Supplemental Adams Tutorial Kit
for Design of Machinery Course Curriculum
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 26 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**

- Example 1: Falling Stone .............................................. 6
- Example 2: Inclined Plane ........................................... 12
- Example 3: Lift Mechanism - Geometry .......................... 22
- Example 4: Lift Mechanism - Simulation ....................... 28
- Example 5: One-degree-of-freedom Pendulum ............... 36
- Example 6: Projectile Motion ..................................... 44
- Example 7: Spring Damper - Part 1 .............................. 50
- Example 8: Spring Damper - Part 2 .............................. 56
- Example 9: Suspension System 1 ................................. 62
- Example 10: Suspension System 2 ............................... 70
- Example 11: Four Bar Velocity ................................... 76
- Example 12: Cam-Follower ......................................... 80
- Example 13: Crank Slider ......................................... 86
- Example 14: Controls Toolkit in ADAMS/View ................ 92

**Section II: Intermediate Level**

- Example 15: Valvetrain Mechanism .............................. 98
- Example 16: Cam-rocker-valve ................................... 106

**Section III: Textbook Problems**

- Example 15: Valvetrain Mechanism ................................ 98
- Example 16: Cam-rocker-valve ................................... 106

**Section IV. Adams/Machinery Applications**

- Example 15: Valvetrain Mechanism ................................ 98
- Example 16: Cam-rocker-valve ................................... 106
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

\[ g = 9810 \frac{mm}{s^2} \]

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. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
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 $y = - \frac{1}{2} gt^2$

b. Substitute: $g = 9810 \text{ mm/s}^2$, $t = 1 \text{ s}$, 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

$g = 32.2 \text{ ft/sec}^2$ or $386.4 \text{ in/sec}^2$

$\mu_s = 0.3$
$\mu_d = 0.25$

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 ($\mu_s$) is 0.3 and the coefficient of dynamic friction ($\mu_d$) is 0.25 and gravity is $32.2 \text{ ft/sec}^2$. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
Step 1. Create a New Adams database

a. To import a file.
b. Click on Create a new model.
c. Under Start in, browse to the folder where you want to save your model.
d. Type the name of the new Model name as inclined_plane and click OK.
e. 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. Then under Orientation, input 15,0,0 and click Apply and OK.
b. Input Top Conv as the New Set Name, set the Target Element Type to 2D, and then click on Input Data.
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 of the crate slides off the ramp. For an end time of 0.5 seconds, verify that the crate 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 $\theta = 15^\circ$, $a = 0$

At $\theta = 20^\circ$, $a = -41.35$ in/sec$^2$.

Max Angle for Crate to Slip ($\theta_{\text{max}}$) = 16.8°. $a = -19.19$ in/sec$^2$.

Closed-form solution

Without friction:

$$\Sigma F_x = m a_x \quad - mg \cdot \sin \theta = m a_x$$

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

For $\theta = 15^\circ$, $a_x = -32.2 \sin(15^\circ)$

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

With friction:

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

$N = mg \cdot \cos \theta$

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

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

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

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

$$\mu_x \cdot \tan \theta_{\text{max}} = 0$$

$$\theta_{\text{max}} = \arctan(\mu_x) = \arctan(0.30) = 16.7^\circ$$

Once the crate starts sliding,

$$\Sigma F_x = m a_x : F_x - mg \cdot \sin \theta = m a_x$$

$$\mu_k \cdot N - mg \cdot \sin \theta = m a_x$$

$$\mu_k \cdot mg \cdot \cos \theta - mg \cdot \sin \theta = m a_x$$

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

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

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

$$a_x = -40.3 \text{ in/sec}^2 (-3.45 \text{ ft/sec}^2)$$
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. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
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 direction 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.
Section I: Beginner's Level | Example 3: Lift Mechanism - Geometry

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. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
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 constrained properly.

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

Step 8. Add Joint Motions.

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

a. Under Speed, enter \(360d \times \text{time}\).

b. Then select the Mount-to-Base revolute joint (JOINT_2) to apply.

c. Now add a motion to the Shoulder-to-Mount joint by entering \(-\text{STEP(time,0,0,0.10,30d)}\) in the Speed Box.

d. Then select the Shoulder-to-Mount revolute joint (JOINT_3) to apply.

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

e. Enter \(-\text{STEP(time,0.8,0,1,5)}\) for the Speed.

f. Then select the Boom-to-Shoulder translational joint (JOINT_4) to apply.

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

g. Under Speed enter \(45d \times (1 - \cos(360d \times \text{time}))\).

h. Then select the Bucket-to-Boom revolute joint (JOINT_5) to apply.
Step 9. Verify the Joint Motions.

a. Check to see if the functions were properly entered for each joint by going to Modify 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

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. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
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 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. 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 Sphere Part 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 stone 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_5 and Create Strip Chart are selected, and click Apply and Cancel.
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_5 and Create Strip Chart are selected, and click Apply and Cancel.

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, and using the mouse or Location Event select 450,0,0 as the marker location.
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 select New.
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.
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.
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 -30 in the text box. Select Rot.Velo and enter -300 in the text box. Then click OK and OK.
Step 11. Add the Initial Conditions to the Pendulum Model.

a. From the Status bar, click on the Verify tool, and right-click the Information tool stack. Close the window once your model has been verified.
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 \( \text{pivot\_force\_x} \) 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 \( \text{pivot\_force\_y} \).
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 \( \text{pend\_angle} \).
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 Horizontal Force supported by the pivot A is equal to \(-A \cos 30\).
- The Vertical Force supported by the pivot A is equal to \(-A \sin 30\).
- Compare your results from MD ADAMS with the analytical solution.

\[ I_A = I_{zz} + ml^2 \]
\[ I_A = 0 + mL^2 \]
\[ \omega_0 = 300^\circ/\text{sec} \]
\[ \omega_0 = 5.24 \text{ rad/sec} \]

\[ \Sigma M_A = I_A \alpha - mg(L \cos 30) = (ml^2) \alpha \]
\[ g \cos 30 = L \alpha \]
\[ \alpha = \frac{-2}{L} \cos 30 \]
\[ \alpha = -18.88 \text{ rad/sec}^2 \]

\[ \Sigma F_t = m \alpha \quad mg \cos 30 - A_t = mL \alpha \]
\[ A_t = m(g \cos 30 - L \alpha) \]
\[ A_t = 6N \]

\[ \Sigma F_n = mr \omega^2 \quad A_n - mg \sin 30 = mL \omega^2 \]
\[ A_n = m(g \sin 30 + L \omega^2) \]
\[ A_n = 34.53N \]
Example 6: Projectile Motion

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. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
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 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.


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 \(6 \cos(60^\circ) \text{(m/sec)}\), and then check Y Axis and enter \(6 \sin(60^\circ) \text{(m/sec)}\). 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 -4903.

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

\[
\begin{align*}
x_o &= 0 & x_f &= R \\
y_o &= 0 & y_f &= 0 \\
V_{x_o} &= 6000 \times \cos 60^\circ = 3000 \text{ mm/sec} \\
V_{y_o} &= 6000 \times \sin 60^\circ = 5196 \text{ mm/sec} \\
y_f &= y_o + V_{y_o} t - \frac{1}{2} g t^2 \\
0 &= 0 + 5196 t - 0.5 \times 9806 \times t^2 \\
0 &= (5196 - 4905) t \\
t &= 1.06 \text{ sec} \\
x_f &= x_o + V_{x_o} t \\
R &= 0 + 3000 \times 1.06 \\
R &= 3180 \text{ mm}
\end{align*}
\]
Example 7: Spring Damper - Part 1

Software Version
Adams 2013.2

Problem Description
Find the force in spring damper at static equilibrium. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge

M: 187.224 Kg
K: 5.0 N/mm
C: 0.05 N-sec/mm
L0: 400 mm
F0: 0
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:
  - $187.224 \text{ kg} \times 9806.65 \text{ mm/s}^2 = 1836.04 \text{ N}$
- 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. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
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 the y-axis on the grid for the action point.
**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. Go to the **Simulation Tab** and click on **Run a Scripted Simulation** (Calculator Icon).
f. Click on **Interactive**.
g. Change the **End Time** to 2.
h. Change the **Steps** to 50.
i. 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**.


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:
  - \[187.224 \text{ kg} \times 9806.65 \text{ mm/s}^2 = 1836.04 \text{ N}\]
- 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. Note: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
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.
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 Joint.

a. First, click on Motion Driver and choose Point Motion.
b. Choose the Spindle_Wheel.Center when selecting the location.
c. 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.
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. Modify the Translational Joint to be a Fixed Joint.
   a. Right-click on the Join: `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: If you need assistance on a step, just click on it for more information. Note: Click on images to enlarge.
Step 1. Open an Existing Database.

a. First, choose Existing Model.
b. Under File Name, locate the suspension.bin file.
c. Then click OK.

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

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_

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

c. For the Dependent Axis under Measure choose Toe_Angle and then click on Data for the Independent Axis.

d. Choose Last_Run for Simulation and choose Wheel_Height for Measure.

e. Click OK then click on Add Curves.

f. 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 \( \omega = 10 \text{ rad/s} \) during a short interval of motion. When the link CB passes the vertical position shown, point A has coordinates \( x = -60 \text{ mm} \) and \( y = 80 \text{ mm} \). 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 (1 rad) 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**

\[
\begin{align*}
\omega_A &= \omega_{AO} \times \frac{r_A}{r_A} \\
\omega_B &= \omega_{BC} \times \frac{r_B}{r_B} \\
\omega_C &= \omega_{CA} \times \frac{r_C}{r_C} \\
\omega_D &= \omega_{DA} \times \frac{r_D}{r_D}
\end{align*}
\]

\[
\begin{align*}
\text{Dimen. in mm} \\
\omega_A &= 10 \text{ rad/s} \\
\omega_B &= -0.6 \text{ rad/s} \\
\omega_C &= 0.8 \text{ rad/s} \\
\omega_D &= 2.5 \text{ rad/s}
\end{align*}
\]

**Adams solution**

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.25</td>
</tr>
<tr>
<td>0.0</td>
<td>0.5833</td>
</tr>
</tbody>
</table>
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.

![Create New Model](image)

**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

![Working Grid Settings](image)

**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.**
- 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**

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>0</td>
<td>-50</td>
<td>-70</td>
<td>-80</td>
<td>-70</td>
<td>-50</td>
<td>0</td>
<td>50</td>
<td>70</td>
<td>80</td>
<td>70</td>
<td>50</td>
<td>0</td>
</tr>
<tr>
<td>Y</td>
<td>0</td>
<td>-50</td>
<td>-70</td>
<td>120</td>
<td>150</td>
<td>190</td>
<td>180</td>
<td>160</td>
<td>120</td>
<td>70</td>
<td>50</td>
<td>-30</td>
<td>0</td>
</tr>
<tr>
<td>Z</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
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.

<table>
<thead>
<tr>
<th>Points</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>-80</td>
<td>-200</td>
<td>-150</td>
<td>-100</td>
<td>-50</td>
<td>0</td>
<td>50</td>
</tr>
<tr>
<td>Y</td>
<td>50</td>
<td>40</td>
<td>40</td>
<td>30</td>
<td>10</td>
<td>0</td>
<td>10</td>
</tr>
<tr>
<td>Z</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Points</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>100</td>
<td>150</td>
<td>200</td>
<td>250</td>
<td>220</td>
<td>190</td>
<td>160</td>
</tr>
<tr>
<td>Y</td>
<td>30</td>
<td>40</td>
<td>40</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
</tr>
<tr>
<td>Z</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Points</th>
<th>15</th>
<th>16</th>
<th>17</th>
<th>18</th>
<th>19</th>
<th>20</th>
<th>21</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>150</td>
<td>100</td>
<td>70</td>
<td>40</td>
<td>10</td>
<td>-30</td>
<td>-50</td>
</tr>
<tr>
<td>Y</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
</tr>
<tr>
<td>Z</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Points</th>
<th>22</th>
<th>23</th>
<th>24</th>
<th>25</th>
<th>26</th>
<th>27</th>
<th>28</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>-80</td>
<td>-110</td>
<td>-140</td>
<td>-170</td>
<td>-200</td>
<td>-230</td>
<td>-260</td>
</tr>
<tr>
<td>Y</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
<td>50</td>
</tr>
<tr>
<td>Z</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
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 close 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.

![Part Measure dialog box](image)
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.

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


Copyright © 1992, by John Wiley & Sons, Inc.

This material is used by permission of John Wiley & Sons, Inc.
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 GlobalZ 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

\[ r = 2.266\text{m} \]
\[ r - \text{dot} = 3.58\text{ m/s} \]
\[ r - \text{double dot} = 316\text{ m/s}^2 \]
\[ \theta = 11.46\text{deg} \]
\[ \theta - \text{dot} = 17.86\text{ rad/s} \]
\[ \theta - \text{double dot} = -1510\text{ rad/s}^2 \]

ADAMS solution

\[ r = 2.266\text{m} \]
\[ r - \text{dot} = 3.58\text{ m/s} \]
\[ r - \text{double dot} = 316\text{ m/s}^2 \]
\[ \theta = 11.46\text{deg} \]
\[ \theta - \text{dot} = 17.86\text{ rad/s} \]
\[ \theta - \text{double dot} = -1510\text{ rad/s}^2 \]
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 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
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: _____________________________

| Frame <F> |
| Top <T> |
| Right <R> |
| Isometric <I> |
| Rotate XY <> |
| Translate <> |
| Zoom in/out <> |
| Zoom Box <> |
| Probe <> |
| Origin |
| Fit to View <> |
| Fit to View - No Ground <Ctrl> |
| Align to 3 Point |
| Align to Object XY <> |
| Working Grid On <> |
| Working Grid Off <> |
| Toggle Icon Visibility <> |
| Save View Settings |
| Restore View Settings |
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.
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 Adam/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. \( \text{STEP}(\text{time}, 0.4, 0, 0.6, 13) + \text{STEP}(\text{time}, 0.6, 0, 0.8, -13) \).
   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).

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

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 new Function 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. If 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/Example13.
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_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_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_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?