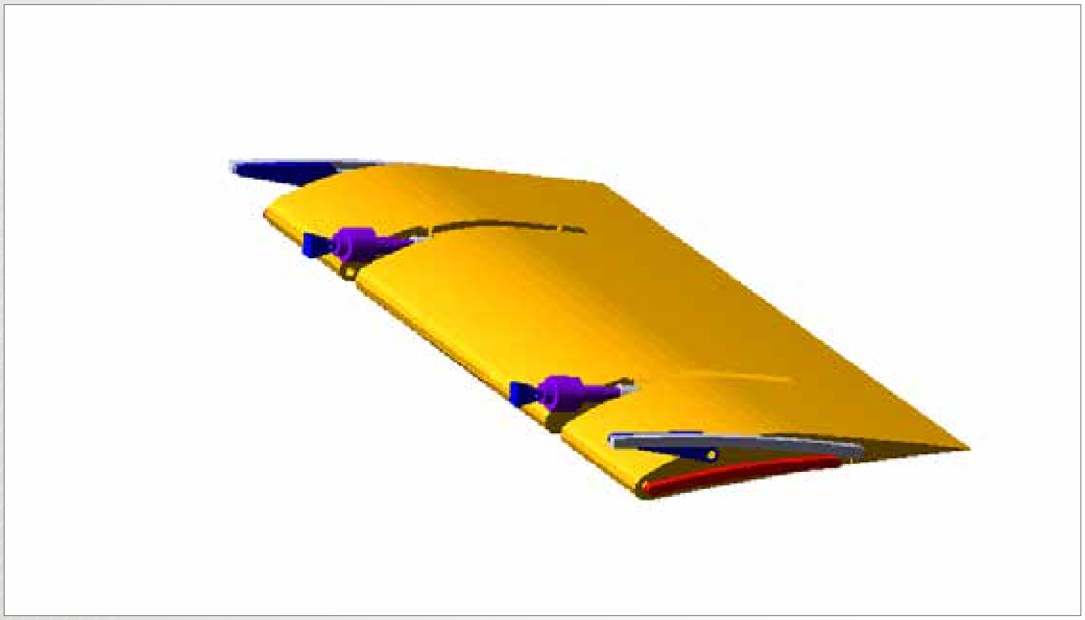



## Step 14: Compare Simulation Results

a. Run post_processing_plot.m.

b. It is found that the two simulation results are identical.

# Example 35: Airplane Control Surface



## Software Version

Adams 2013.2

## Problem Description

In Adams/View section, we have successfully built a model of airplane control surface flap wing. In real world application, after the position command has been sent by the pilot, onboard computer will control the hydraulic system to actuate the flap wing to control the roll of the plane. In this example, we will simulate this event by Adams/Matlab Cosimulation.

## Step 1. Open Adams Model.

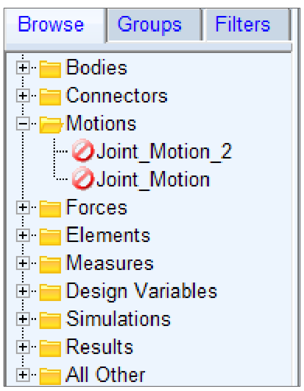
a. Start Adams/View 2013.2.

b. Select **Existing Model** and browse in the exercise directory for **Control_Surface_cosim_start.cmd**. You can also use the model you have built. If you are prompt to see if you would like to use the .mnf file copy in working directory to replace the .mnf file that can't be found, click OK.

c. Click **OK**. Before you start, you can check the video in exercise directory to view the final result.

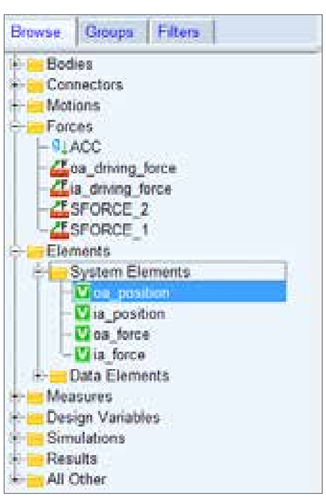
## Step 2. Check Motions.

a. Make sure that the two joint motions are **deactivated**.



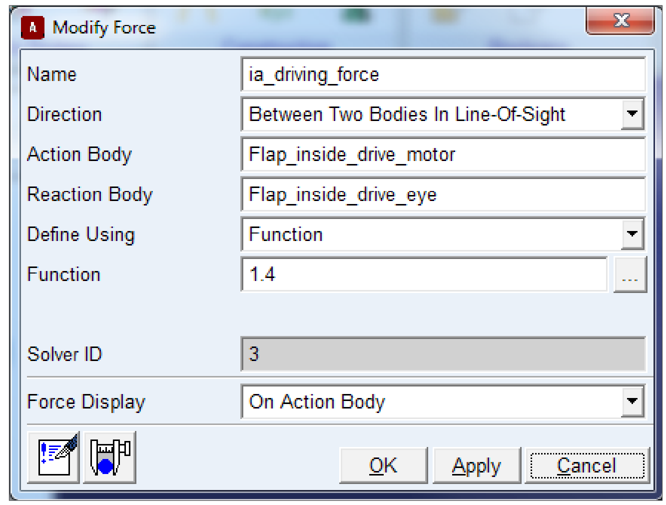

## Step 3. Check State Variables.

a. We have created four state variables already. Among them, **oa_force** and **ia_force** will later be used as input from the Simulink model to Adams model and **oa_postion** and **ia_position** are the output from Adams model to Simulink controller.

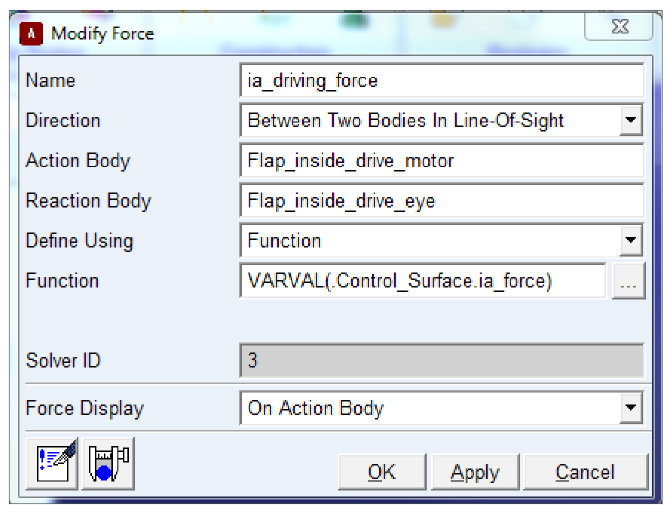



b. Double click **ia_driving_force**.

c. Change Function from 1.4 to **VARVAL(ia_force)** and click **OK**. **ia_driving_force** is the force the motor applied to the drive eye that is connected with the flap wing. This function relates the force the value which is controlled by Simulink as an input to the Adams model.




d. Similarly, change **oa_driving_force** to **VARVAL(oa_force)**.



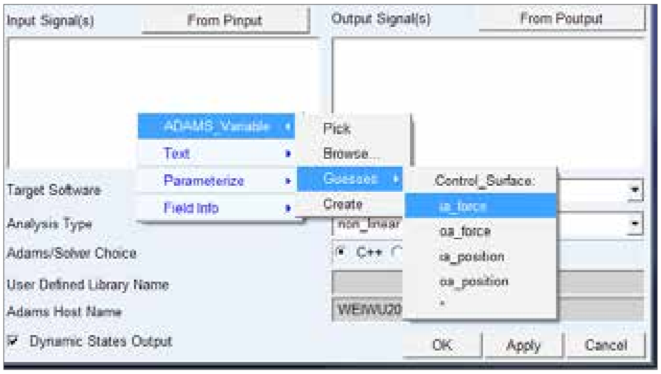

## Step 4. Export Control Plant

a. If Adams/Controls plugin has not been loaded already, go to **Tools-> Plugin Manager** and check **Adams/Controls.**

b. From **Plugins** ribbon, click **Controls->Plant Export**.

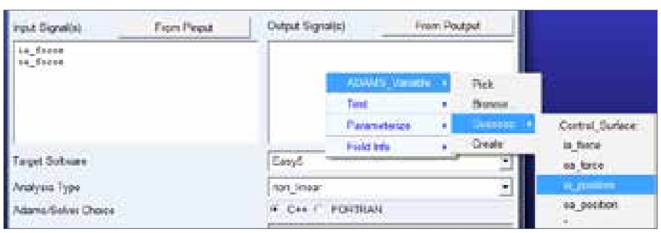
c. Change the **File Prefix** to **Control_Surface_Plant**.

d. Right click in the **Input Signals** field, go to **ADAMS_Variable->Guesses->ia_force**. Similarly, add **oa_force** to the input as well.

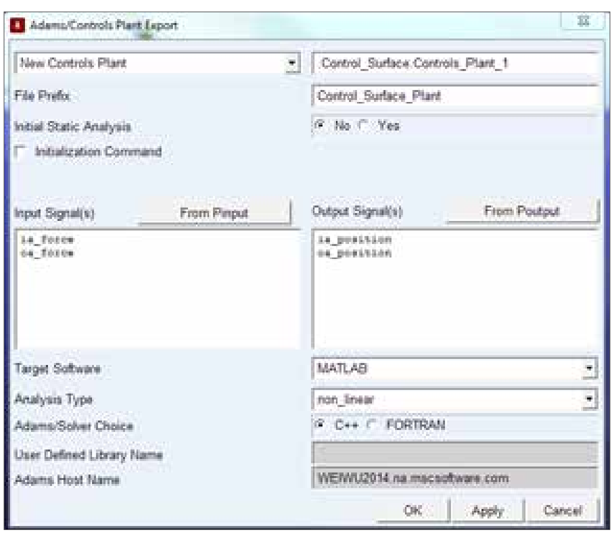



e. Right click in the **Output Signals** field, go to **ADAMS_Variable->Guesses->ia_position**. Similarly, add **oa_position** to the output as well.

f. Select **MATLAB** as Target Software.



g. Click **OK**.




h. In your working directory, you should find several files with Control_Surface_Plant prefix.

## Step 5. Build Simulink Model

a. Start MATLAB and open Simulink UI as well.

b. Change your MATLAB directory to your working directory.

c. Make sure that you have copied **initialize.m**, **Control_Surface_Cosim_start.slx** from start file directory to your working directory.

d. Run **initialize.m**.

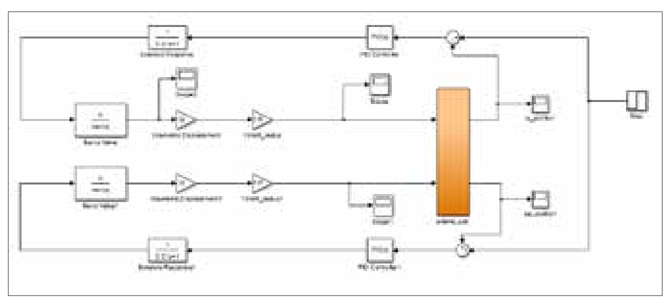
e. Run **Control_Surface_Plant.m**. You should see the following lines in Command Window.

```
%%% INFO : ADAMS plant actuators names :
1 ia_force
2 oa_force
%%% INFO : ADAMS plant sensors   names :
1 ia_position
2 oa_position
```

f. Enter **adams_sys in** the command window. Another Simulink window will open with an Adams subsystem in it.

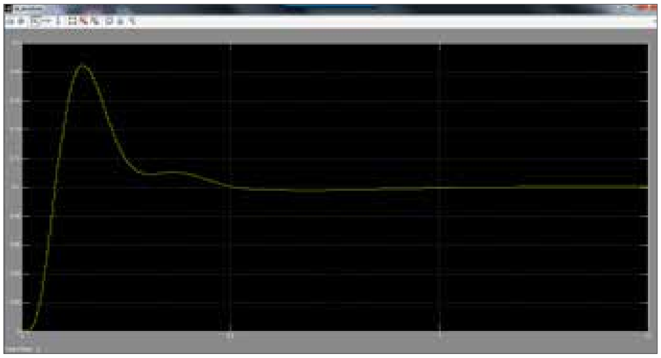
g. Open **Control_Surface_Cosim_start.slx**.

h. Copy the subsystem from **adams_sys.slx** to **Control_Surface_Cosim_start.slx** and connect the proper signals to complete the model.



## Step 6. Run Simulation

a. Run the simulation for 1.5 seconds. After the simulation, you can double click each scope to check the simulation results. Better control can be achieved by fine tuning the PID gains or implementing other control strategy but they are beyond the scope of this tutorial.



## Step 7. Postprocessing the Simulation

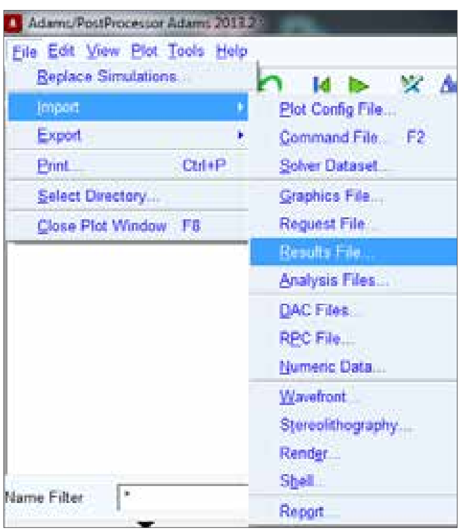
a. The scopes in Simulink model won't provide us with much insight of the mechanism. We can postprocess the result in Adams to get more understanding of the operation of the control surface.

b. In Adams/View, go to **File->New Database**, then select **Existing Model**.

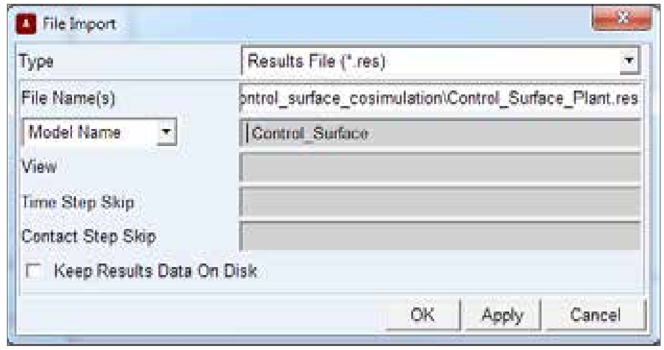
c. Browse in your working directory, find and pick **Control_Surface_Plant.cmd**. This file is generated when you export the control plant in Step 4.

d. Press **F8** to enter postprocessing window.

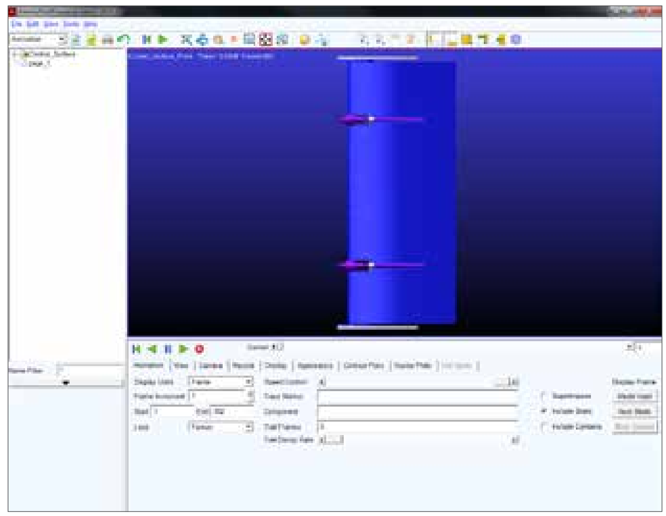
e. Go to **File->Import->Results File**.




f. Select **Control_Surface_Plant.res** in your working directory. This file is created after the simulation is finished. Right click in the field of Model Name and select **Model->Guesses->Control_Surface**. Click **OK**.

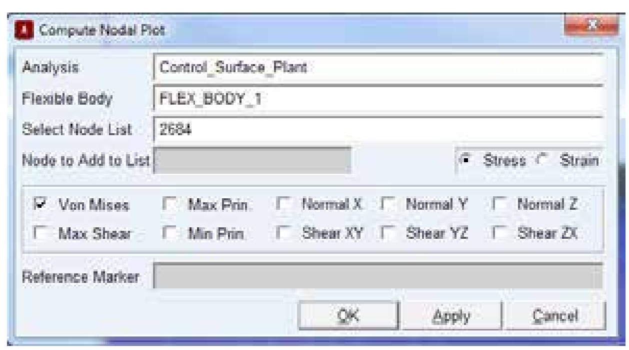



g. Select **View->Load Animation**. Now you can view the animation with the control buttons.



## Step 8. View Stress Contour Plot

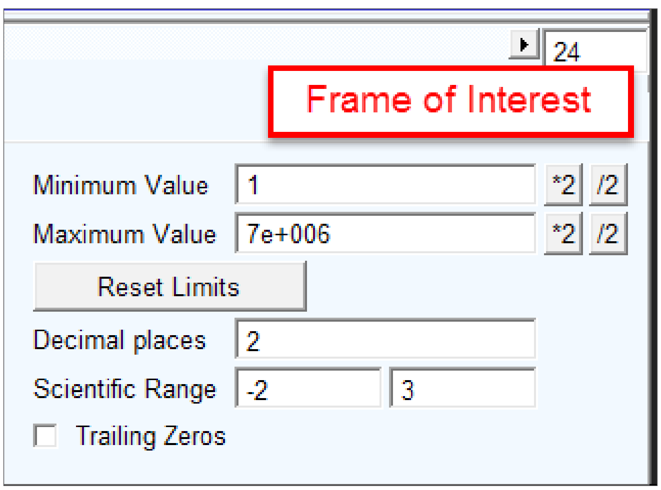
a. Return to Adams/View, select **Plugins->Durability->Nodal Plots**. If you haven't loaded the Durability plugin, go to **Tools->Plugin Manager** and check **Adams/Durability**. Right click in Flexible Body field and select **FLEX_BODY_1**. Enter **2684** in **Select Node List**. This could be any node on the flexible body. Click **OK**.

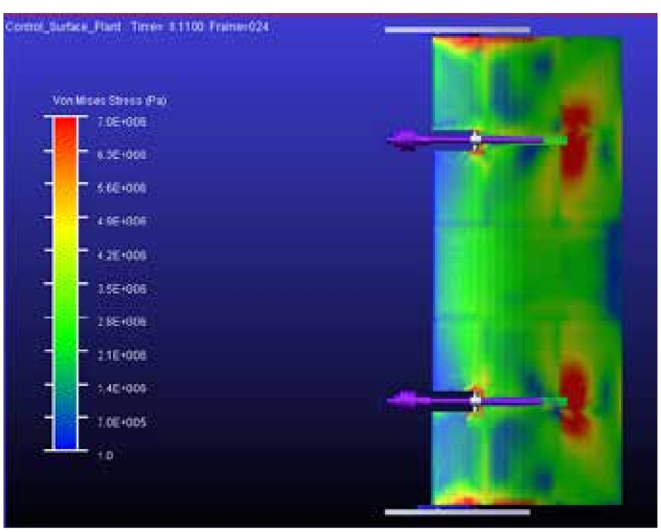


b. Now, switch back to postprocessing window. Click **Contour Plots** tab and select **Von Mises Stress** as **Contour Plot Type**. Now the contour plot in the animation shows the stress distribution in the flexible


body.

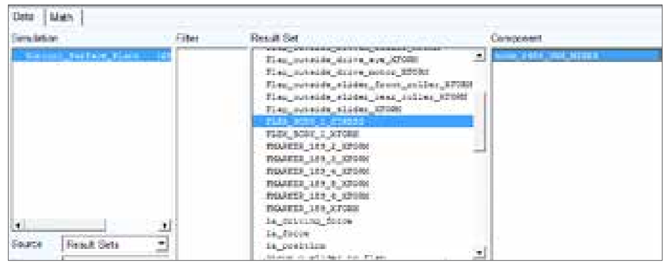
c. As we deactivated the external load on the wings, the stress distribution doesn't show dramatic distinctions at different spots. We can change the Minimum and Maximum value of the legend to show more distribution of stress. Change Minimum Value and Maximum Value to 1 and 7e6 respectively. Change the frame to 24. Then we can get to know the distribution of stresses

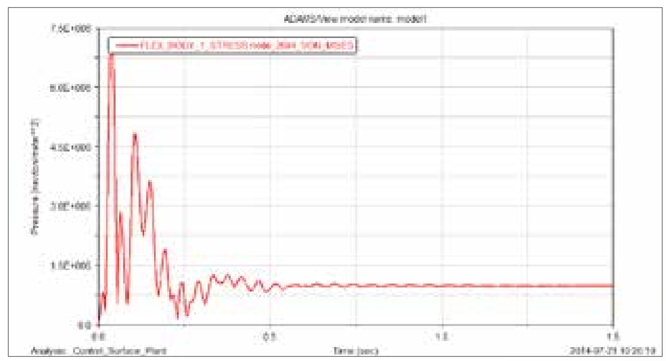



more clearly.

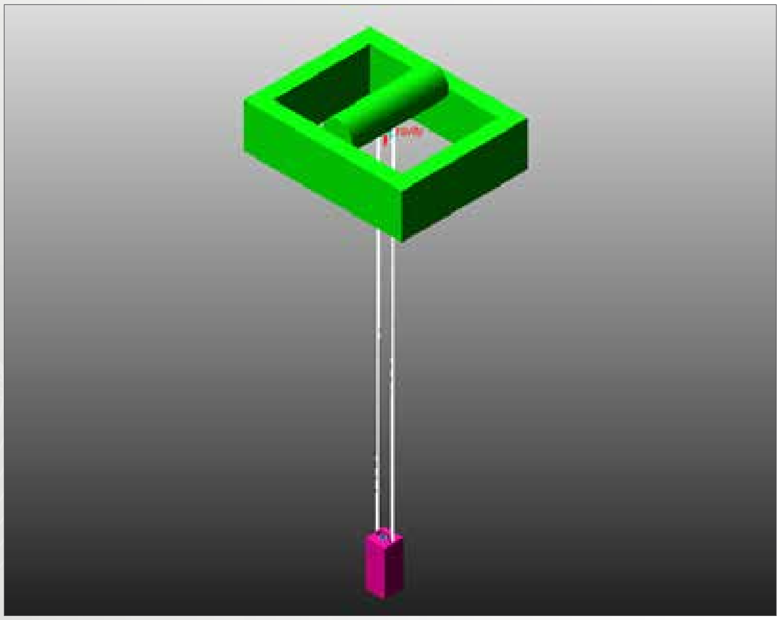
d. Go to **View->Page->New** and create a new page. Right click in the window and select **Load Plot**. Select **FLEX_BODY_STRESS** Result Set and pick **node_2684_VON_MISES** Component. Then click **Add Curves**. Now the time history of the stress at a




specific node can be studied.

# Example 36: Bridge Crane



## Software Version

Adams 2013.2

## Problem Description

Bridge Crane is a commonly seen machinery in warehouses, ports and factories. A major problem that engineers are concerned with is the swing of the cargo when the trolley accelerates and decelerates. Usually, controllers are integrated into bridge crane system to reduce the sway angle of the load. In this example, a bridge crane system will be modeled using the Cable Module of Adams/Machinery, and then be incorporated into a MATLAB/Simulink model. Before you start, copy all the files in example start folder to your working directory.

This example is created in collaboration with Prof. Frank Owen at California Polytechnic State University and Michele Ermidoro at Università degli studi di Bergamo.

## Step 1. Start Adams/View 2013.2

a. Start **Adams/View 2013.2**.

b. Select **Existing Model** and browse in the example start folder for **BridgeCrane_start.cmd**.

## Step 2. Build Cable System

a. From **Machinery** ribbon, click **Cable: Create Cable**.

b. At Anchor Layout page, change the **Number of Anchors** to **0**. In our case, the cable is closed so we don't need to define any anchors.

c. Click Next.

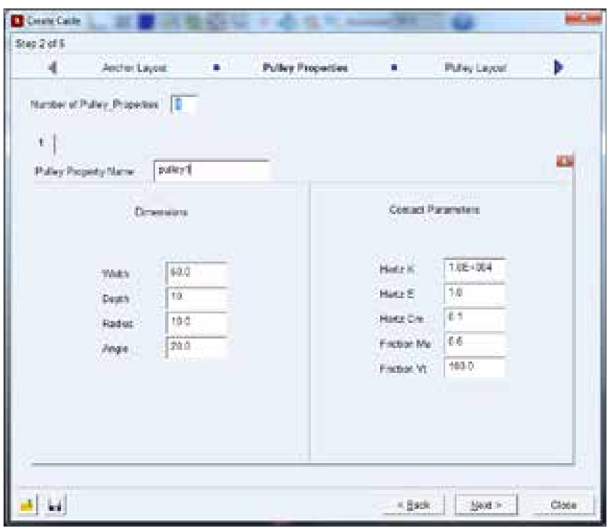
## Step 3. Define Pulley Properties

a. In this page, the shape and contact properties of pulleys are defined. You can define multiple pulley properties which you can select in later steps.

b. Enter **pulley1** as Pulley Property Name. And then enter the dimensions according to the table below.

| | |
|---|---|
| Width | 50 |
| Depth | 10 |
| Radius | 10 |
| Angle | 20 |

c. The definition of each dimension can be found in Adams Help.




d. Keep the contact parameters as default and click Next.

## Step 4. Define Pulley Layout

a. Enter **2** in **Number of Pulleys**. There will be one pulley at the trolley and one pulley at the load.

b. Select **Global Z** as **Axis of Rotation**.

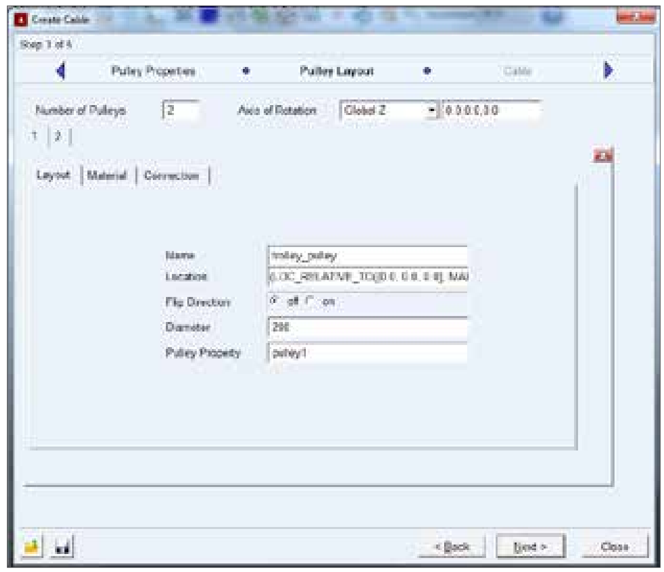
c. At the first page, Name the pulley "**trolley_pulley**".

d. Enter " **(LOC_RELATIVE_TO({0.0, 0.0, 0.0}, MARKER_5))**" at location.

e. Turn off Flip Direction.

f. Enter **200** as **Diameter**.

g. Right click in Pulley Property and go to **cable_pully_props->Guesses->.BridgeCrane_SimpleCable.Cable_Sys_1.pulley1**.




h. Click on **Connection** tab. Select **Revolute** Connection Type. Right click at Connection Part and go to **PART->Guesses->Part 2**.

i. Click on **Pulley page 2**.

j. Name the pulley **load_pulley**.

k. Enter **(LOC_RELATIVE_TO({0.0, 0.0, 0.0}, MARKER_6))** at Location. Leave the rest the same as trolley_pulley and click on Connection tab.

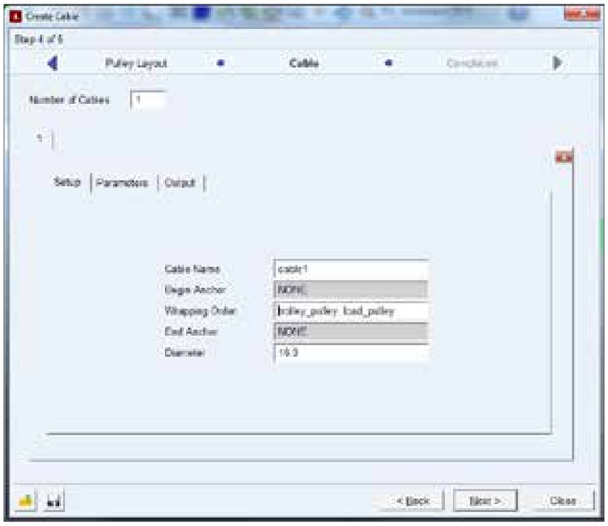
l. Select **Revolute** Connection Type. Right click at Connection Part and go to **PART->Guesses->Part 3**. Click Next.

## Step 5. Define Cable

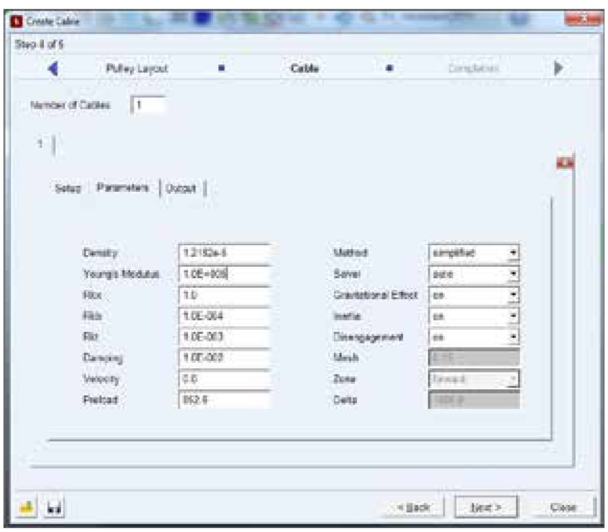
a. Name the cable "**cable1**".

b. Enter "**trolley_pulley, load_pulley**" as **Wrapping Order**. You can also specify this order by right click and select them from Guesses list.

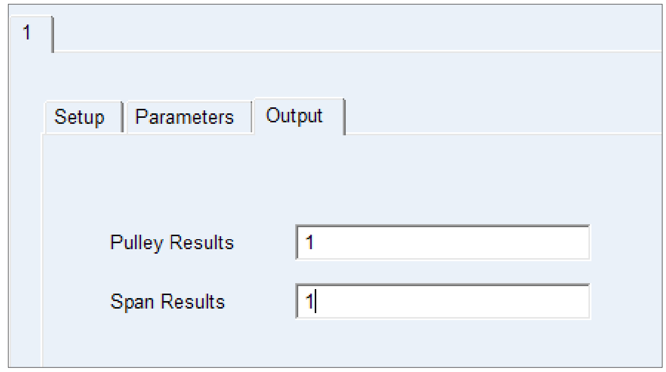
c. Enter **16** at **Diameter**. Click on **Parameters** tab.



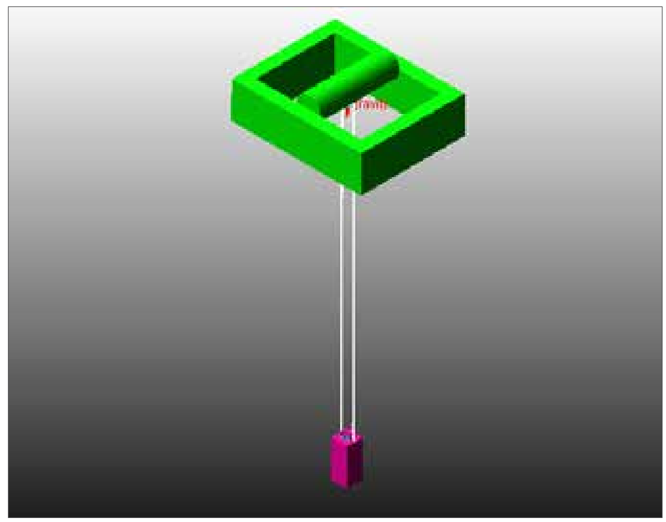

d. Enter **1.2182e-5** in **Density** and **852.6** in **Preload**. Leave the rest as default. It is important to get the preload correct so that the model can run through equilibrium phase.




e. Click on **Output** tab. Enter **1** for both Pulley Results and Span Results. This means that we are getting the result of the first pulley and the first span. Specific details of the results that can be obtained in cable module can be found in Adams Help.



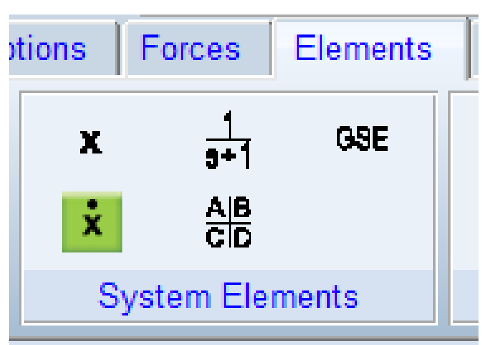

f. Click Next and then Finish. The finished system should look like the one below.



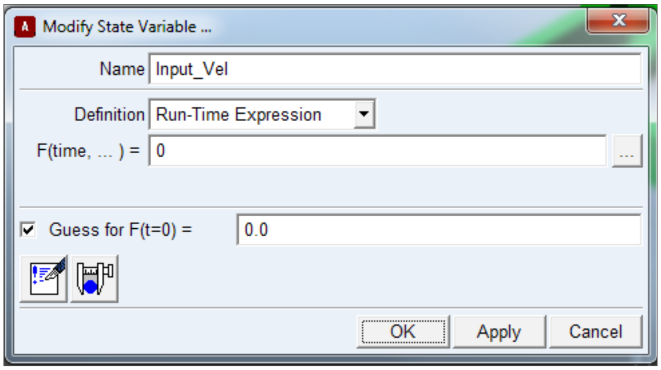
## Step 6. Create State Variables

a. To set up the model for co-simulation, input and output variables should be created. In this model, the input is the translational motion of the trolley. The output of the model is the sway angle of the load.

b. From **Elements** ribbon, select System **Elements: Create a State Variable**.

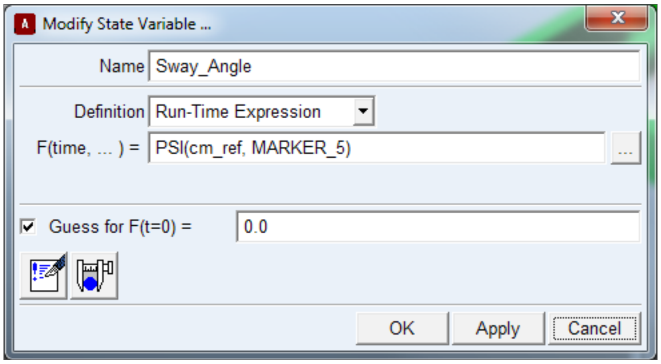


c. Change Name to **Input_Vel**. Leave the Function as 0 because this will be replaced by MATLAB input during simulation. Click Apply.




d. While the Create State Variable window is still open, change the name to **Sway_Angle**.

e. At function, enter **PSI(cm_ref, MARKER_5)**. The function PSI measures the first rotation of the body 313 system. Cm_ref is called To Marker which is the marker being measured and MARKER_5 is called From Marker whose body 313 system is referenced. More detailed information can be found in Adams Help. Click OK.

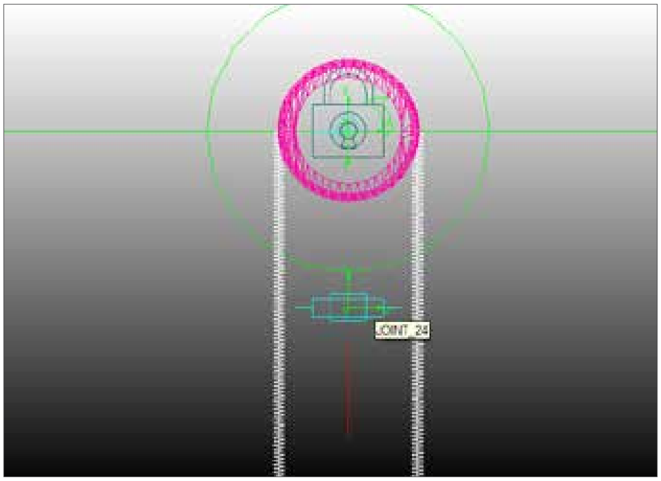



## Step 7. Create Translational Motion

a. Now we need to add a joint motion to the translational joint between the trolley and the ground. This motion drives the trolley forward.

b. From **Motions** ribbon, select **Translational Joint Motion**.

c. Click on **Joint 24**.

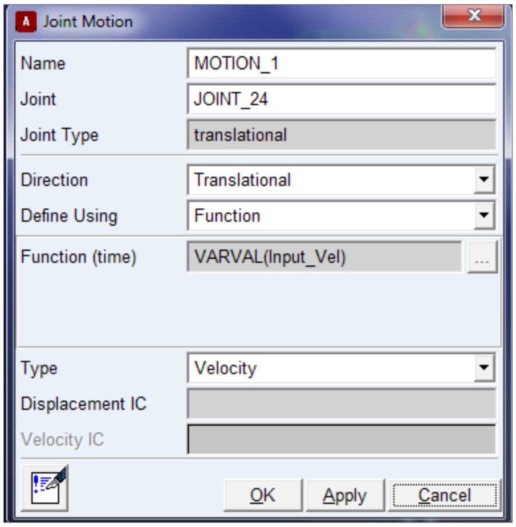



d. From the model tree, right click on **Motion_1**.

e. At Function, enter **VARVAL(Input_Vel)**. This means that the translational speed of the motion is determined by the value of the variable Input_Vel.

f. Change Type to **Velocity**.

g. Click **OK**.




## Step 8. Export Control Plant

a. If you haven't loaded Adams/Controls plugin, go to Tools->Plugin Manager and then check Adams/Controls.

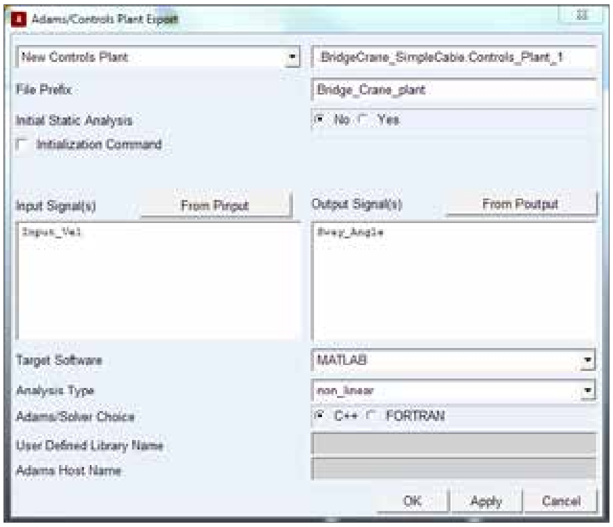
b. From **Plugins** ribbon, select **Controls: Plant Export**.

c. Enter **Bridge_Crane_plant** in **File Prefix**.

d. Check **No** for **Initial Static Analysis**.

e. Enter **Input_Vel** and **Sway_Angle** at **Input Signals** and **Output Signals** respectively.

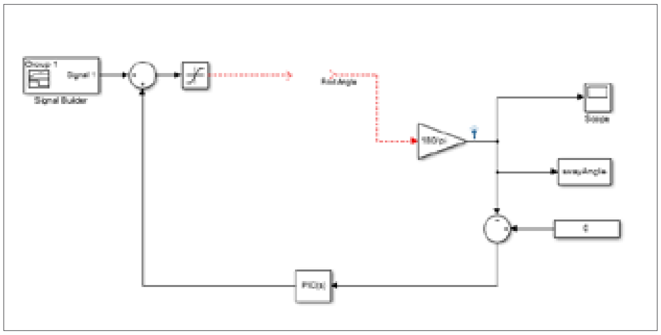
f. Select **MATLAB** as **Target Software**, **non_linear Analysis Type** and **C++ solver**.



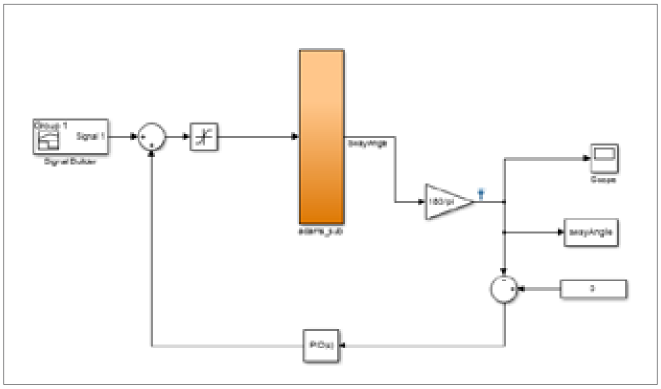

g. Click OK. There should be four files with the defined prefix generated in your working directory.

## Step 9. Build Simulink Model

a. Start **MATLAB**.
b. Change the directory to your working directory.
c. Run **Bridge_Crane_plant.m**.
d. Enter **adams_sys** in the command window. A Simulink model with adams_sub subsystem is created.
e. Open **model_start.slx**.

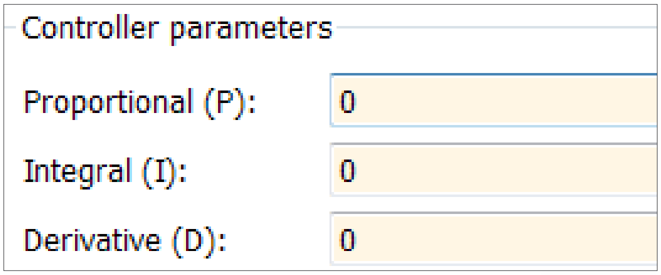


f. Copy the adams_sub subsystem and paste it in model_start.slx. Connect the appropriate signals.

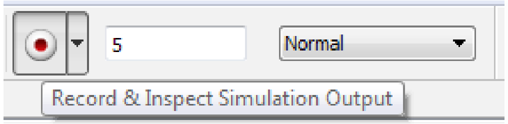


## Step 10. Run a Baseline Simulation

a. Double click on the **PID controller** block.
b. Change all PID controller parameters to **0**.




c. Make sure that Record & Inspect Simulation Output is on.



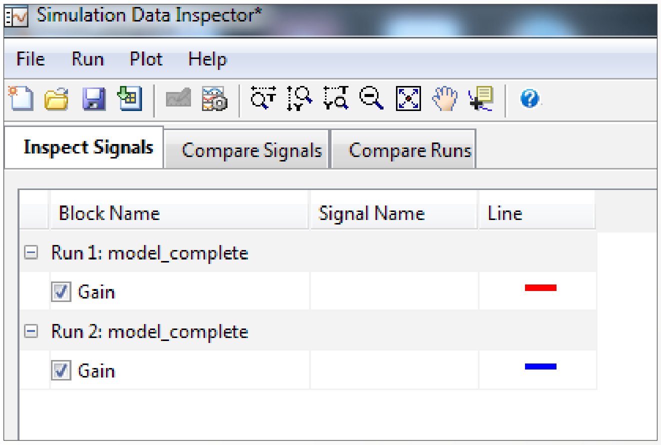

d. Run the simulation for 5 seconds..

## Step 11. Run Simulation with Controller

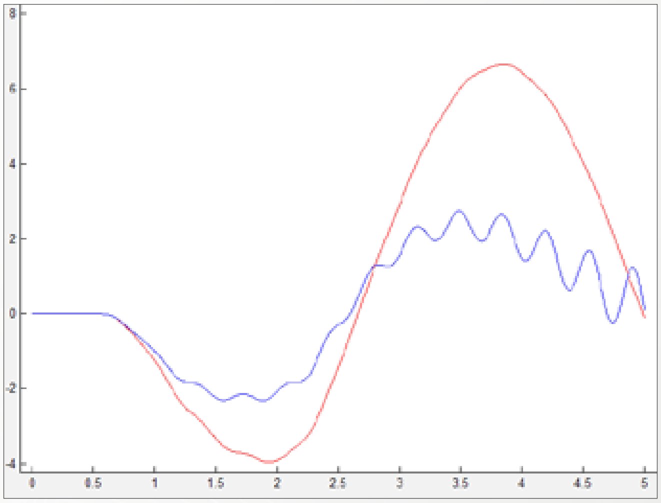
a. Double click on PID block again.
b. Change the proportional gain, integral gain and derivative gain to **-100, -10, -10** respectively.
c. Run the simulation for 5 seconds.

## Step 12. Compare Simulation Result

a. Open Simulation Data Inspector.
b. Check Inspect Signals of Run1 and Run2.




c. On the right hand side a plot of these two runs is shown. It can be found that the sway angle of the load has been reduced significantly after a PID controller is implemented.



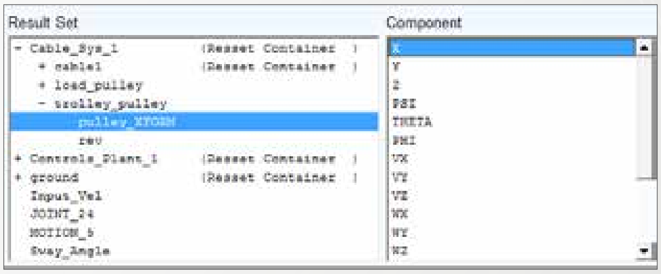
## Step 13. Postprocessing in Adams/View

a. In the Adams model in Simulink, only the sway angle of the load is explicitly exported. When the simulation is finished, we can also study other simulation results in Adams Postprocessor.
b. Start **Adams/Postprocessor**.
c. Go to **File->Import->Results File** and browse in your working directory for the .res file generated after simulation.
d. Click OK.

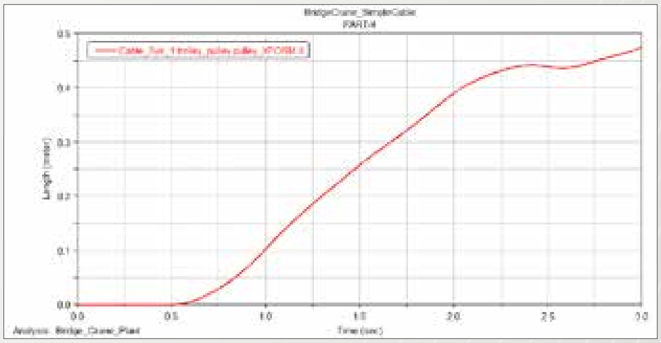
## Step 14. Plot Results of Interest

a. From the **Result Set** column, expand **Cable_Sys_1** and then **trolley_pulley**. Highlight **pulley_XFORM**.

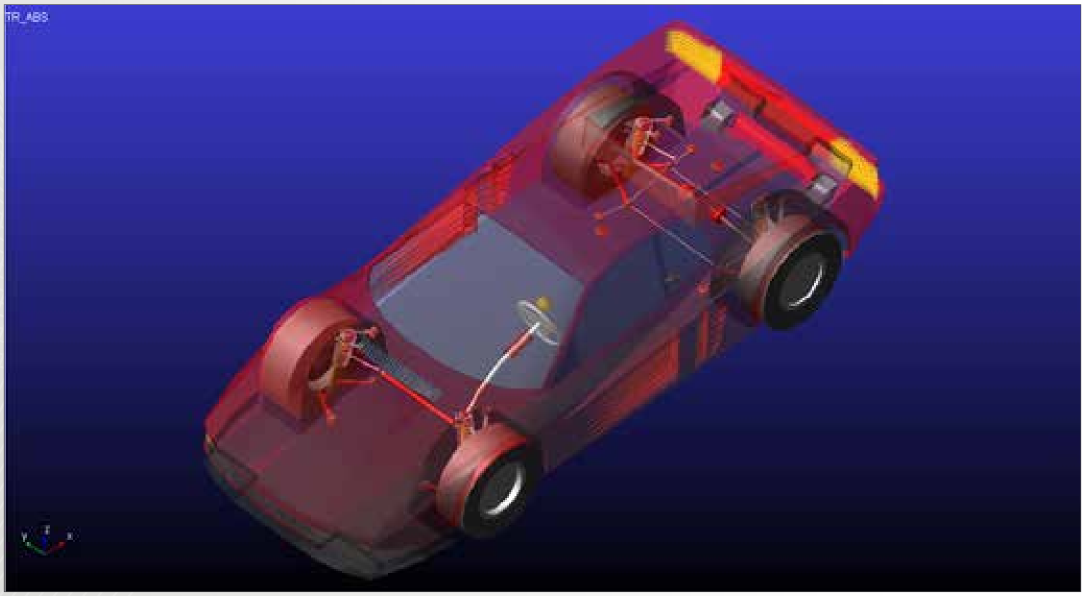
b. Select **X** Component from the right column and click on **Add Curves**.




c. A plot showing the horizontal motion of the trolley pulley, i.e. the trolley will be plotted.



# Example 38: Vehicle ABS System




## Software Version

Adams/Car 2013.2

## Problem Description

This a quick example to show the workflow of Adams/Car and MATLAB/Simulink cosimulaiton. A vehicle is subjected to a split-mu brake test (the friction significantly differs between the left and the right wheel path). An ABS controller is designed in Simulink. A cosimulaiton is performed to verify the controller. Note: you may need to finish Example 39 first before you proceed. You should copy all the files under exercise start folder to your working directory. Since Adams/Car and MATLAB can have their working directory set up separately, it is advised to select a common folder as your single working directory.

## Step 1. Open the Full-Vehicle Assembly

a. Start Adams/Car 2013.2.

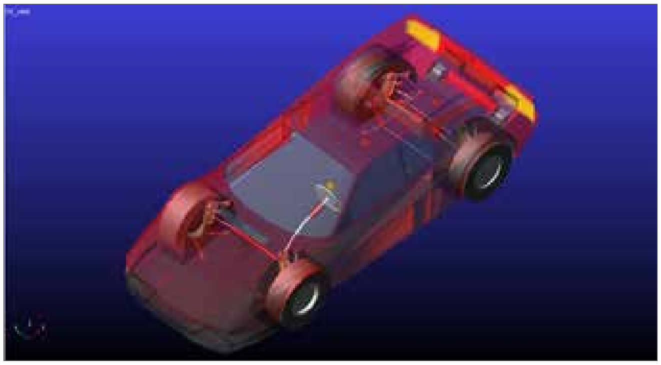
b. If you are prompted to select user interface, select **Standard Interface**.

c. From the menu bar, select **File->Open->Assembly**.

d. Right click in the field and select **TR_ABS.asy** from ABS_2013 database. If the database has not been added to session already, please refer to step 2 in example 39 to add the database to session.

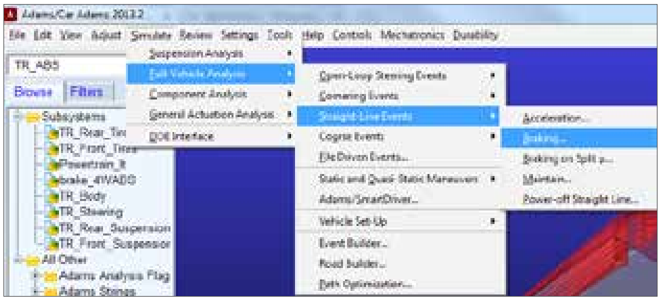
e. The vehicle assembly should be like the one shown here.

f. Go to **Tools->Plugin Manager**. Check **Load Adams/Controls**.



## Step 2. Run a Files_Only Simulation

a. Select **Simulate->Full-Vehicle Analysis->Straight-Line Events->Braking**.




b. Enter **abs** in Output Prefix.

c. Enter **4** and **400** in End Time and Number of Steps respectively.

d. Select **files_only** for Simulaiton Mode.

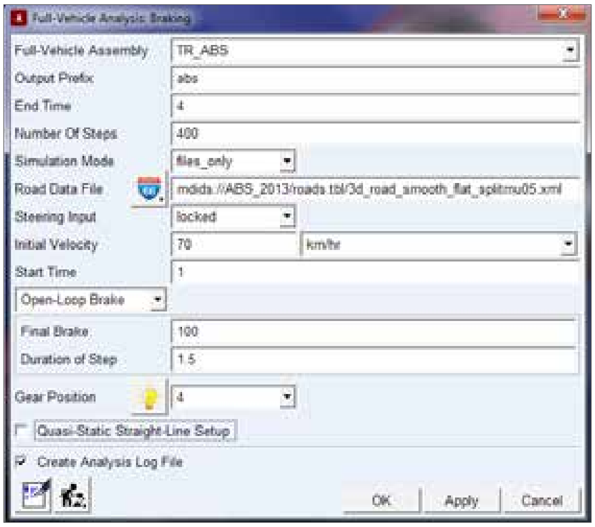
e. Right click in the Road Data File field and go to search. Pick **ABS_2013.tbl** and select **3d_road_smooth_flat_splitmu05.xml**.

f. Select **locked** for **Steering Input**.

g. Enter **70 km/h** in Initial Velocity.

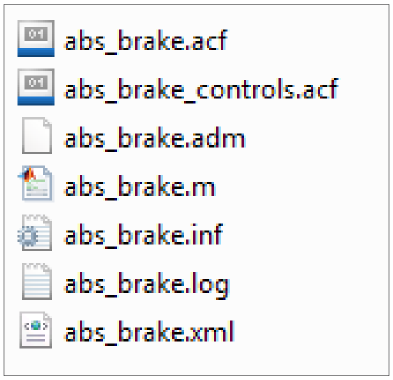
h. Set the rest of the parameters as shown in the figure. **Uncheck Quasi-Static Straight-Line Setup**.

i. Click **OK**.




## Step 3. Check the Files

a. After step 2, you should be able to find a bunch of files under your working directory.




b. Notice that we have a .m file generated but we will replace this file using Adams/Controls plant export.

## Step 4. Export the Plant

a. Make sure you have loaded the Controls plugin. If not, go to **Tools->Plugin Manager**, check Adams/Controls and click OK.

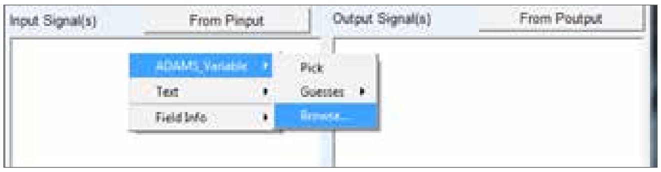
b. Go to **Controls->Plant Export**.

c. Select **New Controls Plant**.

d. Enter **abs_brake** in File Prefix. You should enter the same name as that in step 3. In this way, the .m file generated in this step will replace the .m file from the last step.

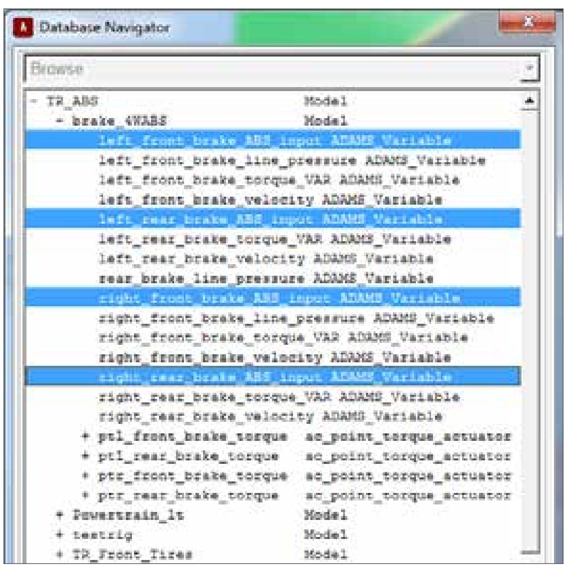
e. Select **No** for **Initial Static Analysis**.

f. Right click in the blank field on the left and select **ADAMS_Variable->Browse**.




g. Expand brake_4WABS. Hold ctrl and highlight four ABS input variables. These are the values that will be passed into Adams by Simulink. In this example, the necessary input variables has been created for you. When you are building your own model for other applications, you should create them using state variables before you export the plant.

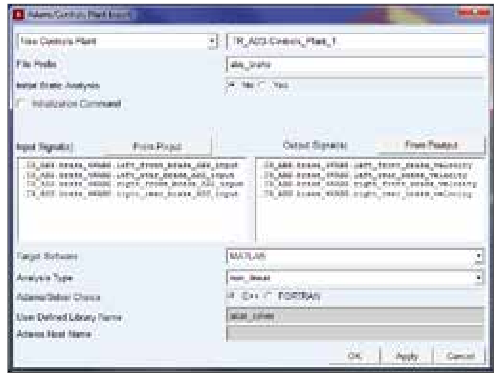
h. Click **OK**.




i. Similarly, right click in the blank field on the right and select four brake velocities.

j. Select **MATLAB** as Target Software and leave the rest as default.

k. If you check the working directory, you should find that abs_brake.m has been replaced (new modification date).



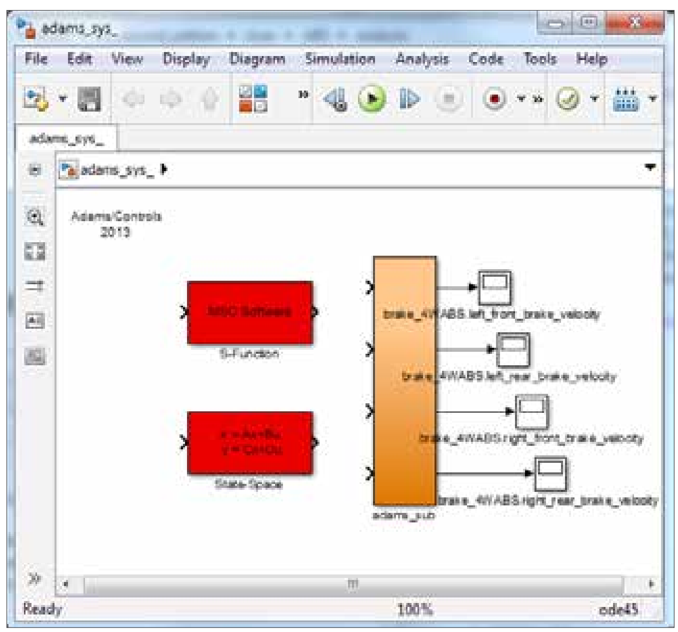
## Step 5. Build Simulink Model

a. Start MATLAB/Simulink. Copy all the files under exercise start folder to your MATLAB working directory.

b. Run **abs_brake.m** by dragging it from the current folder window to command window. This sets up the cosimulation and builds up required inputs and outputs as is shown in the command window.

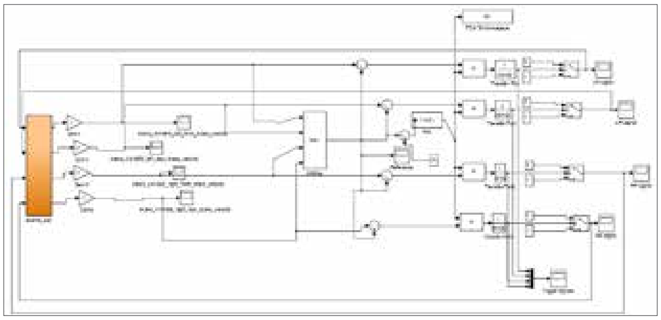
c. Run **abs_setup.m**. This sets up the necessary parameters for abs controller and **opens abs_controller.mdl.**

d. Enter **adams_sys** in the command window. This will create an adams subsystem in Simulink which uses the inputs and outputs in step 5b. Note that this .m file can be used to initiate any cosimulation.




e. Replace the subsystem adams_sub in abs_controller.mdl with the one in adams_sys. You can click on adams_sub in adams_sys and press Ctrl+ C to copy it and press Ctrl+V to paste it in abs_controller.mdl. Observe the signal connection before you delete the original adams_sub block in abs_controller.mdl. Connect the corresponding signals.

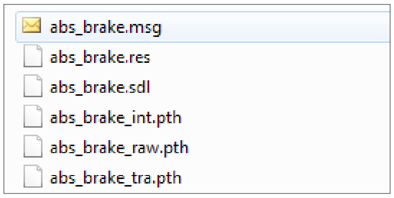
f. Now the cosimulation has been setup and we can run the simulation.



## Step 6. Run Cosimulation

a. Run the Simulink model for 4 seconds. At each communication interval, inputs (brake signals) from Simulink is fed into Adams/Solver while the wheel velocities are fed into Simulink. They exchange these information at every simulation step to finish the cosimulation.

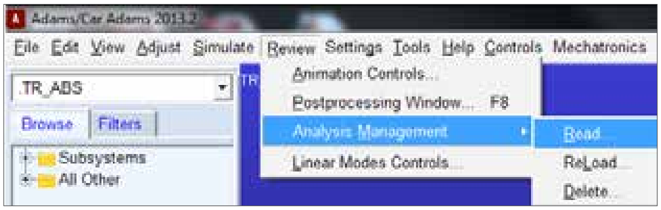
b. After the simulation, you will find that several new files have been generated, including abs_brake.res. This is the Adams result file that can be imported for postprocessing.




## Step 7. View the Animation

a. Make sure that your assembly TR_ABS is open.

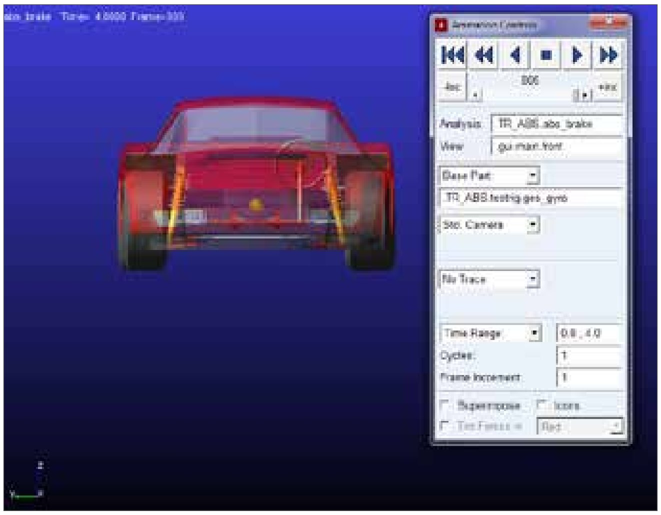
b. Go to **Review->Analysis Management->Read**.




c. Check Results file and right click in Analysis Name. Select Browse and look for **abs_brake.res** in your working directory. Then click **OK**.

d. Go to **Review->Animation Controls**.

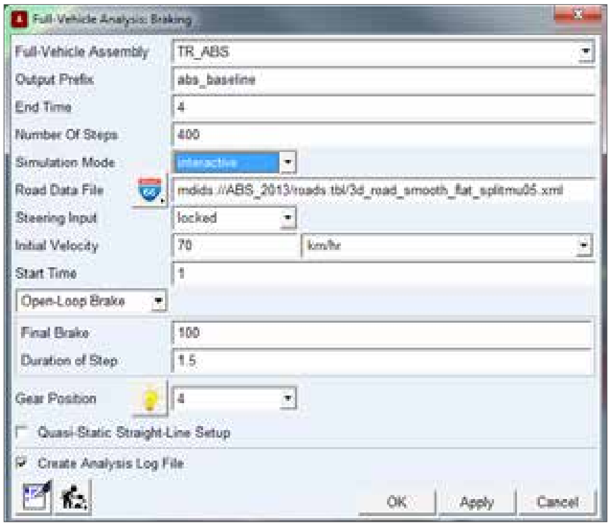
e. Play the animation and it can be found that the vehicle maintains straight path on the split mu road very well



## Step 8. Run a Baseline Simulation

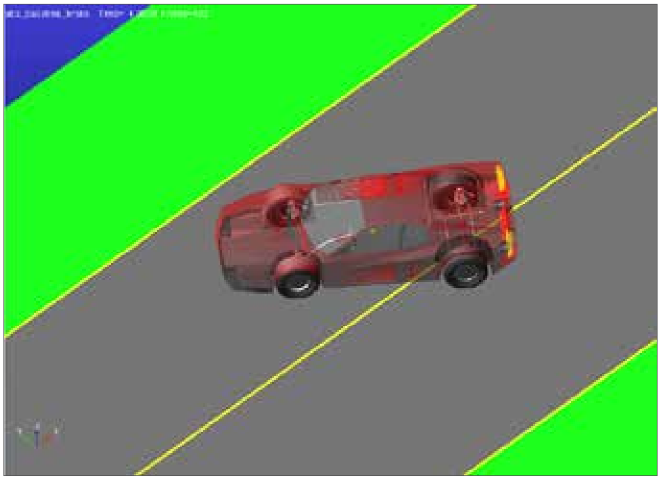
a. Now we will run a baseline simulation without abs controller.

b. Create another braking simulation using the same set of settings. Note that Simulation Mode should be interactive this time.




c. Go to **Review->Animation Controls**.

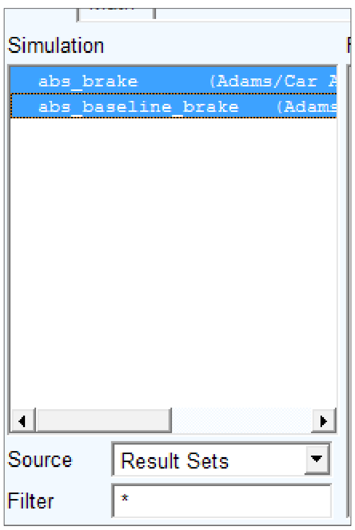
d. Play the animation and you will find that the vehicle spun at the end of simulation.



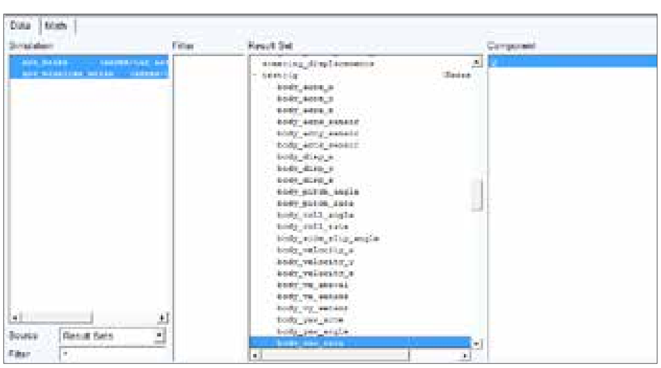
## Step 9. Compare the results

a. Press **F8** to open postprocessing window.

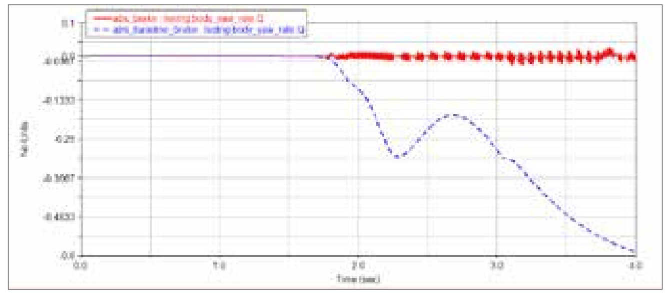
b. Press **Ctrl** and highlight both simulations.




c. Browse for **body_yaw_rate** under testrig and select component Q.



d. Click **Add Curves**. Judging from the plot, the effect of ABS controller is rather significant. The yaw rate is controlled about zero very well while the vehicle without ABS spins off. In the completed folder of this example, you can find a video showing the comparison of these two simulations.



# Example 37: Windshield Wiper Mechanism

## Software Version

Adams 2013.2

## Problem Description

In Adams/Machinery Section, we have built a Windshield Wiper Mechanism using motor and gear module. In this example, we will built a basic UI in MATLAB/Simulink so that we can control the speed of the windshield wiper continuously while the simulation is running. Before you start, copy all the files in the exercise start folder to your working directory.

## Step 1. Start Adams/View

a. Start Adams/View 2013.2.

b. Select **Existing Model** and browse for **Windshield_ Wiper_cosim_start.cmd** in your exercise start folder.

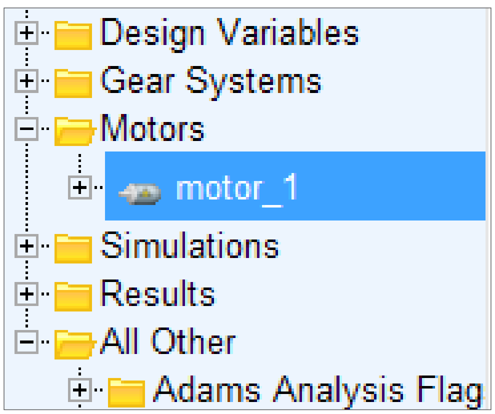
c. Click **OK**.

d. It may take a while for Adams/View to import the model because the model contains worm gear and motor.

## Step 2. Export Control Plant

a. Make sure that the Adams/Controls Plugin has been loaded. If not, go to **Tools->Plugin Manager** and then check Adams/Controls.

b. From the model tree, double click on **Motors->motor_1** to modify the motor. The motor is currently built as an analytical motor which calculates its torque output by built-in formulas. We will replace it with an external one which receives its torque command from MATLAB. Click Next until **Inputs** Page.



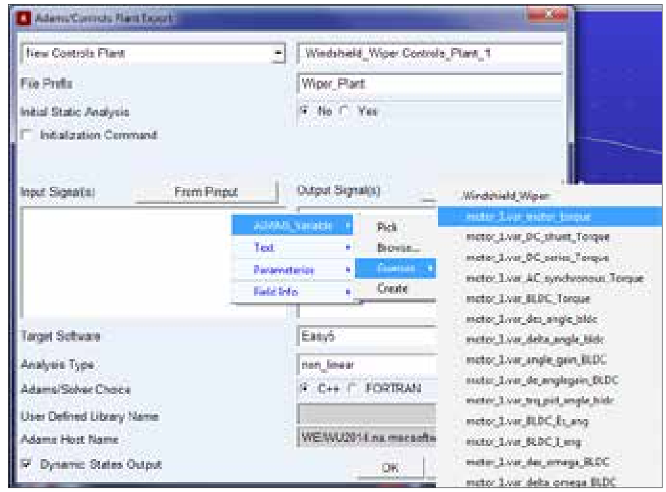

c. Select **Co-Simulation** as External Method.

d. From the drop down menu of **Plant Input/Output**, pick **User Defined**.

e. Enter **Wiper_Plant** at **File Prefix**.

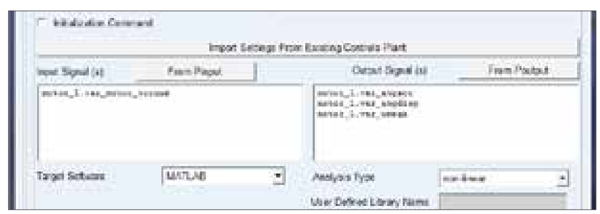
f. Pick **No** for **Initial Static Analysis**.

g. Right click in the field of **Input Signals** and select **Adams_Variable->Guesses->var.motor_torque**.



h. Right click in the **Output Signal** field and select **Adams_Variable->Browse**. In the Database Navigator, expand motor_1 and select **var_angacc**, **var angdisp** and **var_omega**.

i. Select **MATLAB** as Target Software. Click Next and then Finish.



j. Now, if you check your working directory, you will find that several new files with Wiper_Plant prefix has been created.

## Step 3. Build Simulink Model

a. Start MATLAB and open Simulink UI.

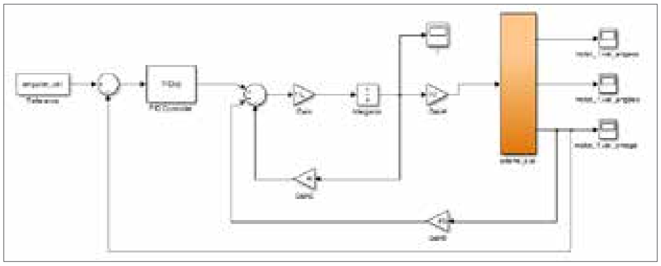
b. Run **Motor_initialize.m**. This file sets up several parameters of the motor.

c. Run **Wiper_Plant.m**. This file sets up the simulation environment and creates necessary inputs and outputs for Adams Subsystem.

d. Enter **adams_sys.m** in the command window. A Simulink UI will be brought about and an adams_sub block is contained in this Simulink model.

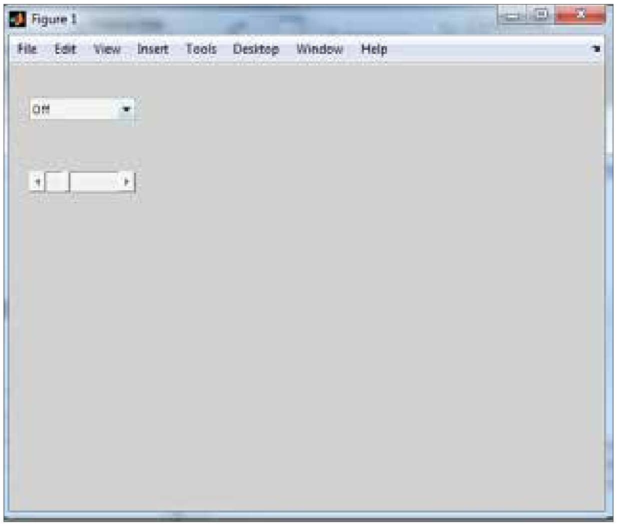
e. Open **wiper_control.slx**. When the model is opened, you will notice that there is a block missing.

f. Copy the adams_sub block from adams_sys_.slx and paste it into wiper_control.slx. Connect matching signals. The completed model will look like the one



below.

g. Run **wiper_control_ui.m**. This file will create a UI with a slider bar. With this slider bar, you can adjust the



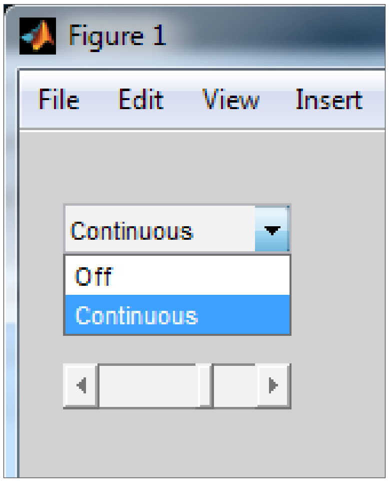

speed of the wiper any time during the simulation.

## Step 4. Run the Co-simulation

a. Set a 50 seconds stop time for the Simulink model.

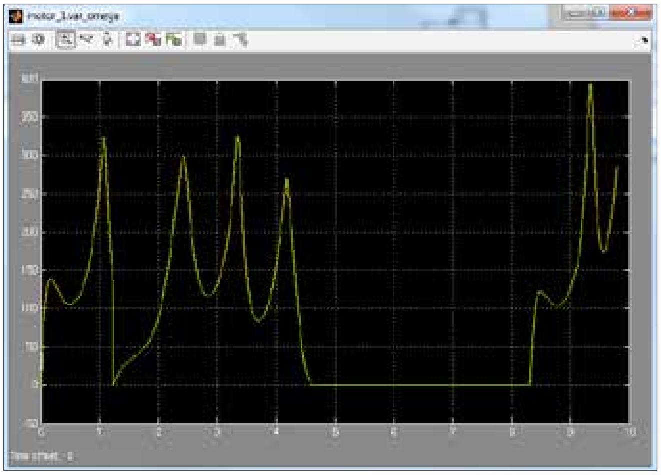
b. Run the simulation.

c. While the simulation starts, from the drop down menu in our control UI, select **Continuous**.




d. Now the wiper starts to move. You can double click on the scope of **var_omega** to see the change in speed.

e. Move the slider bar left and right to see the motor speed changes simultaneously. Note that you may want to stay at certain speed for 2-3 seconds just to see the change in the magnitude of the curve.

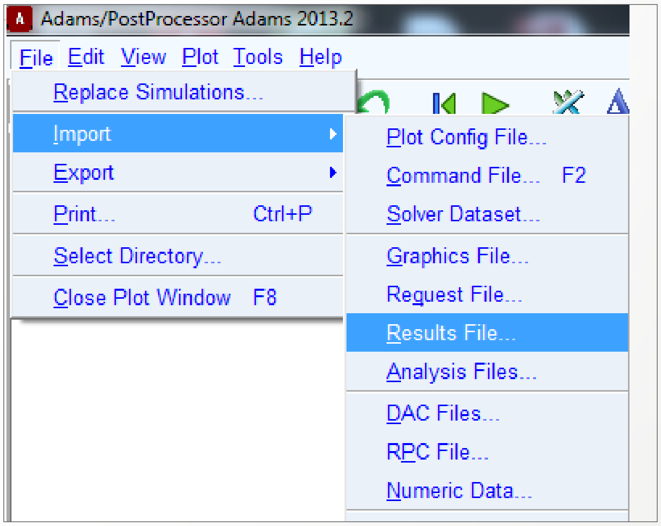


## Step 5. Postprocessing the Simulation.

a. If you have closed the Adams/View window before simulation, open Adams/View 2013.2 and select Existing Model. If not, go to **File->New Database->Existing Model**. Browse in your working directory for Wiper_Plant.cmd. It may take a while for the .cmd file to load. If you are warned that the worm and worm wheel are not inplane, just click OK.

b. When the model is loaded, press **F8** to enter postprocessing window.

c. Go to **File->Import->Results File**.




d. Right click in File Name and browse for **Wiper_Plant.res** in your working directory.

e. Right click in Model Name and select **Model->Guesses->Windshield_Wiper**.


f. Click **OK**.

g. Right click in the page and select **Load Animation**.

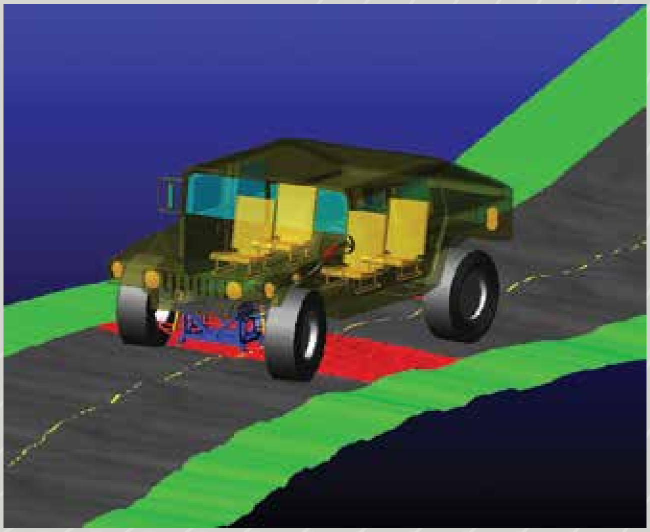
h. Click the play button and you can review the animation of the co-simulation we just performed.

i. You can also plot curves like we did in Example 32

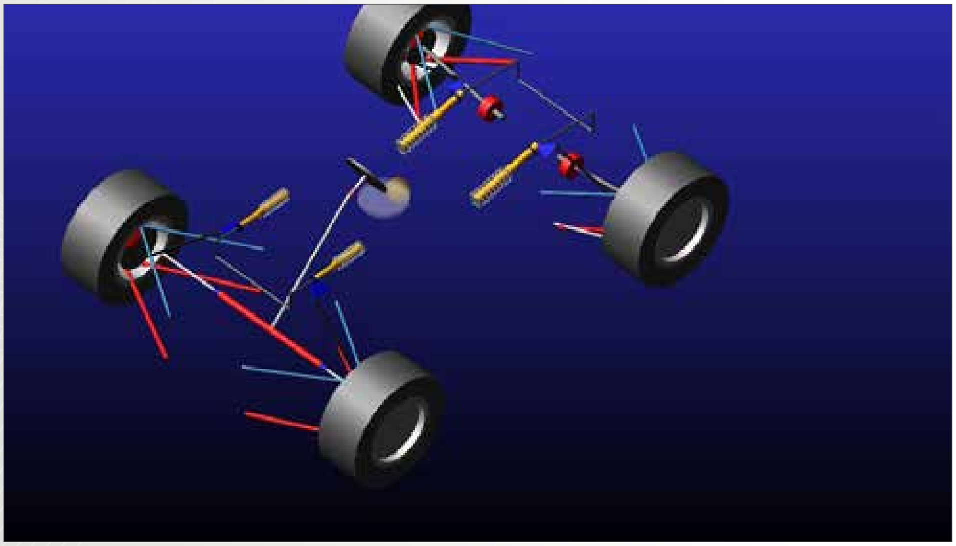
# Section VI: Vehicle Dynamics

*Adams/Car is one of the most popular dynamic simulation package for vehicle dynamics. Adams/Car provides template based strategy for vehicle design with seamless flexible body integration and co-simulation capability. The simulation result can also be used for durability analysis and all kinds of postprocessing.In this section level, you will learn:*

- *How to load Adams/Car database*
- *How to run full-vehicle simulation*
- *How to incorporate flexible body into full-vehicle simulation*
- *How to postprocess the full-vehicle simulation result*



# Example 39: FSAE Full Vehicle Simulation Events



## Software Version

Adams/Car 2013.2

## Problem Description

Use the Formula SAE template to learn the basic full-vehicle simualtion work flow of Adams/Car

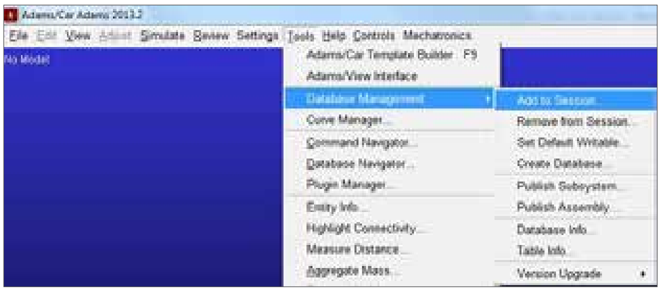
## Step 1. Start Adams/Car.

a. Start Adams/Car 2013.2.

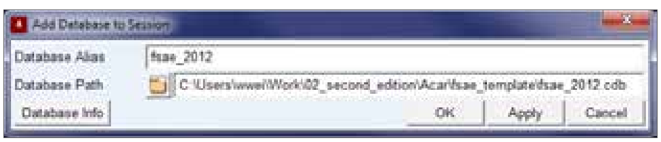
b. If you are prompted to select user interface, select Standard Interface.

## Step 2. Add FSAE Database to Adams/Car.

a. From the toolbar, select Tools->Database Management->Add to Session.



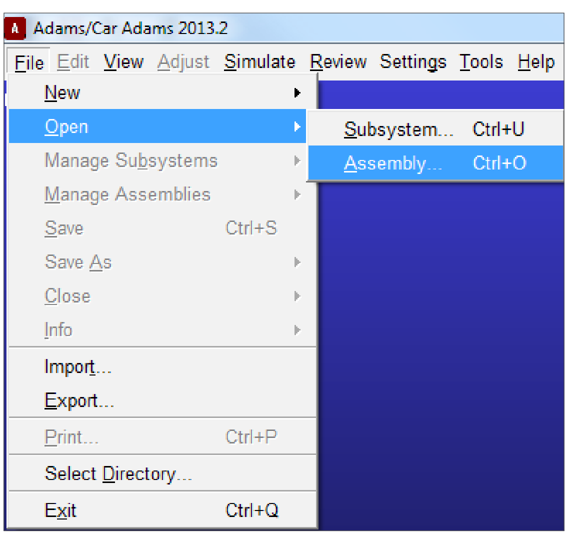

b. Make sure that the Database Alias is set to fsae_2012 and then select the Database Path as the path where the .cdb folder is located.



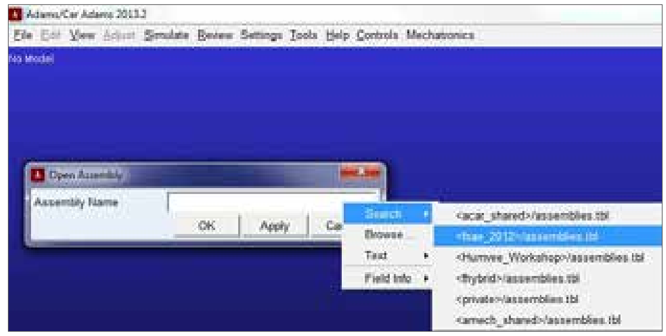

c. Click OK.

## Step 3. Open Full Vehicle Assembly from the FSAE Database.

a. From the toolbar, select File->Open->Assembly.




b. In the Assembly Name field, right click and select Search->fsae_2012/assemblies.tbl.




c. From the database, select fsae_full_vehicle.asy.

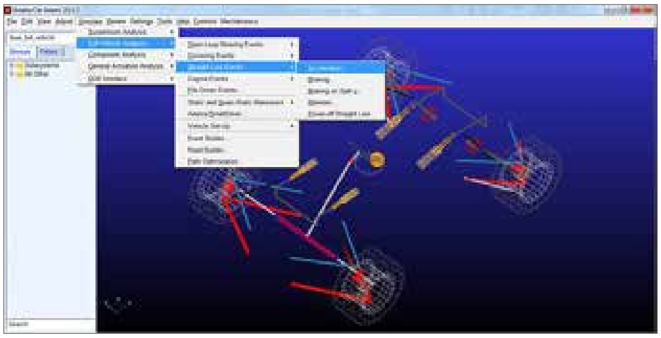

d. Click Open and then OK.

e. You may get error messages when the assembly is opened. It is not a problem because the message simply tells you that there is unassigned communicators. Subsystems talk to each other through communicators. Unassigned communicators means that they are not able to find the input/output.


f. Close the message window and take your time to have a look at the assembly. You can use shift + S to switch between different rendering modes.

## Step 4. Run a Full-vehicle Simulation.

a. From the toolbar click Simulate->Full-vehicle Analysis->Straigh-Line Events->Acceleration. Acceleration event is one of the most basic simulations in Adams/Car. Use this simulation to check if your full-vehicle assembly is working fine and then proceed to more complicated tasks.



b. Before you start your simulation, go to File->Select Directory. Make sure that the working directory where the simulation data are stored is your desired one.

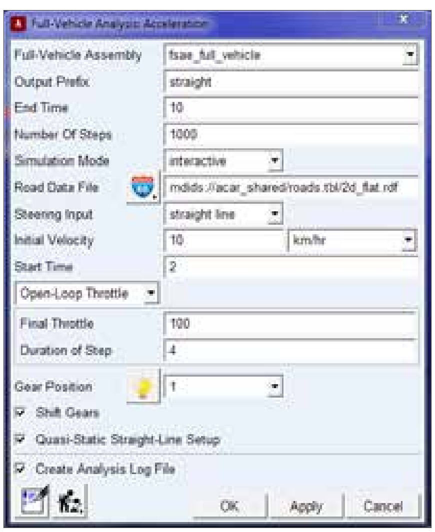
c. Now you can setup the acceleration simulation.

d. Set the output prefix as “Straight”. This will be added to all the files that the simulation creats.

e. Set the end time to 10 seconds and increase the steps to 1000.

f. Continue to set the rest of the parameters as shown. Note that when you are not sure what a particular parameter does, simply hover your mouse over the field and Adams will give you a hint. If no hint appears, click in the field and when the cursor in the current field, press F1.

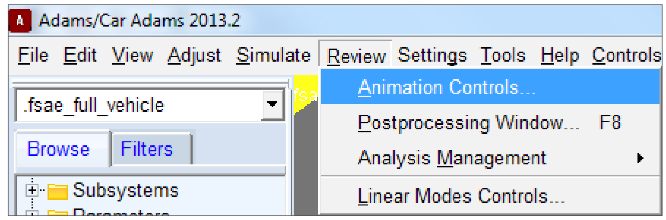
g. Click OK.



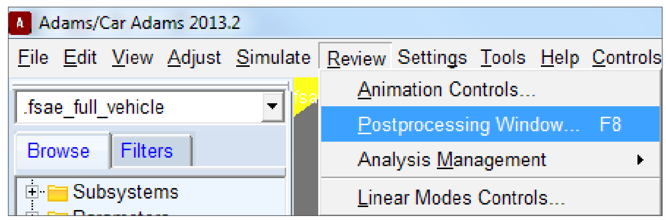

## Step 5. Review Simulation Result.

a. From the toolbar, click Review->Animation Controls.

b. You can now review the animation.




c. From the toolbar, click Review->Postprocessing Window.




d. In the postprocessing window, choose data source as Requests.

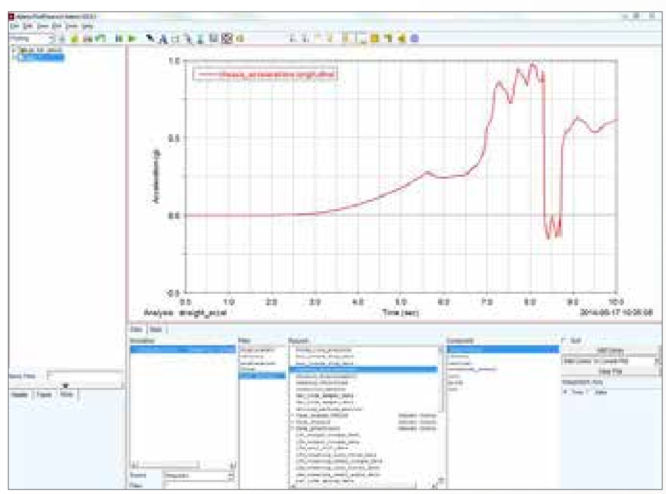
e. Under Filter, pick user defined.

f. Under Request, pick chassis_accelerations.

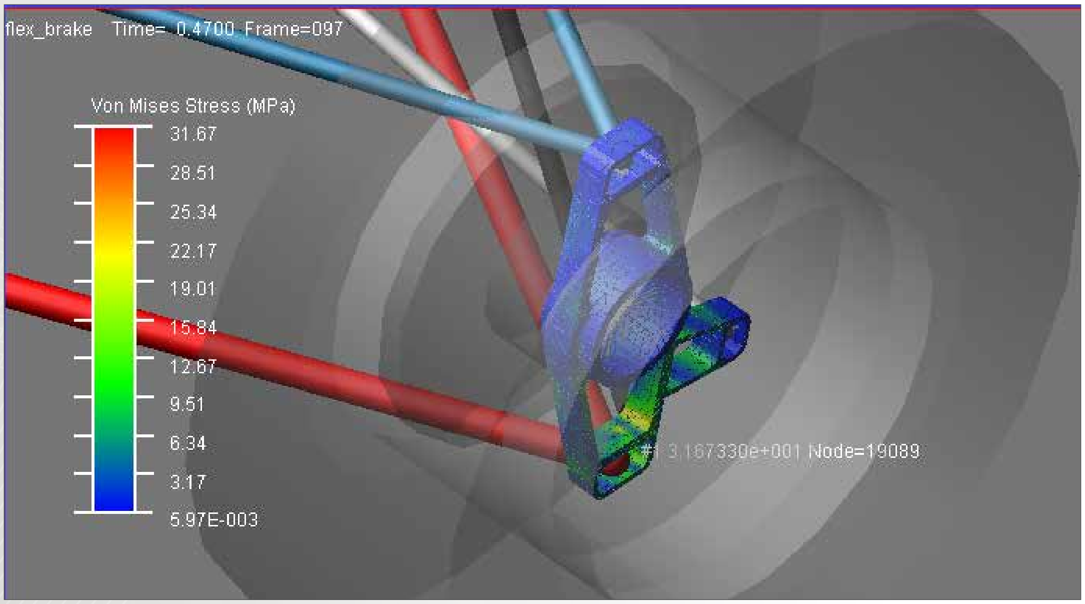
g. Under Component, pick longitudinal.

h. Click Add Curves.

i. You can review other simulation results.



# Example 40: FSAE Flex Body Tutorial




## Software Version

Adams 2013.2

## Problem Description

In this example, we will replace the upright of a Formula SAE race car with a flexible body generated in ViewFlex. After the swap, a braking event will be run and we will find the spot where the stress is maximum during the simulation event.

## Step 1. Start Adams/View.

a. Start **Adams/View 2013.2**.

b. Select **New Model**, and name the model as **flexible_upright**.

c. Select a path of your choice for working directory.

## Step 2. Import Parasolid Model.

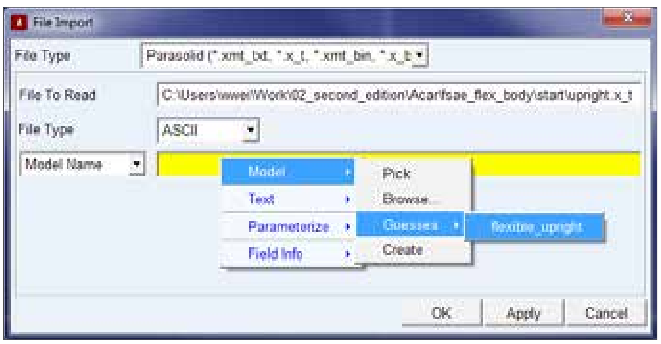
a. From the toolbar, select **File->Import**.

b. Select **Parasolid** as **File Type**.

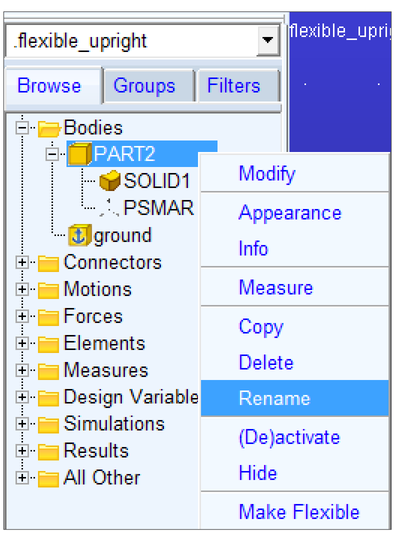
c. Browse to pick **upright.x_t** located in the start folder of this example.

d. Right click in the model name field and pick **flexible_upright**.

e. Click **OK**.



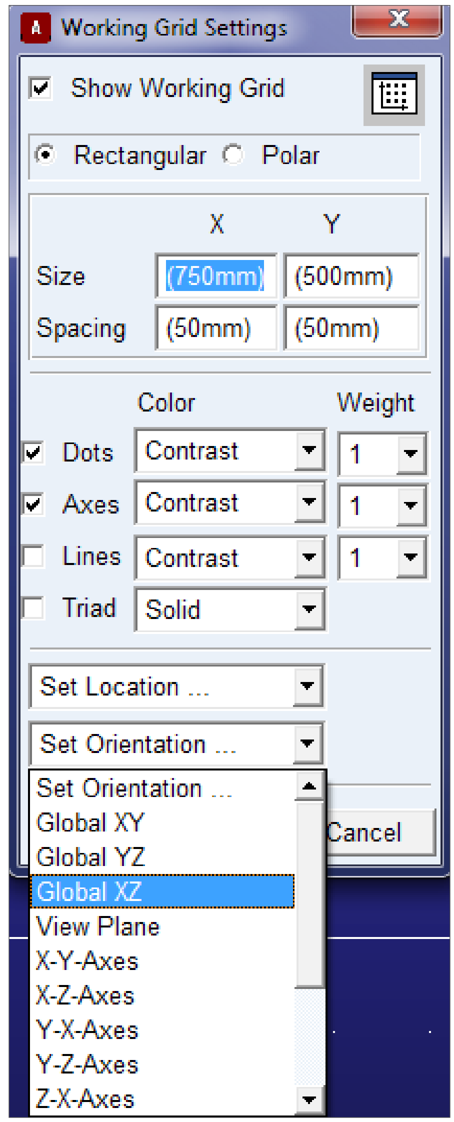

f. Rename **PART_2** as **Upright**.




## Step 3. Create the Flexible Body Mesh.

a. The first step of creating a flexible body is to create all of the attachment points as joints in order for ViewFlex to generate an .MNF file.

b. Select **Settings->Working Grid**. Set the orientation to Global XZ plane.




c. Create fixed joints between upright and ground at the following locations:

| X | Y | Z |
|---|---|---|
| 0 | 0 | 0 |
| 0 | 0 | 115 |
| 0 | 0 | -115 |
| 85 | 0 | -72.6 |

Note that after you have selected upright and ground, right click in the background to invoke location event and type the coordinates. The joints must be created in the same order as the table.

d. Right click on upright and select **Part: upright->Make Flexible**.

e. Select **Create New**.

f. Change the material to aluminum by right clicking the box and selecting **Materials -> Guesses -> Aluminum**.

g. Check **Stress Analysis**. If this is left unchecked, we will not be able to get the stress contour in postprocessor.

h. Check **Advanced Settings** to show the full parameters.

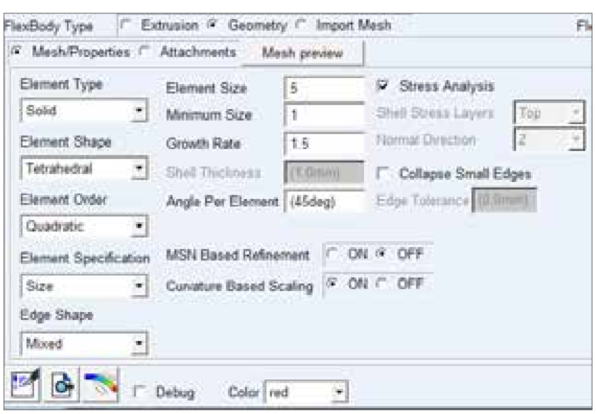
i. Select **FlexBody Type** as **Geometry**.

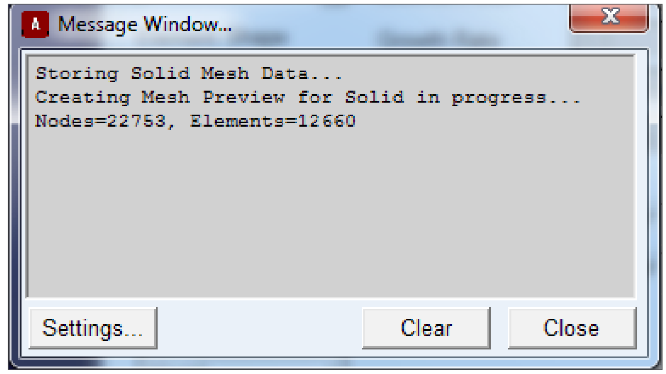
j. Pick **Solid Element Type**, and change **Element Specification** to **Size**, then set **5mm** and **1mm** for Element Size and Minimum Size respectively.

k. Pick **Quadratic Element Order**.

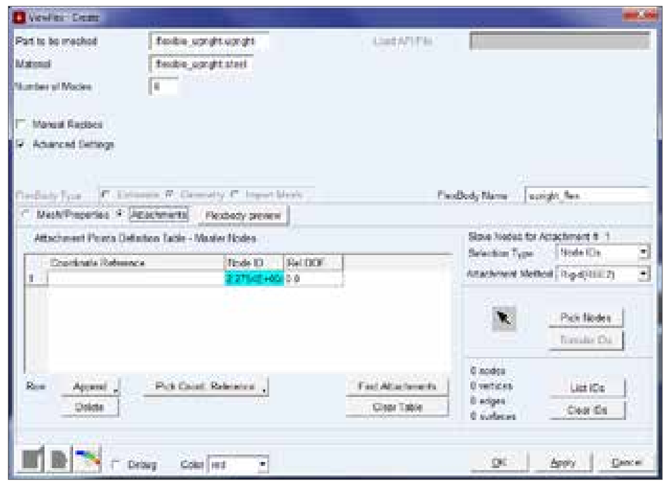
l. Use **Mixed Edge Shape** and turn on **Curvature Based Scaling**.

m. Click **Mesh preview**. A message window will show the number of nodes and elements.




n. Close the message window and select **Attachments**.



## Step 4. Define Flex Body Attachments.

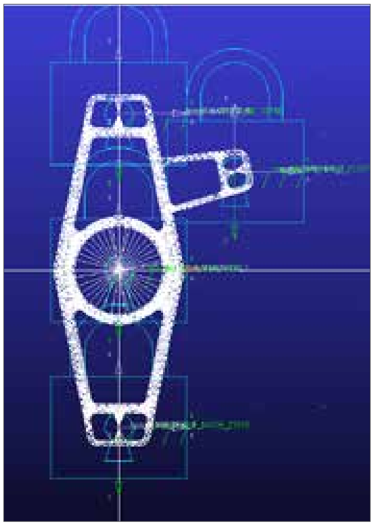
a. Click **Find Attachments**.

b. For **Coordinate Reference 1** (MARKER_2): pick Select Type: **Closest Nodes**, pick Attachment Method: **Compliant (RBE3)**, set Number of Nodes to **40** and then click **Transfer IDs**.

c. For **Coordinate Reference 2** (MARKER_4): pick Select Type: **Cylindrical**, pick Attachment Method: **Rigid (RBE2)**, set Radius to **5**, set End Location to **0, 0, 90**, select **Symmetric**, and then click **Transfer IDs**.

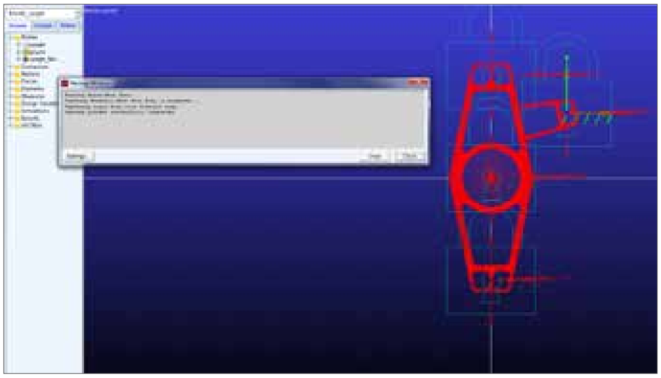
d. For **Coordinate Reference 3** (MARKER_6): Use the same settings as Coordinate Reference 2 except for using **0, 0, -90** as End Location.

e. For **Coordinate Reference 4** (MARKER_8): Use the same settings as Coordinate Reference 2 except for using **96.26, 0, -74.93** as End Location.



f. Select **Flexbody preview** to make sure everything looks as expected.

g. When the upright show up like this, you can close the message window and click **OK**. The creation may take some time depending on the configuration of your machine.



h. Check your working directory and you will find a file named **upright_0.mnf** has been created.

## Step 5. Copy mnf file.

a. Since we need mnf file for both left and right side, we need two mnf file. In our case, as the upright itself is symmetric about its own midplane, we can simply use the same mnf file.

b. Copy and paste upright _0.mnf in the same directory and rename it to **upright_0_2.mnf**.

## Step 6. Start Adams/Car.

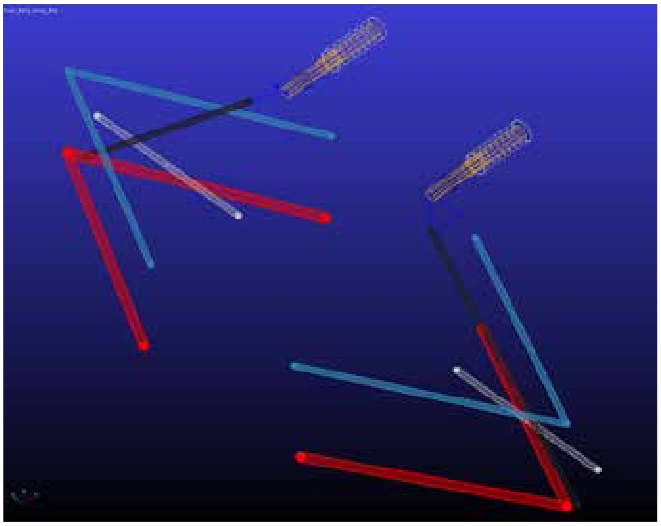
a. Start **Adams/Car** and select **standard interface**.

b. Set the FSAE database as your default writable database by selecting **Tools -> Database Management -> Select Default Writable** and select the fsae_2012 database.

c. Copy and paste the templates _fsae_frontsusp_flex.tpl and _fsae_frontsusp_noflex.tpl to the templates.tbl folder in the FSAE database. Copy and paste the subsystems fsae_front_susp_flex.sub and fsae_front_susp_noflex.sub to the subsystems.tbl folder in the FSAE database. These files have been created to adapt the FSAE front suspension to accommodate the upright design. They are located in the example start folder.

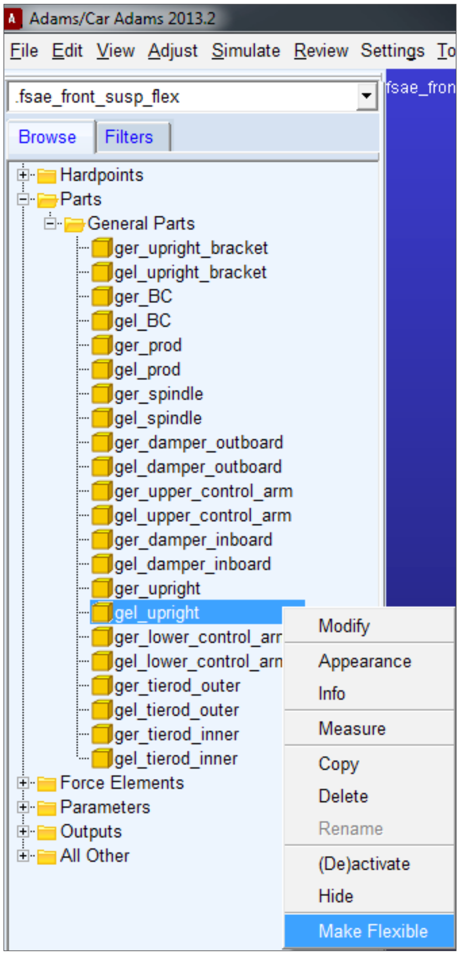
## Step 7. Use the MNF File.

a. From the toolbar, select **File->Open->Subsystem. Browse** to fsae_2012 database and pick **fsae_front_**



**susp_flex.sub**.

b. From the model tree, expand **Parts->General Parts** and right click on **gel_upright** and then select **Make Flexible**.



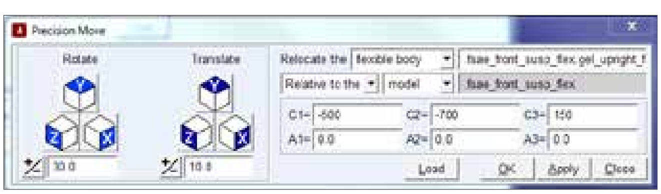

c. Select **Import MNF**.

d. Right click in **Current Part** field and select **gel_ upright**.

e. Right click in **MNF File** field and browse for **upright_0. mnf**.

f. Click **Launch Precision Move Panel**.

g. Enter **-500, -700, 150** for C1, C2 and C3 to bring the

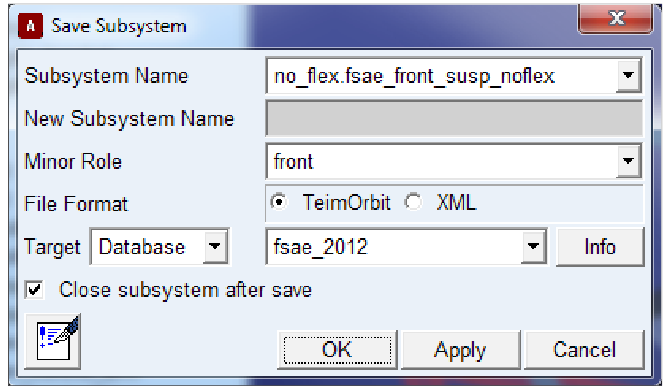



upright to position.

h. Click **OK**.

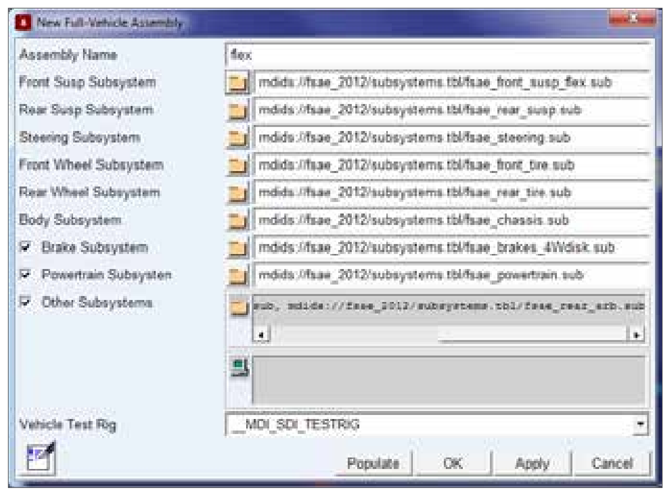
i. Now do the same for the right upright but using the upright_0_2.mnf file and making **C2=700**.

j. Save the subsystem with a **minor role: front**.




## Step 8. Create New Full-Vehicle Assembly.

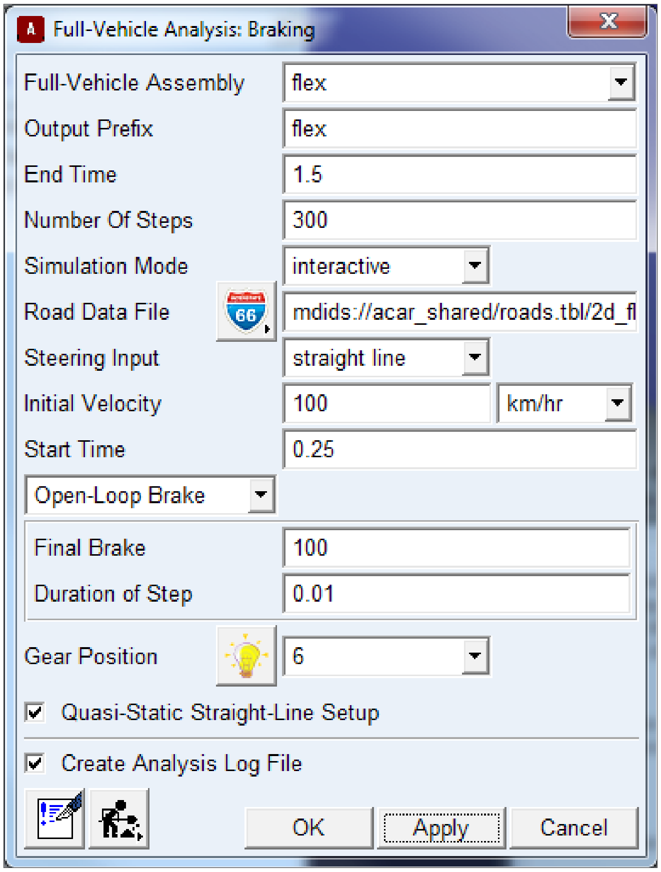
a. Select **File -> New -> Full-Vehicle Assembly** and create the following assemblies.




## Step 9. Run Full_Vehicle Simulations.

a. We will now look at the effect of flexible uprights on a harsh braking event.

b. Select **Simulate -> Full-Vehicle Analysis -> Straight-Line Events -> Braking** and run the two events shown below:




## Step 10. View Hot Spots Table.

a. Switch to the Post-Processor by pressing F8.

b. If you haven't loaded Durability Plugin, go to **Tools->Plugin Manager** and then check Adams/Durability.

c. Go to **Durability->Hot Spots Table.** Hot Spots Table enables us to investigate the nodes where the stresses are maximum.

d. Right click at **Flex Body** and then go to **Flex_Body->Guesses->gel_upright_flex**.

e. Right click at **Analysis** and select **flex_brake**.Set Type to **Von Mises Stress**.

f. Leave the rest as default and click **Report**.

| Model= flex fsae_front_susp_flex | | Analysis= flex_brake | | Time = 0 to 1.5 sec | | |
|---|---|---|---|---|---|---|
| Top 10 Hot Spots | | | Abs | Radius= 0.0 mm | | |
| Hot Spot | Stress | Node | Time | Location wrt LPRF (mm) | | |
| # | (newton/mm**2) | id | (sec) | X | Y | Z |
| 1 | 31.6733 | 19089 | 0.285 | 5.8831 | -2.17259 | -105 |
| 2 | 23.3033 | 22250 | 0.285 | 18.7112 | 5.48705 | -97.1135 |
| 3 | 23.0585 | 22256 | 0.285 | 17.538 | -3.74435 | -97.7696 |
| 4 | 22.8461 | 22248 | 0.285 | 18.7219 | -2.48669 | -97.1049 |
| 5 | 21.9769 | 22281 | 0.285 | 20.1169 | -5.002 | -94.8116 |
| 6 | 21.9498 | 22251 | 0.285 | 19.3506 | 7.62673 | -96.4646 |
| 7 | 21.0723 | 22249 | 0.285 | 18.7176 | 2.16893 | -97.1084 |
| 8 | 20.9825 | 18568 | 0.285 | 20.1169 | -10 | -94.8116 |
| 9 | 20.3536 | 22164 | 0.285 | 7.13313 | 1.0245 | -98 |
| 10 | 20.1382 | 16823 | 0.285 | 5.09887 | 1.10649 | -129 |

g. The top 10 hot spots are summarized in a table. The node having the maximum stress is node No. 19089.

h. Click Close.

## Step 11. View Stress Contour during Simulation

a. Right click at plotting page and select **Load Animation**.

b. From the Database Navigator, double click **flex_brake**.

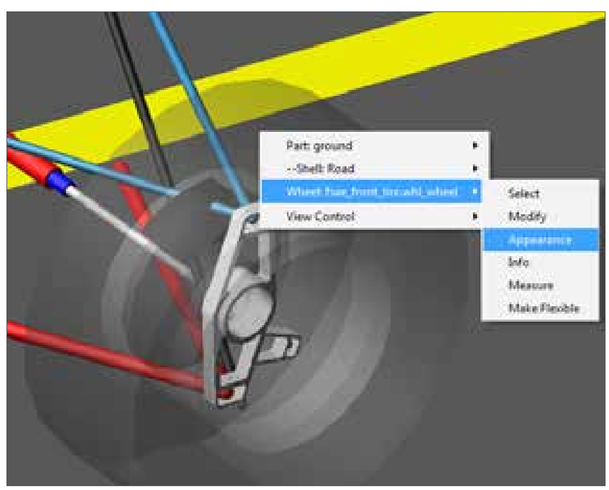
c. Click on **Contour Plots** tab. Change Contour Plot Type from Deformation to **Von Mises Stress**.

d. Click on **Camera** tab.

e. Right click at **Follow Object** and go to **Flexible_Body->gel_upright_flex**.

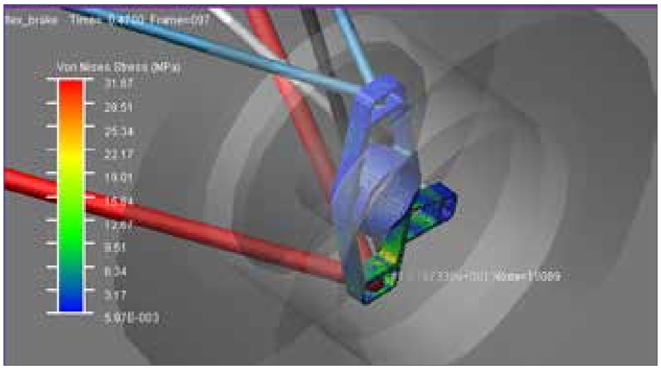
f. Click on **Hot Spots** tab and check **Display Hotspots, Rank, Value** and **Node ID**. This will display the maximum stress point in the animation with its value and node ID beside it.

g. Now switch back to Adams/Car window and right click at the left front wheel. In **Appearance** settings,



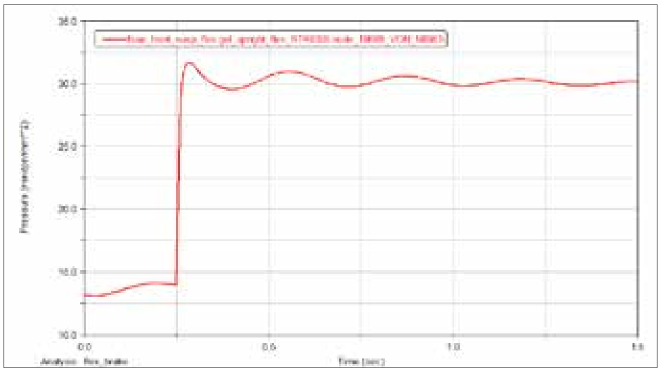
change the **Transparency** to **80**.

h. Switch to postprocessor and press Shift + I to enter ISO view. Press W and drag the mouse to enlarge the left front wheel assembly.
i. Your window should be similar to the figure below.



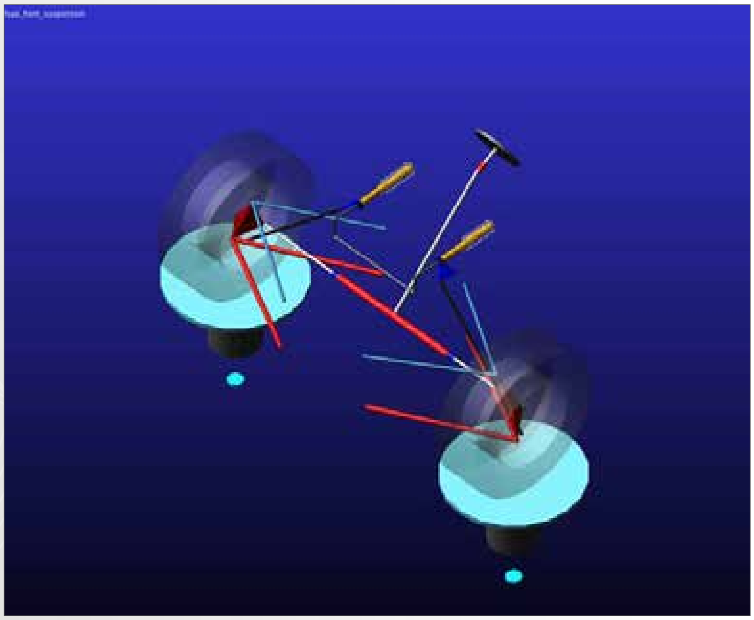

## Step 12. Plot the Time History

a. It is helpful to know the moment when the stress at the hot spot is at its maximum.
b. Go to **Durability->Nodal Plot**.
c. Right click at **Flexible Body** and go to **Flexible_ Body->Guesses->gel_upright_flex**.
d. Enter 19089 at select Node List. This is the No.1 hot spot.
e. Click OK.
f. Create a new page.
g. Make sure the **Source** is set to **Result Sets**. Enter ***flex*** at the Filter.
h. Double click **fsae_front_susp_flex** on the right and highlight **gel_upright_flex_stress**.
i. Highlight **node_19089_Von_MISES** and click **Add Curves**.



j. There is a sharp increase of stress at the initiation of brake action.

# Example 41: Adams/Insight Example



## Software Version

Adams/Car 2013.2

## Problem Description

Use Adams/Insight to arrange suspension hard points in order to achieve desired roll center height. Note: you should have finished the first example in this section to proceed.

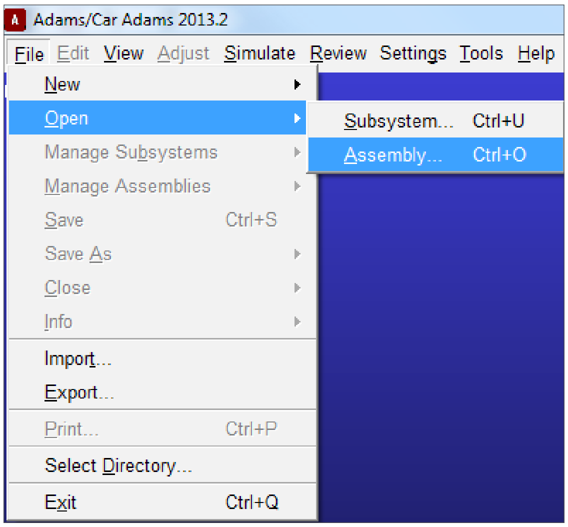
## Step 1. Start Adams/Car.

a. Start Adams/Car 2013.2.

b. If you are prompted to select user interface, select **Standard Interface**.

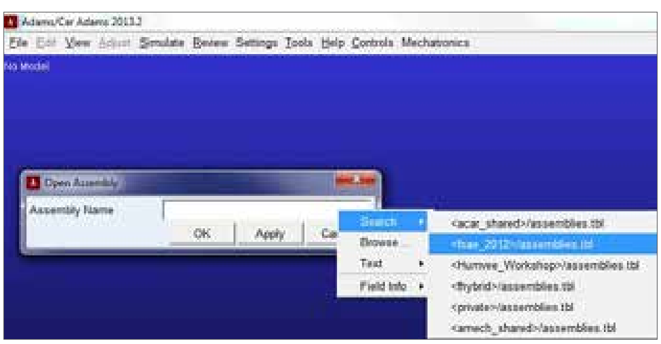
## Step 2. Open Front Suspension Assembly from the FSAE Database.

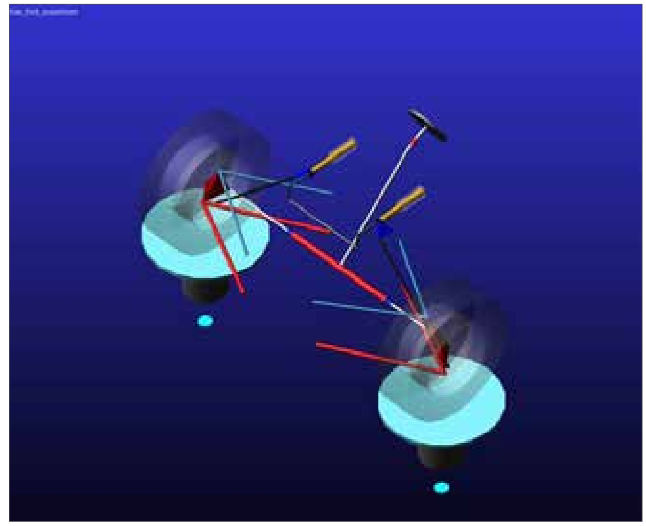
a. From the toolbar, select **File->Open->Assembly**.




b. In the Assembly Name field, right click and select **Search->fsae_2012/assemblies.tbl**.

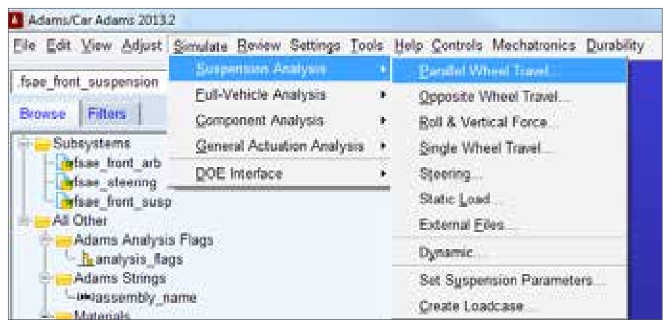
c. From the database, select **fsae_front_suspension.asy**



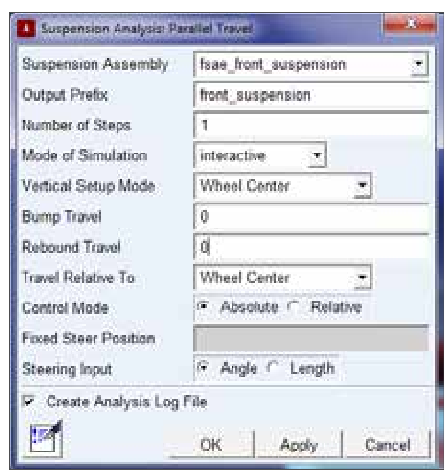



## Step 3. Run a Trial Parallel Travel Suspension Analysis.

a. From the toolbar, select **Simulate->Suspension Analysis->Parallel Travel**.



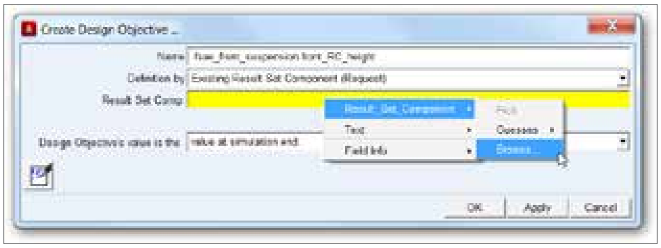

b. Set up a simulation with zero bump and rebound travel with 1step. This helps us to know if the suspension is at design postion.

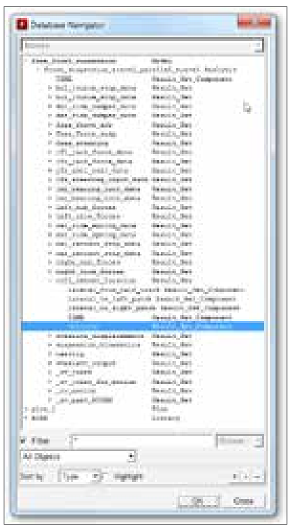


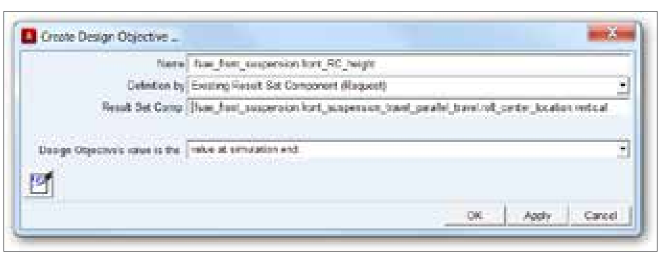
## Step 4. Create a Design Objective.

a. From the toolbar, select **Simulate -> DOE Interface -> Design Objective -> New**.

b. Change the name from OBJECTIVE_1 to **front_RC_height**. Right click in the field of Result Set Comp. and then browse for **roll_center_location/vertical**.







c. Change “Design Objective’s value is the” to **value at simulation end**.

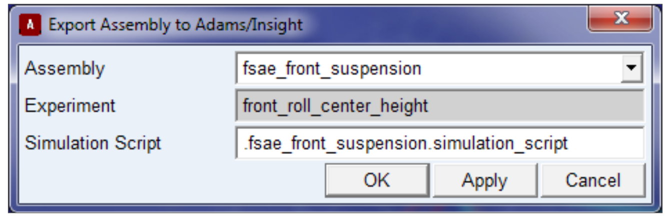
## Step 4. Export the Simulation to Adams/Insight.

a. Click **Simulate -> DOE Interface -> Adams/Insight -> Export**.

b. Change Experiment name to **front_roll_center_height**.

c. Change Simulation Script to **simulation_script**.

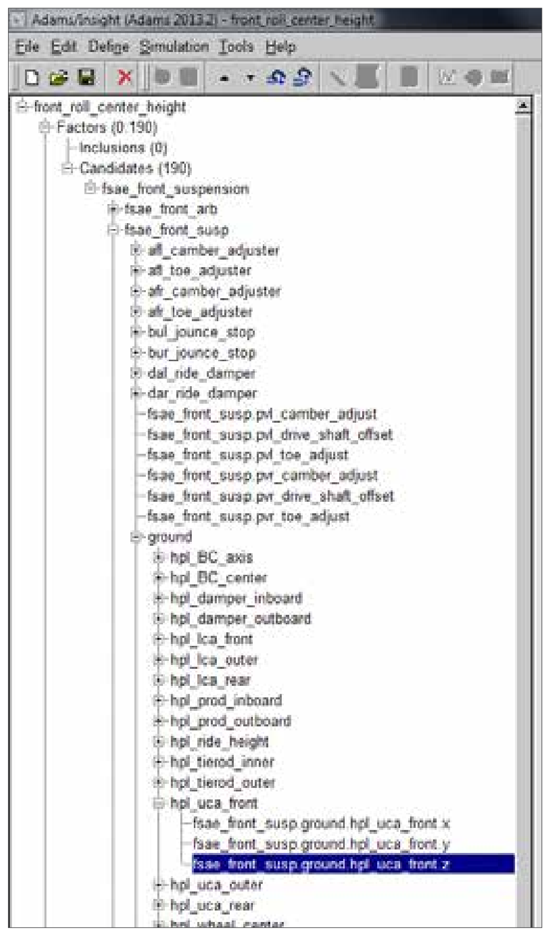
d. Click **OK**.




e. Now Adams/Car will close and Adams/Insight will open with a new experiment called front_roll_center_height.

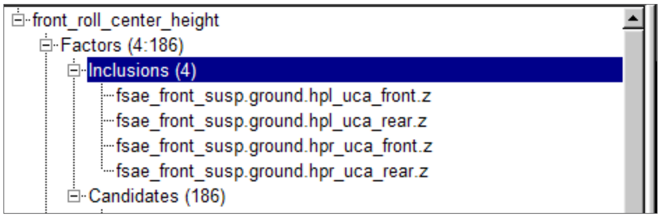
## Step 5. Promote Hardpoints to Factor Inclusions.

a. Expand the model tree to locate the candidates: **hpl_uca_front/rear.z, hpr_uca_front/rear.z**.

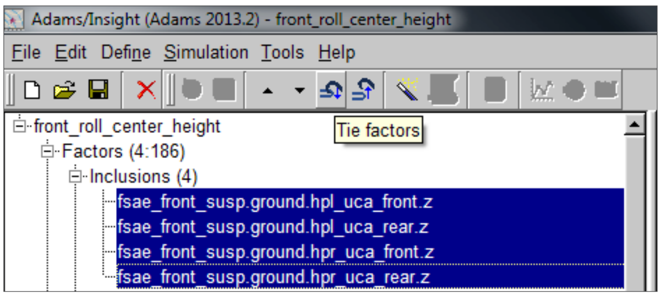


b. After selecting each one, press the **promote** button:

c. The final model tree will look like this.




d. Highlight all of the points and press the **tie** button.

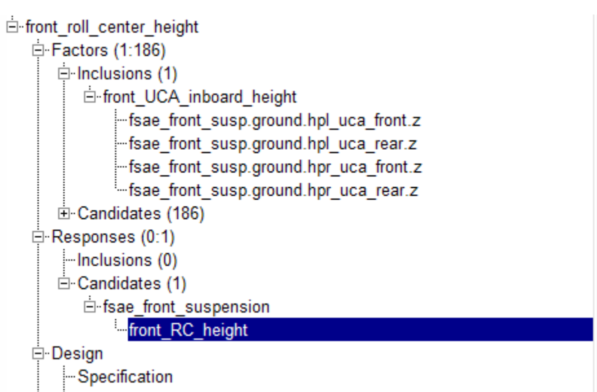



e. Highlight **Tie_01** and change the settings as follows. Keep the original nominal value and set the range

(settings) to -100,100. Pick Relative Percent as Delta Type. Click Apply.

## Step 6. Promote RC Height to Response Inclusions.

a. Locate **front_RC_height** in the model tree.

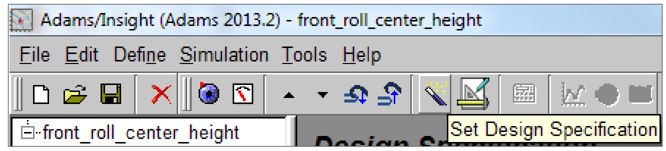



b. Promote it the same way as previous step.

c. Change the Abbreviation to front_RC_height. And then click **Apply**.

## Step 7. Set Design Specification.

a. Click Set Design Specification.




b. Under Investigation Strategy, pick **Sweep**.

c. Under Model, pick **Cubic**.

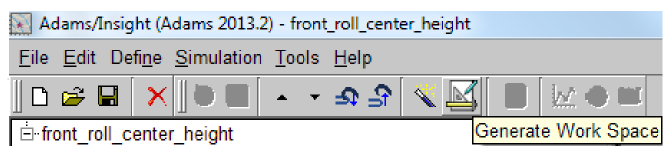
d. Under Run Order, pick **Standard**.

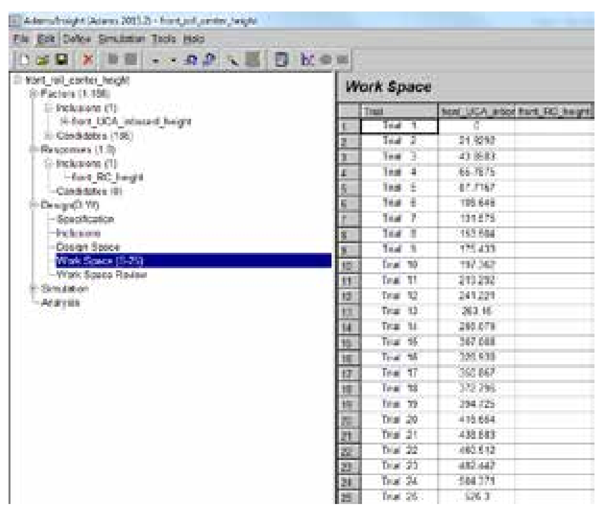
e. Use 25 steps.

f. Click **Apply**.

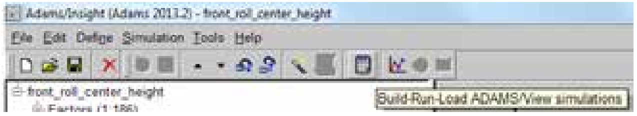
## Step 8. Generate Workspace and Run.

a. Click **Generate Workspace.**

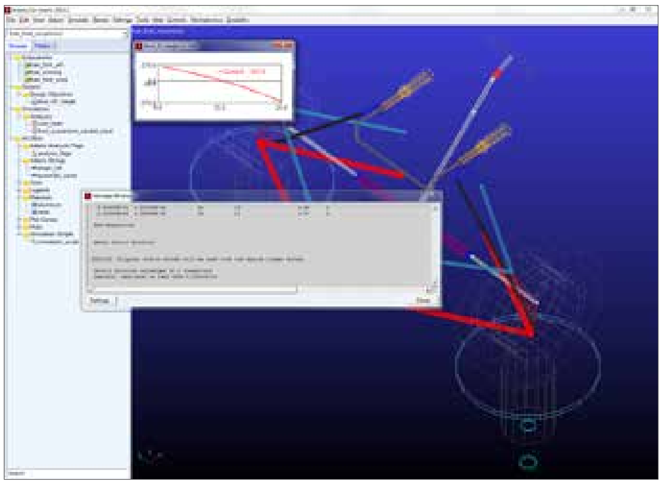


b. Run the design study.



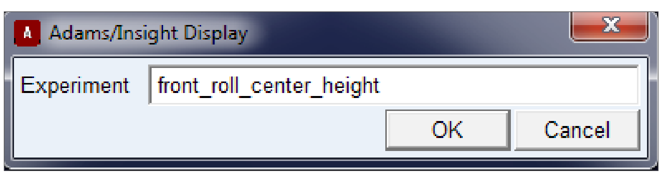

c. Now we switch to Adams/Car again and as the simulation running, you can observe the inboard hardpoints of upper control arms changing with each



iteration.

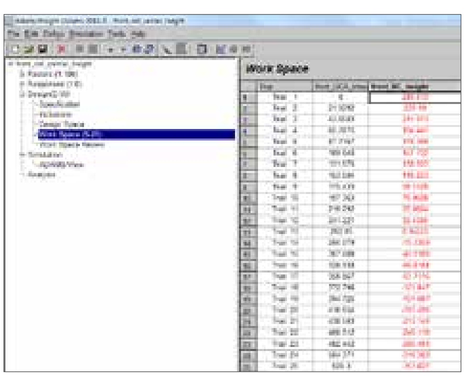
## Step 9. View DOE Results.

a. Go to **Simulate->DOE Interface->Adams/Insight->Display**.

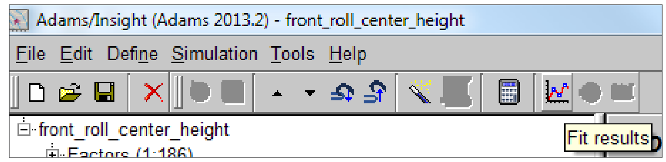



b. Fill in **front_roll_center_height**.

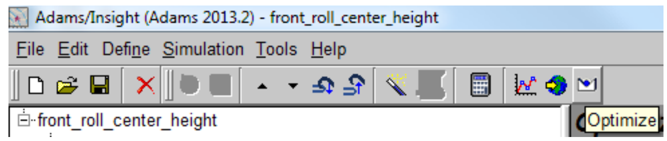
c. Highlight Work Space and notice that the column of



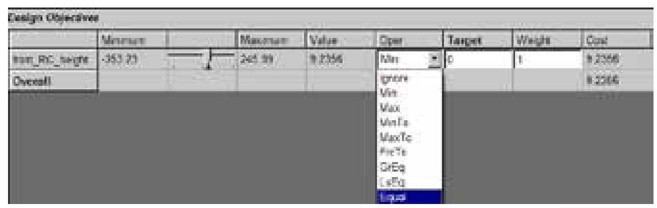

front_RC_height has been filled.




## Step 10. Fit and Optimize Results.




a. Click Fit **Results**.



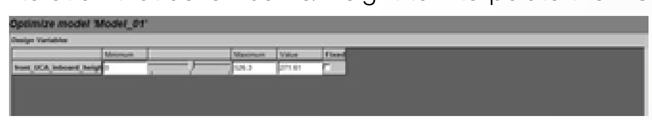

b. Click **Optimize**.

c. Under Design Objectives, change Oper to Equal.

d. Keep Target as 0.

e. Click Run.

f. In Design Variables window, notice that column Value has changed to 271.61. Step 10c to Step 10e is an iteration that asks Adams/Insight to interpolate the DOE




result and find the height of UCA inboard hardpoints that will give us a roll center height of 0.

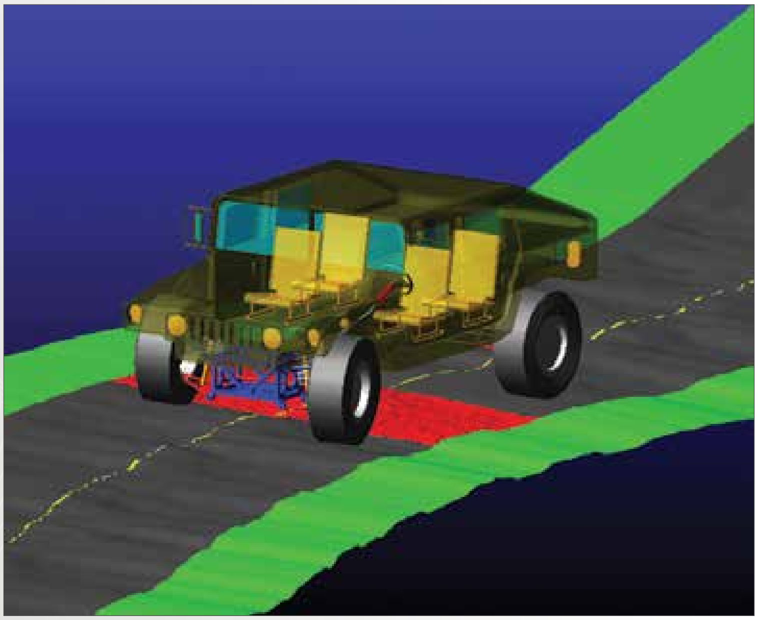
g. Keep changing the value of Target and record how the value of hardpoint height changes. You should get the following results.

| Roll Center Height: | UCA_Inboard_Height: |
|---|---|
| 0 | 271.61 |
| 25 | 248.5 |
| 50 | 224.49 |
| 75 | 199.5 |
| 100 | 173.5 |
| 125 | 146.44 |

| | |
|---|---|
| 150 | 118.28 |
| 175 | 89.018 |
| 200 | 58.643 |
| 225 | 27.19 |

h. Notice that these values are the result of fitting the DOE result. As the problem gets more non-linear, the interpolated result is not necessarily exact.

# Example 42: Humvee Durability Analysis



## Software Version

Adams/Car 2013.2

## Problem Description

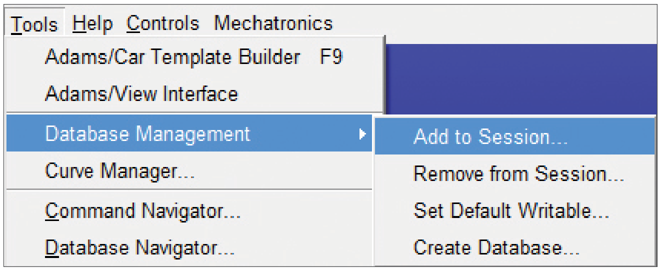
Find the load history on the front subframe of a Humvee cruising across a rough road.

## Step 1. Add Database to Session.

a. Start Adams/Car 2013.2.

b. Choose Standard user interface.

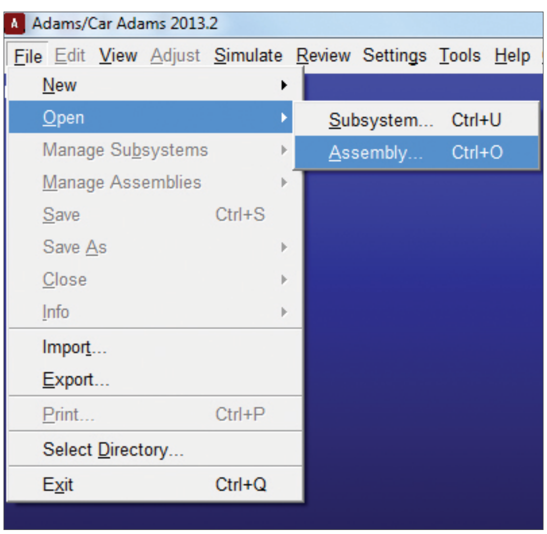
c. From the tool bar, click **Tools->Database Management->Add to Session**.

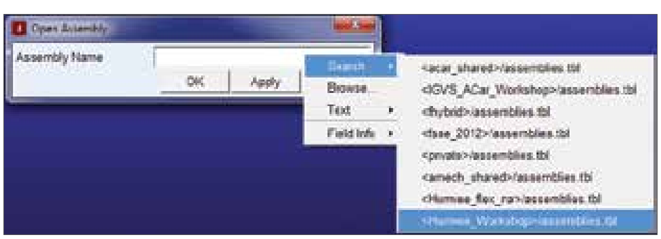



d. Type **Humvee_Workshop** as Database Alias and choose the location of .cdb folder as the Database Path.


## Step 2. Open Assembly.

a. From the tool bar, click **File->Open->Assembly**.



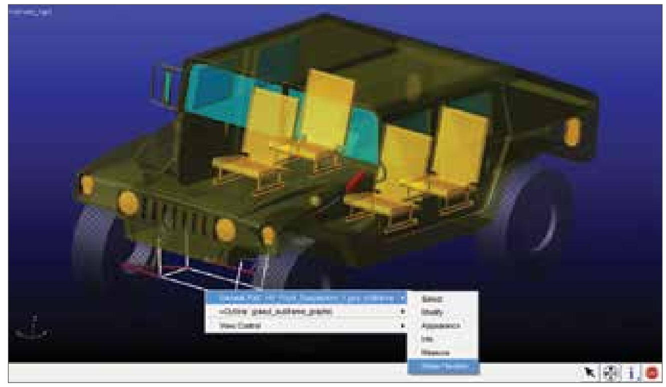

b. Right click in the field and search from the database added in previous steps.

c. Select **Humvee_Rigid.asy**.

d. Click **Open** and then click **OK**.

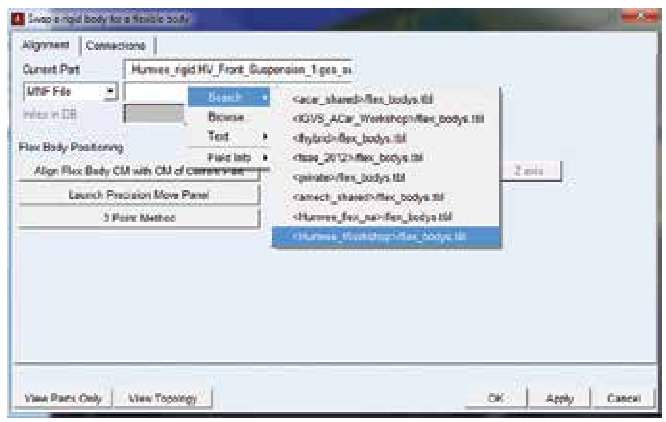
## Step 3. Replace Subframe with a Flexible Part.

a. Right click on the subframe and select **Make flexible**.



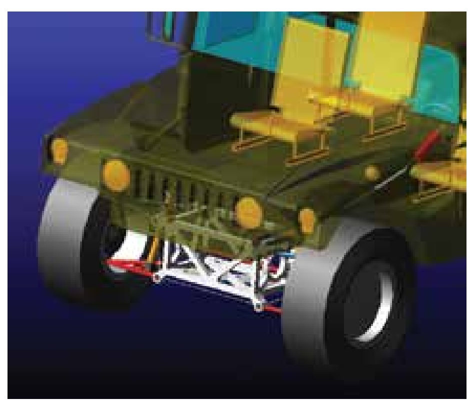
b. Then choose **Import MNF**.

c. Right click to search the database and select the MNF file- **HV_demo_subframe_front.mnf**




d. Click **Align Flex Body CM with CM of Current Part**.

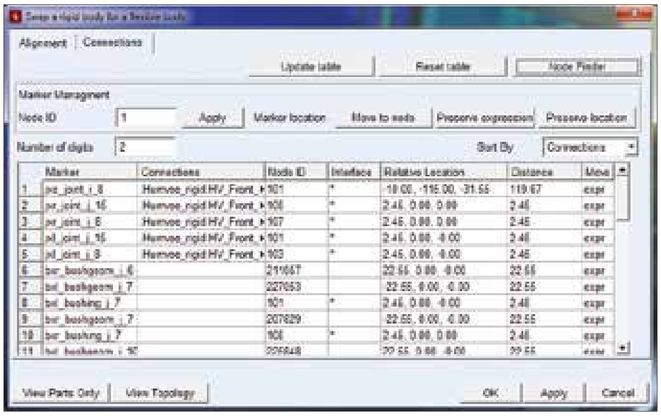
e. The flex body is now aligned with the subframe.



f. Select **Connections** tab.

g. It is found that the connection has already been done in Adams.

h. Click **OK**.

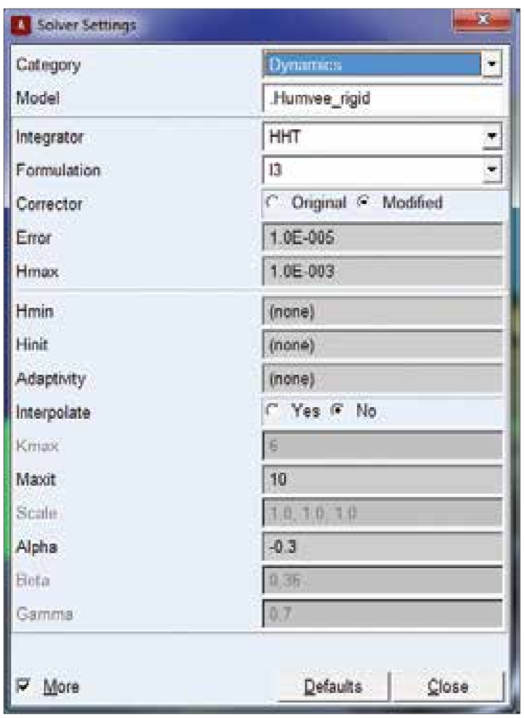


## Step 4. Configure Solver Settings

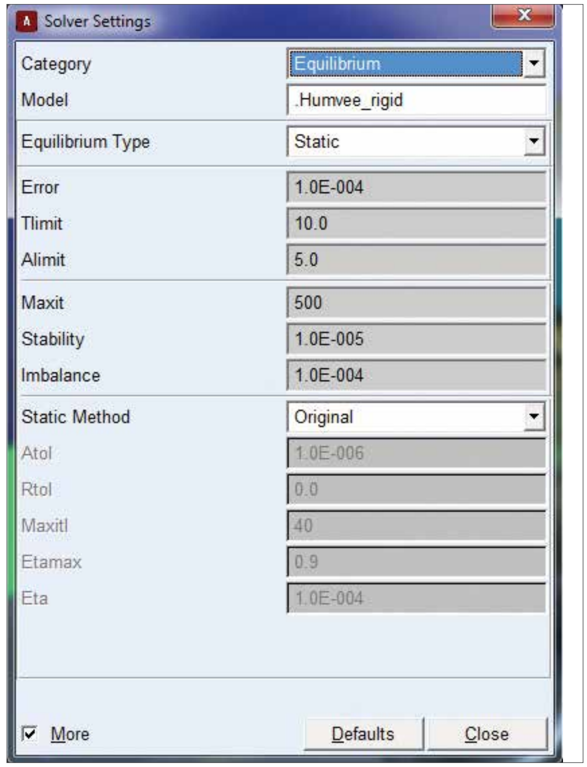
a. From the tool bar, click **Settings->Solver**.

b. From **Category**, select **Dynamics** and choose **HHT** as Integrator. Leave the rest of the settings as default.

c. From **Category**, select **Equilibrium** and reduce



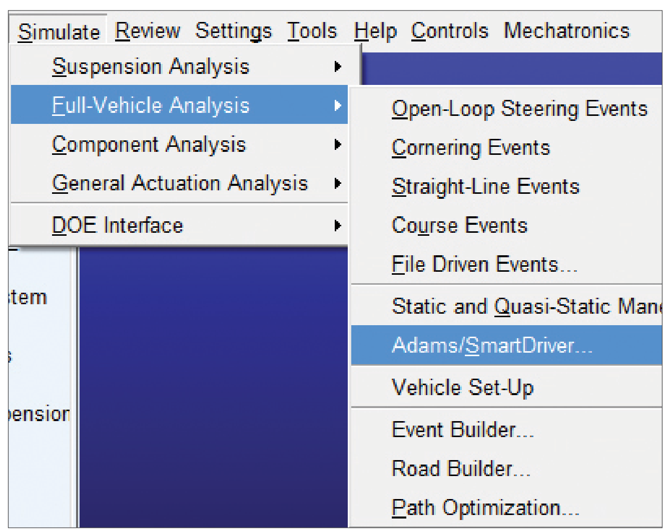

**Tlimit** and **Alimit** to **10** and **5** respectively. This would help to find equilibrium but may take more iterations. So change the **MAXIT** to **500** as well.




d. Click **Close**.

## Step 5. Run an Adams/SmartDriver Event.

a. From the tool bar, click **Simulate->Full-Vehicle Analysis->Adams/SmartDriver**.



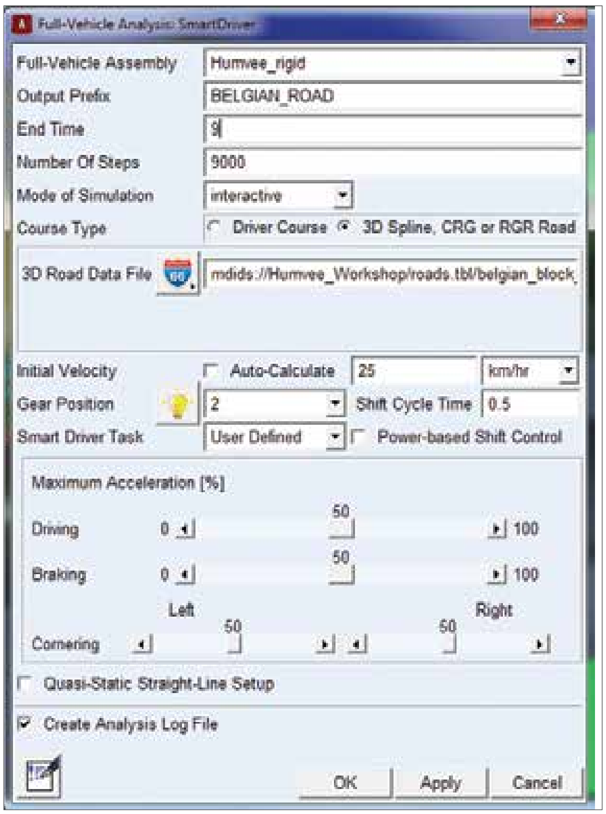
b. Set the **End** time to **9** and output **9000** steps.

c. Then choose **3D spline** for **course type**. For the road data file, search from the current database and select **Belgian_block_extended_2.crg**

d. Use **25 km/hr** as **initial velocity** and choose 2nd gear.

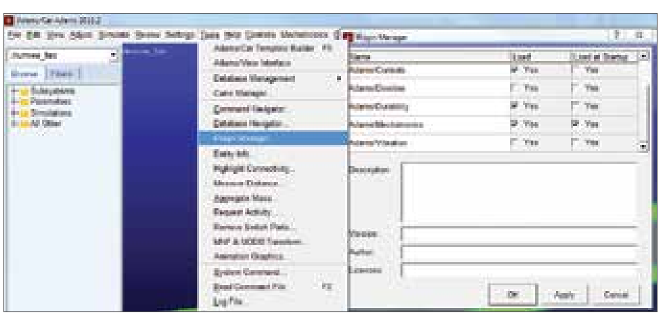
**e. Uncheck** Quasi-Static Straight-Line Setup.

f. Click **OK**.

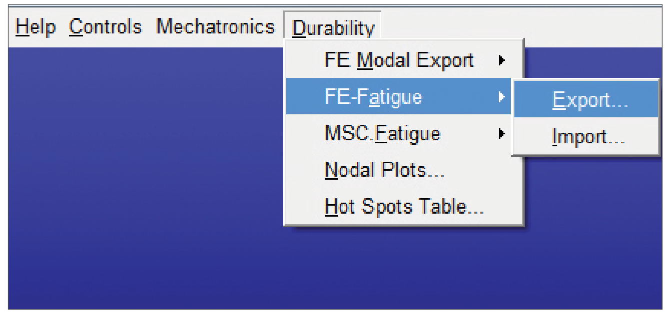



## Step 6. Export.DAC File.

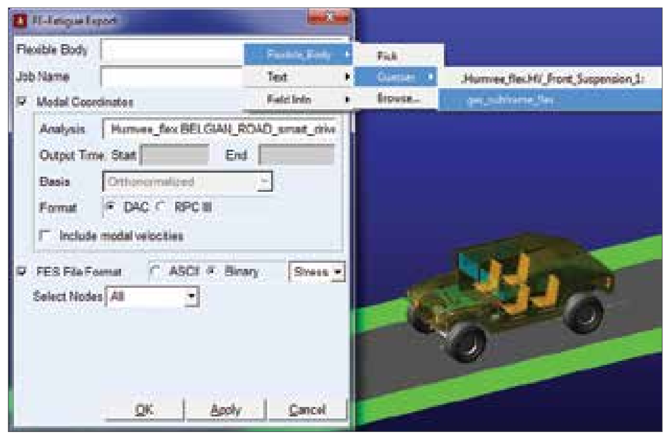
a. From the tool bar, click **Tools->Plugin Manager**, then check **Adams/Durability**. Click **OK**.



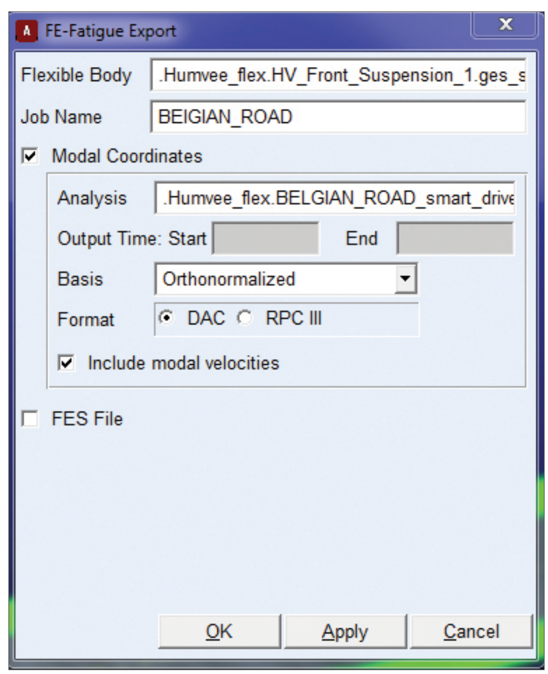
b. From the tool bar, click **Durability->FE-Fatigue->Export**.




c. Choose the flex body in the assembly.

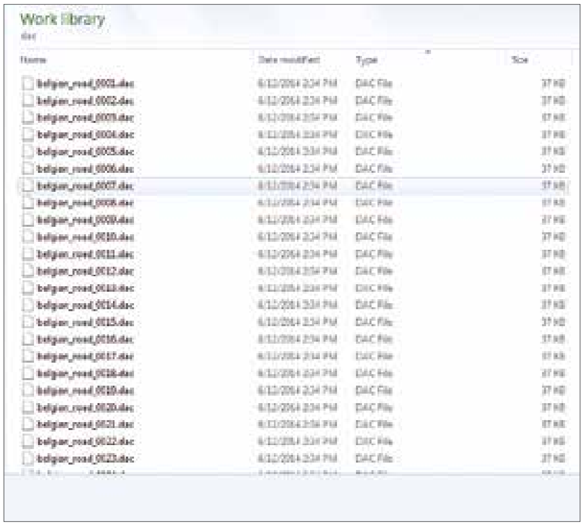


d. a. Uncheck FES file, then click **OK**.

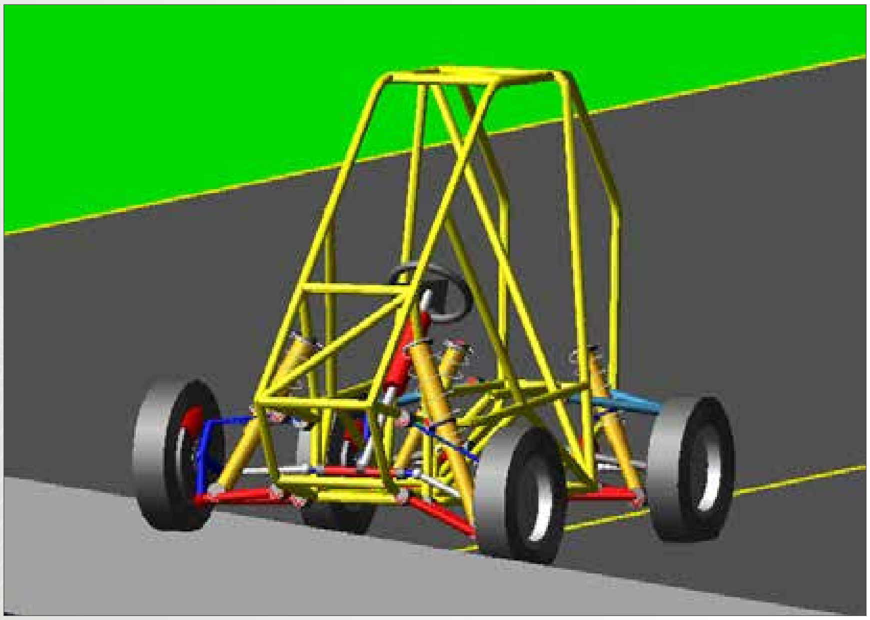


e. Now in your working directory, you should be able to find a list of .dac files.

f. Now the files are ready for fatigue analysis.



# Example 43: Mini Baja Example



## Software Version

Adams 2013.2

## Problem Description

In previous examples, we have studied Formula SAE race cars using Adams/Car. Another popular racing competition among students is the Baja SAE Series. Unlike Formula SAE, Baja race cars run on rough roads and hill climbs. Thus, Baja teams are faced with different problems. In this example, we will show how to use Adams to create flexible bodies and incorporate them in the simulation. This tutorial is intended for intermediate to advanced Adams user. Please finish basic examples of Adams/View and Adams/Car before you proceed.

## Step 1. Start Adams/View 2013.2

a. Start Adams/View 2013.2.

b. Select Existing Model and browse for UCA_Flex_start.cmd.

c. Click OK.

d. Expand the model tree to locate the ground points.

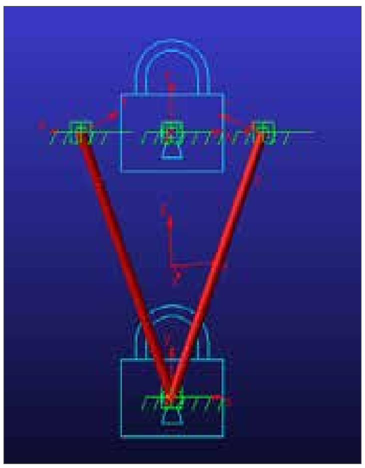
## Step 2. Create Fixed Joints

a. We now need to create all of the attachment points as joints in order for ViewFlex to generate an MNF file.

b. From Connectors ribbon, select Joints: Fixed.

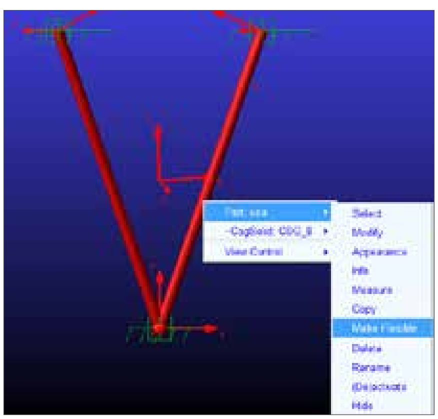
c. Click on uca and then ground. Click on Point_3 to set the location.

d. Create fixed joints at Point_4 following the same procedure.



## Step 3. Create Flexible Body-Meshing

a. Right click on part: uca and select Make Flexible.

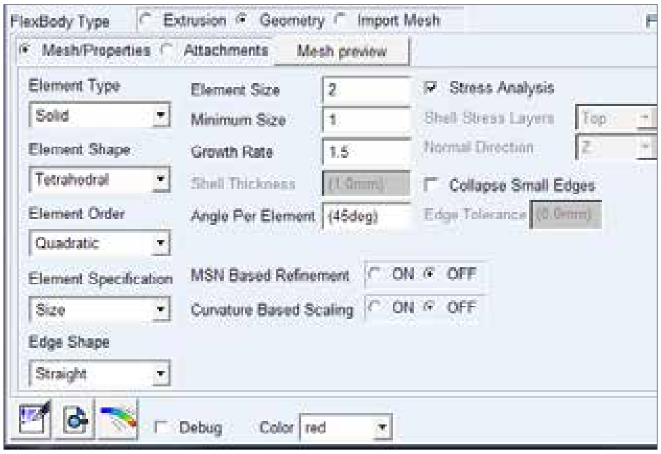


b. Select Create New.

c. Make sure the material is set as steel and then check Stress Analysis and Advanced Settings.

d. Make the following setups.

| | |
|---|---|
| Element Type | Solid |
| Element Shape | Tetrahedral |
| Element Order | Quadratic |
| Element Specification | Size |
| Edge Shape | Straight |
| Element Size | 2 |
| Minimum Size | 1 |

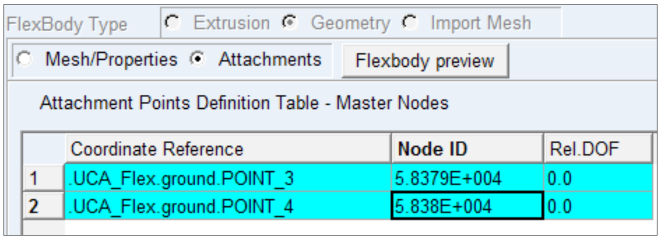



e. Click on Mesh Preview. It can take a while depending on the configuration of your machine.

## Step 4. Create Flexible Body-Attachements

a. When the mesh is generated, click on Attachments.

b. Click on Find Attachments.

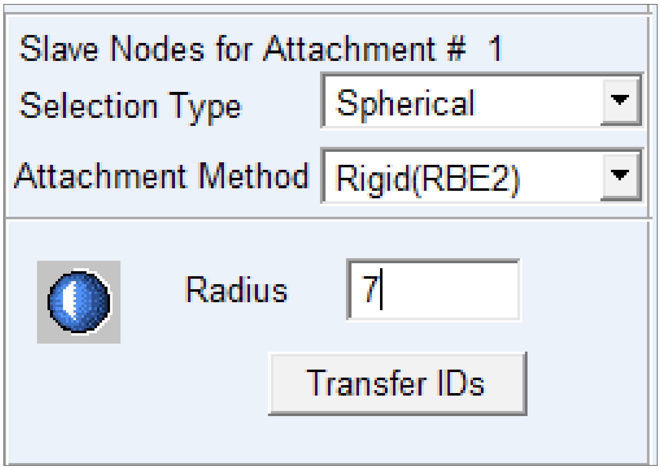



c. Adams will automatically find the attachment points we defined previously.

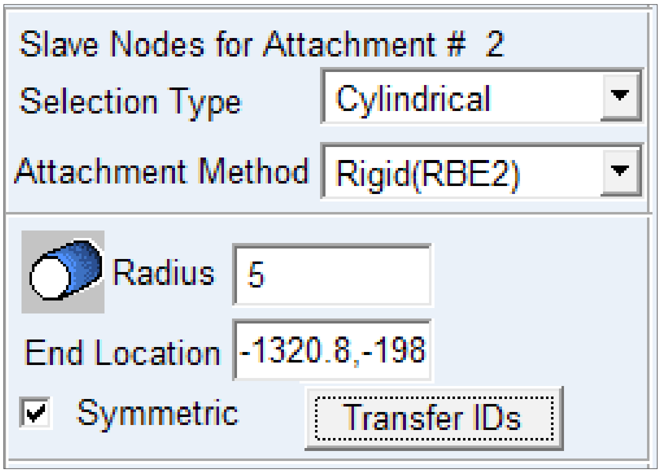
d. Click on Coordinate Reference 1.

e. Change Selection Type to Spherical.

f. Set Attachment Method as Rigid (RBE2).
g. Set Radius to 7.
h. Click Transfer IDs.



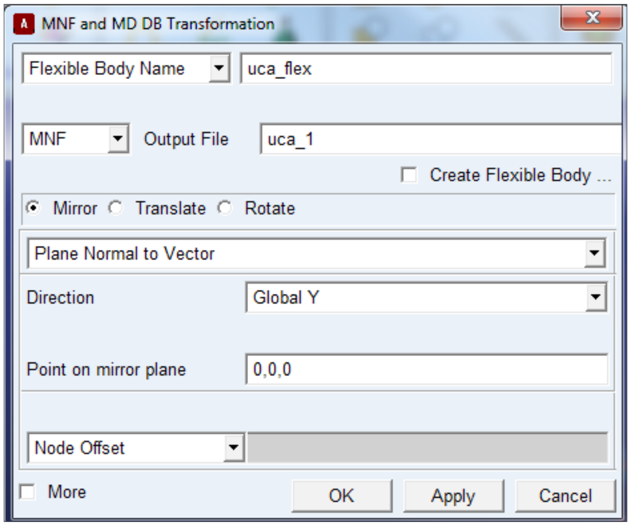

i. Click on Coordinate Reference 2. Change Selection Type to Cylindrical.
j. Set Attachment Method as Rigid (RBE2).
k. Set Radius to 5.
l. Enter -1320.8, -198.12, 353.86 at End Location.
m. Check Symmetric
n. Click on Transfer IDs.




o. Click on OK. The final generation of MNF File may take a while.
p. There should be a file named uca_0.mnf created in your working directory.

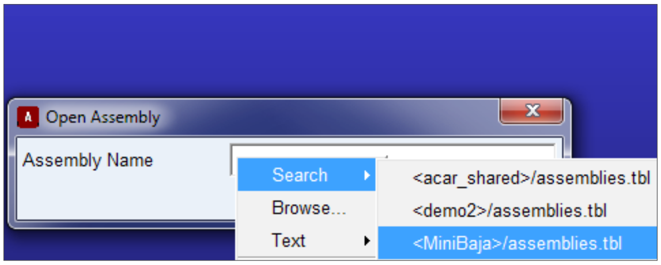
## Step 5. Create a Mirror Mesh

a. Since there are a set of control arms, it is necessary to create a mirror of the current mesh.
b. From Bodies ribbon, select Flexible Bodies: MNF XForm.
c. Right click at Flexible Body Name and go to Flexible_ body->Guesses->uca_flex.
d. Enter uca_1 at Output File and uncheck Create Flexible Body.
e. Select Mirror: Plane Normal to Vector.
f. Select Global Y as Direction.
g. Enter 0,0,0 at Point on mirror plane.
h. Click OK.
i. At your working directory, a new file named uca_1.mnf has been created.




## Step 6. Load MiniBaja Database.

a. Start Adams/Car 2013.2.
b. Load the MiniBaja database using the steps in the first example of this section.
c. Go to File->Open->Assembly.
d. Right click at Assembly Name and search in <MiniBaja>/assemblies.tbl.

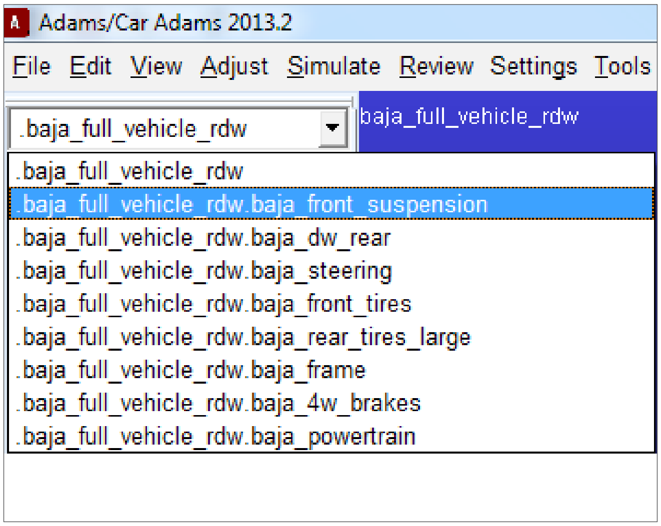


e. Double click on baja_full_vehicle_rdw.asy.

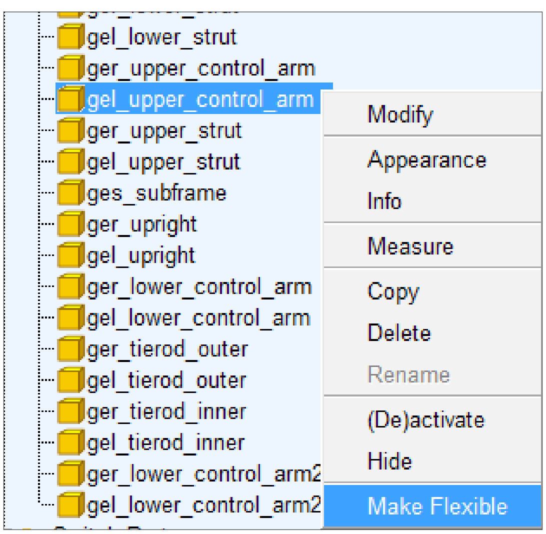
f. Click OK.

## Step 7. Replace Rigid Parts with Flexible Parts

a. When the assembly is loaded, switch to view the front suspension subsystem.



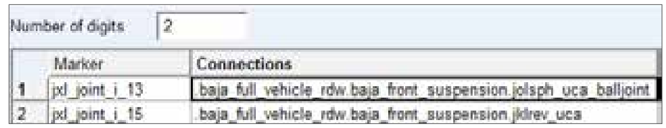

b. Right click on gel_upper_control_arm and select Make Flexible.




c. Select Import MNF.

d. Right click at MNF File and browse for uca_0.mnf you have created.

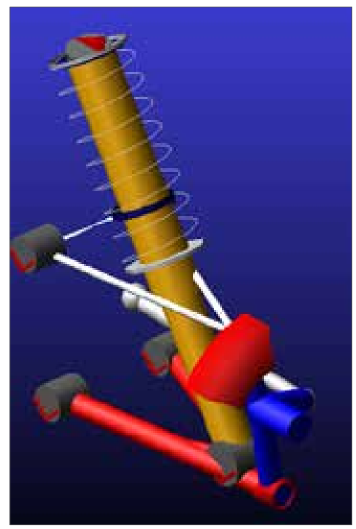
e. Click on connections, make sure that the connections are shown like the figure below.




| | Marker | Connections |
|---|---|---|
| 1 | jxl_joint_i_13 | .baja_full_vehicle_rdw.baja_front_suspension.jolsph_uca_balljoint |
| 2 | jxl_joint_i_15 | .baja_full_vehicle_rdw.baja_front_suspension.jklrev_uca |

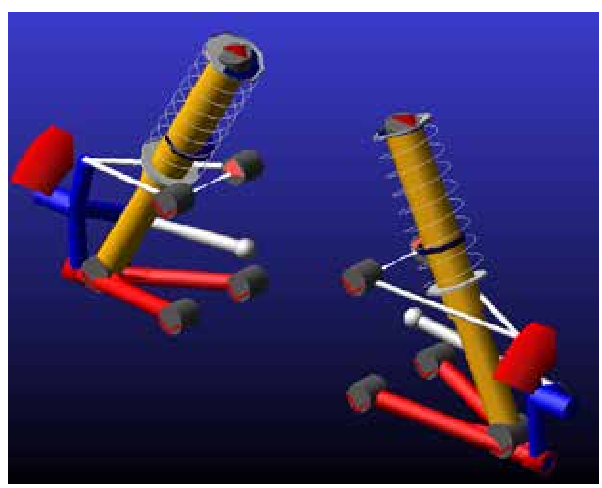
f. Click OK.

g. Now the left upper control arm has been replaced by a flexible body shown in white.



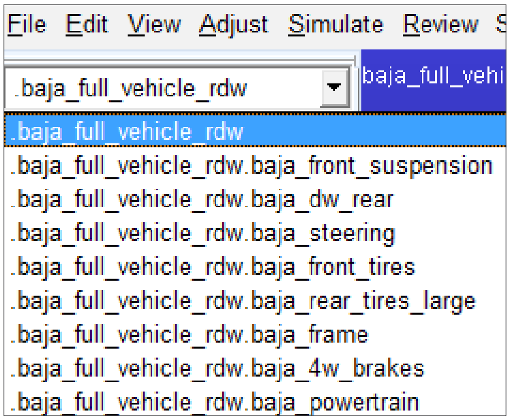
h. Repeat the steps for ger_upper_control_arm. This time, browse for uca_1.mnf.

i. The completed suspension will look like the figure below.

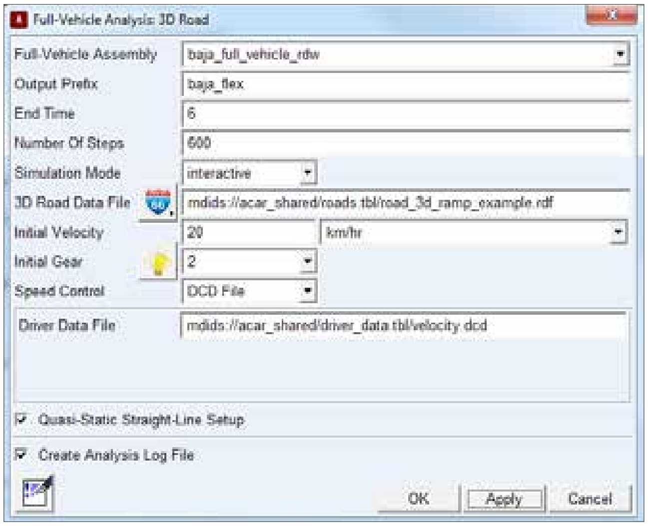


## Step 8. Run Full-Vehicle Simulation

a. Switch back to Assembly view.

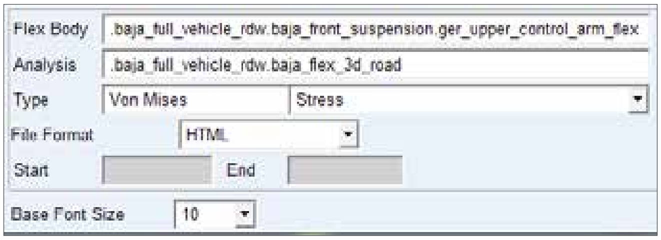


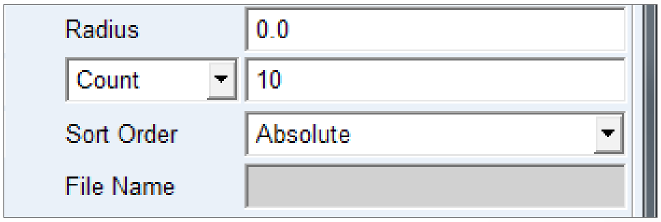

b. Go to Simulate->Full-Vehicle Analysis->Course Events->3D Road.
c. Enter baja_flex at Output Prefix.
d. Enter 6 at End Time and 600 at Number of Steps.
e. Select Interactive Simulation Mode.
f. Right click at 3D Road Data File, go to Search-><acar_shard>roads.tbl and browse for road_3d_ramp_example.rdf.
g. Enter 20 as Initial Velocity.
h. Choose 2 as Initial Gear.
i. Choose DCD File for Speed Control.
j. Right click at Diver Data File and go to Search-><acar_shard>driver_data.tbl and browse for velocity .dcd.
k. Check Quasi-Static Straight-Line Setup and click OK.




## Step 9. Check Hotspot

a. If Adams/Durability plugin has not been loaded, go to Tools->Plugin Manager and check Adams/Durability.
b. Click Durability->Hotspots Table.
c. Right click at Flex Body and browse for ger_upper_control_arm.
d. Right click at Analysis and browse for baja_flex_3D_road.
e. Select VonMises Stress at Type. Leave the rest as default.
f. Click Report.



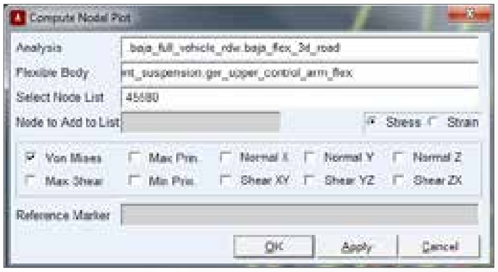




| VON MISES Hot Spots for ger_upper_control_arm_flex Date= 2014-06-04 17:33:07 | | | | | | |
|---|---|---|---|---|---|---|
| Model= .baja_full_vehicle_rdw.baja_front_suspension | | Analysis= baja_flex_3d_road | | Time = 0 to 6 sec | | |
| Top 10 Hot Spots | | | Abs | Radius= 0.0 mm | | |
| Hot Spot | Stress | Node | Time | Location wrt LPRF (mm) | | |
| # | (newton/mm**2) | id | (sec) | X | Y | Z |
| 1 | 131.424 | 45580 | 4.37 | -970.126 | 206.46 | 286.933 |
| 2 | 131.178 | 45584 | 4.37 | -970.77 | 208.334 | 286.658 |
| 3 | 129.18 | 45586 | 4.37 | -971.413 | 210.209 | 286.382 |
| 4 | 128.815 | 45591 | 4.37 | -969.884 | 211.63 | 285.863 |
| 5 | 128.521 | 45587 | 4.37 | -969.241 | 209.757 | 286.139 |
| 6 | 128.502 | 45590 | 4.37 | -972.056 | 212.083 | 286.106 |
| 7 | 128.128 | 45585 | 4.37 | -968.599 | 207.884 | 286.414 |
| 8 | 127.984 | 45592 | 4.37 | -972.699 | 213.957 | 285.831 |
| 9 | 127.467 | 56449 | 4.36 | -967.486 | 210.799 | 295.776 |
| 10 | 127.453 | 45578 | 4.37 | -969.483 | 204.586 | 287.209 |

g. From the table, we find that Node No.45580 of front right upper control arm is the number one hot spot with maximum stress of 131.424 MPa.

## Step 10. Generate Nodal Plots

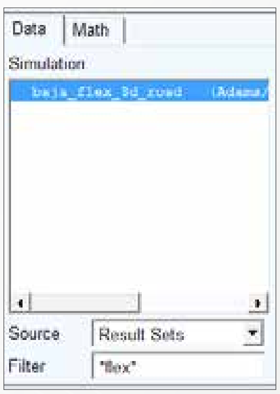
a. Click on Durability->Nodal Plots.
b. Again, Select baja_flex_3d_road as Analysis and ger_upper_control_arm as Flexible Body.
c. Enter 45580 at Select Node List and check Stress and Von Mises.




d. Click OK. This will enable us to plot the time history of stress at Node 45580 of front right upper control arm.

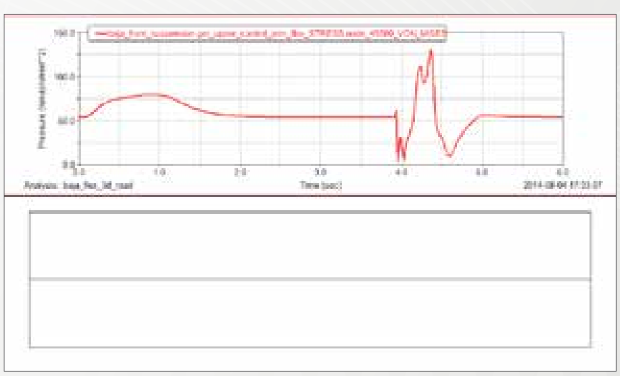
## Step 11. Postprocessing

a. Press F8 to enter postprocessing window.
b. Go to View->Page->Page Layouts.
c. Select 2 Views: Over and Under. The plot page will be divided into two windows.
d. Change Data Source to Result Sets.
e. Enter *flex* at Filter.
f. In the Result Set window, expand baja_front_

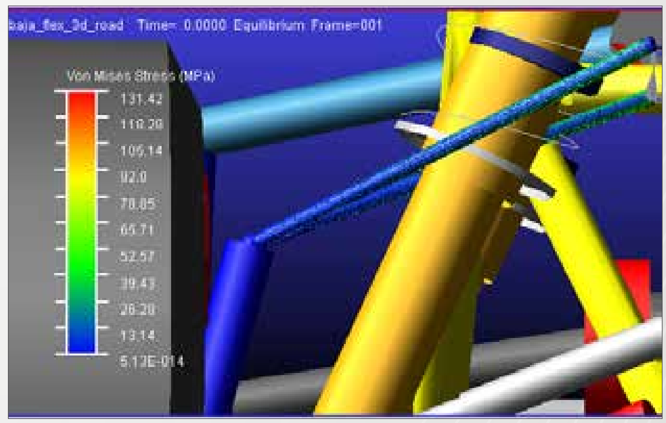



suspension and click on ger_upper_control_arm.

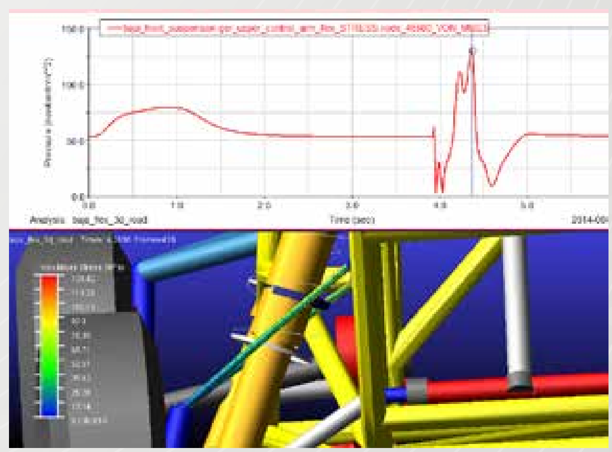
g. Click on node_45580_VON_MISES on the right column and click Add Curves. A plot will be added at the upper plot window.



h. Right click at the lower plot window, select Load Animation. From Database Navigator, double click at baja_flex_3d_road.
i. Click on Camera Tab. Right click at Follow Object and go to Flexible_Body->ger_upper_control_arm.
j. Click on Contour Plots Tab and change Contour Plot Type to Von Mises Stress.
k. Adjust your view to focus on the front right upper control arm.

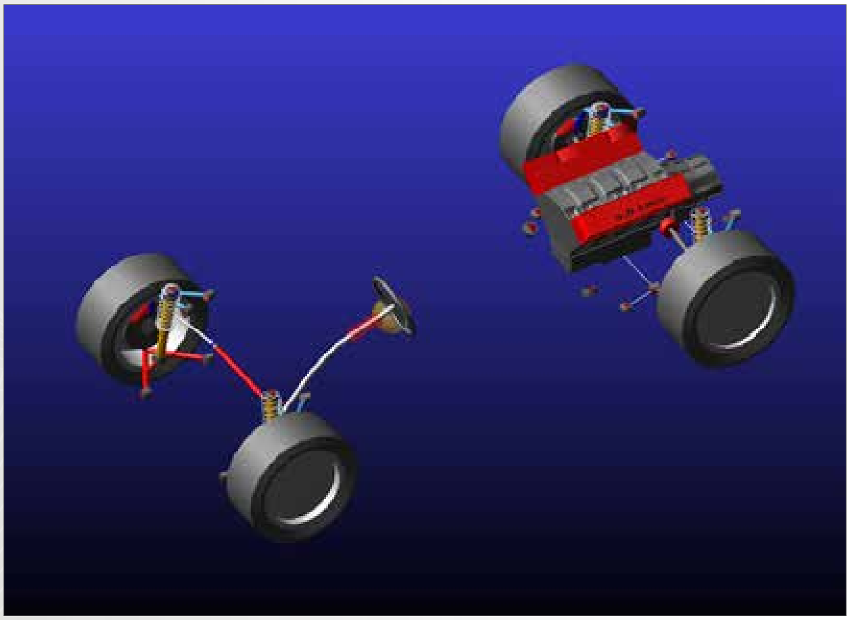



l. Click on the play button of the animation and you can observe the status of the vehicle when the stress at the hotspot is at its maximum.



m. Now, you can either proceed to check other results or save the front suspension subsystem and full vehicle assembly as new files.

# Example 44: Adams for Vehicle Dynamic Course



## Software Version

Adams/Car and Adams/View 2013.2

## Problem Description

Adams/Car is not only a powerful tool for engineering teams in automotive industry, but also a good package for academic purposes. In this example, we will show the benefits of Adams/Car and Adams/View for vehicle dynamics courses. You will learn to customize the vehicle assembly so that you can conduct various standard tests to have a better understanding of vehicle dynamics. Please use the Adams Database we provided in example to follow the steps. You are advised to finish Example 39 first before you proceed.

## Part 1: Using Adams/Car for Longitudinal and Lateral Dynamics Analysis

In this part, you will learn how to change vehicle parameters such as mass, wheelbase, inertia, tire property, etc. Their influence on the vehicle dynamics will be reflected through several simulations including acceleration, braking, constant radius cornering, step steer, etc.

### Step 1: Open Assembly

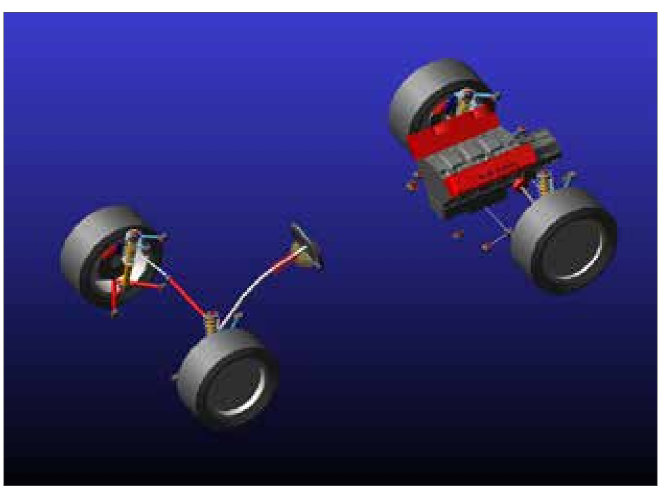
a. Before we introduce the simulation events provided by Adams/Car, it is important to learn how to customize the vehicle assembly so as to conduct design studies of parameter change.

b. Start Adams/Car 2013.2.

c. If prompted, select Standard Interface. Otherwise, follow steps in Example 39 to load the VD_Course and VD_Course_FWD database.

d. Go to **Tools->Database Management->Set Default** Writable and select VD_Course. This will save all the changes we make to the VD_Course vehicle assembly at its own database.

e. Go to **File->Open->Assembly**. Right click in the field and search in VD_Course folder for **VD_Course.asy**. Then Click OK. When the assembly is loaded, you should fine the following configuration.



### Step 2. Things to Do

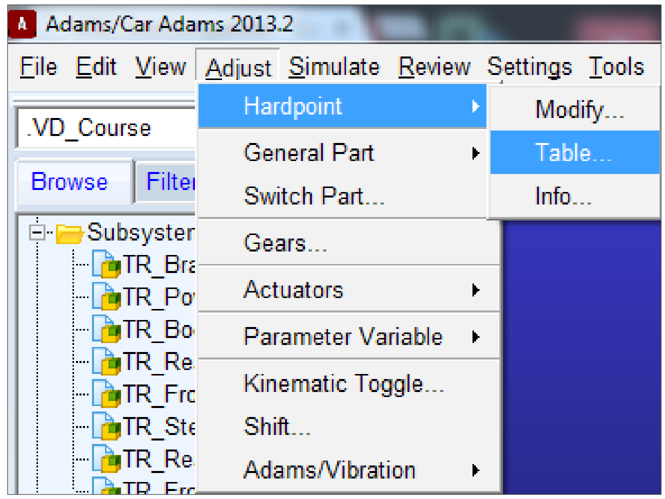
a. In this example, we will focus on a few parameters that are paramount to vehicle dynamics courses. They include: wheelbase, cg location (relative position fore and aft), mass, inertia, tire lateral stiffness.

b. We have also provided FWD vehicle assembly.

### Step 3. Adjust Wheelbase

a. To adjust wheelbase, it is necessary to know the current default wheelbase.

b. Go to **Adjust->Hardpoint->Table**.




c. In the dropdown menu, select **VD_Course.TR_Front-Suspension**.

d. Note that the x location of hpl_wheel_center is 267.

e. From the dropdown menu, change the subsystem to **VD_Course.TR_Rear_Suspension**.

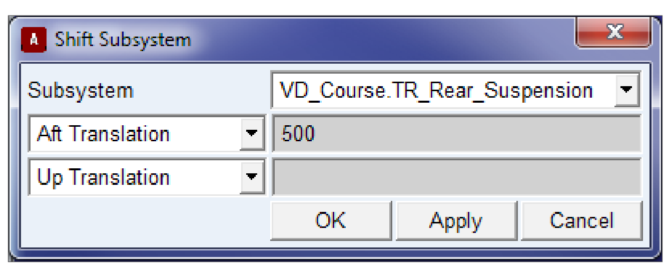
f. Note this time that the x location of hpl_wheel_center is 2827.

g. Thus, the current wheelbase is 2827-267=2560.

h. In order to change the wheelbase, go to **Adjust->Shift**.

i. From the Subsystem dropdown menu, select **rear suspension**.

j. Select **Aft Translation** and enter **500**.



k. Click OK and you will find that the rear suspension assembly has been moved backwards. You can check this by looking at the wheel center position of rear suspension. Notice that the engine block hasn't changed. This is not a big issue since the engine block is merely a graphical representation. Now restore the rear suspension to its original position by doing a Fore Translation shift.

## Step 4. Adjust Mass Properties

a. In this step, we will adjust mass properties, inertia properties.

b. Go to **Simulate->Full-Vehicle Analysis->Vehicle Set Up->Adjust Mass**.

c. In the window, enter **0, 0, 0** at **CG Location**.

d. Right click at **Relative to Marker** and go to **Marker->Browse**.

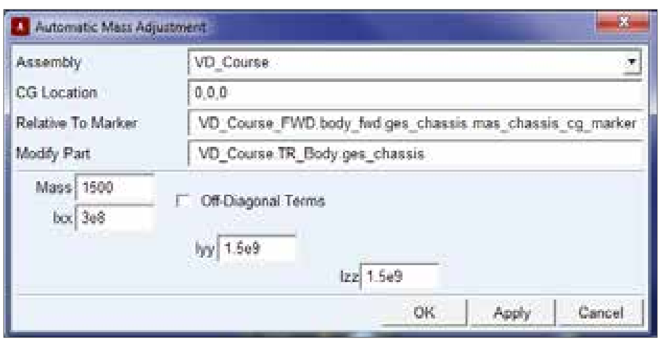
e. In the Database Navigator, expand **TR_Body->ges_chassis** and double click **mas_chassis_cg_marker**.

f. ges_chassis has already been selected automatically as Modify Part.

g. After the above steps, we have relocated the cg position to mas_chassis_cg_marker, which can be changed later.

h. Enter **1500** at Mass, **3e8** at Ixx, **1.5e9** at Iyy and **1.5e9** at Izz.

i. Click **OK**.



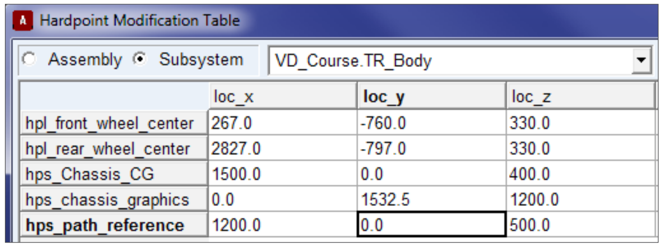

## Step 5. Adjust CG Location

a. Go to **Adjust->Hardpoint->Table**.

b. From the dropdown menu, select TR_Body subsystem.

c. Change hps_Chassis_CG to **1500, 0, 400**.

d. Click OK. It is advised that you input a reasonable value of CG location with respect to vehicle mass and inertia properties. Otherwise it will cause the static equilibrium at initial simulation to fail.




| | loc_x | loc_y | loc_z |
|---|---|---|---|
| hpl_front_wheel_center | 267.0 | -760.0 | 330.0 |
| hpl_rear_wheel_center | 2827.0 | -797.0 | 330.0 |
| hps_Chassis_CG | 1500.0 | 0.0 | 400.0 |
| hps_chassis_graphics | 0.0 | 1532.5 | 1200.0 |
| hps_path_reference | 1200.0 | 0.0 | 500.0 |

## Step 6. Adjust Tire Properties

a. Another factor affects vehicle performance is tire property.

b. Right click at one of the front tires graphics and select Modify.

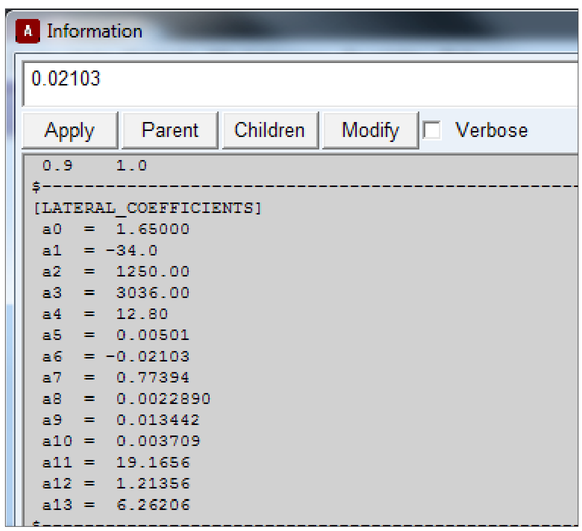
c. Notice that in the property file field, a tire property with Pacejka '89 is selected.

d. Click at to view the property.

e. You will find a list of coefficients that define the tire property curve shape.

f. Under Lateral Coefficients, we find coefficients from a0 to a13. From our knowledge of Pacejka tire model, changing a3, a4 and a5 will affect the initial slope, i.e. lateral stiffness of the tire within linear region.

g. Close the information window.




h. Right click at Property File and search in **<VD_Course>/tire.tpl/ for TR_front_pac_89_stiff.tir**.

i. Check the property file again and it can be found that a3 has been increased to 8000. To create your own tire property file, open .tir file with any kind of text editor and change any of the value of your interest. More information about Pacejka tire model can be found both in Adams Help and other source of literature.

## Step 7. Longitudinal Dynamics Events

a. After you have customized the vehicle to your intention, it is time to perform full-vehicle analysis. Adams/Car provides a number of Full-Vehicle Simulations. For longitudinal dynamics, typical events include acceleration test and braking test.

b. Go to **Simulate->Full-Vehicle Analysis->Straight-Line Events->Acceleration**.

c. Enter **VD_Course** at Output Prefix.

d. Enter **10** at End Time.

e. Enter **500** at Number of Steps.

f. The default Road Data is 2d_flat.rdf. This is a simple straight line course. You can right click at the field and search in acar_shared database for other road profiles such as 2d_ramp. Rdf.

g. Enter **40 mph** at Initial Velocity.

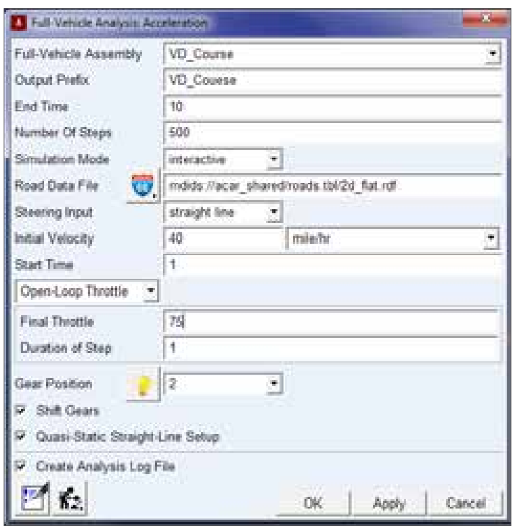
h. Enter **1** at Start Time. This is the time that the acceleration starts.

i. Enter **75** at Final Throttle.

j. Enter **1** at Duration of Step. These two parameters defines how intense the acceleration maneuver is.

k. Select **2nd gear**.

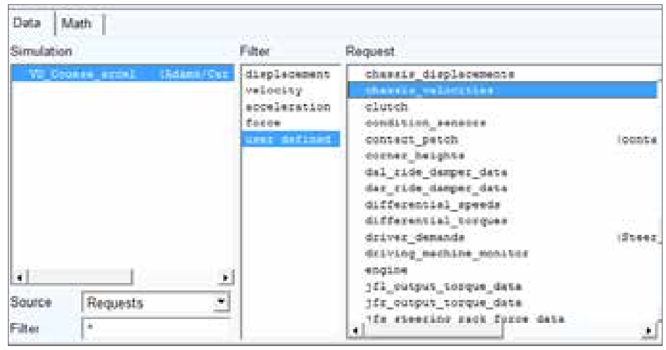
l. Click **OK**.




## Step 8. Postprocess the Result

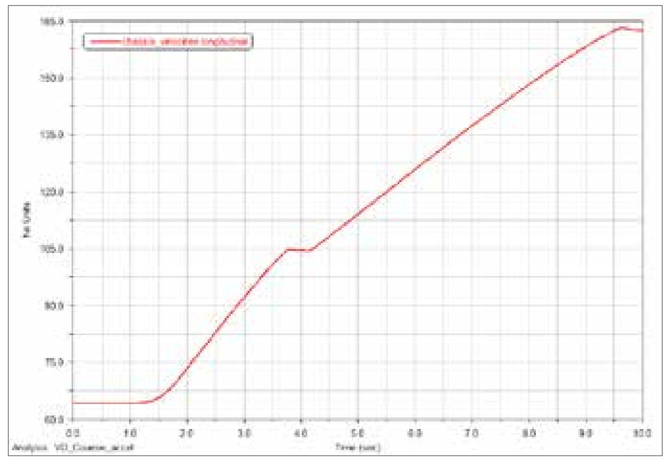
a. Press **F8** to enter postprocessor.

b. Under the **Request** column, locate and highlight **chassis_velocities**.




c. Select **longitudinal** under **Component**.

d. Click **Add Curves**.




e. Notice that it took the vehicle about 4 seconds to accelerate from 40 mph to 70 mph. The plot unit is km/hr. Plot any other curves of your interest.

## Step 9. Braking Analysis

a. Go to **Simulate->Full-Vehicle Analysis->Straight-Line Events->Braking**.

b. Enter **VD_Course** at Output Prefix.

c. Enter **5** at End Time.

d. Enter **250** at Number of Steps.

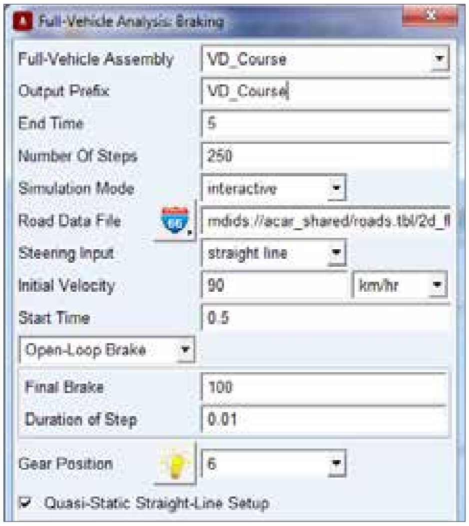
e. Keep the Road Data as the default 2d_flat.rdf.

f. Enter **90 km/hr** at Initial Velocity.

g. Enter **0.5** at Start Time.

h. Enter **100** at Final Brake and 0.05 at Duration of Step.

i. Select **6th gear** and click **OK**.



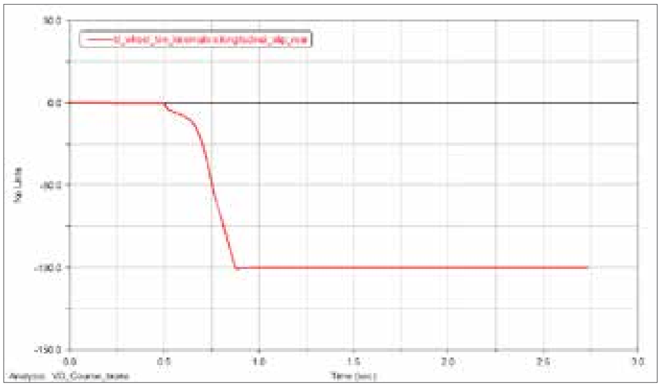

## Step 10. Postprocess the Result

a. Press **F8** to enter postprocessor.

b. Highlight **VD_Course_brake** under **Simulation**.

c. Change **Source** to **Result Sets** and highlight **til_wheel_tire_kinematics** on the right.

d. Select **longitudinal_slip_rear** under **Component** and click **Add Curve**.



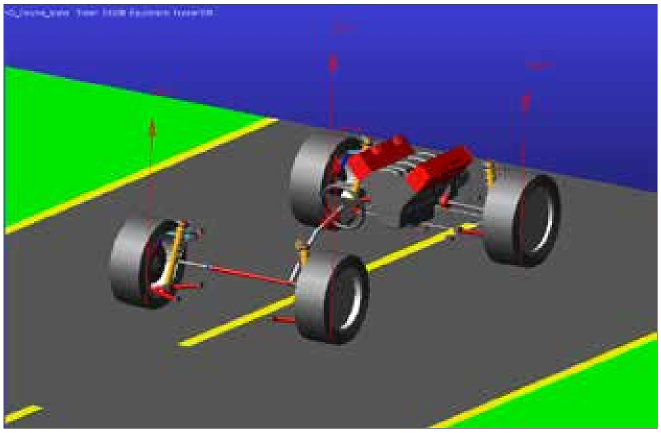
e. Immediately it is spotted that the rear wheels locked up (-100 percent slip) during the braking event.

f. Create a new page. Right click in the page and select **Load Animation**.

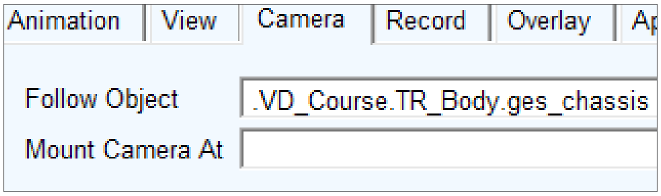
g. Select **VD_Course_brake** from the Database Navigator.

h. Switch back to Adams/Car window.

i. Go to **Review->Animation Controls**. Check **Tire Forces** at the bottom and then switch to postprocessor. Now you can find the tire forces in red.

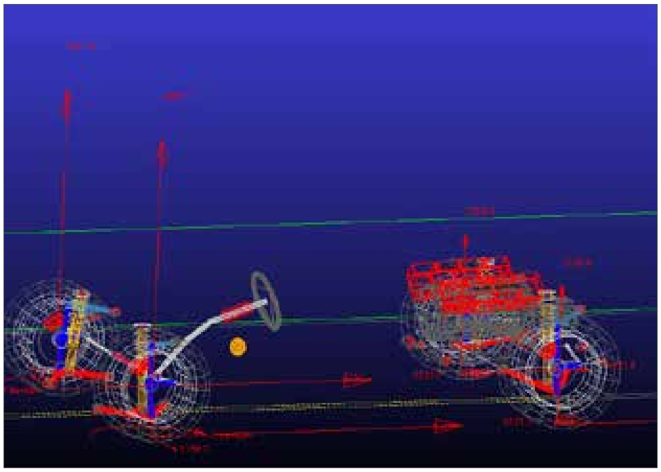


j. Click **Camera** tab. Right click at Follow Object and go to **Part->Browse**, select **ges_chassis** under **TR_Body**.




k. Press shift+S to change render mode to wireframe. This will help us to spot the wheel lock up.

l. Play the animation and you can find the large amount of weight transfer occurred during the event.



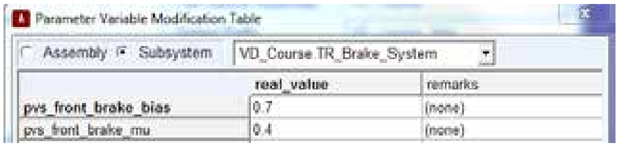
## Step 11. Change the Brake Bias.

a. The wheel lock up could be the wrong brake bias distribution with respect to specified cg location and road friction coefficient.

b. Go to **Adjust->Parameter Variable->Table**.

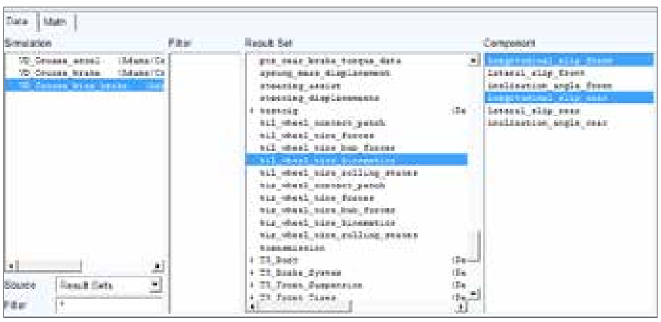
c. From the dropdown menu, select **TR_Brake_System**.

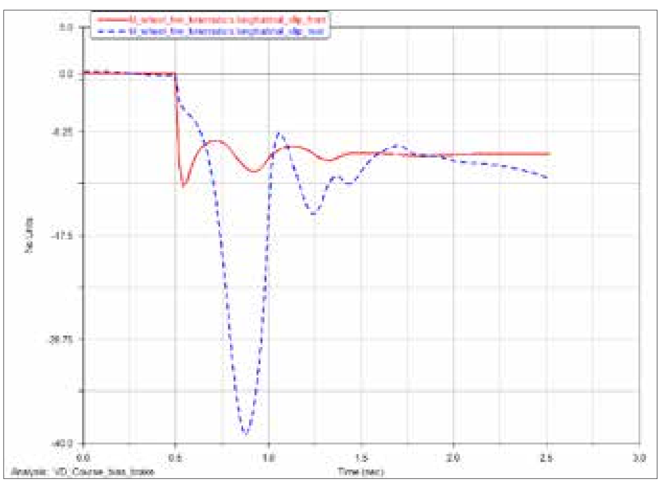
d. Increase pvs_front_brake_bias from 0.6 to **0.7**.




e. Run the braking event with output prefix VD_Course_bias and the same set of settings.

f. In the postprocessor, in a new page, plot the tire slip curve of both front and rear tire.




g. This time, none of the tires lock up.

h. In fact, there are quite a number of parameters that you can adjust through Adjust->Parameter Variables. Change any one of your interest and check how it affects the dynamics of the vehicle!

## Step 12. Lateral Dynamic Events

a. Adams/Car also provides tests for lateral dynamics analysis.

b. We will run a **Constant Radius Cornering** event first.

This skidpad event is a helpful test to determine the vehicle's capability to negotiate a corner and its cornering characteristics.

c. Go to **Simulate->Full-Vehicle Analysis->Cornering Events->Constant Radius Cornering**.

d. Enter **VD_Course** at Output Prefix.

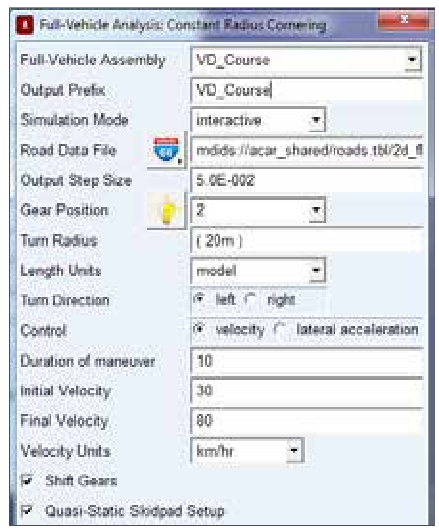
e. Enter **0.05** at Output Step Size.

f. Choose **2nd gear**.

g. Enter **20m** at Turn Radius.

h. Enter **10** at Duration of Maneuver and 30 km/hr, 80 km/hr at Initial and Final Velocity respectively.

i. Check **Shift Gears** and click OK.

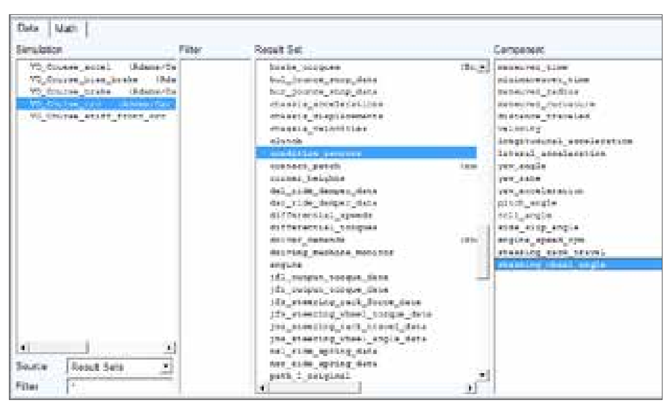



## Step 13. Postprocess the Result

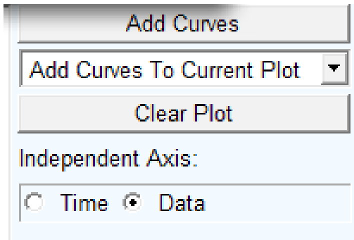
a. Press **F8** to enter postprocessor.

b. Highlight **VD_Course_crc** under simulation and change Source to Result Sets.

c. Highlight **condition_sensors** in Result Set and select **steering_wheel_angle**.



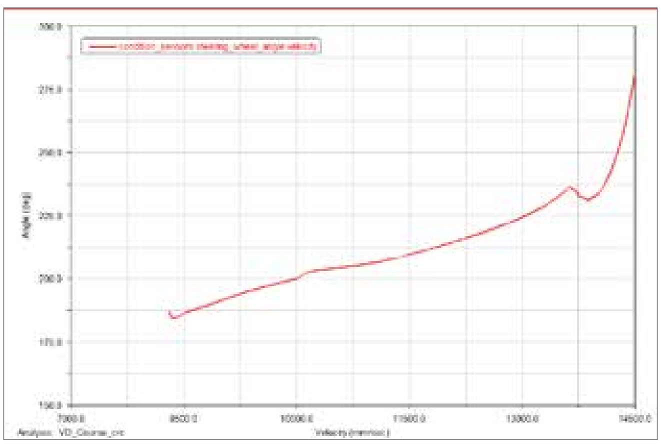
d. Click at Data on the right.




e. In the **Independent Axis Browser**, highlight **condition_sensors** and select **velocity** as component.

f. Click **OK** and then **Add Curve**.

g. Zoom into the region where the vehicle hasn't spun.

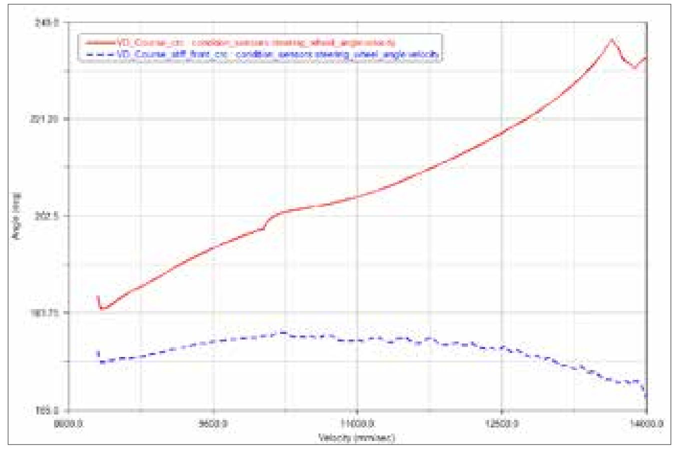


h. The steering angle increases as the vehicle velocity to maintain a constant radius. Apparently the vehicle has an understeer characteristic.

## Step 14. Change the Tire Property

a. Now, change the front tire property as instructed in previous step. Select **TR_front_pac_89_stiff.tir**.

b. Run the simulation again and plot the same curve in the same plot.



c. This time, the steering angle needs little increase and even decreases a little as the speed increases. This indicates that the vehicle has geared towards neutral and slightly oversteer characteristic because of the increase of front tire lateral stiffness.

## Step 15. Single Lane Change

a. Another commonly seen testing event is single lane change. The steering wheel is subject to a sinusoidal input in a short time. This is useful for testing ABS, ESP system because it replicates a maneuver similar to obstacle evasion.

b. Change the front tire property back to default. Go to **Simulate->Full-Vehicle Analysis->Open-Loop Steering Events->Single Lane Change**.

c. Enter **VD_Course** at Output Prefix.

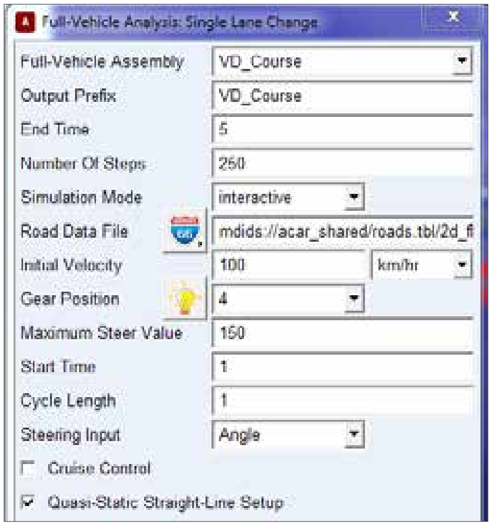
d. Enter **5** at End Time and 250 at Number of Steps.

e. Enter **100 km/hr** at initial Velocity and select 4th gear.

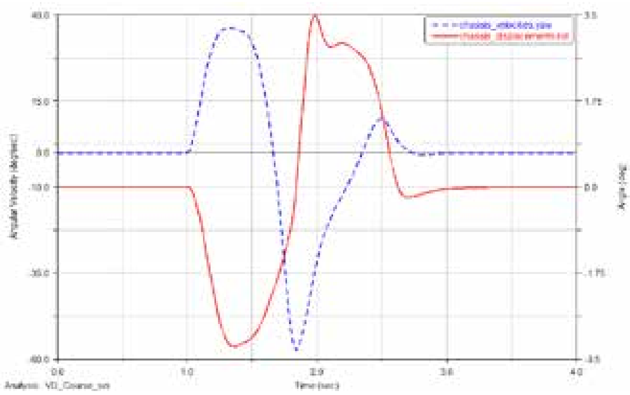
f. Enter **150** at Maximum Steer Value.

g. Enter **1** at Start Time and 1 at Cycle Length. The cycle length determines how fast the steering maneuver is.

h. Click **OK**.

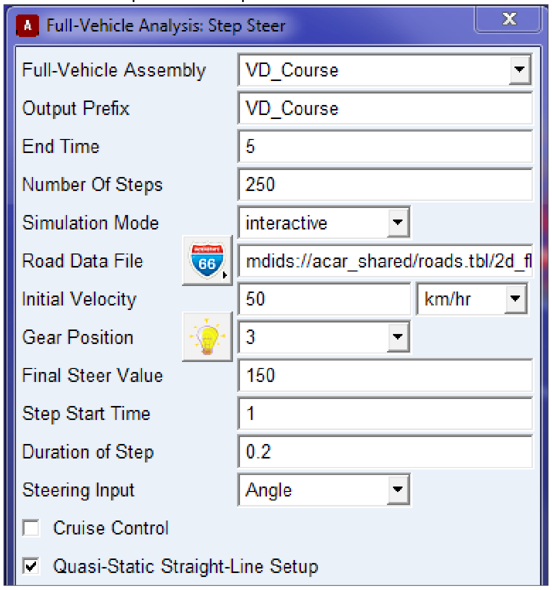



i. A number of plots can be generated such as yaw rate and roll angle. They are useful for investigating the effect of cg height, cg position, mass and other properties on the response of a vehicle considering DOFs including roll.

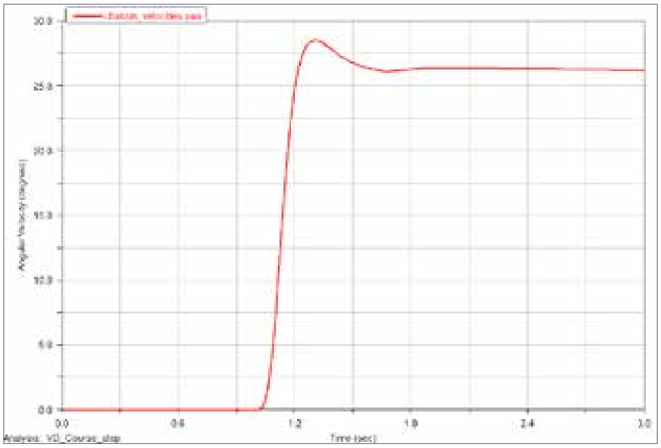



## Step 16. Step Steer

a. Engineers are always interested in the step response of any kind of system. Vehicles are no exception.
b. Go to **Simulate->Full-Vehicle Analysis->Open-Loop Steering Events->Step Steer**.
c. Enter **VD_Course** at Output Prefix.
d. Enter **5** at End Time and 250 at Number of Output Steps.
e. Enter **50 km/hr** at Initial Velocity and select 3rd gear.
f. Enter **150** at Final Steer Value.
g. Enter **1** at Step Start Time.
h. Enter **0.2** at Duration of Step. Again this value determines how fast the step steer is going to occur. There is no ideal step steer input.




i. Check the yaw response in postprocessor and it can be found that the vehicle yaw response to step steer input is satisfactory in both settling time and maximum overshoot.

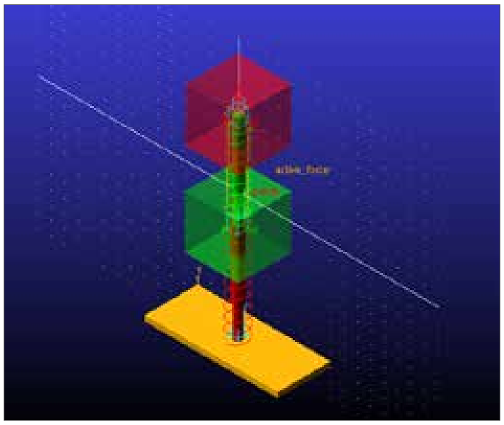


## Part 2. Using Adams/View for Quarter Car Model Analysis

In this part, a quarter car model in Adams/View is provided. Instructions on how to modify parameters of the model so as to investigate their impact on vertical dynamics are given.

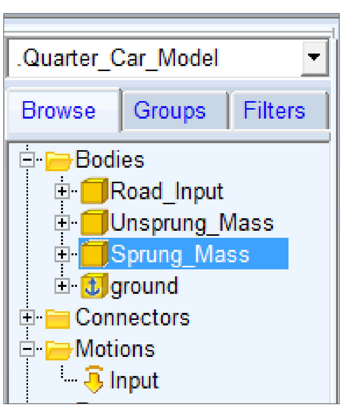
## Step 1. Start Adams/View

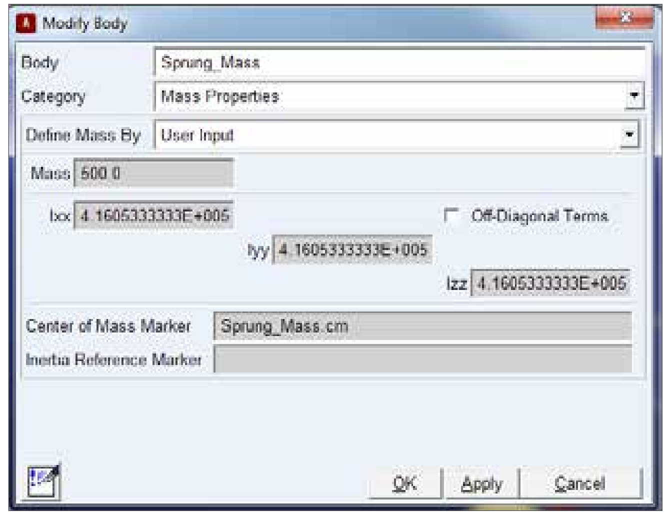
a. Vertical dynamics, i.e ride analysis is also an interesting topic in vehicle dynamics. A quarter car model is commonly used for this type of analysis. Although Adams/Car provides four post analysis testrig, we can also build a standard quarter car model in Adams/View.
b. Start Adams/View 2013.2 and select **Existing Model**.
c. Browse in example start folder for **Quarter_Car_Model.cmd**.
d. Click OK.



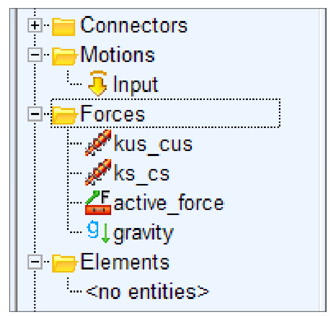
## Step 2. Examine the Model

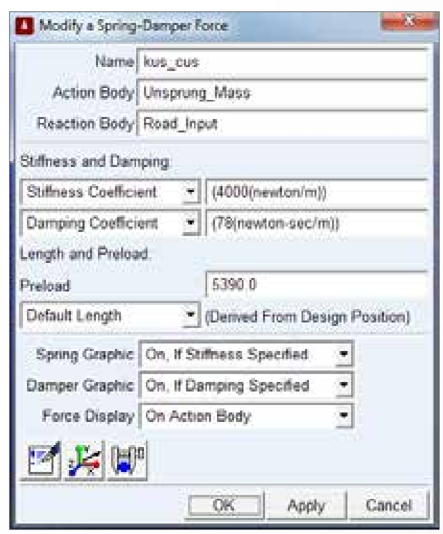
a. The model contains an unsprung mass (green), a sprung mass (red) and a road input (maize). They are interconnected by two sets of spring-dampers. There is also an active force acting between the sprung and unsprung mass for active suspension analysis. Its default value is set to zero.
b. In the model tree on the left, under Bodies folder, double click either Sprung_Mass or Unsprung_Mass.
c. You can find that we have set the mass of the body to be defined by user input. Hence, the mass property can be modified independent of the geometry.



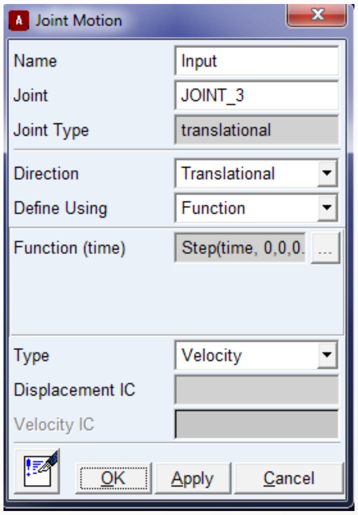




d. Similarly, double click the spring-dampers under Forces folder and then you can define preload, spring stiffness and damping coefficient.




e. There is a Motion called Input. It is applied at the translational joint between the Road_Input and the ground. The input type can be displacement, velocity or acceleration depending on the problem you are studying.




## Step 3. Instructions on How to Use the Model

a. The model can be used for all kinds of quarter car model ride analysis. Simply change the values of mass, stiffness and damping and the difference will show in postprocessing. Please refer to Section I of the tutorial kit to learn more about building measurements and postprocessing.

b. The model can also be used for Adams/MATLAB co-simualtion. The active force can be input from Simulink. Please refer to Section V of the tutorial kit to learn more about how to build inputs and outputs, export control plants, set up cosimulation and postprocess the simulation.

MSC Software