

# 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	7
Example 1: Falling Stone	8
Example 2: Inclined Plane	14
Example 3: Lift Mechanism - Geometry	
Example 4: Lift Mechanism - Simulation	
Example 5: One-degree-of-freedom Pendulum	38
Example 6: Projectile Motion	
Example 7: Spring Damper - Part 1	
Example 8: Spring Damper - Part 2	<u>5</u> 8
Example 9: Suspension System 1	<u></u> 62
Example 10: Suspension System 2	
Example 11: Four Bar Velocity	
Example 12: Cam-Follower	
Example 13: Crank Slider	
Example 14: Controls Toolkit in ADAMS/View	
Section II: Intermediate Level	97
Example 15: Valvetrain Mechanism	
Example 16: Cam-rocker-valve	
Example 17: Stamping Mechanism	116
Example 18: Robot Arm	122
Example 19: Adams Optimization	
Example 20: Airplane Control Surface	
Example 21: More on Pendulum	
Example 22: Gyroscope	154
Example 23: Power Hacksaw Mechanism	
Section III: Textbook Problems	169
Example 24: Walking Beam Indexer	
Example 25: Watt's Linkage in a Steam Engine	
Section IV. Adams/Machinery Applications	187
Example 26: Open Differential	188
Example 27: Planetary Gear Sets Modification	196
Example 28: Bearing System Workshop	200
Example 29: Simple Belt Example	206
Example 30: Serpentine Belt System	214

Example 31: Gear Train	222
Example 32: Windshield Wiper Mechanism	
Example 33: Windturbine	232
Section V: Adams/MATLAB Co-simulation	241
Example 34: DC Motor	_242
Example 35: Airplane Control Surface	
Example 36: Bridge Crane	256
Example 37: Windshield Wiper Mechanism	262
Example 38: Vehicle ABS System	
Section VI: Vehicle Dynamics	273
Example 40: FSAE Flex Body Tutorial	
Example 39: FSAE Full Vehicle Simulation Events	282
Example 41: Adams/Insight Example	286
Example 42: Humvee Durability Analysis	292
Example 43: Mini Baja Example	298
Example 44: Adams for Vehicle Dynamic Course	



# 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**.



	C Ritswillings							1	10.4
Die Eik	falling state								1 ( P
Ender	Acry Past	0604	Modify T	Veboue	Cen	Read from The	Seetsfie	China	-
	Ngant Ree Upon Tra Sector Tra Sector	7.011 7.011 7.011 8.01,07							
	e Grunbler C. 5 Berling Fart	ette Sappris	Lasta degrar oling given	n. (7. 21040)	25				
	The dollarser of three office of the there are to fart further, for	arto hare an a then. Tro abid mathematica tona RART_1	Construction	The sector					
	Basen are no all		to sales adapt						
Seath	Notest Institutes of	ninees fol 1g						-	
2				_	_	_	- 111		0.0

### 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  $y = -(1/2) gt^2$
- 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 ( $\mu$ s) is 0.3 and the coefficient of dynamic friction ( $\mu$ d) 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, Ibm, Ibf, 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.





AdamsTrine Adams 2012		261
a ba yee Series Data 国品のの文白色のの。	22日前後留泉 = ◆ Q U. ト	campet M11
ntes  Conscion   Mattee   Farras   Dimens   Despi Explosion   Ph	apter, 1 Mediation   Mesure	
। ।	<b>9</b> 🖉 🔍	
14401 B24 A41 62V	40	
Solds Paulik Bolins Construction Peakeese	Forum	
and a long a		
Trees [Press   Press		
Colo	-	
Manary Mana Provation		
Trine Name By Liter Inc.		
		and a second
0	-Built 500	Parking and
CHOADEN CHOADEN	Ves-Count	Aquestance
and her the second seco		200
the prototest prototest and the prototest prototest and the prototest prototest and the prototest prototes		Messare
Cantor of Mana Manar I TH	14233	Kupy Number
with Patricke Parker		Owinia
		Deterro
		Colections
OK ANN DACK	And the second se	1843
Hech Difference in the second s		
		C 10 4 D

#### 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.



🖬 hanne 🗄 ene det an				2 T 2
DV 247 14	- set In Plan	CXC222.X	■김희원의 ● 수	Q EL iconort #11
Massas Isans Pat Chalacterate Caracterate	an Savet prime Crater, MCA_1 Crate Child acceleration ] = Y = Z = ming (Crat			
Represent a constant Do time distribution of Constant Data Chap Constant Data Chap Constant Data	Con MARGER -	taan Ind Partnelicte Partnelicte Partneli	AX Barrens BAERE 7 Dante	nolvez piere grund Michael piere grund Michael e
Deers 2				- <b></b>

#### 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.



ten alline C	1 K K K					
Reaction Arm	1.0					
Initial Owinap	1000.0					
With Pestine Joint Display	atast .					
Centap Will	Patron Coestant ·					
Shates Transition Victority	at.	1.1	Contract Allowers		Caller	
Mex Shoton Deformation	0.01	Careful and	Ident Christen		and a state of the	
Friction Force Prelant	00	New J0107_1	100 10000	_	Junior Manager	1.00
Ebut	Stirten and Skiling	First Bady Crate			-storage.	12
Input Farons to Friction	100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100 - 100	General Gody ground		1	And a second	11
P Deloat	P Dearton Face 🦳 Reading Movem	Type Tuesday	100 ×		PADERAN	
F. Tarasanal Memore	10		-		cità.	ш.
Friction Inactive During		Force Display Hore	-	_	Hate Flexible	Ц.
T State Excellence		ingener.	Manual I		Dobrite	11
CV SEX STREET		antial C	and internal land		Renaced	
		and and some Party		1.00	(Delection)	Ц.
-	and stated and the	日本美义	0	3.0	Hide	1.0
	OK Appy Curces	OS ANY	Canad I	~		nd.
2	- 20 12 2	hard and a set of	and benerative		BETOL	000
						_

## 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.
- 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.
- 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.





a freshes kines hitt.					del
Tens, 2016/20 Took	···	× BBB .	常調さ物図	Bogan	benament 300
Dodes Desectors Make	rs First Devents	Conter Expension Po	and Scalaton   P	and a	
San G. C. P. a	के आ है	<u>A</u>			
隆温·····王 王 日	然				
Data Kamatra Dp	nam Damenta   Pipel	East Correct Turké			
ncined plans	100101-01010				
Berne Denes Films					
a Qualities stands					
Pres hive been		() (S		18	
Rath	Trunckete Palscatt	the prop .	interaction .	23.9	
0	Parada	to the model .	Text C	Alexant.	
AN I	00 "	4	Abranatara	Guesses	
N N N		42-1	Field Into	. Quete	
2 21 2	100	<b>Lini</b>		CHICK .	
				- 1074	
				10	
			and the second	Time 1.000	-Curam: 4.005+00
				19	1
Sec. 4	14				
and a second sec			- 13	35	25 10
A					





## 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  $\theta = 15^{\circ}$ , a = 0

At  $\theta = 20^{\circ}$ , a = -41.35 in/sec2.

Max Angle for Crate to Slip ( $\theta$ max) = 16.8°. a = -19.19 in/sec2.



#### Closed-form solution



#### Without friction:

$$\Sigma F_x = ma_x : -mg \cdot \sin\theta = ma_x$$
$$a_x = -g\sin\theta$$
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  $(\Theta_{max})$  at which the crate will not slide:  $\Sigma F_{x} = 0: F_{f} - mg \cdot \sin\theta_{max} = 0$   $\mu_{z} \cdot N - mg \cdot \sin\theta_{max} = 0$   $\mu_{z} \cdot mg \cdot \cos\theta_{max} - mg \cdot \sin\theta_{max} = 0$   $\mu_{z} - \tan\theta_{max} = 0$   $\theta_{max} = \tan(\mu_{z}) = \tan(0.30) = 16.7^{\circ}$ 

Once the crate starts sliding,

$$\Sigma F_x = ma_x : F_f - mg \cdot \sin\theta = ma_x$$
$$\mu_k \cdot N - mg \cdot \sin\theta = ma_x$$
$$\mu_k \cdot mg \cdot \cos\theta - mg \cdot \sin\theta = ma_x$$
$$\mu_k \cdot \cos\theta - \sin\theta = \frac{a_x}{g}$$
$$a_x = (\mu_k \cos\theta - \sin\theta) \cdot g$$
For  $\theta = 20^\circ$ ,  $a_x = (0.25 \cdot \cos 20^\circ - \sin 20^\circ) \cdot 32.2$  ft/sec<sup>2</sup>

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

### 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 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.

Name  Matrix  Forward (Design Control	a ferril for Arters 1111	RHDOS	( <b>SA</b>			O Lincomet	100
Image: Provide and Provid	adare Deseactore Adatas	ra   Furse   Deterior	Design Explanature	Phone Mart	arey Diversion	Bearts	
Total of the second		1 🖓 🔸 🕹 🖉	► ₽ ₽ 5	P 🥵 🗞 e	8		
		1 4 A 44	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	C C C			
ementer, Biol Der Piter + Lengen (Kolen) Lengen (Kolen) Le	TENDONS	Anna -	1	-			
	entrality; Dec						
	Landth (M.Sed		4				
Angle (inc)	Halger (Can)	a series and					
Parts	Digitite (Merc)						
mit							
nest • • • • • • • • • • • • • • • • • • •							
east • • • • • • • • • • • • • • • • • • •							
ne la serie de la	reit .	12 22					
1152 Ta Daye	an Iferen (Phese)	125 11					
Ta Dage 📑 🧿 🖓 👘 🖓 👘	0.92	the second second					
	Te Dage	1630 ·····					
		1 Miles					
	en casta		1.1				

## 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 rightclick to hollow it out. You may want to rotate the view to make this task easier.









#### Step 10. Final Model – Compare your Model



🧏 view fit fit month adapts late the other also

■**三國在<b>→**◎ <u>4</u>0 **◎** 

### 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" lcon larger, right-click on Joint:JOINT\_5 and then go to Appearance and then increase the lcon Scale to 15 and click OK.





#### Step 7. Verify your Model.

- 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\*(1cos(360d\*time)). in the Speed Box (Function(time)).



Patatanal Matter: Select the sate local or syledical pr







#### 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 rightclicking 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 rightclick 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.



anteria/Versi Add	ees Ritti				<b>-</b>
De Ein Yew D Debre (Deered	antings Tools 2		ales   Pages   Medicing   Sing	· Caus	S 22.402
And Matters	14 14 Canana Mariana				
H_HACK Showes Usedan	1				
e Bodes Connectore Motore Fators			MIRRO +		
			-Back DOV 2		
Name :	MOTION_1		-Dilate CYUNER 2 +		
and in the	JOB(T 2	1.1.	ELENING NOTICE COMPANY	Soliei	
Avet Type	(weekster)	B.B.	Answer MOTION 2	Mada	
Evertien	fractional +	All and a second se	Auro 20107_2 +	Арреанка	
Extentions	Function +	100 C	Ave: JON1 3	Auto .	
Fundam direct	DECTS " Same	1	View Control .	Cont	
	and the second sec		0.000	Delete	
				Renative	
Tex	Digitizement +			Department	
Uninemet C	1			244	
WERE AN AL	1	1			
De la compañía de la	Party and a state of the	_			
C	Chest Chest				日本 二 年(6)



#### 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.

-		
	Anna an anna an anna an anna an anna an an	
lant.	-lar	

	A A A A A A A A A A A A A A A A A A A	4 4	
	Benefation (and a second seco		
744	le.		

## 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.

- 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  $T = 2\pi (L/g)^{0.5}$ . Plug in L=0.45m and g=9.8m/s<sup>2</sup>, 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**.
- 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**.





La Da Sana Latar Conne La Da Conne La Da Conne La Da Conne La Da Conne	Tende Entre		ti komuti 303 🧣
Disectly marks Rody Calegory Defice Mass Dy Marce (1004) inv (2004)340	Drove Nexs Properties Interpret Inte		
Centre of Maria & Institu Reference Institu	laher Cm. Natar Chi Anny Carel	-Blyand DLIPSO(),2 + Vies Casoli	Appendix Construction Appendix Construction Main Funding Dence Relation Colorcado Hole Hole Colorcado Hole Colorcado

#### 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.

$$\begin{aligned} x_o &= 0 \quad x_f = R \\ y_o &= 0 \quad y_f = 0 \\ V_{x_o} &= 6000 \times \cos 60^\circ = 3000 \text{ mm/sec} \\ V_{y_o} &= 6000 \times \sin 60^\circ = 5196 \text{ mm/sec} \\ y_f &= y_o + V_{y_o}t - \frac{1}{2}gt^2 \\ 0 &= 0 + 5196t - 0.5 \times 9806 \times t^2 \\ 0 &= (5196 - 4905t)t \\ t &= 1.06 \text{ sec} \\ x_f &= x_o + V_{x_o}t \\ R &= 0 + 3000 \times 1.06 \\ R &= 3180 \text{ mm} \end{aligned}$$

### Example 7: Spring Damper - Part 1



M: 187.224 Kg K: 5.0 N/mm C: 0.05 N-sec/mm L0: 400 mm F0: 0

### 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



M: 187.224 Kg K: 5.0 N/mm C: 0.05 N-sec/mm L0: 400 mm F0: 0

### 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.

Channellere Altern 2014				
Ele 26 yes Salings Josh Rober   Consider   Malaer	Turns   East	ins (Pages ) 3	nanco (Mende	6 G () iceard #1
Parag mass 2 Down Quant Flow Common Control Contrel Control Control Control Control Control Control	Houniri I		View Control	Eveni Modily     Apprentin     Monoure     Casp     Monoure     Casp     Fornane     Prentin     Prentinute     Plate
Enven				
*		-		■ <u>□</u> <u></u>

#### 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.
- 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.
- Click on HP8 when selecting the first direction vector. f.
- g. Click on HP13 when selecting the second direction vector.









No cocation, Direction aligned to generatly features. Delect the se









#### 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.



		Contraction of the states	
⇒ L. bertikeiner Germetike	-	Name Money Port Reference Fired	Included) Nonexte (12 Nonexte (12)
Ingenia		Denter Deter Using Deter Using Denter Street	Avg 2 Freder RD <sup>1</sup> bes
Wath Frantises A A References they Barran Tores Reserved Heart 100 1000 B 1000 B	an j r	ing t	Art Dest
nin nio Intyresent Marie Response Trat	Sectory Open Data		



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

- 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

Norking Grid Settings								
Show Working Grid								
Rectangular O Polar								
	Y							
Size	Size (750mm) (500mm)							
Spacing	(10mm)	(10mm)						
	Color	Weight						
Dots	Contrast	• 1 •						
Axes	Contrast	• 1 •						
Lines	Contrast	▼ 1 ▼						
Triad	Solid	•						
Set Loca	Set Location							
Set Orie	Set Orientation							
OK	Apply	Cancel						

A Units Settings							
Length	Millimeter -						
Mass	Kilogram 💌						
Force	Newton 💌						
Time	Second 💌						
Angle	Radian 💌						
Frequency	Hertz						
MMKS MKS CGS IPS							
	OK Cancel						

#### 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 pointsA 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

0

D

Theoretical Solution



Adams solution

X:	Y:
0.0	0.25
V	V.
¥ -	v ·

# 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 Ne	w Model				
Model Name	Model Name cam_follower					
Gravity	Earth Nor	mal (-Global Y	1	-		
Units	MMKS - r	nm,kg,N,s,de	g	•		
Working Directory	D:\Docum	ent\MSC\Can	n Follower\co	mpl		
		OK	Acebu	Canca		

#### 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									
Show Working Grid									
Rectangular O Polar									
	X Y								
Size	(750mm) (500mm)								
Spacing	ng (10mm) (10mm)								
Color Weight									
Dots	Contrast	• 1 •							
Axes	Contrast	▼ 1 ▼							
Lines	Contrast	▼ 1 ▼							
Triad	Solid	•							
Set Loca	ation	•							
Set Orie	ntation	•							
ОК	OK Apply Cancel								

#### Step 3. Closed Body Spline

- a. Under the Bodies ribbon, click on Spline xyz
- 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
х	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



- 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 xyz
- 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
х	-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.

care_follower							-
Apply Parent	Children	Medify	Verbose	Clase	Read from File	Seve to File	Cleve
<ul> <li>Grundlas Jours</li> <li>Moving Jurys</li> <li>Oplinations J</li> <li>Develates Jury</li> <li>Nothers</li> <li>Destance Jurys</li> <li>Degenes of St</li> </ul>	t lappened net south taks tak t	inte deg dang geo	lane of Clandia Gdd1 Joner	e:			

#### 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

A Part Measure			x					
Measure Name:	PART_3_MEA_1							
Part:	PART_3	PART_3						
Characteristic:	CM position	CM position						
Component:	Component: CX CY CZ C mag Cartesian -							
		0	rientation					
Represent coordinate	ates in:							
Create Strip Ch	art							
	OK	Apply	Cancel					

#### 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.



**101** 

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 betadot = 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, thetadot, 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.

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

This material is used by permission of 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.



Joint Motion	×
Name	MOTION_1
Joint	JOINT_1
Joint Type	revolute
Direction	Rotational
Define Using	Function -
Function (time)	-30d - 60.0 * time
Type Displacement IC	Displacement <
<u></u> к	<u>Apply</u> <u>Cancel</u>

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

- a. From **Design Exploration** ribbon, select **Pointto-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,** rightclick 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.**

Point Measure			×			
Measure Name:	cm_R_dot	cm_R_dot				
Point:	piston.cm					
Characteristic:	Translational velo	Translational velocity				
Component: CXCYCZC mag Cartesian 🗸						
From/At:	From/At: © piston.cm © ground Orientation					
Represent coordina	ates in: cylinde	er.cm				
Do time derivatives in: .slider_crank.ground.MARKER_9						
Create Strip Chart						
	OK	Apply	Cancel			

#### 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

slider_cra	ink.joint_2				
Apply	Parent	Children	Modify	Verbose	Clear
Object Object Parent Adams	Name Type Type ID	: .slider : Spheric : Model : 2	r_crank.JO cal Joint	INT_2	

Tunction Builder						
Grade of Modify a Function Measure at manufact, manning11)		C. Fals	aries 🕈 Shot	патез Г	Adama ida	
Deplacement • Assist	Measure Name silder_cra	A FUNCTION	MEA_INEA		1	•.]
Distance along H Distance along T Distance slong T Distance Heplitude Angle about H Angle about T Martin Avent T	General Attributes Units angle Legend	Asis At Label Typo Lower	inkures delault	Cerve Atte Color Thickness Line Type	bules default	
1931 Seguence- Let Schedon 1931 Seguence: Tod Transfor 1932 Seguence: Sed Schedon 1932 Seguence: Sed Schedon 1933 Seguence: Sed Schedon 1934 Seguence: Sed Schedon 1935 Seguence: Sed Schedon 1935 Seguence: Seg	Create Strip Chart     Getting Object Data     Varians	Upper		Bymbel Insert Doge	default 	
AZ( To_Marker , From_Marker )			ОК	Apply	Garce	

#### 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  $\ensuremath{\text{Plot}}$  tracking tool
- e. Follow the plot curve. Find the size measurement at X = 0.0



#### Results

Theoretical Solution



#### ADAMS solution

r = 2.266mr-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

File Type	Adams/Vie	w Command File	(*.cmd)	•
File To Read	D:\Docu nands sen del Upon Comple	ment\MSC\Conto	rls Toolkit\cor	npleted\Lift_M
On Error:	ommand ເ Ign	nore Command C	Abort File	
		OK	Annhy	Canaal

#### 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**
- I. 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**



C Function Rolling	Contract on the local division of the				- E
Defee a runtime function		8.3	full names (* . Sti	otrates C Al	411 KB
AllTong_I_MAG, Sol_Mag)					
Displacement · Assist	1				· .
Distance elong H Distance along T Distance along Z Distance Maptimude Angle about X Angle scout Y					
1011 Sequence: Let Potetion 2011 Sequence: Dod Potetion 2011 Sequence: Dod Potetion	Getting Object Data				
Nill Sequence: Ist Addation Bill Sequence: Ist Donation Dif Sequence: Ist Donation Encluded Angle Nodal Displacement	Markers	•		weet Object	Garne   Vesity
A21 To_Marker , Proro_Marker			DK	Acply	Cancol

#### Step 3. Create a Summing-Junction Block

Σ

Κ

- a. Click summing-junction block tool
- b. Enter .Lift\_Mechanism.theta\_sum
- c. Right-click in Input 1 text field, select controls\_input  $\rightarrow$  Guesses  $\rightarrow$  theta\_desired
- d. Right-click in Input 2 text field, select
   controls\_input → Guesses → theta\_actual
- e. Click OK

Create Controls Block							
f <sub>i</sub> →	Σ	K	1/5	$\frac{1}{s+a}$	s+b s+a		
<u>n(s)</u> d(s)	2nd- order filter	PID	<u></u>				
Name	: Lift_	Mecha	inism.t	heta_s	um		
Input	Su 1:	ımming	g <mark>Junc</mark> t	ion			
+	theta_	desired	į				
Input	2:						
+	theta_a	actual					
→f, ✓ i							
0	ĸ	Ap	ply	CI	ose		

#### 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  $\rightarrow$  Guesses  $\rightarrow$  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**  $\rightarrow$  **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  $\rightarrow$  Guesses  $\rightarrow$  theta\_gain
- e. Click Insert Object Name
- f. The name of the measure should appear in the editor above
- g. Click OK



I anchas Balder					
Define a natime function		C. Full same	es IP, Shot	names C. A	derra ida
Lift_Verhause theta_pain the	a juint				
Math Functions	-				•••
Add Adda Adda Adda Adda Adda Adda Adda	Getting Object Data Messoures	•_ (theta_gam			
Possier Conice Series Possier Size Series	Plot Flat Limits	ii ii		men object	Verty
AZ( To_Marker , From_Marker )	-		OK	Apply	Carcel

#### 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

End Time 💌	2.0
Steps 💌	100
Sim. Type:	Default -
No Debug	
<ul> <li>Update graphics of</li> <li>Interactive C Sc</li> <li>E</li> <li>E</li> <li>Simulation</li> </ul>	display ripted





# 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 rightclick 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:

Front <f> Top <t> Probe <f> Rotate XY <r> Iso <b> Rotate XY <r> Translate <t> Zoom In/Out <z> Zoom In/Out <z> Zoom Box <w> Probe Origin Fit to View <b> Fit to View <b> Fit to View <b> Fit to View <b> Fit to View <b> Hidden Line <b> Working Grid On <g> Working Grid Of <g> Working Grid Off <g> Toggle Icon Visibility <w> Save View Settings Restore View Settings</w></g></g></g></b></b></b></b></b></b></w></z></z></t></r></b></r></f></t></f>	
Top <t> Right <r> Iso <d> Rotate XY <r> Translate <t> Zoom In/Out &lt;2&gt; Zoom In/Out &lt;2&gt; Zoom Box <w> Probe Origin Fit to View <d> Fit to View <d> Fit to View <d> Fit to View <d <="" td=""> Align to 3 Point Align to 3 Point Align to 0bject XY <e> Wireframe <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Off <g> Toggle Icon Visibility <w> Save View Settings Restore View Settings</w></g></g></h></s></s></e></d></d></d></d></w></t></r></d></r></t>	Front <f></f>
Right <r>         Iso <i>         Rotate XY <r>       Translate <i>         Zoom In/Out &lt;2&gt;         Zoom In/Out &lt;2&gt;         Zoom In/Out &lt;2&gt;         Zoom In/Out &lt;2&gt;         Probe          Origin         Fit to View <d>         Fit to View <d>         Align to 3 Point         Align to 3 Point         Align to 0bject XY <e>         Winuframe <s>         Shaded <s>         Hidden Line <h>         Working Grid On <g>         Working Grid Off <g>         Toggle Icon Visibility <v>         Save View Settings         Restore View Settings</v></g></g></h></s></s></e></d></d></i></r></i></r>	Top <t></t>
Iso Iso Rotate XY <r> Rotate XY <r> Rotate XY <r> Translate &lt;&gt; Zoom InvOut <z> Zoom Box <w> Probe  Origin Fit to View   Fit to View   Fit to View   Fit to View <no <ctrl-f="" ground=""> Align to 3 Point Align to 3 Point Align to 0bject XY <e> Wirufname <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings Restore View Settings</v></g></g></h></s></s></e></no></w></z></r></r></r>	Right <r></r>
Rotate XY <r> Rotate XY <r> Translate <t> Zoom In/Out <z> Zoom Box <w> Probe  Origin Fit to View <t> Fit to View <t> Rotate XY <e> Wireframe <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h></s></s></e></t></t></w></z></t></r></r>	lso <b< td=""></b<>
Translate <t> Zoom liv/Out <z> Zoom liv/Out <z> Zoom Box <w> Probe  Origin Fit to View <t> Fit to View <t> Fit to View <t> Align to 3 Point Align to 3 Point Align to 0bject XY <e> Wireframe <s> Shaded <s> Hidden Line <t> Working Grid On <g> Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></t></s></s></e></t></t></t></w></z></z></t>	Rotate XY <r></r>
Zoom In/Out <2> Zoom Box <w> Probe Origin Fit to View &lt;5&gt; Fit to View <no <ctrl-f="" ground=""> Align to 3 Point Align to 3 Point Align to Object XY <e> Wireframe <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Of <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h></s></s></e></no></w>	Translate <t></t>
Zoom Box <w> Probe Origin Fit to View <b> Fit to View   No Ground <ctrl-b> Align to Object XY <e> Wirufname <b> Shaded <s> Hidden Line     Working Grid On <g> Working Grid Of <g> Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</br></v></g></g></g></s></br></b></e></ctrl-b></b></b></b></b></b></w>	Zoom In/Out <z></z>
Probe Origin Fit to View <f> Fit to View <f> Fit to View - No Ground <ctrl-f> Align to 3 Point Align to Object XY <e> Wireframe <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Of <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h></s></s></e></ctrl-f></f></f>	Zoom Box <w></w>
Origin         Fit to View <f>         Fit to View <no <ctrl-f="" ground="">         Align to 3 Point         Align to Object XY <e>         Wireframe <s>         Shaded <s>         Hidden Line <h>         Working Grid On <g>         Working Grid Off <g>         Toggle Icon Visibility <v>         Save View Settings         Restore View Settings</v></g></g></h></s></s></e></no></f>	Probe
Fit to View <5> Fit to View < No Ground <ctrl-f> Align to 3 Point Align to Object XY <e> Wireframe <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Of <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h></s></s></e></ctrl-f>	Origin
Fit to View - No Ground <ctrl-i>         Align to 3 Point         Align to Object XY <e>         Wireframe <s>         Shaded <s>         Hidden Line <h>         Working Grid On <g>         Working Grid Off <g>         Toggle Icon Visibility <v>         Save View Settings         Restore View Settings</v></g></g></h></s></s></e></ctrl-i>	Fit to View <f></f>
Align to 3 Point Align to Object XV <e> Wireframe <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h></s></s></e>	Fit to View - No Ground <ctrl-f></ctrl-f>
Align to Object XY <e> Wireframe <s> Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h></s></s></e>	Align to 3 Point
Wireframe <s>         Shaded <s>         Hidden Line <h>         Working Grid On <g>         Working Grid Off <g>         Toggle Icon Visibility <v>         Save View Settings         Restore View Settings</v></g></g></h></s></s>	Align to Object XY <e></e>
Shaded <s> Hidden Line <h> Working Grid On <g> Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h></s>	Wireframe <s></s>
Hidden Line <h> Working Grid On <g> Working Grid Off <g> Taggle Icon Visibility <v> Save View Settings Restore View Settings</v></g></g></h>	Shaded <s></s>
Working Grid On <g> Working Grid Off <g> Taggle I can Visibility <v> Save View Settings Restore View Settings</v></g></g>	Hidden Line <h></h>
Working Grid Off <g> Toggle Icon Visibility <v> Save View Settings Restore View Settings</v></g>	Working Grid On <g></g>
Toggle Icon Visibility <v> Save View Settings Restore View Settings</v>	Working Grid Off <g></g>
Save View Settings Restore View Settings	Toggle Icon Visibility <v></v>
Restore View Settings	Save View Settings
The second se	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.

# Image: Table (1980) Image: Table

#### 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 lso shortcut <l> 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 rightclicking 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.



1 Islamator						
Apply Parent Child	dee   Musify  - Vete		Dear   Read for	ofile   3	Space to Film	Ome
Populary of small reals Second State ground Dame, St. Carrow, Social Tanae, Social, 1 Soc., Social, 3 Soc., Social, 3 Soc., Social, 3 Soc., Social, 3 Soc., Social, 3 Soc., Social, 3 Soc., Social, 3 Social, 4 Social, 3 Social, 3 Social, 4 Social, 3 Social, 4 Social, 3 Social, 4 Social, 3 Social, 4 Social, 4 Social, 3 Social, 4 Social,	encentra 2no en Gonorita 7527e es comento 2notar es manatos 2no es consento 2no es consento 2no es consento 2no es consento 2no es consento 2no es consento 2no es	th Care : th garward to th growth to th growth to th factor : th factor : th factor : th garward :	lieve (Leve) Constantional Para Sevelate Johns Dylane Printran Dylane Printrany Relevant Paral	ant Salasa Jalasa Jalasa Jalasa Jalasa		
SALANCES PERFECTION Constitute of the second Constitute of the seco	tanantalar terjan prostanta depace of initiality grand dara dara dara dara dara dara dara	C)	development first	2001		
These are no reductant	enerrent epatore					
Rodel vesifies ascess	NCAY-					

#### 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.

Bodies Connectors Motions Forces Elements Design Exploration Plagms Seru	lation Results
Situp Simulate	
Simulation Control	
End Time  2	
Steps 🔽 100	
Sim. Type: Default 💌	
☐ Start at equilibrium	
Reset before running	
No Debug	
Nastran 🗮	
Update graphics display	
Interactive C Scripted	
톤 名 📕 🖽 🔛	
Simulation Settings	

#### 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.

Save Run Results	x
Name: first_results	
Auto-Increment Name	
OK Cancel	
Simulation Control	
End Time  2	
Steps	
Sim. Type: Default 💌	
Start at equilibrium	
Reset before running	
No Debug	
Nastran 🗮	
Update graphics display	
Interactive C Scripted	
🖹 😤 📕 🗠	
Simulation Settings	

#### 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  $\ensuremath{\textbf{Reset}}$  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.

<u>F</u> ile	<u>E</u> dit ⊻iew Settin <u>g</u> s <u>T</u> ools	
-	<u>N</u> ew Database Ctrl+N	
	<u>O</u> pen Database Ctrl+O	
	<u>S</u> ave Database Ctrl+S	
	Save Database <u>A</u> s	
	Sim <u>M</u> anager	×.
	Import	
	<u>E</u> xport	
	Print Ctrl+P	
	Select <u>D</u> irectory	
	E <u>x</u> it Ctrl+Q	

File Type	Adams	View Comm	and File	-
File Name	valve1			
Model Name	valve			
Use Parasolid	As is	-		

#### **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



- 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.

	Open an Existing Model	
File Name	nod_21_camrocker/walve_train_start.cmd 🔗	
	Use File Directory as Working Directory	
Working Directory	s_2013/exercise_dir/mod_21_carrrocked	
		-
F Echo Command	is	
□ Update Screen		
Display Model	Jpon Completion	
On Error		
C Continue Com	nand 🙆 Ignore Command C Abort File	



#### 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, 100step** 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.



#### valve\_train • Browse Groups Filters 🗄 🔚 Bodies 🗄 🧀 Connectors 🖻 🦳 Motions Modify MOTION 3 Appearance cam\_motion Info 🗄 🔚 Forces Measure 🗄 🤚 Elements 🗄 🧀 Measures Copy ÷.. Design Variables Delete 🗄 🦳 Simulations Rename 🗄 🧀 Results All Other (De)activate ÷... Hide



#### 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



Select

QK Apply Close

- 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.

Friction Force	Coulomb
Coulomb Friction	On 🔽
Static Coefficient	0.08
Dynamic Coefficient	0.05
Stiction Transition Vel.	(1(mm/sec))
Friction Transition Vel.	(2(mm/sec))
	OK Apply Close

Contact Name	.valve_train.cam_contact
Contact Type	Point to Curve
Marker(s)	ref_marker
Curve(s)	GCURVE_201
Direction(s)	GCURVE_201 -





A Activate/E	A Activate/Deactivate				
Object:	.valve_tra	iin.CVCV_1			
🗖 Ођ	Dbject Active				
🗖 Ођ	ect's Depen	dents Active			
	<u>0</u> K	<u>A</u> pply	<u>C</u> ancel		

#### 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.**



Name	SPRING_1
Action Body	valve
Reaction Body	ground
Stiffness and Dan	nping:
Stiffness Coeffici	ent • (20(N/mm))
Damping Coeffici	ent • (2.0E-003(N-sec/mm))
Length and Preloa	d I too
Preload	100
Default Length	(Derived From Design Position)
Spring Graphic	On, If Stiffness Specified
Damper Graphic	On, If Damping Specified -
	On Action Body
Force Display	a with r deployed we way
Force Display	

#### 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.



A Joint Motion	×
Name	cam_motion
Joint	Cam_Ground_Jt
Joint Type	revolute
Direction	Rotational 💌
Define Using	Function 💌
Function (time)	-50*360d*time
Туре	Displacement 🔻
Displacement IC	
Velocity IC	
<u>о</u> к	Apply Cancel

# 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.



#### Assist Measure Name Adve\_Hain FUNCTION\_MEA\_1 Force in Object Joint Bares Joint Force House Force Point-for-Corne Force Course-to-force Force Charge to -force Engle compenses Force Three-component Force Three-component Force Three-component Force Holispector Force Basis force Basis force Basis force Contacts Force General Attributes Axis Attributes Curve Attributes • Label Coinr Units force - Thickness Legend Type default Line Type default Lover P Create Ship Chart Symbol default Upper **Getting Object Data** Markers • insert Object Varie field Rocce furing-bangwir focom Friesion Forom Verily

OK Apply Cancel

#### 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.

A Modify a Spring-Damp	er Force		×
Name SPRI	NG_1		
Action Body valve			
Reaction Body groun	d		
Stiffness and Damping:			
Stiffness Coefficient	▼ (110(N/r	mm))	
Damping Coefficient	▼ (2.0E-00	03(N-sec/mm	))
Length and Preload:			
Preload	( 400 N	)	
Default Length	(Derived)	From Design	Position)
Spring Graphic On, If	f Stiffness Sp	ecified -	]
Damper Graphic On, If	Damping Sp	ecified -	
Force Display On A	ction Body	•	]
1			
	ОК	Apply	Cancel
	-		

CONTACT( Contact\_Name On\_Body Component Ases )

#### 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



Import MNF	Create New

ViewFlex - Create	×
Part to be meshed	valve_train.valve
Material	.valve_train.steel
Number of Modes	6
Stress Analysis	
Manual Replace	
Advanced Settings	
QK	Apply Cancel

- 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



Veullei - Ceste			
Pat to be meshed	salve_base set	14	Lood A/17 in
Material	salve train.et		
Number of Modes	16		
Manual Replace			
Adveced Settings			
PauBody Type 17.1	Establish F. Gelated	ing C. Import	Mesh Paulbols Name Value fee
* Mesh-Freperties	Attachments M	web percent	
Element Type	Electrent Size	12	C Stress Artifices
Daid .	Mmmm Size	0.4	That Sowy Laws Tag 1
Element Shipe	Crewth Rate	15	terral Devices
Tetahedral •	Shell Papiness	(thereas)	Calapse Shall Edges
Element Order	Angle Per Elemen	(46degi	Edge Telesures (1) linear
Lung .	1 B	Section.	Construction of the second
	· · · · · · · · · · · · · · · · · · ·	arrest C.T.	DVI # CPF
Depert Specification	MSN Based Right		
Depert Specification	Curreture Based Fight	kaleg C.G	24 × 049
Denert Specification	Currenture Based S	Icales (Fil	or + Ore

- i. From the Tools menu, select Database Navigator.
- j. Change Browse to Graphical Topology.
- k. Highlight **Valve\_flex** part.
- I. Notice that the joints and spring are now attached to the flexible valve part.

Tools	
Comma	nd Navigator
Databas	e Navigator.
Eunction	Builder
Table Ed	Sitor
Plugin M	lanager.



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

	L K	× -
End Time	• ( 1/50 )	
Steps	• 100	_
Sim. Type:	Default	٠
<u>▶</u> <u>₹</u>	7 <u></u> <u>.</u>	
Nata		Ħ
Update graphic	s display	_
B. A	Scripted	

#### **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.

## **Demonstration Information**

Setting Up Model

Simulate

Investigate Results



#### 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.

# Demonstration Information Setting Up Model

### Simulate

### Investigate Results

Stamper_Simulate
Step 2: Simulate the Model
Single C Design Study C Optimization
Output Step Size: Large
Model Update: At Every Output Step 💌
Apply Close Help
Output Step Size: Large

# Apply Close Help

Never

#### Step 4. Investigate the Results

To investigate the results:

Model Update:

- 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**.

Demonstration Information	
Setting Up Model	
Simulate	
Investigate Results	
Stamper_Investigate	





#### 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.

por	Martine .						
2.1928							
Λρρίγ	Parent Children	s Neddy -	Verbooa	Clear Rea	el from File	Save to File	Clees
2milije	Buaty Samery		-			-	
Nodel 1 Date D	Cause : starp	4-88-11F		Information			
Object.	1744		13	Design	n Study op	eration was	s successful.
					1000 C 1000 C 1000	(1997) (1997)	
047	Hitisian of starg in	41304		~			
01)	Minimum of stamping Dains : int Manimum Walker : Minimum Walker :	40.0765 (%+5	al 1)	~			
01) Devige	Histon of starp in Date : int Masteur Yelver History Yelver Vertables	40.1765 (%45 -8.42215 (%45	al 11 al 5)	~	Clo	se	
01) Device V33	Histonia of stang is Date - as Mastenia Talus - Mastenia Talus - Martables contest_libb _ senot Date - as	eligini 60.3766 (huis -4.62225 (huis h	al 1) al 5:	~	Clo	se	
01) Devior V33 Trae3	Hintows of stang in Dates	40,0766 (1446 -9,40215 (1446 -9,40215 (1446 1	al 1) al 5) Denelairta	~	Clo	se	
OLI De+1gn VAI Trise3	Hintower of steam is Diskse : an Hantower Viles : Hintower Viles : Vertebles envert, tild jengel Disks : an steamy_knight 23.774	40,0766 (144) -4,42215 (144) -9,42215 (144) 19 19	al 1) al 5) Denelniminy -L. 1427	~	Clo	se	
01) Devige Vik) Trisel 1 2	Hardman of stear, by Hardman Viller Hardman Viller Hardman Viller Workshes entropy, stable, server Dates entropy, based at starts the starts the starts th	42, 3793 (144) -42, 42213 (144) -42, 42213 (144) 0 79, 299, 65 244, 61	al 1) al 5) Genetativity -1.347	~	Clo	58	
01) Design Vill Trifel 1 1 2	Hintman of stamp, by Datase - an Hantanan Talast Hantanan Talast Unrealites entropy_lation assay_lation at 23, 274 47, 318 33, 354	40.0116 (144 -4.40016 (144 -4.40016 (144 ) -4.40016 (144) -4.40016 (144) -4.400016 (144) -4.4000000000000000000	at 1) at 5) Genetativity -1.847 -1.847 -4.120	~	Cio	se	
01) Devign Vill Tratel	Hintman of stamp, by Datase - an Hantana Tribusi Hantana Tribusi Unites - the senergilander di 214 ar 216 ar 216 a	61.011 69.0116 144 49.0215 144 71. 79. 79. 79. 79. 79. 79. 79. 79. 79. 79	at 1; at 6; Genetativity -1, 4027 -1, 4027 -1, 4027 -1, 4027	~	Clo	se	



#### 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.



Your optimization has been successful. Control Link Length = 267.88 Maximum Penetration = 4.0
OK

#### **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.
- 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.

Morkin	ng Grid Settin	igs	x
Show	Working Grid		Ш
<ul> <li>Recta</li> </ul>	angular 🤉 P	olar	1
	х	Y	
Size	(750mm)	(500mm)	)
Spacing	(10mm)	(10mm)	
	Color	Weig	ht
✓ Dots	Contrast	• 1	•
Axes	Contrast	• 1	•
Lines	Contrast	• 1	٠
Triad	Solid	•	
Set Loca	ation	•	
Set Orie	ntation	•	
ок	Apply	Cance	ei i

### 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 dropdown 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.
- 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.

1

 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 x20. Switch to top view.
- b. Use the **Rigid Body: Box** to build boxes as shown below by selecting the appropriate corner locations.
- 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.



Modify (	Coupler			×
Name	OUPLER_1			
Т	wo Joint Coupler	By Scales	•	
	Joint	Freedom T	ype	Scale
Driver	JOINT_5	Rotational	· -1.	D
Coupled	JOINT_6	Rotational	· -1.	0
-				
12		ОК	Apply	Cancel
_				

### 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.
- 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.

Modify Torque	×
Name	SFORCE_3
Direction	Between Two Bodies
Action Body	manipulator_base
Reaction Body	gripper_right
Define Using	Function
Function	0.0
Solver ID	1
Torque Display	On Action Body
	<u>QK</u> <u>Apply</u> <u>Cancel</u>

### 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=x) and x=x1.

#### Format

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

#### Arguments

x	The independent variable. It can be a function expression.
×0	A real variable that specifies the x value at which the STEP function begins.
×1	A real variable that specifies the $\boldsymbol{x}$ value at which the STEP function ends.
h0	The initial value of the step.
hi	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:

$$\alpha = h_1 - h_0$$
  

$$\Delta = (x - x_0) / (x_1 - x_0)$$
  
(h\_0)

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

٦

#### 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.

Modify Contact		
Contact Name	gripper_contact_left	
Contact Type	Solid to Solid	
I Solid(s)	LINK_6	
J Solid(s)	BOX_10	
Force Display	Red	
Normal Force	Impact	
Stiffness	1.0E+005	
Force Exponent	2.2	
Damping	1000.0	
Penetration Depth	0.1	
<ul> <li>Augmented Lagrang</li> </ul>	ian	
Friction Force	Coulomb	•
Coulomb Friction	On	-
Static Coefficient	1.0	
Dynamic Coefficient	0.7	
Stiction Transition Vel.	100.0	
Friction Transition Vel.	1000.0	



#### 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)

		. 4	Iread	y Don										
Movement Description	Si Mov	ep1 ve Into sition	Sto Grig Cu	np2 o the iba	Ste U	ip 3 ift	Ste Mov platf	re to orm2	Sto Lowe pla	ep5 er into acc	Ste Rel	op6 case	Sto Ra mani c	ise pulat r
Time	жO	*1	80	81	×0	81	ж0	*1	ж0	*1	89	*1	жŨ	81
Values	0	1	1.1	2	2	3	2.5	3.5	3	4	4	4.1	4.1	5
Motion/ Torque Values	hO	hi	hQ	hā	hQ	h1	hQ	hi	hQ	hī	ы	hi	hØ	hi
meter_3	0	-40d	N/A	N/A	•	20d	N/A	N/A			N/A	N/A		
mator_2	0	-110d	N/A	N/A	•	40d	N/A	N/A			N/A	N/A		
metor_3	0	60d	N/A	N/A	0	-60d	N/A	N/A			N/A	N/A		
metor_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
arlip 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

		۵	head	y Don	<b>e</b> )(		1							
Movement Description	St Mov	ep1 re into sition	Sto Grig cu	ep2 o the ibe	Ste Li	p3 ft	Sto Mov platf	ep4 ve to orm2	Sti Lowe pli	ep5 er into ace	St Rel	ep6 ease	Sto Ra man to	rp7 ise ipula or
Time	жD	<b>x1</b>	жÐ	*1	жD	xt	жО	xt	жD	*1	ж0	*1	жО	<b>x1</b>
Values	0	1	1.1	2	2	3	2.5	3.5	3	4	4	4.1	4.1	5
Motion/ Torque Values	hO	h1	hØ	h1	h0	h1	h0	h1	hū	h1	hū	h1	hū	h1
motor 1	0	-40d	N/A	N/A	0	20d	N/A	N/A	0	-20d	N/A	N/A	0	20d
mattra 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
sustor_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
arity 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.

Create New Model	×
	Create New Model
Model Name	Basketball
Gravity	Earth Normal (-Global Y)
Units	MKS - m,kg,N,s,deg
Working Directory	C:\Users\wwei\Work\02_second_edition
	OK Apply Cancel

#### 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  ${\bf F}$  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.

Create Contact	X
Contact Name	.Basketball.CONTACT_1
Contact Type	Solid to Solid
I Solid(s)	ELLIPSOID_1
J Solid(s)	TORUS_2
<ul> <li>Force Display</li> </ul>	Red
Normal Force	Impact 💌
Stiffness	1.0E+008
Force Exponent	2.2
Damping	1.0E+004
Penetration Depth	1.0E-004
Augmented Lagrang	ian
Friction Force	None
	OK Apply Close

### Step 7. Create Design Variables.

a. From **Design Exploration** ribbon, select **Design Variable: Create a Design Variable**.

- b. Name the variable  $\boldsymbol{x}\_\boldsymbol{vel}.$
- 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.

Name x_vel	
Type Real	▼ Units velocity ▼
Standard Value	5.0
Value Range by	Absolute Min and Max Values 🔹
Min. Value	5.0
Max. Value	10.0
Allow Optimiza	tion to ignore range
List of allowed	values
<b>E</b>	OK Apply Cancel

### Step 8. Impose Initial Velocities to the Ball.

a. From the model tree, right click on part Ball and select Modify.

Modify Body	A DESCRIPTION OF THE OWNER OWNER OF THE OWNER OWNER OF THE OWNER OWNE			*
Body	Ball			
Category	Mass Properties			2
Define Mass By	Name and Position Mass Properties			
Material Type	Position Initial Conditions			
Density	Velocity Initial Conditions Ground Part			
Young's Modulus	2.07E+011 newton/meter**2			
Poisson's Ratio	0.29			
			Show calcu	lated inertia
	0	ж	Apply	Cancel

- b. Under Category popup menu, pick **Velocity Initial Conditions**.
- c. Check X axis and Y axis.

#### Modify Body Body Ball Category . Velocity Initial Conditions Translational velocity along Angular velocity about Ground C Marker Part CM C Marker Q. X axis Clear Ċ, Y axis Cut Сору Z axis Paste **Create Design Variable** Field Info Expression Builder ... Unparameterize OK Apoly Cancel

- 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.

Modify Body		*
Body	Ball	
Category	Velocity Initial Co	nditions
Translational velocity along		Angular velocity about
Ground C	Marker	Part CM C Marker
🗟 X axis (L	Basketball.x_vel)	□ X axis
I⊽ Y axis (E	Basketball.y_vel)	T Y axis
T Z axis		T Z axos
~		
2		QK Apply Cancel

#### 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.

Model:	Basketball
Simulation Script	Basketball Last_Sim
Study a	Measure C Objective
Minimum of	MEA_PT2PT_1
🔿 Design Study 🔿 De	sign of Experiments @ Optimization
Design Variables:	x_vel Y_vel
	Auto. Save Save Restore
Goal:	Auto, Save Save Restore     Minumize Des Meas / Objective
Goal: I‴ Constraints:	Auto. Save Save Restore
Goal: Constraints Settings Displa	Auto. Save Save Restore     Minumize Des. Meas. / Objective      y. Output. Optimizer

- 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.



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.

Ale and 2 bedaudaut unsettation equations.		
This mestaint		unnecessarily measures this 507
Control Judice paper, a bides to flay Control further tofined a bottom piller to flay	(Cylindrical Joint) (Tigad Joint)	Autobium Dervens 21 e NJ Jutation Batrann 21 : 23

f. Close the information window and make the body

flexible again. Pick partial inertia modeling.

Make Flexible	<b>X</b>
Inertia Modeling	
○ Constant ⊙	Partial C Full C Custom
1	OK Cancel

#### 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.

<u> </u>	
➡	Modify
	Appearance
	Measure
	Сору
🗄 🔮 FLEX_BODY_1	Delete
- 💳 Connectors - 💳 Motions	Rename
	(De)activate
- CSFORCE_2	Hide
SFORCE_1	
<sup>I</sup> 9↓ACC	
- Elements	

A Modify Force	×
Name	SFORCE_1
Direction	On One Body, Fixed in Space
Body	FLEX_BODY_1
Define Using	Function
Function	step(time, 0,0,1,-1500)
Solver ID	1
Force Display	On 💌
<b>1</b>	OK <u>Apply</u> <u>Cancel</u>

#### 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.



Save Run Results		
Name:	flex	
	Auto-Increment Name	
	OK Cancel	

- b. Make the body rigid and run the same scripted simulation again. Save the simulation result as rigid.
- c. Press  $\ensuremath{\textbf{F8}}$  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 massspring 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.

A Save	Run Results
Name:	pendulum
	Auto-Increment Name
	OK Cancel

e. Click plotting to enter postprocessing window.



f. In plotting page, make sure that **Objects** is selected as the source.

Data	Math
Model	
. Per	ıdulum
Source	Objects 💌
Filter	*

g. Then, hold Ctrl key and highlight both **bob.cm** and **Mass\_block.cm**.

Model		Filter	Object	
	i lium	body	- bob	
		CONSTRAINT	MADERE_3 MADERE_4 MADERE_11 - Mass_block	
Source	Objects	•	MARXIR_12 MARXIR_13 MARXIR_14 * Yout	
Filter	+		+ fixed_rod_bob	

h. Then select **Translational\_Displacement** and **X** component. Click **Add Curves**.



2.0

25

2014-07

1.0

11

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**.

🚺 Gravity S	Gravity Settings						
Gravity							
X 0		-X*	+Χ*				
Y -1622		-Y*	+Y*				
Z 0		-Z*	+Z*				
* Set value in global	* Set values for Earth gravity in global coordinates.						
ОК	0	ance					

- 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->Gravity 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.

Pendulum	Modify Design Venable
Browse Groups Filters	Name initial_angle Type Real  Units angle
Occas     Connectors     Connectors     Connectors     Connectors     Devents     Design Variables <u>Variables     Variables     Variables </u>	Standard Value 60 Value Range by +/- Delta Relative to Value - Delta -1.0 + Delta 1.0 Allow Optimization to ignore range List of allowed values OK Apply Cancel

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.
- 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.

Data Multi				
Semister .	1 fear	Result Set	Groupered	17 IM
personal de la companya de la compan		Set Print Transform (and Transform) Set (	1	Add Curren To Curren Flor - Ower Flor
al Some Flood for		sa kiini 5 BADA_5 (Remo Jonacor	78.	independent Ann 4 - Sens / Data
File: F		13	41	



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



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.
- 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)\*st ep(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.

Name	SFORCE_1				
Direction	On One Body.	Moving with Body			
Body	InertiaWheel				
Define Using	Function				
Function	step(time, 0 1,0,0.15,100)*step(time,0.15,100,0.2,0)			d	
Solver ID	1				
Force Display	On				
			ж	Apply	Cancel

### 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.

Create or Modify a Function Measure	in Full names (* Shot names (* Adams ids
AF (InertialBool . m) fenné	

## 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**.

Tute					
1144	Speak Sheet				
File Name	ev.		+		
Result Set Name	FUNCTON NEA.1	Think for		Put	a
		Tec	+	Burns	
		Parameterze		George +	Dynian Rat
		QyoLast, Rei FUNCTION, MEALT	۰.	Oosta	FLACTOR NO.
		FullTitle	+		SPORE_1
	3				1_1000
		OK Appy Carcial			2000 2

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.

aytes - Romped	teres de la companya
Tak Life Turonet Tem Help "(No Title)"	14
2014-06- 4 10:41:54 "Result IO" "Result Type" I Marker ID" "Narker ID" "Result Title" "FARTION-VEA_1" (N/A)" (N/A) (N/A)	mient)*
<ul> <li>Gyr 0. LaST_BUR_FUNCTION_MEA_LITIME" Gyr 0. LaST_Ram, FUNCTION_MEA_LITY" Gyr 0. LaST_RAM, FUNCTION_GYR 0. LAS</li></ul>	

# 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.

	Create New Model	
Model Name	Power_Hacksaw	
Gravity	Earth Normal (-Global Y)	•
Units	MMKS - mm,kg,N,s,deg	•
Working Directory	C:\Users\	

### 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.

Workin	g Grid Setting	s X			
Show Working Grid					
<ul> <li>Recta</li> </ul>	angular C P	olar			
	Х	Y			
Size	(400mm)	(150mm)			
Spacing	(5mm)	(5mm)			
	Color	Weight			
Dots	Contrast	• 1 •			
Axes	Contrast	• 1 •			
🗖 Lines	Contrast	• 1 •			
Triad	Solid	•			
Set Location					
Set Orie	Set Orientation				
ОК	Apply	Cancel			

### 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.

File Import	
File Type	Parasolid (*.xmt_txt, *.x_t, *.xmt_bin, *.x_b ▼
File To Read	\\engin-labs.m.storage.umich.edu\shenxin\windat.v2\
File Type	ASCII
Model Name	Power_Hacksaw
	OK Apply Cancel

### 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



# 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  $\ensuremath{\text{Normal Force}}$  to  $\ensuremath{\text{Restitution}}.$
- e. Set Penalty to 1.0E+010. Set Restitution Coefficient to 0.5.
- f. Select Coulomb as Friction Force.
- g. Click **OK**.

Contact Name	CONTACT_1				
Contact Type	Solid to Solid	7			
I Solid(s)	SOLID5				
J Solid(s)	BOX_8				
Force Display	Red 💌				
Normal Force	Restitution	•			
Penalty	1.0E+010				
Restitution Coefficient	0.5				
Restitution Coefficient	0.5				
Restitution Coefficient	0.5 ian				
Restitution Coefficient Augmented Lagrang Friction Force	0.5 ian Coulomb				
Restitution Coefficient Augmented Lagrang Friction Force Coulomb Friction	0.5 ian Coulomb On	•			
Restitution Coefficient Augmented Lagrang Friction Force Coulomb Friction Static Coefficient	I an Coulomb On 0.3	•			
Restitution Coefficient Augmented Lagrang Friction Force Coulomb Friction Static Coefficient Dynamic Coefficient	0.5 ian Coulomb 0.3 0.1	•			
Restitution Coefficient Augmented Lagrang Friction Force Coulomb Friction Static Coefficient Dynamic Coefficient Stiction Transition Vel.	0.5 ian Coulomb On 0.3 0.1 100.0	•			

### 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**...
- 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.



Juntas Balder						
Create or Modily a Function Measure	e -	C Fells	ames # Shot	unes (* .	Adams kits	
aranany, aranya						
Vath Functions 🔭 Ass	et Measure Name Power H	ickaan Angle.)	Lini/2toOase		-	•
285	· General Attributes	Asis Al	litudes	Curve Attr	DURS	
ADAT	Units angle	+ Label		Cotor		
ADIN	Legend	Тури	debuit 👲	Thickness	0.0	1.7
ATMI ATMI		Lower	0.6	Line Type	default	
Chakyshov Inlynamial	P Dyets Skip Charl	Upper	0.0	Symbol	default	*
CORM	Getting Object Data					
LELAY	Markans •				-	-
EXP Reprise fination Barries	X 55			Inset Oby	ct Neve	
Sundier Size Settes Enversion Step	- Plot Plat Limits				Verty	1
A2) To Mather, From Mather ]	an a sendari ma		ox	Acely	Carcel	1

### 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.

T(c)



# 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**.

**[0]** 

- 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  $\ensuremath{\textbf{Plotting}}$  .
- 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  $\ensuremath{\text{Measure}}$  as  $\ensuremath{\text{Source}},$  and then select  $\ensuremath{\text{stroke}}$
- e. Click Add Curve.

Simulation Contro	
	3 <
End Time 💌	15.0
Step Size 💌	0.1
Sim. Type:	Default 🔹
Start at equilibriu  Reset before run No Debug	ning I I P [A B]
Nastran	
Interactive ⊂ So	cripted
<u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u><u></u></u>	
Simulation	n Settings

## **Step 17. Compare Results**

#### **Theoretical Solution**





#### **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
- Iink.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.

	Create New Model	
Model Name	Walking_Beam_Indexer	
Gravity	Earth Normal (-Global Y)	•
Units	MMKS - mm,kg,N,s,deg	•
Working Directory	C:\Users\	

## 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**.

Workin	g Grid Setting	s X
Show	Working Grid	i 🛄
<ul> <li>Recta</li> </ul>	angular C P	olar
	Х	Y
Size	(250mm)	(250mm)
Spacing	(5mm)	(5mm)
	Color	Weight
Dots	Contrast	• 1 •
Axes	Contrast	• 1 •
🗖 Lines	Contrast	• 1 •
Triad	Solid	•
Set Loca	ation	•
Set Orie	ntation	•
OK	Apply	Cancel

## 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
02	57	82	-10
04	-51	82	0
06	-128	0	-10
В	10.517	7.641	0
F	-100	136	10
D	-62.695	120.252	0
С	-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  $\ensuremath{\textbf{Spur}}$  in  $\ensuremath{\textbf{Gear}}$   $\ensuremath{\textbf{Type}}\xspace$  , and then click  $\ensuremath{\textbf{Next}}\xspace$  .
- c. Choose **Detailed** in **Method**, and then click **Next**.
- d. Set the parameters in the  $\ensuremath{\textbf{Geometry}}$  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



File Type	Parasolid (*.xr	nt_txt, *.x_t, *	xmt_bin, *.x_	b 💌
File To Read File Type Part Name	Vengin-labs ASCII Crank	.m.storage.um	ich.edu\shenx	in'windat.v2
Location Orientation	(LOC_RELA	TIVE_TO({0.0.	0}. O2))	
Relative To	emblies			_
		ОК	Apply	Cancel

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

GEAR1	ure Angle 20	Axi	a of Rotation	Giobal Z	+100	10450		
GEARI				1.1.1	-			
			1		GEAR2			
	Driver.		Name		Driver		External	
örg			W New C	Existing				
er Location	0000-50			Center Los	ation	-51.82	5	
Teath	72			No. of Tea	0	72		
Width	10			Gear Widt	h	10		
Radus	5			Dore Radi		10		
	Standard •			Profile		Standar	d 💽	
shit Coof	0.0			Profile Shi	t Cost	0.0		
ndum Factor	1.0			Addendum	Factor	1.0		
ndum Factor	1.25			Dedendur	Factor	1.25	6	
	ting ar Location d'Teath Width Radux e a Shit Caef noum Factor noum Factor	ing ar Location 0.0.8 (C-5.0 af Teeth 72 Width 19 Radus 5 e Standard • a Shift Cael 0.0 robum Factor 10 notum Factor 125	ting er Location 0.0.9.65.0 d Teath 72 Viriah 19 Radux 5 e Standard • e Standard • noum Factor 10 noum Factor 125	ing # New f ar Location 0.0.8.6.5.0 # Teeth 72 Width 19 Radus 5 e Standard • a Shift Coeff 0.0 noum Factor 10 noum Factor 125	ting Pactor 125	ting Pactor 00.000 €.5.0 Profile Prof	ting # New C Existing er Location 0.0.8.6.5.0 d Teath 72 Width 19 Rudux 5 e Standard • s Shit Coef 0.0 noum Factor 1.0 noum Factor 1.25 Center Location 51.82 No. of Teath 72 No. of Teath 72 Cent Width 10 Bore Radius 10 Profile Standar Addendum Factor 10 Dedendum Factor 125	If Standard         If Steve C         Existing           or Location         0.0.8.6.5.0         Center Location         51.82.5           All Teeth         72         No. of Teeth         72           Width         19         Cear Width         10           Rudux         5         Bore Radius         10           e         Standard         Profile         Standard         -           e Standard         Profile         Standard         -           noum Facter         1.0         Addendum Facter         1.0           noum Facter         1.25         Dedendum Facter         1.25

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.
- 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.
- I. Select Platen\_Left.cm.Y and click OK.

- m. Click Unparamerized.
- n. Rename the new part as Platen\_Right.

#### Step 7. Create Eccentric Cam

- 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.

Add to I	Part 🔻	
Profile	Curve	•
TTOMIC	10 10 10 10 10 10 10 10 10 10 10 10 10 1	
Tome		
Path	Forward	•

## 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.

1 <sup>st</sup> Body	2 <sup>nd</sup> Body	Joint Location
Place_Arm	ground	06
Place_Arm	Link	с
Link	Driver_1	В
Walking_Beam	Driven_1	D
Walking_Beam	Crank	Walking_Beam.SOLID15.E64
Crank	ground	02



### 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.





### 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.

walking_t	peam_index	ter.joint_6	
Apply	Parent	Children Modify 🗆 Verbose	Clear
Object	Name	: .Walking_Beam_Indexer.JOINT_6	
Object	Type	: Revolute Joint	
Parent	Type	: Model	
Adams	TD	: 6	
Active		NO OPINION	
I Mark	er	: .Walking_Beam_Indexer.Flace_Arm.	MARKER_10
J Mark	er 1. Candini	: .waiking_beam_indexer.ground.mAs	KKER_1/
Initia	lar Digpl	agement - NOT SET	
	THE DISCI	ICement . NOI SEI	

I function Builder						
Create or Modily a Function Measure		C.Full	urres 🕫 Shot	патиз С	Adama ida	
ALTRADUCT, 18, DARKE, 17						
Deplacement · Assist	Manuse Name Walking_Be	am_hidaser.	Angle_PlaceArm	toGround	7.1	•.
Distance along I	General Attributes	Avia A	linitures	Curve Atte	bure	
Distance along 2	Units ne_units	· Libel		Celor		
Angle about N	Legend	Туре	detault	Thickness	1	
Angle about T		Lower	escore a co	Line Type	default	
8821 Sequence: Let Schanish	Grate Stip Chart	Upper	16	Badrate	default	
Dill Seguence: Ind Interior	Getting Object Data					
2013 Seguence: Let Schabion 2013 Seguence: Ind Schabion	Variers •				325	
2010 Repairies and Antabian Included Angle				Inset Oom	of Name	
Nodel Displayment			~		Verty	6
All To Marker, From Marker I			<i>av</i> 1	And	1 course	1

### 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.

MSC Software | 175





# 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.



#### Stroke\_Walking\_Beam 1.0 25.0 Current 0.9 - Angle\_PlaceArmtoGround 20.0 0.8 15.0 0.7 Length (mm) 0.6 (deg) 10.0 0.5 Angle 5.0 0.4 0.3 0.0 0.2 -5.0 0.1 0.0 -10.0 0.0 2.0 4.0 6.0 8.0 10.0 12.0 14.0 16.0 18.0 20.0 Time (sec)



### 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



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.

The portion of one revolution of Link 2 is

$$1 - \frac{154}{360} = 57.2\%$$

# 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.

Model Name	Watts_Linkage		
Gravity	Earth Normal (-Global Y)	•	
Units	MMKS - mm,kg,N,s,deg	•	
Working Directory	D:\Document\MSC\New Example 1	6\coi	

#### 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.

Workin	ng Grid Settin	igs 🛛 🕅
Show	Working Grid	t 🏢
Recta	angular C P	olar
	Х	Y
Size	(500mm)	
Spacing	(10mm)	(10mm)
	Color	Weight
Dots	Contrast	▼ 1 ▼
Axes	Contrast	▼ 1 ▼
Lines	Contrast	• 1 •
Triad	Solid	•
Set Loca	ation	•
Set Orie	ntation	•
ОК	Apply	Cancel

## 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	Ylocation	Z location
02	-200	350	0
04	200	450	0
08	290	100	-17.5
Α	0	350	0
B	0	450	0
C	300	450	0
D	0	100	0
E	320	110	0
Ρ	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.08** in the working area.
- i. Replace the contents of **Orientation** with **(0.0, 0.0, 18.43)**.
- j. Click OK.

a une moleore				÷	
File Type	Parasolid (*.xn	nt_txt, *.x_t, *	xmt_bin, ".x_	••	
File To Read	D:\Documer	nt/MSC/New E	xample 16\sta	nt/wheel.x_t	
File Type	ASCII	•			
Part Name	• Wheel				
Location	(LOC_RELA	(LOC_RELATIVE_TO((0,0,0), O8))			
Orientation	0.0, 0.0, 18.4	0.0, 0.0, 18.43			
Relative To					
Explode Ass	emblies				
		OK	Apply	Cancel	

### 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**.
- I. 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.
- Enter Piston in the New Name content. f.
- 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	Ρ
Lower_Link	10 mm	5 mm	02	А
Middle_Link	30 mm	5 mm	А	В
Upper_Link	10 mm	5 mm	В	С
Drive_Link	10 mm	5 mm	С	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.







Model         10         Pressure Angle 15.0         Axis of Ritation         Global 2         \$8.0.0.0.0           GEAR1         GEAR2         Name         GEAR2         Extension         Extension         Extension           (# 1-line C Finitumg         (# -1-line C Finitumg)         (# -1-line C Finitumg)         Extension         Extension         Extension           (# 1-line C Finitumg)         (# -1-line C Finitumg)         (# -1-line C Finitumg)         Extension         Extension <th>4</th> <th>dation .</th> <th>Geometry</th> <th></th> <th>Manadat</th> <th></th>	4	dation .	Geometry		Manadat	
Module         10         Pressure Angle 15.0         Axis of Ritation         Global 7         \$8.0.0.0.0           GEAR1         GEAR2         GEAR2         Gear2         Gear2         External         Exte		10000 C	510110)			
GEAR1         GEAR2           Name         Dmmi 2         External           (*) Nami         Dmmi 2         Nami	Module 10 Pre	essure Angle 15.0	Axis of Retu	tion Global Z	. 30.000	0.0
Name Desired External Profile Standard P		GEAR1			GEAR2	
P Hen C Entiting     If Hen C Entiting       Center Location     320.0, 110.0, -5       No. of Trach     31       Gase Width     5.0       Bore Radius     0.0       Frollie     Standard       Prollie     Standa	Name	Chall(2)	Name		Driar_1	External .
Center Location         320.0, 110.0, -5         Center Location         250.0, 100.0, -5           No. of Tradh         31         No. of Tradh         31           Gase Width         5.0         Gase Width         5.0           Bore Radius         0.0         Bore Radius         0.0           Frollie         Standard         •         Prollie         Standard           Prollie         Standard         •         Prollie         Standard           Dedeedwer Factor         1.0         Cedeedwer Factor         1.26           Tooth Modification         Tooth Modification         Tooth Modification	e Non C Haister		# time	C Ridsteg	16 – 7	1
No. of Teach 31 No. of Teach 31 Gear Width 5.0 Gear Width 5.0 Bore Radius 0.0 Bore Radius 0.0 Profile 5.1 Standard • Profile 5.1 Coeff. 0.0 Profile Shit Coeff. 0.0 Addeedum Factor 1.0 Coeff. 1.0 Coeff. 1.0 Coeff. 1.25 Troth Modification Tooth Modification Toot	Center Location	320.0. 110.05	Center Los	cator.	290.0 100.0 -5	
Gear Width 5.0 Gear Width 5.0 Gear Width 5.0 Bore Radius 0.0 Bore Radius 0.0 Profile 5.1 and and Profile Standard Profile Standard Profile Standard Profile Standard Profile Standard 1.0 Addeedum Factor 1.0 Cedendum Factor 1.26 Tooth Modification Tooth Modifica	No. of Teath	31	No. of Teo	dh.	31	
Born Radius         0.0         Born Radius         0.0           Frollie         Standard         •         Prollie         Standard         •           Prollie Ohit Coeff.         0.0         Prollie Shit Coeff.         0.0         •         •           Addendum Factor         1.0         Addendum Factor         1.0         •         •           Dadendum Factor         1.25         Dedendum Factor         1.25         •         •           Tooth Mudification         Tooth Mudification         Tooth Standard         •         •         •	Gear Width	6.0	Gear Web		5.0	
Profile Standard  Profile Standard  Profile Standard  Profile Shit Coeff. 0.0 Profile Shit Coeff. 0.0 Addeedum Factor 1.0 Dedeedum Factor 1.26 Tooth Modification Tooth Modificati Tooth Modification Tooth	Bore Radius	0.0	Bore Radi	15	0.0	
Profile Shit Coviff. 0.0 Profile Shit Coviff. 0.0 Addendum Factor 1.0 Addendum Factor 1.0 Dedendum Factor 1.26 Tooth Modification Tooth Middlication Tooth Middlication	Protie	Standard •	Profile		Standard •	
Addendum Factor 10 Addendum Factor 1.0 Dedendum Factor 1.26 Dedendum Factor 1.26 Tooth Modification Tooth Modification Tooth Modification	Profile Shilt Coeff.	0.0	Prolie Shi	it Coeff.	0.0	
Declarifium Factor 125 Declarifium Factor 125 Tooth Mudification Tooth Mudification Tooth Mudification Tooth Mudification	Addendum Factor	1.0	Addeedum	n Factor	1.0	
Tooth Modification	Decendum Factor	1.25	Declaration	e Factor	1.25	
T-D-COM DA	Tauth Modification		Tooth Mad	dification		
npreser start 10.0	Tip Relief Start	0.0	Tip Relief	Start	0.0	
Tip Relief Coeff 0.0 Tip Relief Coeff 0.0	Tip Relief Coeff	0.0	Tip Relief	Coeff	0.0	
Crown Magnitude 0.0 Crown Magnitude 0.0	Crown Magnitude	0.0	Crown Ma	onitude	0.0	
Geenetry Settings: Profile points 10 Layers 5	Geometry Settings:	Profile points 10	Layers 5	2		

GEAR1	GEAR2	GEAR1	GEAR2
Туре	Fixed	Туре	Rotational
Body	Watts_Linkage.Drive_Lin	Body	Watts_Linkage ground

### 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	02
Middle_Link	Lower_Link	А
Middle_Link	Piston_Link	Р
Middle_Link	Upper_Link	В
Piston_Link	Piston	D
Upper_Link	ground	04
Drive_Link	Upper_Link	С
Gear_Link	Ground	08
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	02
6	Revolute joint	Lower link and middle link	A
7	Revolute joint	Piston link and middle link	Ρ
8	Revolute joint	Upper link and middle link	В
9	Revolute joint	Upper link and drive link	С
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  $\ensuremath{\text{Piston}}$  , and then select point  $\ensuremath{\textbf{D}}$  .
- c. Move upwards and create a force in y direction.
- d. Expand the Forces tree in the Model Browser.
- e. Right-click  $\ensuremath{\texttt{SFORCE\_1}}$  , and then select  $\ensuremath{\texttt{Modify}}$  .
- 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.

	Create New Model		
Model Name	Open_Differential		
Gravity	Earth Normal (-Global Y)		•
Units	MMKS - mm,kg,N,s,deg		•
Working Directory	D:\Document\MSC\New	Example 17	01

### 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**.

Show	Working Grid	igs 🔔
Recta	angular C P	olar
	Х	Y
Size	(150mm)	(150mm)
Spacing	(5mm)	(5mm)
	Color	Weight
Dots	Contrast	▼ 1 ▼
Axes	Contrast	▼ 1 ▼
Lines	Contrast	▼ 1 ▼
Triad	Solid	-
Set Loca	ation	•
Set Orie	ntation	•
OK	Apply	Cancel

### 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
Α	-45	0	0
В	45	0	0
С	0	43	0
D	0	-43	0
E	40	0	137
F	0	50	0
G	0	-50	0
Н	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  $\ensuremath{\text{Bevel}}$  in  $\ensuremath{\text{Gear Type}},$  and then click  $\ensuremath{\text{Next}}.$
- 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  $\ensuremath{\textit{Next}}$  with the default setting of  $\ensuremath{\textit{Material}}$
- f. In the Connection page, select none in Type.
   Only select Rotational Joint and ground for Ring\_ Gear.
- g. Click Finish.

4	Wethod		Geor	neiry		Material	
Pressure Angle	0 Cert	er Location		K			
OEAR1	Name	Lower	Finian_O	OCAR2		Name	Left Side Oea
F New C Existin	U			R New C	Emilia		
Avis of Fictution	Overtation	- 0.0.3	0.0, 10.0	Acia of Rate	017	Orientation	- 270 0. 90 0 0
No of Teets	18	Carrier Contraction		No of Teets	£	18	
Face Width	25.4	_		Face With		25.4	
Back cone distance	0	_		Back cone of	Sistance	0	_
Pitch Angle	45			Pitch Angle		45	
Pitch Apex	0.0			Ptch Apex		0.0	
Bore Radius	0			Bore Radius	Č.	0	1. Augusta
Mean Spiral Angle	0	19 U	HI FRH	Moon Spiral	Angla	0	C TH & S
Outer Trans Modul	4.536773	0173		Outer Tran	s. Module	4.83677391	12
Tooth Depth and We	Cata Typ	+1-150 255	(9 - 65)	Tech Depth	and Width	Data Type I	- 150 23539 •
Prulle Shift Coefficie	ent 0.505		124	Profile Drift	Coefficient	-0.545	
Addendum Factor	1.0			Addendum I	actor	1.0	
Dedendum Factor	1.26			Dedendum I	actor	1.25	1
Thickness Med. Cos	2.7E 002			Thickness N	lod Coet	4.6E-002	
Georeetry Settings	Frafix po	nta 33	Layers				

4 1	Vathod	• Ger	metry a	Statest	•
Freesure Angle 2	Contr	r Location (90.00,0	0		
GEAR1	Name	Upper, Pinton, G	GEAR2	tiane	Lif. Sen the
* New C Existing	1		C New @ Existin	1g Select Gear	Lift_Sice_Gea
Axis of Rotation	Osercation	• 180.90.190	Axis of Retation	Orientation .	270.0.000.000
No. of Testh	18		No. of Teeth	18	March of barriers
Face Width	25.4		Face Width	25.4	-
Back cone distance	0		Back cone distance	0.0	-
Pitch Angle	45	3	Ptch Arge	45.0	
Pitch Apex	0.0		Pitch Apex	0.0	-
Bors Radius			Boro Radius	0.0	
Mean Sprai Angle	1	IF LH C RH	Mean Spiral Angle	0.0	T LH # R
Outer Trans. Module	4.535773	0173	Outer Trans. Modu	le + 4.5357730173	E
Tooth Depth and Wild	th Data Type	et - ISO 23609 ·	Tooth Depth and W	dth Data Type 1 -	150 2 509 •
Profile Shift Coefficier	0.545		Profile Shift Coeffici	nt -0.505	
Addendum Factor	1/2		Addendium Factor	1.4	
Dedendum Factor	1.25		Decendum Factor	1.25	
Thickness Mod. Coef	3.75-032		Thickness Mod. Co	of -5.5E-002	
Geometry Settings:	Profile point	es 10 Leyen	5		





	aw 2	20	2015 II	14.94	
< Met	396	• Geor	nety •	Material	
Pressure Angle 20.0	Center L	cotion 40.0.0.0	0.0		
GEARI	Nana	WAR FAMILY	GEAR2	Name	Hig Cen 1
6 Haw C Existing			C Have C Election		
Axia al Ratation	Global Z .	0.0,0.0,0.0	Axis of Rotation	Osentation •	90.0, 90.0, 3.0
No. of Teach	18	-	No. of Teeth	60	-
Face Width	43.0		Face Width	40.0	
Back cone distance	0.0		Back core distance	0.0	1
Pitch Angle	(ATANE14/6	0)	Pitch Angle	(ATAN(60 r 54)	Ē.
Ptch Apex	0.0		Pitch Apex	0.0	÷
Bore Radius	00		Bore Radius	25.8	
Near Spiral Angle	0.0	F LH C RH	Mean Spiral Angle	0.0	CHER
Outer Trans. Module	4.535773817.	ł.	Outer Trans. Module	• 4.5357738173	
Tooth Depth and Weth	Data Type 1	1SO 23503 *	Toolh Elepth and Width	Data Type I - I	50 23539 •
Profile Shift Coefficient	0.505		Profile Shift Coefficient	0.605	
Addendum Factor	10		Addendum Factor	3.0	8
Dedendum Fector	1.25		<b>Dedendum Factor</b>	1.25	
Thickness Mod. Coeff.	3.76-002		Thickness Mod. Coef.	-5.5E-002	E
Geometry Settings	Profile points	11 Layors	6		
North Contraction	100001000	And a set of the set	Street and Street		



#### 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  $\boldsymbol{C},$  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**.
- I. 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.**
- 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  $\ensuremath{\textbf{point}}\xspace$  H.

192





1 +

### 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	
Upper_Housing	Yellow
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  ${\boldsymbol{\mathsf{F}}}$ 

- 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.

0





## Example 27: Planetary Gear Sets Modification



### Workshop Objectives

- Modify the current planetary model and compare results in postprocessor
- 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

A Simulation Control	×
	<b>5</b> 3
End Time 💌	2
Steps -	200
Sim. Type:	Default 💌
Start at equilibriu	m
Reset before runn	ning
No Debug	-
Nastran	
Update graphics d	lisplay
Interactive C Sc	ripted
P. 6	
Simulation	Settings

### 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.

Sodily Restory Dear			100 March 100 Ma
Thep I of I Mathani	· Owners	• Material	
24		Nero First, M. F	-
Complicates 0.5355.5	Act of S	Cond 2 . 1150	H
Maide Direal)	Provins Jught Stornal) [38	8 Hais Juge [	34
Gewittlath 25.0	Planet Gales	-	
Stander of Teath	1		
Box Gaus 42	Repflear IN	and Gen 19	
Par Dellar	(Augo 1)		
So-Peel [81	Plant - Roy	16	A CONTRACTOR OF A CONTRACTOR A CO
Dow Mag Radoo	1000000000	- HARMAN -	1
But Dear   81	ExpOse  00	Plent Ger   B.B.	
Gerney Setrip Inde	Aven NR.	Layers [ER]	
10 kg		-Bet	Ours

### 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 P
- 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



*	Nethod	•	Тури		Servey.	•
					1	
Method	Oetai Joint Comm	ed Bard	j			
-	Afaros	id is used to repri	ecent the bearing. Sti	these is calcula	ted at every	
0	atap bi is basi charac	ased on the pos of on user-speci teastic geometry	bons and velocities a field factors. Also, a li y values for many cal	t the bearing loc brary can be us alog bearings fr	ation Damping ad to locitum Im several	
9	at the is service	ecturiers. This ma bearing complian life prediction b	ethod provides a far n toe than the complian assed on industry star	sole accurate se it method: it als stards.	o allows for	

- 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.

							1125
4	Type	•	Geomoty	у (	8 (	Comitai	
Dearing Name	Dearing_1	A	us of Rotation	ocel 2 🕑	0.0.0.0.0.0	16	
Dearing Location	0.0, 0.0, -35.0	Be	aning Geometry Si	caling	<u>ا_</u> ا	.1	
Dreate Dearing	From Databa	ace (* With)	User leput				
Heat X 0	0 Offset Y	0.0 09	hat Z Q.C.	Musignme	nt X 0.0	Maalign	ment Y 0.0
Constrant IR	ADIAL AVIAL BOT	H	vine Destance	102			
		1.5		12/1	1		
Annelacturer	P FAG P TM	NEN 17 18	SK P SKF P	NA P K	ovo Pi	BC 17 )	GRVI/
Wanufacturer Stamater 50	P FAG P TM	ven 17 na D Analatik B	ek P skF P Ikarnga <b>Line</b>		ovo 💌	BC P )	GRW (
Vandacturer Stameter 30 Show Geometry	P FAG P TM	KEN P N b Available 5 Geometry I	SK 🗭 SKF 🛱 Isarrigs - 💽	From load rate	ovo 🖂 i	BC P )	GRUI
Vandacturer Stamater 50 Show Geometry Iolier Pitch Diam	P FAG P TM P Box C Hu war (Dow) 37.5	NEN IP N D Available B Geometry I 3285 No	SK P SKF P Skarings Pedigree Demod	Form load rate	ovo 🕞 I	BC 9 1	GW/
Vandacturer Diameter 30 Show Geometry Rober Pitch Diam Directive Rober Lo	P FAG P TM # Boxe C Hu wher (Dpw) 37.5 engli(Law) 14.5	NEN P N D Austable D Geometry I 0285 No 3956111 Rul	SK P SKF P Skarngs Ender Pedigree Demod Inther of Rollers Int Diameter (De)	PUA P 10 Pom haad raile 18.0 2.506992527	CIYO P I	BC P ) SRating ad Limit	GRW 3,138+004 3816-853675
Vandacturer Damater   50 Show Geometry Roler Pitch Diam Directive Roller Lo	P FAG P TM P Box C Hu ster (Dpv) 37.5 mgt(Lev) 14.5	NEN P 16 D Auslatte 5 Geometry 1 0285 Nu 9954111 Rul	SK P SKF P Skarngs Ender Pedigree Demod Inther of Rollers Int Diameter (Dw)	P NA P K	DYO PI	BC P ) SRating ad Limit	GW 3,13E+064 3616.863675
Vandacturor Dameter   30 Show Geometry Inder Pitch Diam Directive Roller Lo Inder Cleanance	© FAG © TM <sup>™</sup> Boxe ⊂ thu ster (Dpe) [37.5 ength(Lee) [14.5 [2.05]	NEN 19 18 D Austate 5 Geometry 1 0285 No 0285 No 0285 No 0285 No	SK P SKF P bearings Pedigree : Demod inther of Rollers Int Diameter (Dw) retrail Clearance	PAA P 16	DYO P I Static Loa Paligue Lo Dynamic La	BC P ) S Rating ad Linit ad Rating	GW/ 3.132+004 3616.863675 1.396+004
Vandacturer Damater   30 Show Geometry Noter Pitch Diam Discline Roller Lo Notel Cleanance	9 FAG 9 TM 8 Box C Hu I Her (Dov) 37.5 mgt(Lee) 14.5 2 05	NEN 19 18 Geometry I 0285 No 9954111 Rul 1400 Diam	SK IP SKF S bearings Intern Pedigree : Demod Inter of Rotlers Ier Diameter (Dw) retrail Clearance	PA P K	OYO P I Satic Loa Palipie Lo Dynamic Lo	BC P 1 SRating ad Limit ad Rating	GTW 3.13E+064 3E16.863671 1.39E+064

- 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

4	Georg	MEY		Connectio		Co	ngénter	
aut	PART 2		Housing	pound				
nposa M	ationa. (P	on c o	e					
DoF Ret Z	Type rtisp(time) =	. 53736	time) 20.1 time	Disp. IC Vale	i IC			
the Z	- (emiligeb	• 0*tm	•					
erte Dis	play Distance							
oolal Dam	iping Factor	0.1	Radal C	Jamping Pactor	0.1			
hending C	lamping Factor	01	Torsion	il Denging Fecto	User Functio	n <u>•</u> l01time		

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.

4	Type	•	Geomany	•	Committee	•
saring Name	Dearing_2	Ada	of Rotation Global 2	- 000.00	0	
naring Location	0.0, 0.0, 35.3	Beat	ing Geometry Scieling	لا	1 1	
eate Dearing	🐨 From Databa	ee 🦳 Willh Us	er laput			
feet X	0 Offset Y	0.0 Ofte	az 0.0 Mirale	primert X 00	Musign	ment Y 0.0
		-			-	a normal de la compañía de
onstraint (K	ADAL AGAL BOT	H + ERAV	ng Calearance	2	-	
	-					CO.M.
andacturer	P FAG P TM	KEN 17 NSK	PSKFPMA	P KOYO F	EC P	ew .
lanuflacturier lamatier 20	P FAG P TM P Bore ⊂ PM	KEN 19 NSK D Available Bea	P SKF P MA	P KOYO S	EC P	evi
andacturer Iamater [20 how Geometry]	P FAG P TM I* Bose ⊂ Hut	KEN IP NSK Prailable Bea Geometry Pe	Gran Damed from laa	P KOYO 6	EC P 1	sw
andacturer ameter [20 how Geometry] oter Pitch Clam	9 FAG 9 TM # Boxe C Hut	KEN IP NSK D Available Bea Geometry Pe	SKF 9 BA 1 Imma Freedotation dense Damed from load	P KOYO S Indept Static	Coad Rating	GRW (2,112+064)
andacturer ameter [20 how Geometry] oter Pitch Diam fective Roller Le	© FAG © TM P Buse ⊂ Mut eter (Dpw) 27.75 math(Lere) 15.25	KEN 12 NGK D Available Dec Geometry Pe 1195 Nam	SKF 9 BA	Fictory Static 1	Coad Pating	GW 2.11E+964 2573-600321
andacturer amater 20 how Geometry der Pitch Diam fective Roller Le	9 FAG 9 TM # Boxe C Hut eter (Dov) 27 71 right(Lee) 1523	KEN 9 16K D Analable Dec Geometry Pe 1955 Num 196322 Robe	D SKF D BA ings Demot Fibre Ka digree Demot Fibre Ka ber of Rothers 12.0 Diameter (Day) 2.53116	P KOYO S Indeps State 1 2295 Falipue	e BC P i	GFW/ 2.11E+004 2573-000321
andacturar amater [20 how Geometry] oter Pitch Diam fective Roller Le coal Clearance	9 FAG 9 TM 8 Box C Hul eter (Dov) 27.77 ngt1(Ler) 1523 205	KEN P 16K D Autoble Dis Geometry Pe 5185 Nam 1196322 Roller 403 Diame	District Cestance Total Cestance	V KOYO S Indiana Static I 2295 Falipus Dynami	Coad Ruting	SW/ 2.11E+064 2573 00022 1.53E+064
andacturer amater [20 how Geometry] other Pitch Diam fective Roller Le ceal Cleanance	9 FAG 97 TMA * Bus 1 Hul eter (Dov) 2775 ngt(See) 1528 206	KEN 17 NSK D Anatobie Bau Geometry Pe 1955 Num 19522 Roller 003 Diame	District Clearance	State 1 229 Falgue	Coad Rating	GW/ 2.11E+004 2573.00032 1.53E+004

### 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.

Bearing Name	Bearing_1				
Motion					
Rotational					
🔽 Disp	placement 🔽 Ve	elocity 🔽	Acceler	ation	
Translation	al				
🔽 Dist	alacement 🔽 Ve	elocity 🔽	Acceler	ation	
Bearing Forces Force IP Rad Detailed K Lubricant Propert Type	ial 🖓 Axial Stiffness ies C Oil & Grease	Torque P Bend Damping Manu	ling 🔽 . 🔽 facturer	Torsion Service Life Arcanol	
Lubric	ants Arcanol MUL	TITOP	0.000		-
Tempe	erature(C.) 75.0	Failur	probabil	ity 10.0	%
Service life is pre- as the predicted t that moment in th	dicted based on IS( bearing service life i le simulation.	D/TS 16281. T n hours under	he curve the spee	points can d and load	be interprete conditions a

### 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.



0.01 Time (see

0.02 2013-07-30 20 13:50

### 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.

5.0E+011 0.0 0.0

## 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.

Create New Model	and the second second	×
	Create New Model	-
Model Name	Belt	
Gravity	Earth Normal (-Global Y)	
Units	MMKS - mm,kg,N,s,deg	
Working Directory	C:\Users\wwei\Work\02_second_edition'	
	OK Apply (	Cancel

### 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.

	ļ				
top 1 of 11					
1	Туре	5 <b>.</b> 5	Mettod	· · · · · · · · · · · · · · · · · · ·	offanos 🕨 🕨
Delt System					
lane	betsys	<u>a</u> t (			
Pulley Dat					
Name	puteys	et_1	Туре	Smooth	-
				Poly-V Greeved	
				Trapezoodar 1 pots	+D
		and the second	half surgery the second	white the sectors	
3	Also to singley sidee v	rown as a far i ys a cord-reinfo round through	belt system, the snoo roedbelt which is sno a suries of smooth po	oth belt system seth on both fleys	
J	Ano ti engloj sidea v	rown as a far ys a cord-reinfo round through	belt system, the smoo roedbelt which is sno a socies of smooth pu	ch left system seh en both føys	
3	Also tu modey sidee v	rown as a far i ys a cord-renfo round through	belt aystem, the smoot recoldent which is a rec a savies of smooth pu	Extremely of best system orth on both deys	

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.

ab e 9, 11						
-4	Туре	٠	Method	•	Opinity Pullips	•
Metrod	2D Letts Constraint 2D Letts 3D Letts 3D Letts Nonp	planar				
-	The belt is a to each other	censtrained to er with stiffnas	a plane. The belt is a elements and ana	insdeled with lytically calo	I planar port obgenerita con ulated contact forces betw	mecte sen
	the segment the acts of t	its and pulleys rotation must b	This modeling met e parallel to one of i	hod is faster De global an	to simulate than 30 links. Va	bit
	the segment the accord	its and pulleys rotation must b	This modeling mer e parallel to one of t	hod is faster the global an	to simulate than 30 links. #3	biđ
	the segment the accs of c	its and pullays	This modeling met	rod is finite die global ao	ts similate than 30 ares.	bat.

g. Name the first pulley **P1**. Enter (-200, 0, 0) for Center Location.

Create Pulleys							
Rap 3 of 11							
4	Antoi	٠	Geometry-Pub	oys -	• No	lanai Pafliya	•
Number of Pulleys	2		Axes	ef Rotation	Gobal Z	1000048	
1  2							
Pater			Name	PT.		1	3.5
Center Location	200.0.0	-1					
Geometry							
Pulay Width	38	Pule	y Pitch Dameter	6	-		
- La					Back	bleet >	Close

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.
- I. 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**.

	Completio	n-Pulleys		Geo	ometry Tens	ioners		Material-Tensione	n 🕨
Number	of Tensioner	with Devia	tion Pul	ley	1				
1									
Type	Ro	ational		2					
Tension	e )	iarre 1	1	7					
Pivet Ce	nter	0.0,-200	0,0.0	-					
Geomet	y								
Length	140.0	Widt	30	0	Depth 30	0	Instal	ation Angle 90.0	
Devation	Pulley 1	tame [1	21	-	Asta Of Rot	ation [	Linear Z	• 00.00.00	
Geomet	y								
Putey R	adus	30.6		Pulley	Width 30.0		int	out ⊂ is i⊂ O	it i
Connect	ion Detween	Teraione	and De	visition Pub	ey 🤄 Yes	/" Ne			
-									

- 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.

Create Belt	1		12		-	
ç3et7						
4	Method		Geometry	•	Centact and Mass	•
kill.				Name	bolt 1	-1
ais of Rotation	Gickel 2	• 0	0.0.0.0	Reference	Location -200 0.0 0.0.0	-1
Total Samuel		Same	or Leven 1			
			and the second sec			
elt Height 3	6	Belt We	80 30.0	Ce	stal Diotance 0.5	
of Stiffness						
Liffrees	Geometry	k Material	-			
egnerit Aree	00.0 Sect	an inefta	0.5 Yeungs I	Adulus 1.08	+0 Damping Rate 1.0E	03
Inter Voduke	2006-0	Shear	Correction F	On T Of	- 935 10	
inglution Stiff	was Mutiplier	10				
annen Ramina						
	P			12.01		
erners secre			Stand Country	Eastle	-	
echecy secry let Graphics	Shidt	-	rerea stapres	C. C. C.		
et Graphics	Shdil		rene andres	(Case)	. 4 :	
et Graphics	(Shrift)	-	role and the	< Bat	a   Next > ]	Cite

- 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.

Contact	and Mana   Wapping Order	Dutyus R	ra.mt	•	
But Whipping Order		Witness	a As		
		Tel	<ul> <li>80</li> </ul>	04.0	
	Specify mapping order clocivates about the posit	Fam. eres			bet:
					palapart) putapart) das j
al sal	7.4				

Des.		
virapping Groer	pulleyset_1_P1 pulleyset_1_P2 pulleyset_1dev_D1	Revers

- 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**.

And the second se	- V/211	A 424 Statistics	1000	1/2-10 Alia	
<ul> <li>wabbed (sige</li> </ul>		Output Request	•	Completion	_
Span Regioal	P 5497	rent Request			
But Span   Bat Sagna	at ( )				
				1	
Eleit Parts					
segnane_10					
1					
Reference Part S	bruch				
Motion Average	Force Ave	rager			



#### 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.

tep 1 of 5	and an an and a second					
4	Actuator	٠	Type	٠	Function	•
Pulley Se	e .					
Name	1	pubey	at 1	Pick	r:	
Belt Syst	em	Ted		Browse	1000	
Name	1	Parame Field be	do e	Create	Jet:	evental.
Actuator						
Name		actuator_1				
Pulley						

c. For the actuator pulley, select P1. Click Next.

tep 1 of 5	ł					
4	Actuator	•	Type	•	Function	•
Pulley Se	e.					
Name		pulleyset_1				
Belt Syst	em					
Name		Beit beitsys	_1			
Actuator						
Name		actuator_1				
Pulley		- UTE-bit	inte à l	Pick	F	
		Text		Browse-		
		Paramet	erize •	Guesses #		
		Elalist had		Create	maile set 1 a	almost 1 P

- 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

- 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.

	S
End Time	• 1
Steps	• 100
Sim. Type:	Dynamic
and the second se	a manufacture summarial damaged summers
Nation	H.
Nation Vipdate gra	aphics display
Vpdate grs	aphics display
Vpdate gra	aphics display C Scripted D L L C

#### 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.



- 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**.

File Import	
File Type	Adams/View Command File (*.cmd)
File To Read	/ork\02_second_edition\Aview\Belt\start\contact.cmd
Echo Comman	ds
Update Screen	
Display Model	Upon Completion
On Error:	
C Continue Com	nand 💿 Ignore Command 🔿 Abort File
	OK Apply Cancel

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.

Script	Belt.SIM_SCRIPT_	1	
Script Type	Adams/View Comm	ands	
Adams/Views Comma	Simple Run Adams/View Comm	ands	
! Insert /View co	Adams/Solver Com	mands	
Append Run Com	mands		
Append Run Com	mands		
Append Run Com	mands	~ 1	L Second

c. Under the menu of **Append ACF Command**, choose **Dynamic Simulation**.

Script	Beb SIM_SCRIP	T_1	
Script Type	Adams/Solver Co	mmands	
Adams/Solver Comma	ands:		
Append ACF Comm	and	<u> </u>	
Append ACF Comma	and	-	
Append ACF Comm Append ACF Comma Initial Conditions	and	-	
Append ACF Commu Append ACF Commu Initial Conditions Transient Simulation	and	-	- Carrol
Append ACF Comma Append ACF Comma Initial Conditions Transient Simulation Kinematic Simulation	and and	• • •	Gancel
Append ACF Comma Append ACF Comma Initial Conditions Transient: Simulation Kinematic Simulation Quasi-static Simulation	and and a	• • •	Cancel
Append ACF Commu Append ACF Commu Initial Conditions Transient Simulation Constitution Simulation Quasi-static Simulation Static Calculation	and and i ion	epty	Cancel
Append ACF Commi Append ACF Commi Initial Conditions Transient Simulation Kinematic Simulation Quest-static Simulatio Static Calculation Nastran Export	and and a	• • • • • • •	Cancel
Append ACF Comma Append ACF Comma Initial Conditions Transient Simulation Kinematic Simulation Quasi-static Simulati Static Calculation Nastran Export Activate	and and a	*	Cance

- 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.

Adams/Solver Commands: ! Insert ACF commands here: SIMULATE/DYNAMIC, END=0.5, STEPS=50

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 Name 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  $\ensuremath{\text{Type}}$  as  $\ensuremath{\text{Fixed}}$  to fix the pulley to  $\ensuremath{\text{crank\_shaft}}$
- 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	Туре	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

	Туре	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.
- 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
- 69
- 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.

							3
•	Method		Geometry			Mass	
Belt				Name	belt_1		_
Axis of Rotat	tion Global Z	- 0.0	0.0.0.0	Reference	location	0.0.0.0.0	_
Geometry							
	-	-	-				
Total seg	pments (0)	Segmen	t length   10				
			-				
Flair hainht	136	Roll works	130.0				
Belt height	3.6	Belt width	30.0				
Belt height	36	Belt width	30.0				
Belt height Belt stiffness	36	Belt width	30.0				_
Belt height Belt stiffness Stiffness	3.6 Geometry 8	Belt width	]30.0	<u> </u>			_
Belt height Belt stiffness Stiffness Segment are	Geometry 4	Belt width & Material <u>•</u> on Inertia	[30.0 ] [0.5 Young's m	rodulus 1.0	E+0 Dan	rping rate 1.0E	-403
Belt height Belt stiffness Stiffness Segment are	Geometry ( Geometry )	Belt width & Material _ on Inertia	[30.0 ] [0.5 Young's m	rodulus 10	E+0 Dan	nping rate 1.0E	5-003

- 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

Contact					
Parameter	s Create		]		
Stiffness	1.0E+004	Scale Facto	of 1.0	Exponent 1.5	
Damping	30.0	Penetration	Depth 0.6	8	
Friction	Create	2			
Static	0.5	Dynamic	0.3	Translation Velocity	10.0

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.

14 017						
4	Geometry	•	Mass	•	Wapping Order	•
elt sogme	at 1					
it Segme	et.					1
855	8.0E-004	-				
enter of M	855					
cation wit	h respect to part	0.08.5,0.0				
ertia						
2.55	E-002					
	hyy	2.6E-002				
			lzz	2.1E-003	-	

#### 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.

<ul> <li>wrapping Ord</li> </ul>	er o	Output Request	•	Completion	_
P Span Request	17 Segn	sent Request			
Belt Span Belt Seg	ment ]				
Belt parts					
segment_E					
Reference part	ground				
Motion average	Force aver	rage			

# 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

 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/chaintype 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  $\ensuremath{\textbf{Withbacklash.cmd}}$  and hit  $\ensuremath{\textbf{OK}}$



### 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.

				Conne	ection	
	Gear type	Method	G	ear 1	G	ear 2
			Туре	Body	Туре	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

	2572275		2.52				-
4	Method	•	Geometry	•	Marena		2
Wodule 2 P	essure Angle 28	Heix Ang	t 14.0 Axis of Ret	ation Global	x • 278.0	0.90 0.90 0	
	GEAR1				GEAR2		
larie	Criver1		Name		Driver2	External	1.0
4 New C. Existe	0		d New	C Existing			
Center Location	0.0.360.0.0.0	5	Center Lo	cator.	0 134 56689	294	
No. of Teath	67		No. of Tex	ath	36	and a	
Gear Width	30	2	Gear Web	th .	30		
Bore Radius	15	T)	Eore Rad	45	15	10	
Hand of Helix	IF LH C RH		Hand of H	lelo.	CUHRR	H	
Prattie	Standard		Profile		Standard		
Profile Shilt Coeff.	0.0	11	Prolie th	it Coeff.	0.0		
Addendum Factor	10		Addeedur	n Factor	1.0		
Decendum Factor	1.26		Dedeadur	n Factor	1.25		
Path Modification			Tooth Ma	dification			
Tip Relief Start	0.0		Tip Relief	Start	0.0		
Tp Relief Coeff	0.0		Tip Relief	Cost	0.0	134	
Crown Magnitude	0.0	I	Crown Ma	ignitude	0.0	10	
Geemetry Settings	Profile points	10	Layers 5	1			

Figure: Geometry Setting of Gear Pair 1



Figure: Geometry Setting of Gear Pair 2

up 5 or 6	22/2015		27772			220335	
4	Method	•	Geom	etry	•	Maceda	
tessure Angle X	00 Mear	Spiral Angle	20.0		Cente	rLocation	00.900.00
	GEAR1					GEAR2	
larie	Chat(_2)			Name		Disar_1	
P Non C Hoim	rg.			e line e	Existen		
tuis of Rotation	Orientatio	m - 90.0.	90.0, 278	Auis of Rotati	00	Orientation	- 0.0, 90.0, 0.0
in of Teeth	56			No. of Teeth		25	-
ace Width	10.0			Face Width		19.6	
lack cone distance	10.0			Back cone de	stance	7.0	
Hich Angle	(ATAN)	55 / 25()		Pitch Argla		(ATAN(25	553)
Hich Apex	0.0			Pitch Apex		0.0	
lors Radies	15.0			Bore Radius		0.5	
land of Helix	♥ LH	RH	-	Hand of Helix		C IH &	RH
Outer Trans Modu	a + 20			Outer Trans.	Module	+ 2.0	
Geemetry Settings	Profile p	onts 10	Layers	5			

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**.

0

d. Run another simulation with **0.2 seconds** and **200 steps** 

A Solver Settings	22				
Category	Dynamics 💌				
Model	.Withbacklash				
Integrator	HHT				
Formulation	13				
Corrector	Original O Modified				
Error	1.0E-006				
Hmax	(none)				
□ <u>M</u> ore	Defaults Close				

### 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 enter10 for Number of nodes.
- k. Apply the same settings for tab 2 and tab 3.
- I. Click OK

Vendfies - Create							1.8
Part is be meshed	Vitteecklash.P	ART_S	10	ed AFIT In			
Material	VillMacklash.sl	and .					
Number of Modes	1						
Manual Replace							
P Advanced Settings							
Summer of the							
	initia V Anardari	C. bound b				DAMES & A.V.	
a Mesh Chronities (	Attachments Mari	- Property of			idooly name	power to him.	_
Report Loss		· period					
Checken ripe	Element Size	(3.6wm)	V Carries Awaysia	In the second second	E Alan	of Operations	
1000	Minimum Sate	(2.0mm)	Date Sorres Layers	100			
Element Seugre	Grouth light	25	the second se				
Lienteens -	Shift Thickness	A lowership of	Calapse Small 2	dges.			
Element Coder	Angle Per Element	(45 de g)	Line Telepice (Bill	NUMBER OF			
[Lees •]							
<b>Element Specification</b>	MES Eased Robners	ant (* 01	1 # OFF				
fire x	Canature David Sco	leg (* 0)	CR OFF				
Edge Shape							
Blongfil +							
of ralmal							
E M T	Delay Calar and	•			QK.	Lasty	Canoe

Vendfins - Consta						H.
Part is be meshed ViktMackies	hPART_6		LINEACHTAN.			
Material Villibockias	(h.pipel)					
Number of Modes						
T Manual Replace						
Adveced Settings						
Pardboly Type P Detention P General	try C. Irgen	Medi	Pa.	doty Name	PART & fee	_
C Mesh/Properties # Attachments (F	indody prese	w l				
Attackment Points Defention Table - Ma	der fileden	583		Sieve Node	n far Attachment # 1	1
Constituente Balianance	Barly 10	Bulloof		Selection T	Yos Closust rock	۰.
9 Witnessland grand offerica. L.	1711.1	Harbor		Atachevet	Methed Rigio(RBE2)	
2 Withheld and provid reference if	7736.0	8.0		Cab Taul	and Inc.	
2 Withteritish goest elevence 3	7729.0	#1		- Norther	of nodes   90	
					Transfer Do	
				10 modes		
Raw Appent . Pick Carol	Relevance .		Fiel Atacherents	0 vertices	Let Ds	1
Datate	10.		Cesr Table	0 solgen 0 suitacee	Citer Ett	1
E Ding Carl	ad a			QK.	Jarky S	ancel.

# 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.

Form Tante Rockster				
Write a design line function expression	R.			
acation/Divitation • Assist.	Feector Value			
OC_NLOWS_11ME	-			
OC BY FARMONY MODELE				
OC_FRAME_NINGS OC_FLORAL				
OC_DESTRE	-			
or_torks. or_wishop	Getting Object Dat			
00_04_3818 00_04_3398	Mariaes		1.6	
OC_PERFERENCES		Get Data O	when the close in	Inset Opent Name
or search to	Cree:			Eshate

From Tanin Rocksler					132
Define a design time handhian expression					
Lor_BHIATTIN_TO( (0,-45,6) , CRIT					-
Lecator/Drietator	Function Value				
LOC_XAMMS_INHE   COC_WILLING_INDOID  LOC_XAT_INDOID  LOC_XAT_INDOID  LOC_XATANA  LOC_YATANACANA  LOC_YATANACANACANA  LOC_YATANACANACANA  LOC_YATANACANACANA  LOC_YATANACANACANACANACANA  LOC_YATANACANACANACANACANACANACANACANACANACAN	Getting Object Data				
DOC_DM_LATE DOC_DM_LINE DOC_MERFENCIOULAS DOC_MERFENCIOULAS	Markars	• Get	lita Cuint By 65	init 🚺 teart Oo	antere .
COLLEGATION COLLEGATION	19960				Exabusta
LOC_RELATIVE_TO( (ARRAY) . OBJECT	1		0	K Apply	Carcel

Fanctum Builder			10
Define a design time function expression			
LOC_RELATIVE_T0( (0,-85,0) , )			
Location/Orientation · Assist	Function Value:		•.
LOC_ALCORD_INFE LOC_NT_XINTENT_PRODEID LOC_NT_XINTENT_PRODEID LOC_NT_XINTENT_PRODEID LOC_NT_XINTENT_PRODE LOC_NT_XINTENT_PRODE LOC_NT_XINTENT_PRODE LOC_NT_XINTENT_PRODE LOC_NT_XINTENT_PRODEID LOC_NT_XINTENT_PRODEI	Getting Object Data	• Dummy_stat.cm Get Data Owned By Obj	tof   Inset: Object Name
LOC SPEEDICAL	Plac		Esaluate
OC_RELATIVE_TO( (ARRAY) . OBJECT	1	ok	Acoly Carcel

From Toris Realider							14
Define a design time has	chan e	-					
TULBRITTIN_TO (0	(-88,1	4) <sub></sub> Maa	Antiating Algorith	ang_staft.cm.)			
Lecitor/Orientation	•	Asset	Function Value:				
LOC ALONG ING LOC BY INFORM FROM LOC DY INSUCCE ROOM LOC INITERION LOC INITERION LOC INITERION LOC INITERI LOC LOC LOC LOC							
LOC_MINDOD			Getting Object De	sta			
200_00_123MB			Markara	· Dunny_shat.	177		
LOC PLANE HIRRON				Get Data Ow	and Dy Object	Insert Object	o Narre
COLUMN TO A COLUMN			19962				Exabute.
							the second se

4 100	or Type	•	Motor Connections		Marial.	Garatalay 7	
Motor Name	mator 1	_					
Meter	New	:	Decive	CW	•		
Location	G.OC_RELA	11/12_701 (2	Roor Atlanti Part	Dammy_shaft		Field	÷
Acts of Relation	Orentation		Stator Atlach Part	gand		Field	
	0.0.90.0.0		Forta DiscRay	Tooms		IT Flp Ge	ametry

- 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.

tep 4 of 11				
Motor Cennochara	Motor Goometry		lipits	•
P Detailed Geometry				
Hotor Longth (C) 05 mj	Stator Long	a (#.1 <i>m</i> )	-	
Rator Diameter 0.01 eg	Date: We	n (# E5 m)		
Rotor Stator				
Define Moza By Geametry and Material Fy	V04		-	
Material Type				
Denity 7.8E1E-006agrice="1				
Young's Machine 2.02E+605readon/rem**	7			
Pequaris Ratio 0.25NO UNITU				
				_
1.11				

I. 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.

4 10	eted ( ) eta	Geor	ietty .	•	Materia	é.	•
Andula 1	Fiesture Argie 73	<b>9</b>					
	Were				Worm Villere		
lativ.	Dher		Name .		Driws		
R New I' Existing			5 flee (* 1	Celify			
Avia of Rotation	(Grobal Y	0.0.00.000	Ania of Becard	10	Gonal Z	+ 0.0.0	0.0.0
Corrier Location	BLOC ABLATVA		Celeor Locaria		mmy_shift.	orma .	
No. of Teeth	1		No. of Toels		50		
Face Witth	58		Face Width		8	-	
Dore Radius	10.0		Dave Rades				
Hond of Holls	H DR.F. BH		Hand of Heles		KUNCE	45	
Reference Damater	11		Parks Angle		6		
Flank Florer	ZA +		1.0				
	1						
	3233 M		Automatica State	51.11			-



g. Fix the worm gear to the rotor of motor and worm wheel to "Shaft". Click Next then Finish.

			Carlo Altabati	140	An and the late	
•	Stacarcal		Connection	•	Catalogue	P
Ware 1	Wore Wheel					
Туре	Field	•				
Bedy	motor_1/star	_				

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.

A Database Navigator	×
Browse	*
- windturbine_start	Model 🔺
+ gear_box	Part
+ ground	Part (ç
+ HSS	Part
+ LSS	Part
- nacelle	Part
cm	Marker
geo_ref	Marker
MARKER_26	Marker
MARKER_40	Marker
MARKER_46	Marker
MARKER_49	Marker
MARKER_51	Marker
MARKER_73	Marker
middle_ref	Marker
middle_ref_2	Marker
ref_stage_1	Marker
ref_stage_1_center	Marker
ref_stage_2	Marker
ref_stage_2_center	Marker
yaw_bearing_marker	Marker
+ blade1	Flexib:
+ blade2	Flexib:
+ blade3	Flexib:
+ hub	Flexib:
+ tower	Flexib:
+ amachinery	Librar

#### 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.

Module (Nor	fodule (Normal) ( windturt Pressure Angle (Norm		(Normal)	20.0		Helix Angle 0			
Gear Width (.windtur		Planet Gears		5 -					
Number of T	eeth				2 3				
Sun Gear	35	_	Ring Gear	105	4	_	anet Gear 35		
and the second second	1.00		1000	1.44	5.		and a second second second		

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**.

4	Method	•	Geometry		Material	•
Set				Name	Planet_set_1	
Center Location	ref_stage_	1_center	Axis of Re	tation Globa	X • 270.0.90	0.90.0
Module (Normal Gear Width	) { windtur	Pressure Angl Planet Gears	e (Normal) 20.0	•	Helix Angle	)
Number of Teet	h 35	Ring Gear	105	Planet G	iear 35	
Pair B Sun - Planet	lacklash	Angle	Planet - Ring	0	1	
Bore/Ring Rade	25 0.0	Ring Gear	0.0	Planet G	ear 0.0	1
Sun Gear						

### 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**.

Sin Gen	Disset General L	Inn Gare I	Carrier		origen	- L
Туре	Rotational	• • • • •	ranian (			
Body	Existing	-	LSS		Pick	F I
			Test		Browse	
			Parameterize		Guessie +	windturbine_star
			windturbine_start1	ss .	Create.	ground
			Field Info			155
						nacene gear,box bladel hub blade2 blade3 tower

- 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.

Type .	Finad	2				
Body	Existing	• USS	Test.		1.4	
			fet		Browse	
			Parameterice		Gammen . +	windtuttine, stat
					Caste	ground .
			Field Info			
				_		natelle

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 to0. Leave the rest settings as default and click Next.

•	Method	•	Geometry		Material	
Set				Name	Planet_set_2	
Center Loca	ition ref_sta	ge_2_center	Axis of Rot	tation Global	X • 270.0.9	0.0.90.0
Module (No	rmal) (.wine	turt Pressure Any	ple (Normal) 20.0		Helix Angle	0
Gear Width	(.wm	turi Planet Gea	n 3	*		
Number of	Teath					
Sun Gear	30	Ring Gear	132	Planet Ge	uar [51]	
Pair	Backlash	Angle				
Sun - Plane	t 0	1	Planet - Ring	0.	1	
Bore/Ring R	ladius					
Sun Dane	0.0	Ring Gear	0.0	Planet Ge	ar 0.0	
own creat	company and	100000000		0/2011		

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.

Rep 5 of 11								
	Material		Connectore		- Devalution	- F	1.	
Tile Dear	Planet Gear 1	ing Dear 1.4	Serier:				-	
707.	Field							
Bidy	Existing			eres -		- 4		
			Tel				Rente.	
			Ferrera					and all a state of the
				nist?ind,	ret,Lour, gårrigen,	part. +	Death.	(present)
			Leibh.					(5)
								1000
								the set
								hab
								twos2
								Manual
						_		100
al ul				mark 1	Liberta	Class		Penet pet Jammer 1
	-	-		100 C	-			Terret Station Strength Pr
								Paret set Ling you per pe

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**.
- 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.

A Database Navigator	X
Browse	-
- windturbine_start	Model 🔺
- gear_box	Part
cm	Marker
hss_a	Marker
hss_b	Marker
MARKER_71	Marker
MARKER_74	Marker
middle_ref	Marker
Planet_set_1_ring_gear	_referend
Planet_set_2_ring_gear	_referend
Planet_set_2_sun_gear_	reference
ref_spur_1	Marker
ref_spur_1_center	Marker
ref_spur_2	Marker
ref_spur_2_center	Marker
+ ground	Part (ç
+ HSS	Part
+ LSS	Part
+ nacelle	Part
+ blade1	Flexib:
+ blade2	Flexib:
+ blade3	Flexib:
+ hub	Flexib:
+ tower	Flexib:
+ Planet_set_1	planet

- 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.

Rep ] of 6							
4	Method		Goometry		Merenia	•	
Module	( whic Pressure Ang	la 29	Axis of Res	mon Global II	₹ 279.3	99.0.99.2	
	OEAR	0		10	o£AR2		
Name		Deter	180/90		Driveri	Extend	
	GetterLocation	U.OC_RELATIN		Center Laca	ten acc	RELATIVA	
	No. of Texts.	50		No. of Teeth	21		
	Gear Width	[##8.66].2		Gear Witth	Luit	divitine_st	
	Dan Padus	0.0		Elve Fachra	0.1		
Goarnerry	r Sittings - Field	e panta 18	Layes (5	1			
					I Francisco	1	

### 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.

	(energing)	270 0				Constraints -	
GEART	GEAR2						
Тура	Fixed	-					
Body	with rore.	and a second		Pek Brock-	1	ndtwitting start: round	
		Parameterize		Guessian		22	
		and along data our		Cheffer:			
		FieldInte	· T		3 Q	adar.	
	-				1	ch.	
						a de l	
						atel	
						No. 10	
						inset_ton_3 comile	
					P	lenet pet Laur, gein gein	part
					- 1	and set, 1 mg gew gew.	peri
						anatural planets, peop	ner, yes
					P	hvet, art Zalavet, Z. genra	per pet
					2	inter_set_1 player_1_perce	per_per
					P	unit,4. gave part part	
						unet, 5, gear gear, purt	
						anet_sat_2 carrier	_
de unite						And the Parameters	pet
<ul> <li>Ind</li> </ul>						number 2 not den den.	pen

- b. Right click in Body and pick the sun gear.
- c. Click Gear 2 tab and right click in Body and browse for **HSS**.

GEAR1	GEAR2	
Туре	Fixed <	
Body	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.

A Database Navigator	×
Browse	Ŧ
- windturbine_start	Model 🔺
- gear_box	Part
em	Marker
hss_a	Marker
hss_b	Marker
MARKER_71	Marker
MARKER_74	Marker
middle_ref	Marker
Planet_set_1_ring	_gear_referend
Planet_set_1_sun_	gear_reference
Planet_set_2_ring	_gear_referend

- f. Pick **Global X** as Axis of Rotation.
- g. Fix rotor and stator to hss (high speed shaft) and gear\_ box repectively. Click Next.

4	Meta	г Турн	•	Motor Connections		Mile Geimetry	•
Mate	or Name	motor_1	_				
	Meter	New	1	Direction	CW	2	
4	ocation	hus_b		Roto/ Attach Part	HSS	Finad	
Asis of F	auton	Global X		Stator Attach Part	geor_box	Fixed	•
					-		

- 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
lxx (kg*m^2)	100	400
lyy (kg*m^2)	50	150
lzz (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.

Variable Name	motor_1.source_voltage
New Variable Name	
Comments	
Index	
Reals	
Real Value	0
Units	no_units
Range	
Allowed Values	
Delta Type	absolute
Use Range	yes
Use Allowed Values	yes

g. From the model tree, expand Motors folder and double

Mudily State Veriable	In the second second second second
Name motor_1 var_omega	
Definition Run-Time Expression -	
Filme wo(motor_1_mar_sho_1_motor_1_mar_sho_3_motor	t ma_slo_j]
Guess for F(t=0) =	
construct designed	
	OK Apply Cancel
Musily State Valuable Name   motor _1 var sciline. Torque	OK Apply Cancel
Musily State Valuate Name inclor_1 var_soline_Torque Defention Run-Terre Expression *	OK Apply Cancel
Musity State Yawase      Name     Netro1 var_spine_Torque     Definition     F(time,) =     ANISPL(60/2*pinamat, windustorestart.motor)	OK Appy Cancel
Muchy State Variable Name inctor_1 var_soline_Torque Definition Run-Tame Expression • F(time,) = ANISPL(60.(2"pinamal, windustore_start.motor "Guess for Fit=0) =	OK Apply Cancel
Muchy State Valuable Name   motor_1 var_schine_Torque Definition   Run-Time Expression P(time,) =   ANSPE(K0r(2* pinkanat, windustore_start.motor Guess for Fit=0) = []	OK Apply Cuncel

#### 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.

a contract of the second se		-
Name motor_1 var_omega		
Definition Run-Time Expression		
F(time,) = rtod'wz(motor_1 mar_sto_1.motor_1 mar_sto_	sto_J.motor_1.mar_slo_J)	1
Guess for F(t=0) =	OK LAnde LCa	ecol
	οκ Αφργ Ca	ioc ei
Modify State Variable - Name motor 1 var. spline. Torque		x
Definition Run-Time Expression		
F(time,) = AR3PL(varval(motor_1 var_omega),0,moto	or_1.spl_name. 0)*motor_1.rotor_cw	
Guess for F(t=0) =		
← Guess for F(t=0) =		

## 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 Cosimulation

This section introduces you the benefits of Adams/MATLAB cosimulation. 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.

A Create State Variable	×
Name .motor_shaft.theta	
Definition Run-Time Expression	
F(time, ) = AZ(shaft.cm, ang_ref)	
Guess for F(t=0) = 0	
OK Apply C	ancel

- 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.

motor shaft	Modify Torque		
Browse Groups Filters	Name	SFORCE_1	
Bodies	Direction	On One Body, Fixed In Space .	
Connectors	Body	shaft	
Porces	Define Using	Function	
-(c)TORSION_SPRING_1	Function	0.21	
Elements	Solver ID	1	
- m Measures	Torque Display	On •	
- Design Variables	e W	QK Apply Cancel	

b. Right click in the Function box and select Function Builder.

Define Using	Function	•
Function		
		Clear
Solver ID	1	Cut
		Сору
Torque Display	On	Paste
P P	OK	Function Builder
		Parameterize 🔹 🕨 上

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

Modify Torque	
Name	SFORCE_1
Direction	On One Body, Fixed In Space
Body	shaft
Define Using	Function
Function	-VARVAL(motor_torque)
Solver ID	1
Torque Display	On 💌
	<u>Q</u> K <u>Apply</u> <u>Cancel</u>

then..

- d. Click OK.
- e. Right click on TORSION\_SPRING\_1 and then select Modify.

Modify a Torsion Spring			×
Name TORSIO	N_SPRING_1	1	
Action Body shaft			
Reaction Body ground			
Stiffness and Damping:			
No Stiffness 💌			
No Stiffness Stiffness Coefficient Spline: T=f(defo)	(1.0E-004(	N*m*s/rad))	
Preload	0.0		
Default Angle	(Derived Fro	om Design P	osition)
Torque Display On Action Body			
	ОК	Apply	Cancel

- 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.

Eurotion Builder		9		
Pluge Mateger		Durability	Mechatron	cs
Measure Distance	Tupit Mate	91		0.0
Merge Two Models	Name		Load	Load at Statup
Model Topology Map	Adams Girth	(c)	Yes Yes	P Yes
Model Verify	Adams/Durab	Adams/Durability		T Yes
System Command	Adams/Mech	Adams/Mechationics		T Yes
Seat Command File F2	Adams/Vibrat	Adams/Vibration		T Yes
Macrg +				
Megu Polog Bas Polog Bas Polog Bas Polog Bas Polog Bas Polog	Description	Allows you to integrate motion simulation and control system design in your model.		
Purge Cache Files				
	Version	Adams 2013-2		_
	Author:	Author MSC Software		
	Licenses			

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\_

Adami/Controls P	lant Export	-	-		-	-
New Controls Plant		motor_shaft.Controls_Plant_1				
File Prefix			Controls_F	lant_1		
Initial Static Analys	(a)		IF No □	Yes		
Initialization Co	mmand					
Input Signal(s)	From Pinput	1	Output Sign	ul(s)	From F	Poutput
	ADAMS_Variable +	P	hck	Ĺ		
	Text +	E	in an			
	Parameterize +	9	ivesses +			
	Field Info +	0	reate	1		
Target Software	23		Easy5			
Analysis Type			non_knear	53. 		
Adams/Solver Chok			G Ces C	FORTRAN		
User Defined Library	y Name					
Adams Host Name			WEWU20	14 na mscsof	tware com	
	11/0/10/			1.4		

Variable->Browse.

c. From the Database Navigator, double click motor\_

Database Navigator	×
Browse	-
- motor_shaft	Model
motor_torque	ADAMS_Var
omega	ADAMS_Var
theta	ADAMS_Var
+ amachinery	Library
+ MDI	Library

torque.

- d. Similarly, choose omega and theta for Output Signals.
- e. Choose MATLAB as Target Software.
- f. Leave the rest fields as default.

New Controls Plant	motor_shaft Controls_Plant_1
File Prefix	Controls_Plant_1
Initial Static Analysis	≪ No ← Yes
1" Initialization Command	
Input Signal(s) From Pinput	Output Signat(s) From Poutput
mitar_tangun	Denga theta
Target Software	MATLAB
Analysis Type	non_linear
Adams/Solver Choice	C++ C FORTRAN
User Defined Library Name	
Adams Host Name	WEIWU2014 na macsoftware.com

- 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.



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.

Bodies     Connectors     Motions     Solution     S	Browse	Groups Filters	
Connectors     Motions     Motions     Forces     Succe     Gagetring_force     G	Bod	lies	_
Motions     Porces     PlaCC     Ga_dning_force     Ga_dning_force     GSFORCE_1     Elements     Vice_position	+- Con	inectors	
Forces     Subscription     System Elements     Gesposition     Gespositi	+- Mot	tions	
System Elements     System Elements     Gas force	For	ces	
Coa_driving_force     La driving_force     SFORCE_2     SFORCE_1      System Elements     Vice_position     Vice_position     Vice_force     Ja_force     Data Elements     Measures     Sesures     Sesures     Simulations     Results     All Other	-91	ACC	
Generation     G	-24	oa driving force	
SFORCE_2  SFORCE_1  Setements  S	12	a driving force	
System Elements     System Elements     System Elements     Orce_position     Orce_fraction     Orce     Data Elements     Measures     Design Variables     Simulations     Results     All Other	17	SEORCE 2	
Elements System Elements System Elements Solution Solutio	1.15	SEORCE 1	
System Elements     System Elements     System Elements     Sis_position     Os_force     Os_force     Data Elements     Measures     Simulations     Results     All Other	Ele	ments	
Ore position     Ore position     Ore or position	145	System Elements	
U is_position     U os_force     U is force     Data Elements     Simulations     Simulations     Add Chor	TT	V pa product	
Ora force     Ora Elements     Measures     Simulations     All Other		Via position	
Oti is force     Data Elements     Measures     Simulations     Results     Results     All Other		Cos force	
Data Elements     Measures     Simulations     Results     All Other		Via force	
Measures     Simulations     Results     Alto Cher	1.0	Data Elements	
Design Variables     Simulations     Results     All Other	+- Me	asures	
Simulations Results	- Des	ion Variables	
Results	Sim	wiations	
All Other	Res	aults	
	All	Other	

- 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.

Modify Force	×	
Name	ia_driving_force	
Direction	Between Two Bodies In Line-Of-Sight	
Action Body	Flap_inside_drive_motor	
Reaction Body	Flap_inside_drive_eye	
Define Using	Function -	
Function	1.4	
Solver ID	3	
Force Display	On Action Body	
	OK Apply Cancel	

d. Similarly, change **oa\_driving\_force** to **VARVAL(oa\_ force)**.

Modify Force	23					
Name	ia_driving_force					
Direction	Between Two Bodies In Line-Of-Sight					
Action Body	Flap_inside_drive_motor					
Reaction Body	Flap_inside_drive_eye					
Define Using	Function 💌					
Function	VARVAL(.Control_Surface.ia_force)					
Solver ID	3					
Force Display	On Action Body					
<b>1</b>	<u>O</u> K <u>Apply</u> <u>Cancel</u>					

## 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.

 Right click in the Input Signals field, go to ADAMS\_ Variable->Guesses->ia\_force. Similarly, add oa\_ force to the input as well.

Input Signal(s) From Pinput		1	Output Signal(s)		From Poutput		
	AD-AMS_Variable Text	•	Pick Browse				
Termet Calkunse	Parameterize +		Gesset +	Control_Surface:		-	
ranges overhare	Field Info		Create	in_forc		1	
Analysis Type Adams/Soher Choice User Defined Library Name Adams Host Name		WEIWU20	oa_force ra_position ca_position		-		
Dynamic States	Output			OK	Apply	Cancel	

- 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.

input Dignal(s)	From Peput	Output Sign	alis) Fro	m Pou	tpd 🔝	
ia_focos na_focos			ADM/S Variable	•	Pick	i.
			Parameterize		Designed at	Control, Surface:
and the second second		-	Pield Into		Creater	ia_fiece
Farget Software		Easy5				an force
Analysia Type		non_imear				10,000000
Adams/Selves Choice		14 C++ C	PONTRAN			ea_potition

#### 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.

Туре	Results File (*.res)
File Name(s)	ontrol_surface_cosimulation\Control_Surface_Plant.res
Model Name 💌	Control_Surface
View	
Time Step Skip	
Contact Step Skip	7
F Keep Results Data C	On Disk

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.

Analysis	Control_Surface	e_Plant		
Flexible Body	FLEX_BODY_1			
Select Node List	2684			
Node to Add to Lis	t i		e	Stress C Strain
🔽 Von Mises	Max Prin.	F Normal X	T Normal Y	□ Normal Z
Max Shear	🗆 Min Prin	☐ Shear XY	☐ Shear YZ	☐ Shear ZX
Reference Marker	-			

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

K 4 II Þ	O Current 4
Animation   View	Camera Record Overlay Appearance Contour Plots Vector Plots Hat Spots
P Display Legend	
Contour Plot Type	Ven Mises 8 -
Legend Placement	Lut -
Legend Title	Von Mises Stress (Pa)
Colors	246
Legend Gradients	10

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

	►	24
	Frame of Intere	est
Minimum Value	1	*2 /2
Maximum Value	7e+006	*2 /2
Reset Limits		
Decimal places	2	
Scientific Range	-2 3	
Trailing Zeros		



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.

40.2 et 5						-
4	And in Lays		Pulley Properties		Putey Layout	•
Nation of	Pulsy Pripertee					
6.12						
PaleyPa	opety Narw	pulkyt	-1			-
	Deve	and the second second		Comact P	randes	
				32274	1105-014	
	Wath	60.0		Harry S.	1.0	
	Depth	10.		Harrow	11	
	Joge	20.0		Factor Me	6.6	
		anner -		Friction Vt	183.0	
_						
					170 - C	
1.4				< Back	No.d >	Clob

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.

4 B	dey Properties	• Pub	ry Layout		Cattle	
Number of Pulleys t   2   Layout   Materia	s [2 /	wa of Rotation	Globel Z	- 0001	.10	24
	Marne Lecation File Directory Digmeter	toley_put (c.CC_REL) (r.gt_C	₽y N7N(F_113(0.0. 01	c a t al mui		
	Putry Property	putey1		_		
						-

- 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.
- 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.

Puley Lapout		Cable .	Despision.	×
Nember of Cables 1				
3215				
. 0				10
Setup   Parameters   (	Notari			
				1
Density	12182+5	Method	simplified +	
Young's Modulus	1.0E+005	Sever	aute 💌	
Hot	10	Gravitational Effect	ce	
Film	1.0E-064	inste	en	
File	1.0E-013	Disasgagement	-	
Damping	1.00-002	Mesh	4.46	
Velocity	0.0	2010	downal	
Preitad	162.4	Ceta	1400.9	
				-

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.

1	
Setup Parameters	Output
Pulley Results	1
Span Results	1

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.

Modify State Variable	×
Name Input_Vel	
Definition Run-Time Expression	•
F(time, ) = 0	
Guess for F(t=0) = 0.0	
	OK Apply Cancel

- 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.

Modify State V	ariable
Name	Sway_Angle
Definition	Run-Time Expression
F(time, ) =	PSI(cm_ref, MARKER_5)
Guess for F(t	=0) = 0.0
1	
	OK Apply Cancel

### **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**.

Joint Motion	×
Name	MOTION_1
Joint	JOINT_24
Joint Type	translational
Direction	Translational
Define Using	Function
Function (time)	VARVAL(Input_Vel)
Туре	Velocity 💌
Displacement IC	
Velocity IC	
	OK Apply Cancel

### 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.

New Controls Plant	•	BridgeCrane_SimpleCable Controls_Plant_1	
File Prefix		Bridge_Crane_plant	_
Initial Static Analysis		F No F Yes	
Initialization Command			
Input Signal(s) From Pinput	1	Output Signal(s) From Poutput	1
Impus_Vel		Brey_Angle	
Target Software		MATLAB	,
Analysis Type		non_knew	
Adams/Solver Choice		C++ C FORTRAN	1
User Defined Library Name			

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**.

- Controller parameters	3
Proportional (P):	0
Integral (I):	0
Derivative (D):	0

c. Make sure that Record & Inspect Simulation Output is on.

5	Normal	
Record & Inspect	t Simulation Output	

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.

Simulation Data Inspector*		
File Run Plot Help		
🗋 😅 🔚 🔚 🖬 📓 🖾 🔅	। स्व 🔍 🔀 🖑 ५	2 0
Inspect Signals Compare Signa	ls Compare Runs	
Block Name	Signal Name	Line
Run 1: model_complete		
🔽 Gain		_
Run 2: model_complete		
✓ Gain		_

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**.

Result Set		Component		
<pre>- Cable_Sys_1     + cable1     + losd_pulley     - scolley_pulley</pre>	(Resset Container (Resset Container	Y 2 251 781	-	
rev + Centrols_Plant_1 + ground Inpus_Vel JOINT_24 MOTION_5 Even inple	(Resset Container (Resset Container	9H2 VX VZ VZ WX WY WZ	_	

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.

Elle Edit View Adjust S	mulate Review Settings To	ch	Belp Controls Mechatronics Durab	lay :	
TR_485	Settersion Analysis		Open-Loop Steminy Dients		
Browse Filters	Component Analysis	•	Comming Events	٠.	
Subsystems	General Actuation Analysis	2	Shinght Live Events	•	Acceleration
ATR Avar Ter Of interface ATR Provention & AProvention & Array Content of the Array Statement of t		Cograe Events	1	Baked	
		Ele Driven Events		gesting on Split p	
		Static and Quark Static Mar Adams/SmartDriver	Static and Quasi-Static Management		Meintain
			Adams/SmartDriver		Power-off Straight Line.
		Vehicle Set-Up			
TR Front Suspension			Event Builder		
All Other			Road Builder-		
Adams Analysi	s Plag		Beth Optimization		100

- 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**.

Full-Vehicle Assembly	TR_ABS				
Output Prefix	abs				
End Time	4				
Number Of Steps	400				
Simulation Mode	files_only •				
Road Data File 🛛 🤠	mdida //ABS_2013/rd	ads tbl/3d_road_sn	nooth_flat_spli	tmu05.xml	
Steering Input	locked •				
Initial Velocity	70	km/br		1	
Start Time	1				
Open-Loop Brake	]				
Final Brake	100				
Duration of Step	1.5				
Gear Position	4 👻				
Quasi-Static Straight	Line Setup				

### 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

- 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**.

Input Signal(s)	From P	input	Output Signal(s)	Frem Poutput
°	ADAMS_Veriet	ie +	Pick	
	Text		Guesses +	
	Field Info		firmese	

- 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.
- 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.

Eile Edit View Adjust Simulate	Beview Settings Iools Help Controls	Mechatronics
TR_ABS	Animation Controls Postprocessing Window F8	
Browse Filters	Analysis Management	Bead
Subsystems	Linear Modes Controls	ReLoad
🗄 🔁 All Other		Delete

- 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

	<b>b b</b>
Ant a BOS	1
Concerning and the second s	9. La
Analysis TR_A85.at	o jorake
Base Patt +	nt.
TR ABS testing ges g	nmb
Sto. Carrera 🔹	
Mar Trace	
Time Range 💽	0.0.4.0
Cycles:	1
Prame Incoment	11
F Ten Forma - 10	ALCOPE
	Vev guman to Dere Pat - TR_ABS testing ges. g Sto. Carves • No Trace • Trans Range • Cycles: Prans Rocentert Pagestryone [7] Transperstone [7]

### 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.

Full-Vehicle Assembly	TR_ABS				1.0
Output Prefix	abs baseline				
End Time	4				
Number Of Steps	400				
Simulation Mode	interactive	•			
Road Data File 🛛 👼	mdids //ABS	2013/roads to	1/3d_road_smoc	th_flat_splits	nu05.xml
Steering Input	locked	•			
Initial Velocity	70	kmh			
Start Time	1				
Open-Loop Brake	1				
Final Brake	100				
Duration of Step	1.5				
Gear Position	4	•			
Cuasi-Static Straight	Line Setup	222			
Create Analysis Log	File				
No. of the second s					

### 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.

New Controls Plant   File Prefix Initial Static Analysis		Windshield_Wiper Controls	Plant_1			
		Wiper Part				
Input Signatia: From Prop	4	Output Signal(s)	Winddhield Wipen			
	43444	Neistle Pick	enter Lus entre break			
	Ted Bowie_ moto_lve		motor_1.var_DC_beries_Torque			
	Parame Field Int	terior Cherte	motor_Liver_AC_synchronous_Torqu motor_Liver_BLDC_Torque			
Target Software Analysis Type Adams/Solver Chocs User Defined Library Name		Exty6	meter, I.var, dez, angle, Side meter, I.var, deta, angle, Bide meter, I.var, angle, gain, BLDC			
		non_linear				
		IF C++ ← FORTRAN	motor_1.var_de_anglegein_BLDC			
			motor 1 var BLDC Et ang			
Adame Host Name		WE/WU2014 na macaafte	exctor_1.var_BLDC_1_eng			
C David David David			mater loar day ornana B.CC			

- 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.

repet Signal (s) Fill	in Papul	Octour Deput iss	From Po	dist.
manuf_low_move_becam			and the second s	
		servel, 1. Thi, Kryper Marbia, 1. VHF, Angeling Marbia, 5. VHF, MARBA		
Tanai Gatuma IIII	75.548	 Australia	-	_

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**.

File	Edit	View	Insert
Co	ntinuou	s 🔻	
0	ff		
C	ontinuou	IS	
			1

- 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  $\ensuremath{\textbf{F8}}$  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**.

Туре	Res	Results File (".res)				
File Name(s)	dition\AviewWiper_cosimulation/test\Wiper_Plant.res			er_Plant.res		
Model Name	1	Model	3	Pick	<u> </u>	
Time Stan Skin		Text		Browse		
Contact Sten Skin	÷.	Parameterize	1	Guesses +	Windshield	Wiper
I <sup>™</sup> Keep Results Data	On Disk-	Field Info	1	Create		
Keep Results Data	On Disk-			- Annta -	Concol	

- 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.

Elle Edit View Adapt Simulate Beliew Settings winder	Tools Help Controls Machatronics Adams/Car Template Builder F9 Adams/View Interface			
	- Database Management +	Add to Section :		
	Conve Manager	Remove from Session Set Default Writable Create Database		
	Command Navigator			
	Database Navigator			
	Plugin Manager	Publish Subsystem		
	Entity Info	Publish Assembly Database Info		
	Highlight Connectivity			
	Measure Distance	Table Info		
	Appregate Mass	Version Upgrade		

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.

Database Alias	fsae_2012		
Database Path	C Wsers/wwei/Work/02_second_edition/Acaritsae	templateidsae	_2012.cdb
Database Info	OK	Apply	Cancel

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.

Re Edit Vew Adust S	rtulate Berley	· Settings	Tools	Help	Controls	Mech	havonica
s Mecdei							
Constanting of the local division of the loc		_	_		-		
Dpen Assembly					1.347		
Cpen Assembly Name	OK	Acci	x ()		2 carch		<aca:_shared>/assembles.tbl</aca:_shared>
Cover Assembly Assembly	ОК	Appl	v ]	C.	Error Browse Text		Cacia: _shared>/assembles.tbl that _2012/ assembles.tbl charave: Workshappo/assembles.tb

c. From the database, select fsae\_full\_vehicle.asy.

Teoples	Nerve	Data modified	Type	See
RE Desktop	trave, front, suspension	12/8/2012 2:23 PM	ANTIN	412
a Downloado	fine full vehicle	13/5/0012 2:33 PM	AGr File .	
The Recent Places	II has not suspension	12/3/2012 2/31/944	251 File 1	21448

- 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.

Full-Vehicle Assembly	fsae_full_vehicle		
Output Prefix	straight		
End Time	10		
Number Of Steps	1000		
Simulation Mode	interactive	*	
Road Data File 🦁	mdids://acar_sha	ared/roads.tbl/	2d_flat.rdf
Steering Input	straight line		
Initial Velocity	10	km/hr	
Start Time	2		
Open-Loop Throttle	]		
Final Throttie	190		
Duration of Step	4		
Gear Position	1	•	
Shift Gears	1	20	
Quasi-Static Straight	-Line Setup		
Create Analysis Log	File		
The server a constrained by the server			

Step 5. Review Simulation Result.

- a. From the toolbar, click Review->Animation Controls.
- b. You can now review the animation.

Adams/Car Adams 2013.	2					
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>A</u> djust	<u>S</u> imulate	<u>R</u> eview	Settings	<u>T</u> ools	<u>H</u> elp	Controls
face full unbiolo	_   <mark>fsa</mark>	<u>A</u> nir	mation Co	ntrols		
[.isae_iuii_venicie		Pos	tprocessi	ng Win	dow	F8
Browse Filters		Ana	Iysis <u>M</u> an	ageme	nt	•
E Subsystems		<u>L</u> ine	ar Modes	Contro	ls	

c. From the toolbar, click Review->Postprocessing Window.

Adams/Car Adams 2013.	.2					
<u>F</u> ile <u>E</u> dit <u>V</u> iew <u>A</u> djust	<u>S</u> imulate	<u>R</u> eview	Settings	<u>T</u> ools	<u>H</u> elp	Controls
C	irsa	<u>A</u> nir	mation Co	ntrols		
.fsae_full_vehicle		Pos	tprocessi	ng Win	dow	F8
Browse Filters		Ana	Ivsis Man	aneme	nt	•
		7 414	iyolo <u>ivi</u> an	ageme		
E Subsystems		<u>L</u> ine	ar Modes	Contro	ls	

- 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.

Working Grid Settings						
Show Working Grid						
<ul> <li>Recta</li> </ul>	angular O P	olar				
	Х	Y				
Size	(750mm)	(500mm)				
Spacing	(50mm)	(50mm)				
	Color	Weight				
🔽 Dots	Contrast	• 1 •				
🔽 Axes	Contrast	• 1 •				
🗆 Lines	Contrast	• 1 •				
🗆 Triad	Solid	•				
Set Loca	ation	•				
Set Orientation						
Set Orie	ntation	<b>_</b>				
Global X	Y	Cancel				
Global Y	Global YZ					
Global X						
X-X-Avor	ne					
X-7-Axes						
Y-X-Axes	, 5					
Y-Z-Axes	5					
Z-X-Axes	3	-				

c. Create fixed joints between upright and ground at the following locations:

Х	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.
- I. 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.

fart to be mechod	feable uprate	worght		Clast W1744				
latend	feable agright	steel						
further of Macien	4							
Manual Repbox								
Advanced Dettings								
terdady figs F Er	Internet W. Dermothy	C Input la	data la	C.	dbody Name	a cristi	(fex.)	
Mish/Preparties 9	AEachmental (Plea	decy preven						
Attachment Points De	fatton Table - Maste	rNotes			Sizue Node	a for Acts	chment # 1	-
Chartenale Robert	x:+	Rode ID	Sel DOF	-	Atachment	Method	10041L4	ģ
					×	1	Pain Nodes Translar Da	
Ros Append	Put Goat, Re			Fad Atachmerts	C sodes C vetices		List ICs	
Dolete		- 255		Clear Table	0 aciges 0 surfaces	11 E	Clear Cit	

### **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.

Assembly Name	flex						
Front Susp Subsystem	mdids //fsae_2012/subsystems.tbl/fsae_front_susp_flex.sub						
Rear Susp Subsystem	mdids://fsae_2012/subsystems.tbl/fsae_rear_susp.tub						
Steering Subsystem	mdids://fsae_2012/subsystems:tbl/fsae_steering.sub						
Front Wheel Subsystem	mids://fsae_2012/subsystems.tbl/fsae_front_tire.sub						
Rear Wheel Subsystem	mdids.//fsae_2012/subsystems.tbl/fsae_rear_tire.sub						
Body Subsystem	mdids://fsae_2012/subsystems:tbl/fsae_chassis.sub						
Brake Subsystem	mdids //fsae 2012/subsystems tbl/fsae brakes 4Wdisk sub						
Powertrain Subsystem	mdids //fsae 2012/subsystems tbi/fsae powertrain sub						
Other Subsystems	pub, mdide://fsee_S013/subsystems.tbl/fsee_sear_erb.s						
Vehicle Test Rig							

### 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:

A Full-Vehicle Analysis: Brak	cing 💌
Full-Vehicle Assembly	flex 💌
Output Prefix	flex
End Time	1.5
Number Of Steps	300
Simulation Mode	interactive 💌
Road Data File	mdids://acar_shared/roads.tbl/2d_fl
Steering Input	straight line
Initial Velocity	100 km/hr 💌
Start Time	0.25
Open-Loop Brake 🔻	
Final Brake	100
Duration of Step	0.01
Gear Position	6
Quasi-Static Straight-L	ine Setup
Create Analysis Log Fi	ile
	OK Apply Cancel

### 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 fs	sae_front_susp_flex	Analysis=	flex_brake	Tim	Time = 0 to 1.5 sec				
	Top 10 Hot Spots		Abs	R	Radius= 0.0 mm				
Hot Spot	Stress	Node	Time	Locat	on wrt LPR	F (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.

Adams/Car Ad	dams 2013	.2					
<u>File</u> dit <u>V</u> iev	<b>v <u>A</u>djust</b>	<u>S</u> imulat	e <u>F</u>	<u>R</u> eview	Settings	<u>T</u> ools	<u>H</u> elp
<u>N</u> ew			۲				
<u>O</u> pen			Þ	<u>S</u> ub	system	Ctrl+	U
Manage Su	<u>b</u> systems	6	F	<u>A</u> ss	embly	Ctrl+	0
<u>M</u> anage As	semblies		F				
<u>S</u> ave		Ctrl+S					
Save <u>A</u> s			Þ				
<u>C</u> lose			Þ				
Info			Þ				
Impor <u>t</u>							
Export							
<u>P</u> rint		Ctrl+P					
Select Dire	ctory						
E <u>x</u> it		Ctrl+Q					

- 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**.

Elle Edit View Adjust	Simulate Review Settings Too	ls t	elp Controls Mechatronics Durabilit
free boot sussession	Suspension Analysis		Bandlel Wheel Travel
Browse Filters	Eull-Vehicle Analysis Gomponent Analysis	э 9	Opposite Wheel Travel Boll & Vertical Force
Subsystems	General Actuation Analysis	٠	Single Wheel Travel
fisse front arb	DOE Interface		Steering
fsae_front_sus;	1		Static Load
All Other	i Same		External Edes
Adams Analysi	s Flags os		Dynamic
E Adams Strings			Set Syspension Parameters
-wassembly_n	ame		Create Loadcase

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.

Netw 1	fram_fram_suspension.torx_RC_height						
Definition by E	Evening Result Sat Component (Request)						
Result Set Comp	ake at semulation and	Real Post Correso Test Field Mo		Fiel Quesses +			
21							



Net	<ul> <li>fraiduit cooperation fort_RC height</li> </ul>	
Definition b	y Existing Result Sat Component (Request)	
Result Set Carry	(head and uncertain hort as a person transformed and transford pertoducation)	vertcal
Dange Objective's value is the	Take at simulation and	

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.

Export Assembly to Adar	ns/Insight					
Assembly	fsae_front_suspension					
Experiment	front_roll_center_height					
Simulation Script	.fsae_front_suspension.simulation_script					
	OK Apply Cancel					

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**.

- Adams/Shisk	pht (Adams 2013	2) - fron	t_roil_cent	er_height	_	
Ede Edit De	eline Simulation	n <u>I</u> oola	Help			
	×		23	N. 1		医金属
e⊢nors_roil_e ⊕ Factor ⊨incl ⊕ Car ⊕	senter_neight s (0 190) usions (0) didates (190) fsae_front_susy if sae_front_susy if sae_front_susy if sae_front_s if af_toe_i if af_toe_i if bul_joun if af_toe_i if af_toe_i if bul_joun if af_toe_i if bul_joun if hpl_i if hpl_i i	ension b usp et_adju idjuster c_stopp c_stopp damper t_susp. t_susp	ster ster pd_camb pd_drove_ pd_drove_ pr_camb pr_ca	er_adjust shaft_offse djust er_adjust djust djust shaft_offse djust	et et ca_fror ca_fror	2 × 2 y

b. After selecting each one, press the **promote** button:

٠
c. The final model tree will look like this.

⊡-front_roll_center_height	•
⊨-Factors (4:186)	
⊨-Inclusions (4)	
fsae_front_susp.ground.hpl_uca_front.z	
fsae_front_susp.ground.hpl_uca_rear.z	
fsae_front_susp.ground.hpr_uca_front.z	
fsae_front_susp.ground.hpr_uca_rear.z	
⊡ Candidates (186)	

d. Highlight all of the points and press the **tie** button.

💽 Adams/Insight (Adams 2013.2) - front_roll_center_height
<u>File Edit Define Simulation Tools H</u> elp
」 D 😂 🖬 🗙 ] D 🗉 🔺 - 💁 🗣 🔍 📕 🔝 0 🕬 👘
id-front_roll_center_height id-Factors (4:186) id-Inclusions (4)
fsae_front_susp.ground.hpl_uca_front.z fsae_front_susp.ground.hpl_uca_rear.z fsae_front_susp.ground.hpr_uca_front.z
fsae_front_susp.ground.hpr_uca_rear.z

e. Highlight **Tie\_01** and change the settings as follows. Keep the original nominal value and set the range

Rade Section	net, regi ant, regit		
Settion   Variables   Ta	Devision		
Tippe	totings.	Ka. 90	
T Depty	Televence (	a (Petrola Despe	11
Balla Tara		Ease of Adjustment	
C.Aleiten		C Day	
C Robert		C Molecule	
-Felder Person		C Net	

(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.

🔝 Adams/Insig	ht (Adams 2013.	2) - front_roll_	center_heigh	ıt	
<u>F</u> ile <u>E</u> dit De	fi <u>n</u> e <u>S</u> imulation	<u>T</u> ools <u>H</u> elp	)		
🗅 🚅 日	2 6 🛛	<b>≁ → </b>	S 🕺 🛓	<b>3</b> 📾	₩ • ■
	enter_height	Deci	an Cr <sup>Se</sup>	t 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**.

Design Specificanse		
Exclude forms · Role Terrare · Role Terrare	an the second	

#### Step 8. Generate Workspace and Run.

a. Click Generate Workspace.

ile Edit Define Simulation To	ols Help			
	+		<b>7</b>	W
	+ Di			Suites 🔍 –
∃~front_roll_center_height			Generate	e Work Sp
Els Rid Talks Constitute Tarls Man		_		
	m Hawl			
IC G M N H H H K M K K K K K K K K K K K K K K K	CF D. S M			
in Pactors (1.156)	We	rk Space	6	
E Pickalore (1)		Trail	NUM SICA #50	fort RC hegel
Here DCA enaded height	1 C	Teal 1	d.	
(i) Cos Sates (198)	2	Teal 2	21,9292	
G. Inclusion (1)	3	Yest 3	43 (613	
-fact RC locate	4	Test 4	66-7675	
-Candidates (0)	5	Ind 5	47.7167	
IP-Design(D-W)	6	14# 8	125.648	
- Specification	7	16# 7	101275	
-inchestore	8	164.5	152.604	
-D06-g1 Spoce	5	164 B	175.433	
Wak Spece (5-25)	10.3	Dr.# 10	505,781	
- TYDK SQUED HIDABA	\$31	Test TI	210,292	
- ACREANCE	12	Trar 12	241,221	
	80.1	True 13	263.46	
	14	Tra ti	280.078	
	10.1	Tear 16	347.048	
	16	Teur 16	225.938	
	17	Tes 17	365 867	
	.10	Tela 18	372 795	
	100	Trim 19	294,725	
	251	Dia 20	418.664	
	21	Dia 21	438.683	
	22	Dis 22	482.512	
	23	Dia 23	482.447	
	28	True 24	584.375	
			and the second sec	

b. Run the design study.

Ele E	dt D	efge	Simulat	ion Io	als	Heip	,				
0.		×	00			-	3	1	W.	1.111	

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

Trans. In Sect. Mal.	an dan balan dan kasa balan dan b	2.8.3
In the summer	Ministry Concerns	
Anna (Marcold		
and the second	110-	
and her and		
Particular States		
and the second second	174 St. 34	
- Barrison		
and the second second		
A protoco, hep-		
and the second		
a designed of the local data		
Balance		
Participant in the local interview		
Contract loga		
	N N N	
	The second secon	
	Sense and an an an an an and an an an	
	tena l	
		Sec. 1
		Market and a second sec
	1	
	- Contraction of the second	
	IX III	
	1.542	
and the second s		

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

Adams, Warpin Dogues, 2022. R. March and particular	what .			
129 X	E B MAN			
In Frances of the	10	ork Space		
(P. Rampinson (710)	1.1	lise	And Inc. stand	Band M. Smith
D Dream D We	1	Teal 1	÷.	10.11
- Specification	1	Seat 2	211049	234.44
TELEP	1	Test 2	40.60	341 341
State States of States	1	he d	48.38%	The series
What Brack Same		Tau I	812167	TTL OWN
in Terratane	100	Ital 6	100 644	60.722
Applet View	10.0	Tes 7	20125	110.107
	1.1	That 8	16/1244	111.001
	10	714.1	111.03	28-1488
	10	- Trail 15	+0.20	10,000
	111	746.11	29240	27 10 14
	31	141	201229	20.4284
	111	1.1411	261 H	2 6 2 1 1
	94	True to	384.079	/5.004
	41	That N	3710	41716
	191	748.16	108.118	46.8 18
	10.0	748.1	268.567	45万马
	10	14.10	777.744	101847
	10	24.10	264725	101.00
	100	1 m 21	1864	211 214
	13	Dei 21	438.140	211.544
	10	74122	46.5.0	245.118
	100	12 Wei 21	100 281-	
	10	True (54	184 371	255363
		3 mil 25	626.3	30.401

front\_RC\_height has been filled.

Adams/Insight (Adams 2013.2) - front_roll_center_height	
<u>File</u> <u>E</u> dit Define <u>Simulation</u> <u>T</u> ools <u>H</u> elp	
🛯 🗁 🖬 🗙 🖉 🖷 🔺 🛥 🕾 🖉 🗐	M • m
È-front_roll_center_height	Fit results

#### Step 10. Fit and Optimize Results.

📓 Adams/Insig	3ht (Adams 2013.2	2) - front	t_roll_cent	er_height			
<u>F</u> ile <u>E</u> dit De	fi <u>n</u> e <u>S</u> imulation	<u>T</u> ools	<u>H</u> elp				
] 🗅 😅 🖬		• •	an ∋	1	处 😒	M	
□ front_roll_c	:enter_height						otimize

#### a. Click Fit Results.

	Mesesare :		Maxman	Value :	Ciper.		Target	Weight	Cast
ton_RC_height	-353 29		245.99	9.2356	Ma	10	0	1	9,2556
Overall		1-2-21			Ducus.				8 2366
					Max				
					Visila				
					MaxTo				
					THE IS				
					LsEq		2		
					taa				

- 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

	Memum		Maximum	Maximum Value Fr		
UCA Indexed heigh	2		01.3	271.61		

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**.

Tools Help Controls Mechatronics	
Adams/Car Template Builder F9	
Adams/View Interface	
Database Management	Add to Session
Curve Manager	Remove from Session
Command Navigator	Set Default Writable
Database Navigator	Create Database

d. Type **Humvee\_Workshop** as Database Alias and choose the location of .cdb folder as the Database Path.

🚺 Add Detebase to	Session		*
Database Alias	Humvee_Workshop		
Database Path	C:Wsers\wwei\Wark\humvee_warksh	op/Humvee_V	Vorkshop.cdb
Database Info	ок	Apply	Cancel

#### 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.

		22	Update I	able	Fie	set table		Node	Finder	
Main	er Utaragment e ID 1	Apply A	/arker locatio	n Mave	to neda	Properve exp	ession	Prese	ve locat	en
Nani	ber of sights 2					Sat	By .	Corre	ctiona	
	Market	Connections	Node E	Interface	Ratative L	ecition	Distor	nes .	Move	
1	at jant i 8	Harryos rigid HV Front	¥ 101		-10.001	15.0031.55	119.67	7	CIPI	
2	per joint   16	Humoe rigid HV Front	× 106	•	2.45.0.00	0.00	2.45		enpr	
3	per joint i B	Harryos, rigid HV, Front,	× 907		2.45.0.00	0.00	2.45		CIPI	-
4	al cirt j 15	Humon rigid HV Front	+ 101	<ul> <li>.</li> </ul>	2.46.0.00	-8.00	2.46		expr	
5	pl_icim_i 8	Hernot_rigid HV_Front_	× 103		2.45, 0.00	0.00	2.45		cape	
6	ber beshprom i 6		29166T		22 55 0 8	0.00.0	22.65		expr	
7	bil bushgeom i 7		227053		22.55, 0.0	00, 0.00	22.55		expt	
۵.	bil bushing j 7		901	•	2.45, 0.66	-0.00	2.48		expr	
9	bir bushgeom j.7	÷	207829		22.55, 0.0	00, 40.00	22.55		expi	
10	ber bushing j 7		106	×	245.0.00	0.00	2.46		expr	
111	bit bishingn i S	6	224848	1.1	22 55.0.6	0.04-8	25 55		water.	

#### 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

Category	Dynamics			
Model	.Humvee_rigid			
Integrator	ннт 💌			
Formulation	13 -			
Corrector	C Original @ Modified			
Error	1.0E-005			
Hmax	1.0E-003			
Hmin	(none)			
Hinit	(none)			
Adaptivity	(none)			
Interpolate	C Yes @ No			
Kmax	6			
Maxit	10			
Scale	10,10,10			
Alpha	-0.3			
Eleta	0.36			
Gamma	0.7			

**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.

Category	Equilibrium	-		
Model	.Humvee_rigid			
Equilibrium Type	Static	•		
Error	1.0E-004			
Tlimit	10.0			
Alimit	5.0			
Maxit	500			
Stability	1.0E-005			
Imbalance	1.0E-004			
Static Method	Original	•		
Atol	1.0E-006			
Rtol	0.0			
Maxitl	40			
Etamax	0.9			
	Contraction and Contraction			

d. Click Close.

### Step 5. Run an Adams/SmartDriver Event.

a. From the tool bar, click **Simulate->Full-Vehicle Analysis->Adams/SmartDriver**.

Simulate Review Settings To	ols	Help Controls Mechatronics	
Suspension Analysis	•		
<u>F</u> ull-Vehicle Analysis	►	Open-Loop Steering Events	
Component Analysis	•	Cornering Events	
General Actuation Analysis	Straight-Line Events		
DOE Interface	•	Course Events	
		File Driven Events	
tem		Static and Quasi-Static Man	
		Adams/ <u>S</u> martDriver	
		Vehicle Set-Up	
ension		Event Builder	
		Road Builder	
		Path Optimization	

- 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.

Full-Vehicle Assembly	Humvee_rigid		
Output Prefix	BELGIAN_ROAD		
End Time	9	<u>.</u>	
Number Of Steps	9000		
Mode of Simulation	interactive	•	
Course Type	C Driver Course	<ul> <li>3D Spline, C</li> </ul>	RG or RGR Road
3D Road Data File	[] mdids //Humvee_	Workshop/roads	Itbl/belgian_block
nitial velocity	Auto-Calculate		KITVINF ·
Gear Position	2	Shift Cycle II	me ju.s
Smart Driver Task	User Defined	Power-bas	ed Shift Control
Maximum Acceleration	n (%)		
		5 A	
Driving 0 4	í.		▶   100
Driving 0 <u>•</u> Braking 0 <u>•</u>		50	<ul> <li>▶ 100</li> <li>▶ 100</li> </ul>
Driving 0 <u>•</u> Braking 0 <u>•</u> Lef	J	50	<ul> <li>▲ 100</li> <li>▲ 100</li> <li>Right</li> </ul>
Driving 0 <u>•</u> Braking 0 <u>•</u> Lef Cornering •	」 t 50▲	50 	
Driving 0 Braking 0 Let Cornering	J t 50▲ I-Line Setup	50 	100 100 Night 100
Driving 0 <u>•</u> Braking 0 <u>•</u> Lef Comering • Quasi-Static Straight Create Analysis Log	J t 50 J t-Line Setup File	50 	100 100 Right 100

#### Step 6. Export.DAC File.

a. From the tool bar, click **Tools->Plugin Manager**, then check **Adams/Durability**. Click **OK**.

				_	
alls Dess Bell Domate systematics ()	P Rays Usinger (2) R				
Address Temples Raiser PT	Sara	Lord	It and at Drama		
Existent Veragement +	AlemCyam	W.748	798		
	Risto Donine	E 7m	7 Yes	1	
Comment Emilpator	Nam Duality	W Yes			
Extension Hergelon,	Natiobiotaneses .	P.Yee	P. Yes		
Contract of the second s	Polares Witedian	F 780	17. Yes	24	
Entry M. Holping Convolution Agringuts Mass Departs Activity Entry Activity Entry Activity Amount Studies Experts Communit Experies Communit Experies Communit Experies Communit	Describer Vesser Aufter Lawona			- Martin II	
	reg Dae per present Venetimente d' Alanciè l'empresent alue 19 Adanciè l'empresent alue 19 Cen triage Definis fingen: Definis	rgi Data Strongen Kalan V Adarsch Nampen Kalan V Referen Kalan V Cert Height Det Kalan Strongen Kalan V Cert Height Det Kalan Kalan Det Kalan Kalan Kalan Det Kalan Kalan Kalan Det Kalan Kalan Kalan Det Kalan Kalan Kalan Det Kalan Ka	ng Das Ret Ganzo Mannes ( Taligo Krage Alance Caraon Man, Carao Man, Carao Man, Carao Man, Detain Stangen, Santon Man, Carao Man, Detain Stangen, Santo Man, Carao Man, Detain Stangen, Man, Carao Man, P. Yao Detain Stangen, Man, Nagata Man, Nagata Man, Detain Stangen, Dana Man, Deta	gar Date Daren Malance G Frager Marger 1 2 Adarce Carlos Barlos Carlos Carlo	

b. From the tool bar, click **Durability->FE-Fatigue- >Export**.

<u>H</u> elp	Controls	Mechatronics	<u>D</u> urability		
			FE Modal Export	۲	
			FE-F <u>a</u> tigue	•	<u>E</u> xport
			MSC. <u>F</u> atigue	•	Import
			Nodal Plots		
			Hot Spots Table		

c. Choose the flex body in the assembly.



d. a. Uncheck FES file, then click OK.

	FE-Fatigue Exp	port		×		
Fle	xible Body	.Humvee_flex.H	V_Front_Suspe	ension_1.ges_s		
Job Name		BEIGIAN_ROAD				
	Modal Coord	dinates				
	Analysis	.Humvee_flex.E	BELGIAN_ROA	D_smart_drive		
	Output Tim	e: Start	End			
	Basis	Orthonormalize	ed 🔹	•		
	Format  O DAC O RPC III					
I Include modal velocities						
	FES File					
		<u>0</u> K	Apply	<u>C</u> ancel		

- 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.

lars	Data modified	Type	Sce
Belgier, road 0001.das	6/10/2014 2:04 714	DAC File	37.60
belgies sued 0002.des	6/12/2068 2:04 PM	DACTIN	37.60
beigies, road, 000%, dar	8/12/2084 2/14 7/4	DACKIN	\$7.94
balgar, road, 0004.dac	8/11/2014 2017 PM	DICHI	37.94
belgion road 0005.das	6/12/2014 2/3/ 214	DICFIE	37.83
belgion rund 6006.das	6/12/2054 2:54 PM	DACEN	37.90
belgan, road, 0007 day	8/32/2042/54 PM	CAC File	57.58
helper, mail (0008 day	8/32/284 234 PM	DAC PRE	17 M
belgar, read, 0000.das	8/11/2/84 204 244	EAC Fee	37.95
belgier, read (010,das	6/10/2014 204 744	DACTIN	37 40
belgion road 0011 day	6/12/2064 2:04 PM	DECRIM	37.60
belgies, road, 0012, day	6/12/2014 2:14 714	DACKIN	07.80
belgar road 0013-lier	8/11/2014 EDX PM	DICHIE	17.00
belgion road 0014.das	6/12/2014 2014 244	DICFIE	27.82
belgion road 0015.des	6/12/2064 2:54 PM	DACKIN	37.90
belgan_road_0056.dac	8/52/2014 2:54 PM	<b>DAC File</b>	\$7.85
belger read 0017 day	8/11/284 204 PM	Date Page	57 KJ
belgan_read,0018.dat	8/12/2064 20H PM	EAC Fee	37.99
belgier, read, 0010, das	6/10/2014 204 214	DICHI	37.40
belgion road 0020.das	-6/12/2064 2:04 PM	DECRIM	37.60
belgier yord 0021 der	8/10/2014 2-14 714	DACKIN	(CT A)
belgar, road 9022 dat	6/12/2014 33(794)	DICHE	27.95
belgian road 0023.dec	6/12/2014 2:34 714	DACFIE	27.60
14.0	1.0000000000000000000000000000000000000	B-1-0.001	10.10

## 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

Element Type	Element Size	2	_	I Stress Analysis		
Solid	Minimum Size	1	_	Shell Stress Layers	Top	
Element Shape	Growth Rate	1.5		Normal Direction	Z	
Tetrahedral	Shell Thickness	(1:0r	(m)	Collapse Small E	dges	
Element Order	Angle Per Element	(45d	eg)	Edge Tolerance (0.0	(mm)	
Quadratic •		Contract				
Element Specificatio	m MSN Based Refine	ment	CON	@ OFF		
Size	Curvature Based Se	caling	CON	IF OFF		
Edge Shape						
Straight •	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.

F	FlexBody Type 🔽 Extrusion 🕫 Geometry C Import Mesh							
	C Mesh/Properties  Attachments Flexbody preview							
	Attachment Points Definition Table - Master Nodes							
		Coordinate Reference		Node ID	Rel.DOF			
	1	.UCA_Flex.ground.POINT_3		5.8379E+004	0.0			
	2	.UCA_Flex.ground.POINT_4		5.838E+004	0.0			

- 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.

Slave Nodes for Atta	ichment # 1
Selection Type	Spherical 🔹
Attachment Method	Rigid(RBE2)
Radius	7

- i. Click on Coordinate Reference 2. Change Selection Type to Cylindrical.
- j. Set Attachment Method as Rigid (RBE2).
- k. Set Radius to 5.
- I. 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.

MNF and MD DB Transforma	tion
Flexible Body Name 💌	uca_flex
MNF   Output File	uca_1
	Create Flexible Body
Mirror O Translate O	Rotate
Plane Normal to Vector	•
Direction	Global Y
Point on mirror plane	0,0,0
Node Offset	
More	OK Apply Cancel

#### 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 </Baja>/assemblies.tbl.

Open Assembly		
Assembly Name	Search	<acar_shared>/assemblies.tbl</acar_shared>
	Text •	<oemo∠>rassemblies.tbl <minibaja>/assemblies.tbl</minibaja></oemo∠>

- 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.

A Adams/Car Adams 2013.2
<u>File Edit View Adjust Simulate Review Settings T</u> ools
.baja_full_vehicle_rdw
.baja_full_vehicle_rdw
.baja_full_vehicle_rdw.baja_front_suspension
.baja_full_vehicle_rdw.baja_dw_rear
.baja_full_vehicle_rdw.baja_steering
.baja_full_vehicle_rdw.baja_front_tires
.baja_full_vehicle_rdw.baja_rear_tires_large
.baja_full_vehicle_rdw.baja_frame
.baja_full_vehicle_rdw.baja_4w_brakes
.baja_full_vehicle_rdw.baja_powertrain

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.

Nun	nber of digits	2
<b>F</b>	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.
- 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.

	baja_fuli_vehicle_rdw				
baja_flex	baja_flex				
6					
600	600				
interactive 👻					
mdids://acar	shared	i/roads.tbl/road	1_3d_ra	mp_example.m	df
20		km/hr			•
2	٠	•			
DCD File	٠				
mdids://acar_	shared	/driver_data tb	l'veloci	ty dod	
t-Line Setup					
File					
	6 600 interactive 20 22 DCD File mdids://acar_ 120 DCD File TLune Setup File	6 6 6 6 6 6 6 6 6 6 6 7 7 7 7 7 7 7 7 7 7 7 7 7	6 6 6 6 6 6 6 6 6 6 6 6 6 6	6 600 interactive mdids //acar_shared/roads tbl/road_3d_ra 20 km/hr 2 DCD File mdids://acar_shared/driver_data tbl/veloci t-Line Setup File	6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6

#### 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.

Flex Body	dy baja_full_vehicle_rdw.baja_front_suspension.ger_upper_cor					
Analysis	.baja_full_vehicle_rdw	v.baja_flex_3d_road				
Type	Von Mises	Stress				
File Format	HTML	•				
Start	End					

Radius		0.0	
Count	•	10	
Sort Order		Absolute	-
File Name			

VON I	INSES Hot Spots for ger_uppe	control_arm_f	lex Date= 2014-	08-04 17 33	07			
Model= .baja_full_vehicle	e_rdw.baja_front_suspension	Analysis= baj	a_flex_3d_road	Terr	e = 0 to 5 s	HC .		
	Top 10 Hot Spots		Abs	Ra	Radius= 0.0 mm			
Hot Spot	Stress	Node	Time	Locatio	on wrt LPRI	(mm) -		
	(newton/mmt**2)	bi	(xec)	X	Y.	Z		
10	131.424	45580	4.37	-970.126	206.45	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	286.105		
.7	128.128	45585	4.37	-968.599	207.684	286 414		
8	127.984	45592	4.37	-972.699	213.957	285 831		
9	127.467	56449	4.35	-967.485	210.799	295 776		
10.	127 453	45578	4.37	-969.483	204.585	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.

Analysis	baja_full_vohicl	e_ndw.baja_flex_	3d_road	
Plexible Body	nt_suspension.g	er_upper_contro	al am flex	
Select Node List	45580			
Node to Add to List	t.		4	Stress C Stran
🕫 Von Maes	F Max Prin	□ Normal X	T Normal Y	T Normal Z
F Max Shear	F Me Pris.	F Shear XY	□ Shear YZ	F Shear ZX
Reference Marker	1			

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.



I. 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.

A Adams/(	Car Ada	ms 2013.2		
<u>F</u> ile <u>E</u> dit	<u>V</u> iew	<u>A</u> djust <u>S</u> imulate	Review	Settings <u>T</u> ools
VD Cour	<u>.</u>	Hardpoint	•	Modify
[.vb_cou	1	General Part	•	Table
Browse	Filter	Switch Part		Info
Sub	syster	Gears		
-6	TR Po	Actuators	+	
	TR_Bo	Parameter Vari	able 🔸	
	TR_Re	Kinematic Togg	gle	
- 🗗	TR_Ste	Shift		
	IR_Re	Adams/Vibratio	n 🕨	

- 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.

A Shift Subsystem		×
Subsystem		VD_Course.TR_Rear_Suspension 💌
Aft Translation	•	500
Up Translation	-	
		OK Apply Cancel

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 lxx, **1.5e9** at lyy and **1.5e9** at lzz.
- 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.

A Hardpoint Modification	Table		
C Assembly @ Subsy	stem VD_Course	e.TR_Body	•
	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.

A Informa	ation
0.02103	
Apply	Parent Children Modify Verbose
0.9	1.0
\$	
[LATERA	L_COEFFICIENTS]
a0 =	1.65000
a1 =	-34.0
a2 =	1250.00
a3 =	3036.00
a4 =	12.80
a5 =	0.00501
a6 =	-0.02103
a7 =	0.77394
a8 =	0.0022890
a9 =	0.013442
a10 =	0.003709
a11 =	19.1656
a12 =	1.21356
a13 =	6.26206

- h. Right click at Property File and search in <VD\_</li>
   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.
- I. Click OK.

Full-Vehicle Assembly	VD_Course .
Output Prefix	VD_Course
End Time	10
Number Of Steps	500
Simulation Mode	interactive •
Road Data File 🛛 📆	mdids //acar_shared/roads.tbl/2d_flat.rdf
Steering Input	straight line •
Initial Velocity	40 mile/hr ·
Start Time	1
Open-Loop Throttle	
Final Throttle	75
Duration of Step	1
Gear Position 🍵	2 •
Shift Gears	
Quasi-Static Straight	-Line Setup

#### 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.

Full-Vehicle Assembly	VD_Course	*
Output Prefix	VD_Course	
End Time	5	_
Number Of Steps	250	_
Simulation Mode	interactive ·	
Road Data File 🛛 🦁	mdids //acar_shared/roads.tbl/	2d_1
Steering Input	straight line 💌	
Initial Velocity	90 km/hr	•
Start Time	0.5	
Open-Loop Brake		
Final Brake	100	
Duration of Step	0.01	
CHIPMEN I	-	

#### 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.

Animation View	Camera	Record	Overlay	Ap
Follow Object	.VD_Cours	e.TR_Bod	y.ges_cha	ssis
Mount Camera At				

- k. Press shift+S to change render mode to wireframe. This will help us to spot the wheel lock up.
- I. 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.

Assembly F Subsystem	VD_Course.TR_Brake_System		
	real_value	remarks	
pvs_front_brake_bias	0.7	(none)	
pvs front brake mu	0.4	(none)	

- 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.

Smelation	Rar Republiker	1	Composition
W Grunne annal - Manar/Gr W Grunne krake - Manar/Gr	pro-tear broke toopie date opening mean displacement streating data too		Lational_clip_from inclination_expls_from
	+ tunning bil sheet summary yeark bil sheet une farme bil sheet une has farme bil sheet une has farme	384	lonansi, ship, sona londinansi oʻgʻogʻogʻogʻo
	bil, obset, birs, polling, seams bir, obset, birs, polling, seams bir, obset, birs, bor, forme bir, obset, birs, birs, birs, bir, obset, birs, polling, seams bir, obset, birs, polling, seams birs, birs, birs, billing, seams		
<u>i</u>	+ 13,3003 + 13,3005,27000	(De-La) (De	
Source Result Sets	· 73 Tours Parparetas	0.1	



- g. This time, none of the tires lock up.
- 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 **condi-tion\_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.

Full-Vehicle Assembly	VD_Course •	
Output Prefix	VD_Course	
End Time	5	
Number Of Steps	250	
Simulation Mode	interactive •	
Road Data File 🛛 🤯	mdids://acar_shared/roads.tbl/2d_fi	
Initial Velocity	100 km/hr •	
Gear Position	4 -	
Maximum Steer Value	150	
Start Time	1	
Cycle Length	1	
Steering Input	Angle 👻	
E Onice Control	and the second se	

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.

Full-Vehicle Analysis: St	ep Steer 🛛 🗶
Full-Vehicle Assembly	VD_Course
Output Prefix	VD_Course
End Time	5
Number Of Steps	250
Simulation Mode	interactive -
Road Data File 🐻	mdids://acar_shared/roads.tbl/2d_fl
Initial Velocity	50 km/hr 💌
Gear Position	3 💌
Final Steer Value	150
Step Start Time	1
Duration of Step	0.2
Steering Input	Angle
Cruise Control	
🔽 Quasi-Static Straight	-Line Setup

 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\_Mod-**el.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.

.Quarter_Car_Model
Browse Groups Filters
Bodies
🛄 🖣 Input

Body	Sprung_Mass		
Category	Mass Properties		
Define Mass By	/ User Input		
Mass 500 0			
bx 4 1605333	3333E+0	05 Iyy 4 16053333338+005	Cff-Diagonal Terms
Center of Mass M Inertia Reference	larker Marker	Sprung_Mass.cm	

d. Similarly, double click the spring-dampers under Forces folder and then you can define preload, spring stiffness and damping coefficient.

Name	kus_cus		
Action Body	Unsprung Mass		
Reaction Body	Road_Input		
Stiffness and Dam	ping		
Stiffness Coefficie	ent - (4000(newton/m))		
Damping Coefficie	ent • (78(newton-sec/m))		
Length and Preloa	d		
Preload	5390.0		
Default Length	(Derived From Design Position)		
Spring Graphic	On, If Stiffness Specified		
Damper Graphic	On, If Damping Specified .		
Force Display	Force Display On Action Body		
	a		
second second states			

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.

#### Joint Motion х Name Input JOINT\_3 Joint translational Joint Type Direction Translational • • Define Using Function Step(time, 0,0,0. Function (time) Velocity • Туре Displacement IC Velocity IC 17 <u>0</u>K <u>Apply</u> Cancel

## 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 cosimulation. 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 CCCXV

