ADAMS Guide

Gregory P. Starr

February 2010

Contents

ADAMS Guide

Gregory P. Starr

Department of Mechanical Engineering

The University of New Mexico

1 Introduction

ADAMS (Automated Dynamic Analysis of Mechanical Systems) is a CAD package for mechanical system analysis and design. It is *very* comprehensive, and is currently used by professional engineers. It is representative of this class of CAD tools. ADAMS is used extensively in university education, and is an ideal package for use in ME 314.

1.1 Format of this Document

It is almost useless to sit down and *read* about computer software. One must have a *specific problem* to do—you can then learn about the software as you approach this specific problem.

The specific problem which I've selected is text Problem 3.9, which is shown below. This is a four-bar linkage with given geometry and motion, in which you are to find the angular velocities of the links at a specific location. In solving this problem, we will find *MUCH* more information than simply angular velocities at a specific location—that is one of the many benefits of using a CAD package such as ADAMS.

The mechanism shown in the figure is driven by link 2 at $\omega_2 = 45$ rad/s ccw. Find the angular velocities of links 3 and 4.

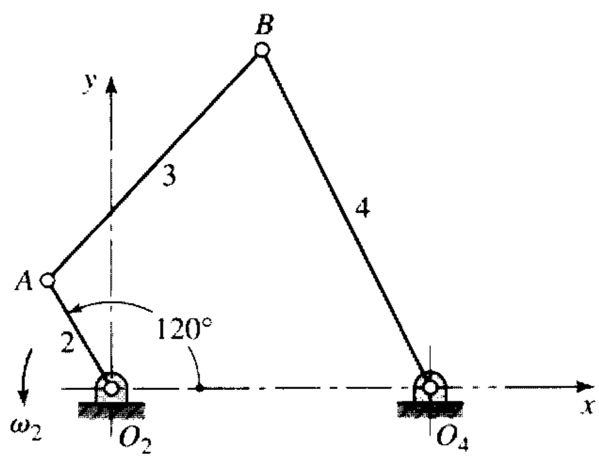


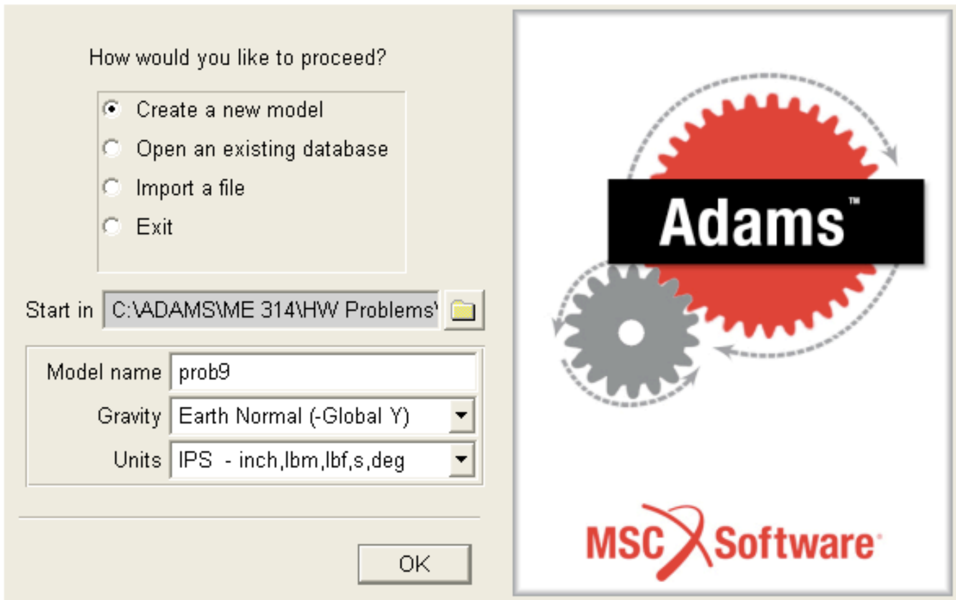
Figure P3.9 $R_{AO_2} = 4$ in, $R_{BA} = 10$ in, $R_{O_4O_2} = 10$ in, $R_{BO_4} = 12$ in.

2 Starting ADAMS/View and Selecting Options

Double-click the ADAMS/Viewicon on the Desktop. ADAMS/Viewis the application in which you construct the mechanism. The other application we will use is the ADAMS/Postprocessor.

2.1 Selecting Options

You will be presented with the dialog box below:



I have specified the following information:

1. Create a new model—we are starting a new model
2. Start in: I've specified the directory in which to create the file
3. Model name: pick something reasonable (no spaces or punctuation, as I recall)
4. Gravity: whatever is appropriate
5. Units: problem 3.9 is given in units of inches, so I selected IPS
6. *NOTE:* Gravity and units can be changed later if needed

2.2 Settings Menu

From the ADAMS/View window, click the Settings menu. The dropdown menu shows many items, but the one you probably always will change is Working Grid. This will depend on the "size" of your particular problem. The two perpendicular lines on the workspace define the (0,0) point.

Referring to Problem 3.9, if we place this (0,0) point at O_2, we will need a Working Grid of extent (remember units are inches):

$$X : -6 \ldots 24 \implies \text{Size} = 48 \qquad Y : -6 \ldots 16 \implies \text{Size} = 32$$
$$\text{Spacing} = 1$$

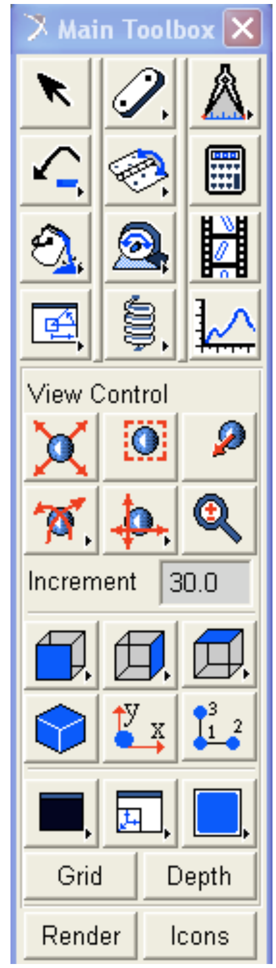
Type these in and click OK. You may need to ZOOM (type "z" and move mouse up/down) and TRANSLATE (type "t" and move mouse around) the window. You can change the grid size and spacing at any time.

Once you click OK the dialog will disappear and the `Main Toolbox` will appear:

2.3 Main Toolbox

You will be interacting with the `Main Toolbox` very frequently.

Some of the key controls in the `Main Toolbox` are:



- First Row
  - Arrow: will get you to ADAMS/View
  - Link: used for constructing part bodies and features
  - Compass: used for constructing linear and angular measures
- Second Row
  - Arrow-minus: undo last action
  - Hinge: here's where you construct joints
  - Calculator: this is how you simulate behavior
- Third Row
  - Paint bucket: used for specifying color
  - Motor-like things: used to specify motion of a joint
  - Film - used to playback the results of a simulation
- Fourth Row
  - Drawing-board icon: used to move bodies around in the workspace
  - Spring: used to specify connectors of various types
  - Plot: used to open ADAMS/Postprocessor for plotting
- View Control: these buttons are used to zoom, translate, *etc.* There are keyboard equivalents for many of these functions.
- Boxes: used to get Front, Right, Top, and Isometric views. These have keyboard equivalents also.
- Next row: background color; toggle main triad visibility, and window layout.
- Final four buttons: toggle grid on/off; toggle between orthographic/perspective; render or wireframe, and toggle marker (frame) icons on/off

NOTE: If you "hover" the cursor above a control, the function of that control will appear. If you right-click on most buttons they will expand to show you more selections.

3 Constructing the Mechanism

Now things get more interesting—we are going to construct the four-bar linkage of Problem 3.9.

3.1 Working Grid

Zoom and translate the Working Grid such that the "origin" is near the lower-left, and there is enough "room" to construct the mechanism. Since it's trivial to zoom and translate, feel free to initially make the grid whatever size you want.

3.2 Link 2

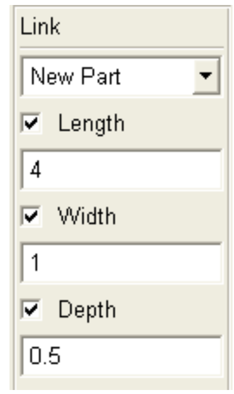
The "ground" is always present; it's automatically constructed for you, even though it's invisible. So we start with link 2. Select the "link" tool:

and you will be presented with the following:

Here I've selected "New Part" since it *is* a new part.

The "length" is known: 4 inches, so check the box and enter the length.

Although the "width" and "depth" are not given, I've assumed reasonable dimensions and entered those.



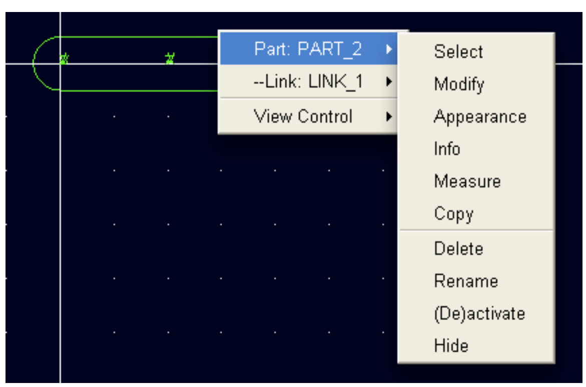
3.2.1 Creating Link 2

I'm going to place link 2 horizontal (*i.e.* at 0° instead of at 120°) then during the simulation phase apply angular velocity of $\omega_2 = 45$ rad/s to move it through its cycle. The initial geometry is simpler.

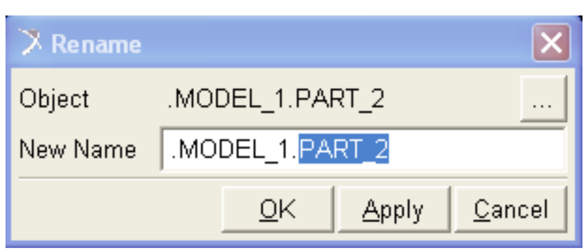
Select the Link icon on the first row of the Main Toolbox, fill in the information as shown at right, then notice at the bottom of the ADAMS/View window there is the message "Link: Pick one end, then drag to other end"...

Place the cursor at the (0,0) point, then click and drag upwards. You'll see the outline of the link. Release the button and the link remains as shown below (your color may be different):

If you right-click on this part you will see a drop-down menu, as shown below:



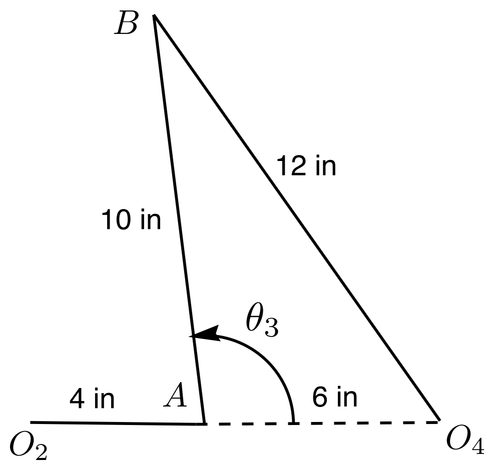
The NAME of this part is PART_2. If you wish to change the name (highly recommended), select the RENAME option, select PART_2, and type in the name you wish (body_2?).



From the PART_2 drop-down menu notice the item "–Link: LINK_1"...that's the *GEOMETRY* of the part. The geometry of a part contains its dimensions, and gives it a shape on the screen.

3.3 Adding Links 3 and 4

To build the rest of the linkage in ADAMS/View, you need to perform some geometrical analysis to find the orientation of either link 3 or 4. Specifically, with link 2 horizontal, we have the situation below:



where I am going to solve for angle θ_3, the orientation of link 3.

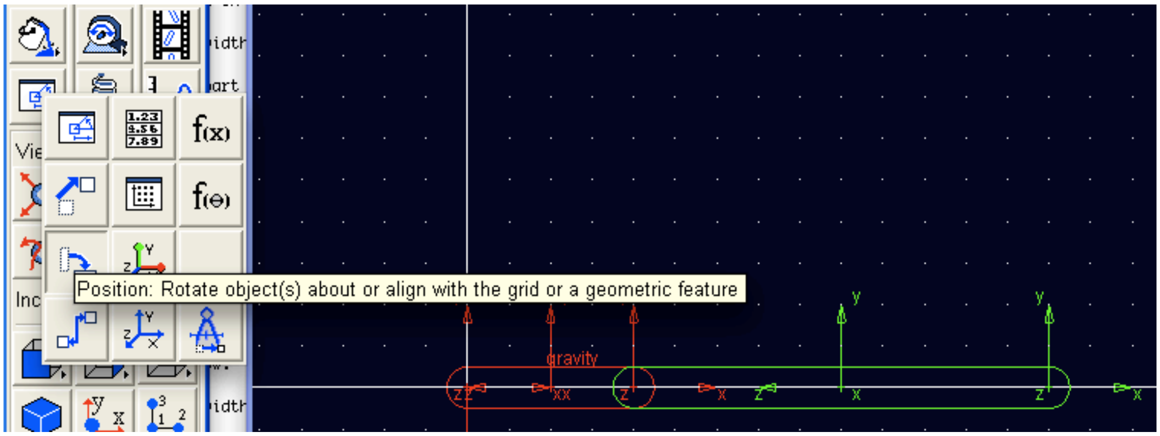
This geometry can be solved using a **Loop Closure Equation** of the sort described in text Section 2.6, although I think it is easier with the **law of cosines**:

$$\cos\theta_3 = \frac{6^2 + 10^2 - 12^2}{2(6)(10)} \implies \theta_3 = 93.8226^\circ \tag{1}$$

3.3.1 Link 3

We will initially place link 3 horizontal in ADAMS/View, as shown below

To rotate link 3 about point A, right-click the (4,1) element in the Main Toolbox:



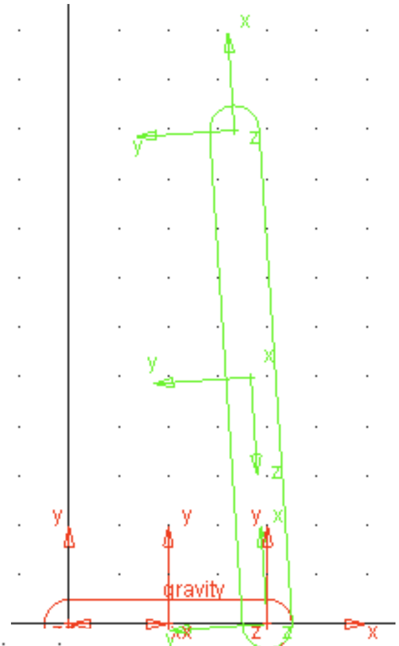
Select the button showing the part rotating from a vertical to a horizontal position.

Click "selected" and enter the angle of 93.8226°.

As you move the cursot into the workspace at the bottom you will see "Rotate: select the object to rotate." When you see "link3" click to select.

Then you will see "Rotate: select the direction vector to rotate about." You want to select the Z axis at the left end of the link, but it's coming directly out of the screen. To see the TOP VIEW, press "T" (Shift-T), then select the vector.

Press "F" (Shift-F) to see the front view, and link3 is rotated, as shown below (I changed to white background):



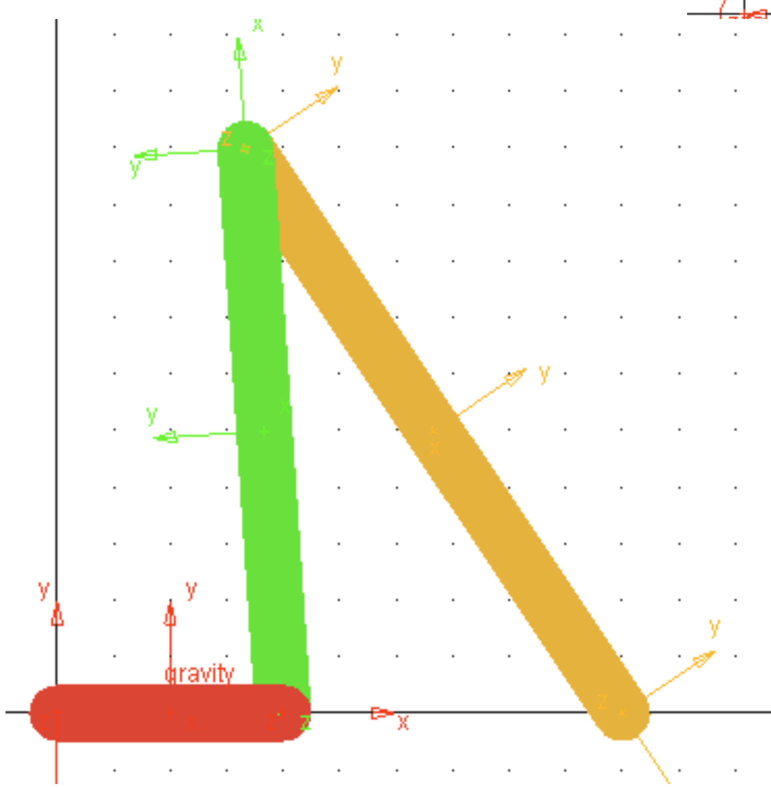
If you make a mistake you can use the "UNDO" button to retrace your steps.

3.3.2 Add Link 4

Now you can add link 4, which should connected the end of link 3 and the ground at O_4.

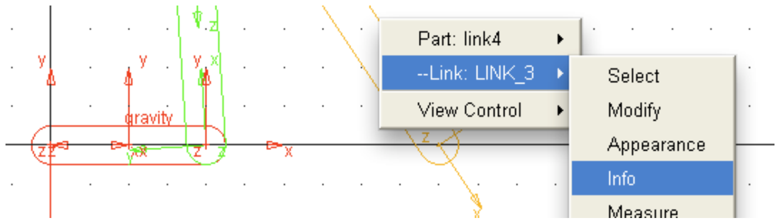
If you press the F4 key (while ADAMS/View window is active) a "coordinate" window will open, showing you the coordinates at each grid point. Point O_4 is at (10, 0).

Note that you should "uncheck" the length box for link 4, since our geometry was *INEXACT* (angles to only four decimal places), and a link of *EXACTLY* the desired length won't *QUITE* fit.

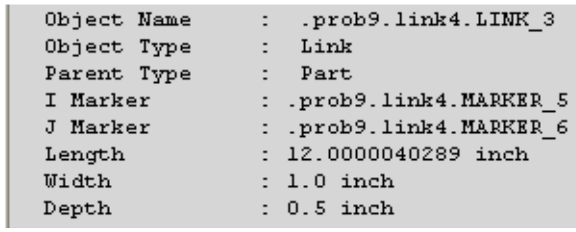


I shaded the bodies in the figure above by typing "S" (Shft-S). I also renamed all the parts as link2, link3, and link4.

To check the length of link 4—which should be exactly 12 inches—right-click on link 4, then the geometry (the child of the part name), then INFO:



You will see the following information:



Object Name	:	.prob9.link4.LINK_3
Object Type	:	Link
Parent Type	:	Part
I Marker	:	.prob9.link4.MARKER_5
J Marker	:	.prob9.link4.MARKER_6
Length	:	12.0000040289 inch
Width	:	1.0 inch
Depth	:	0.5 inch

The final length of link 4 was 12.0000040289 inches; pretty close!

3.4 Adding Joints (Constraints)

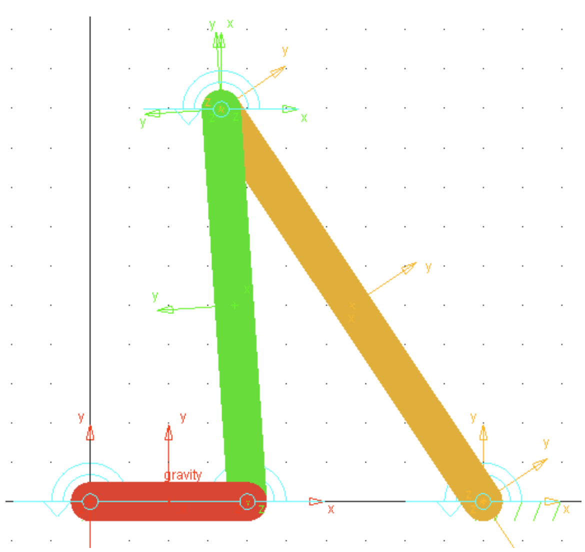
If we simulated the mechanism now everything would fall down off the screen. There are no connections anywhere.

3.4.1 Simulating the Behavior

You can verify that by pressing the CALCULATOR icon, select DURATION instead of END TIME, enter 1 as duration, select STEP SIZE, enter 0.01, then press the PLAY button. To replay the simulation, click the FILM icon, then PLAY. To “slow down” the simulation, select a smaller time step.

3.4.2 Hinge Joint

Right-click the (2,2) element in the Main Toolbox, and select the “hinge” joint (it may already be displayed). All the settings in the joint pane should be okay. In the workspace, select the two links you wish to connect (one may be “ground”) then click where you want the axis. Do this for all joints. It may be interesting to perform simulations as you go.



3.5 Adding Motion

One of the final steps in building a model is to add motion to the model. This is typically done in one of two ways:

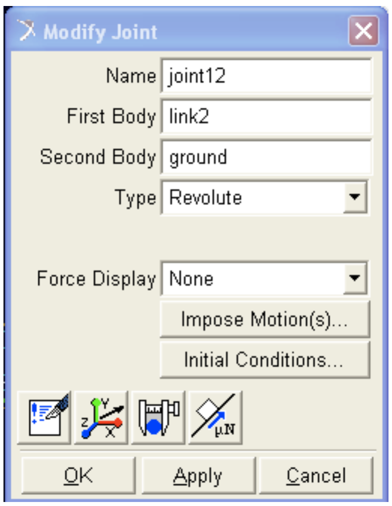
1. Adding motion to a *JOINT*
2. Adding motion to a *LINK* (body)

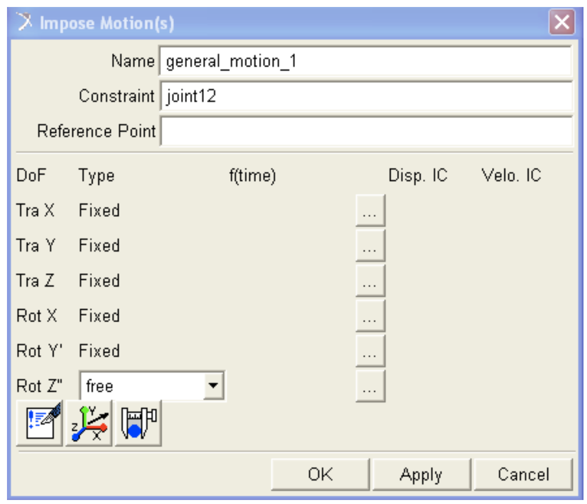
Both of these methods are described below.

3.5.1 Adding Motion to a Joint

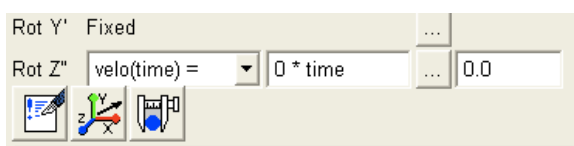
In this case link 2 has an angular velocity of 45 rad/s. Since link 2 is connected to ground (link 1) by a pin (hinge) joint, one can simply specify the motion of that joint. I renamed the joints to be the following: `joint12, joint23, joint34, joint14`, so I want to add an angular velocity of 45 rad/s to `joint12`.

Right-click on the joint, then select MODIFY. This will bring up the box at left below:



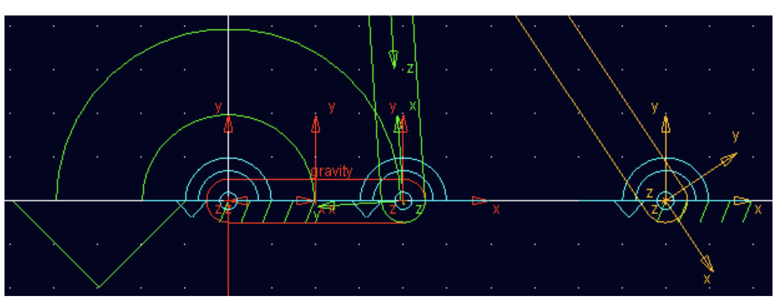


Click "Impose Motion(s)" and you will get the box at right above. Under ROT Z click on "free" and select "velo(time)="



Enter the angular velocity you want in rad/s (45) and leave the Disp. I.C. at 0.0.

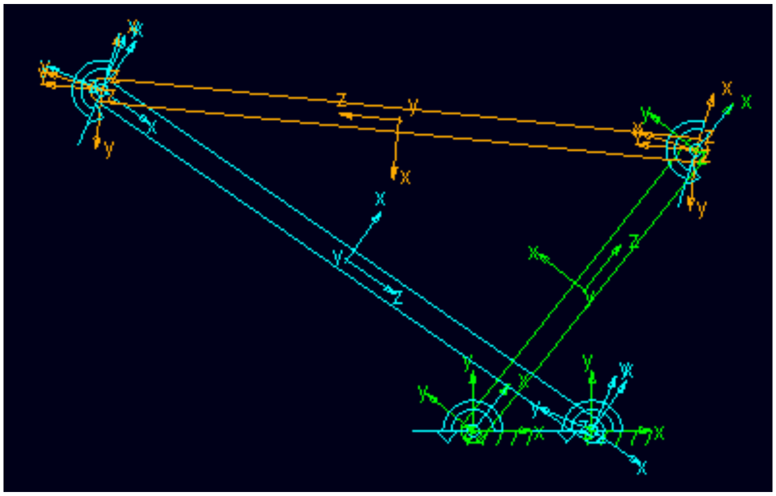
Click OK twice, and now there should be a large "angular arrow" indicating the applied motion:



3.5.2 Addding Motion to a Link (body)

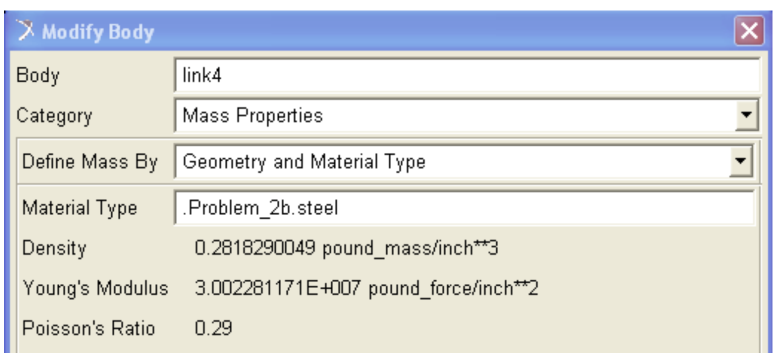
If a mechanism is such that one must specify the motion of a link that is *NOT* attached to ground via a joint, then you must specify the motion (typically the velocity) of the body directly using an **initial condition**.

In 4-bar linkages that satisfy Grashof's Law, sometimes the crank cannot rotate continuously, but the coupler can (two of the inversions of HW Problem 1.2 are like this). In these cases, to animate the linkage it is simplest to specify the angular velocity of the coupler link directly.

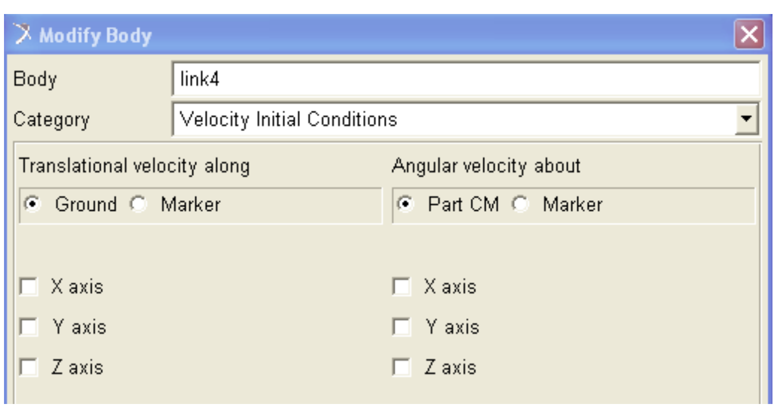


The figure above shows the ADAMS/View model of Problem 1.2 with link 2 fixed. In this case it is link 4 (the amber-colored link) that can rotate continuously. However, link 4 is not connected to ground, so we must specify an angular velocity of link 4 directly. Here's how:

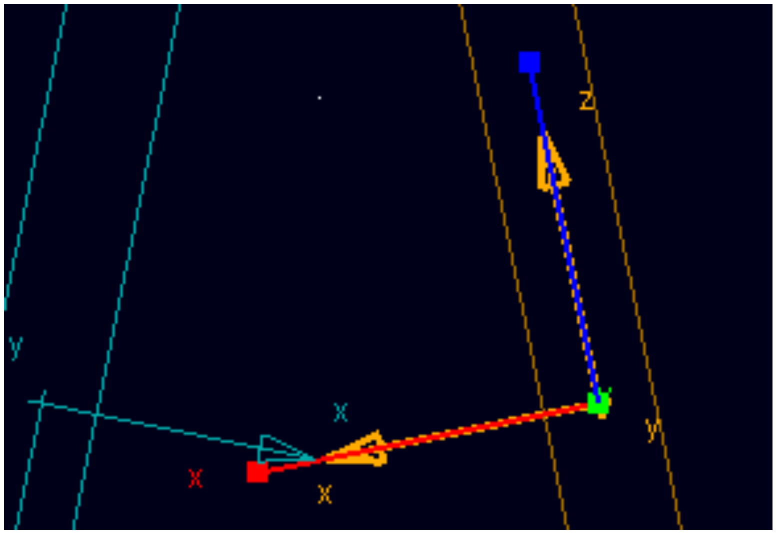
1. Right-click link 4, then select **Modify**. You will get the following box (not all of it is shown below):



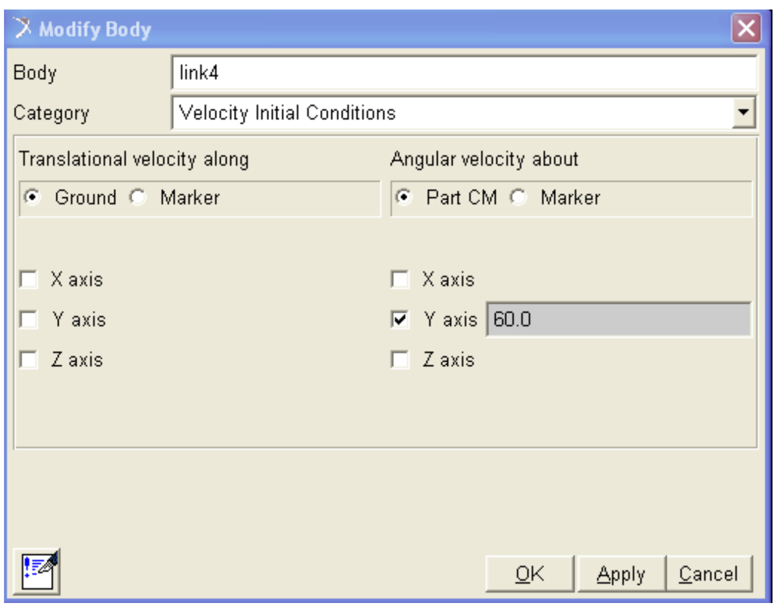
2. Under **Category** select **Velocity Initial Conditions**. You will get the following box:



3. You want to apply an **Angular Velocity** about the **Part CM** (Corner Marker, not center of mass, BTW). The question is: which axis? I usually look at the **center of mass marker**, since it has the same orientation as the CM, and is easier to see. If you zoom in on the center of the link, then select the **c.m. marker**, you should see something like this (I reset the mechanism so link 4 now has a different orientation):



The axes of the selected marker will be colored as: X (red), Y (green), Z (blue). This is in the order $XYZ =$ RGB, which I find easy to remember. So the axis perpendicular to the plane is the GREEN (Y) axis. Thus we should apply an angular velocity around Y axis. Here the units are DEG/sec, and I'm going to select 60 (then I'll simulate for around 6 seconds to get one rotation). The window will look like the following:



4. Click **OK** and you're done.

3.6 Saving the Model

Anytime the workspace window as active, typing CTRL-S will save your model as a "binary" file with the name you initially specified. This can be quite a large file (several Mb); this is because **EVERYTHING** is saved...simulation results, *etc.* There is another way to EXPORT your model as a CMD file which is **MUCH** smaller. Saving periodically is obviously a good idea. Each time you save you're prompted whether you want to preserve the previous copy as a backup. I usually select "No" (which is the default).

Now we're ready to actually run the thing!

4 Simulating the Final Mechanism

You've already (perhaps) done some simulation, but now we need to specify the simulation parameters a little more carefully.

4.1 Simulation Parameters

Click the "CALCULATOR" icon to bring up the simulation tool: the two things you probably want to change are:

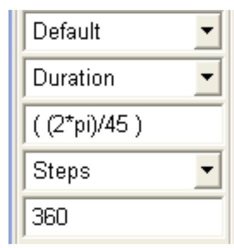
1. Duration
2. Stepsize (or number of Steps)

4.1.1 Simulation Duration

It makes sense to conduct the simulation for one cycle of motion. At an angular speed of 45 rad/s, we have

$$\text{Period } \tau = \frac{2\pi}{\omega_2} = \frac{2\pi}{45} \text{ sec} \tag{2}$$

Since this duration corresponds to 360°, and the original problem statement asked for behavior at $\theta_2 = 120°$, it makes some sense to specify the number of Steps to be 360. Later we can modify the plot to show crank angle in DEG explicitly. This part of the Main Toolbox is shown below; you can type in the scale factor of $(2 * \text{pi})/45$ directly:



4.2 Running the Simulation

In ADAMS/View, you first **compute** the simulation in **CALCULATOR** mode. After that, the simulation can be re-played in **FILM** mode. For more control in **FILM** mode, the **FILM** panel can be opened; this is also the way to trace the path of particular point(s) in the model.

4.2.1 Computing the Simulation

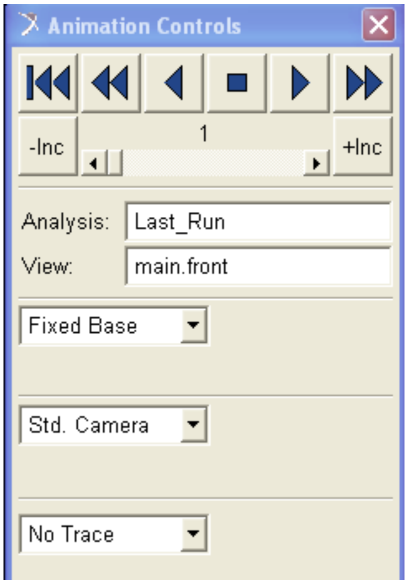
Click, the **CALCULATOR** button, then click the PLAY button. The linkage should move through one cycle.

4.2.2 Playing the Simulation

Once the simulation has been computed, there's no need to compute it again to view it. To play the simulation again, click the **FILM** icon and *then* click **PLAY**. No need to simulate again unless you change parameters. While in **FILM** mode, the results of the most recent simulation will always be played back.

4.2.3 The "Animation Controls (AC)" Panel (needed to trace paths)

To open the **AC** panel, first click the **FILM** button, then click the "..." button at the lower right of the panel. the **AC** panel will then appear, as shown (lower portion of panel not shown):



To trace the path of a point, there must be a **marker** at that point, so you may have to go back and place a marker at the desired point. Then click the **No Trace** item in the panel, and set to to **Trace Marker**.

To select the marker, RIGHT-CLICK in the box below Trace Marker, and use either **Marker/Pick** (you then click on the marker directly) or use **Marker/Browse** (you select from a list in the Database Navigator; it's helpful if you've named your marker something meaningful).

Then use the **PLAY** control in the **Animation Controls** panel to play the animation, and the path of the selected point will be drawn. Additional points may be selected in the same manner; you may have multiple paths shown.

4.2.4 Changing Views

At any time you can change from FRONT (F) view to TOP (T), RIGHT (R) (neither very interesting), or ISOMETRIC (I). You can also press R (Shift-R) then use the mouse to orient the linkage.

To make the animation continuous, click the "loop" button in the lower left corner of the Main Toolbox "animation (FILM)" window.

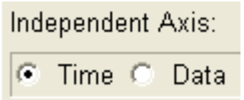
You can change views (as well as ZOOM) while the animation is running.

5 Plotting the Results

Switch to the ADAMS/Postprocessor by clicking the "plot" icon in the Main Toolbox. At the lower left, under "Source," select "Objects." Under "Filter," selecting "Body" or "Constraint" will limit the items displayed to those categories (there is no applied "Force" in this model).

5.1 Plotting Variables with respect to Time

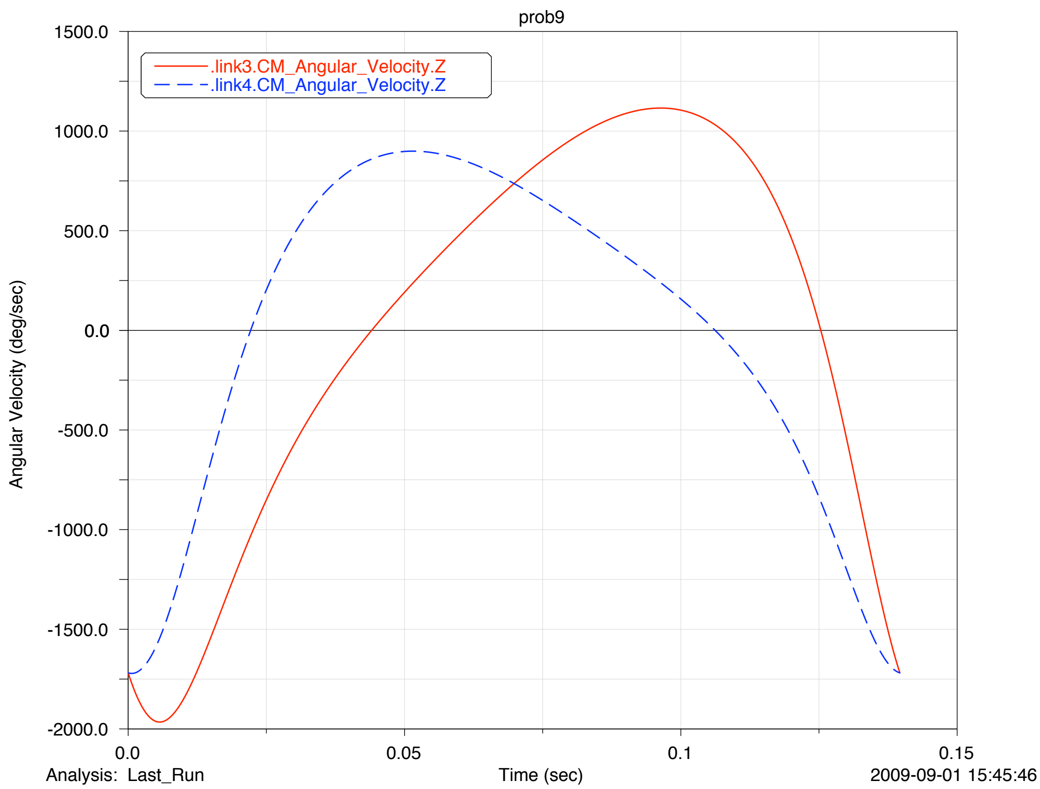
We often wish to plot variables with respect to `Time`, and this is the default setting for the ADAMS/Postprocessor, as shown in the lower right corner of the window:



You simply select the variables you want to plot. Here's how:

To find the angular velocity of links 3 and 4 (as specified in the HW assignment), select "`link3`" then "`CM_Angular_Velocity`" then "`Z`" then "`Add Curves`" and you'll get it. Same idea for link 4 (can both be on the same plot), and also for angular acceleration.

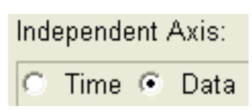
The plot below is the "default;" it's quite easy to add/modify titles, labels, line styles and colors, *etc.*



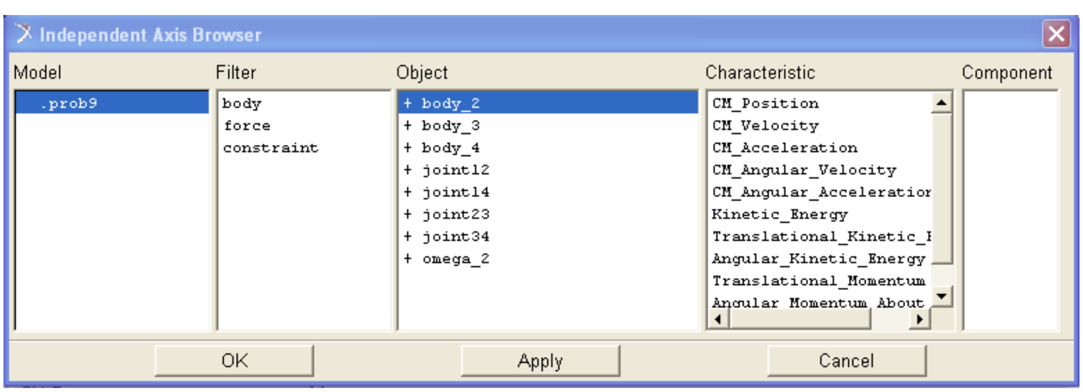
Using ADAMS/View you get the behavior of the mechanism over the full cycle of motion, not just one snapshot. It is equally easy to plot accelerations, forces, *etc.*

5.2 Plotting Variables with respect to Other Variables

We often want to plot a quantity *vs* the angular displacement of a body, *e.g.* crank angle. To do this simply check the `Data` button at the lower right of the window:



The "`Independent Axis Browser`" window will appear,



Now you select which variable should be the independent (horizontal) axis. Unfortunately, the "angle" of body 2 will **NOT** be available. You really need to place a "`ANGLE MEASURE`" to obtain the angle of body 2. I'm going to defer that until later.

If you want to change the independent axis from "`TIME`" to "`Theta 2`" then that can be done by axis scaling, which is in Section 5.3.

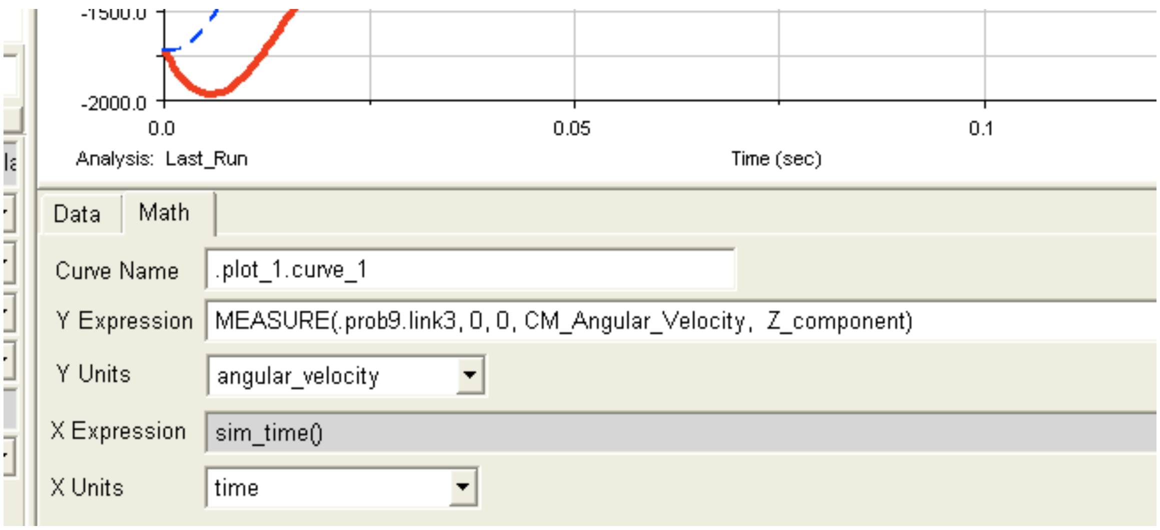
5.3 Changing Axis Scaling

After performing a simulation and plotting the results, you may want to modify the plot axis scaling. The most common changes are:

- Changing the `haxis` scaling from time (`sec`) to an angle (`DEG`).
- Changing the `vaxis` scaling from—for example—`DEG` to `RAD`.

Consider the plot on page 13, suppose that we want to change the horizontal (`haxis`) scaling from time (`sec`) to angle (`DEG`).

Select the `MATH` tab at the bottom left of the plot, then click on the "RED" curve (this is "`curve_1`" in the selection window to the left of the plot; you can also click on this curve in the LEGEND). Below the plot you will then see this:



Note that the "RED" curve is thicker (it's selected). We will change the horizontal axis from "sec" to "deg". We will also change the angular velocity units of the vertical axis from "deg/sec" to "rad/sec." Here are the steps:

1. You need to reverse the time scaling you imposed in equation (2) on page 11, and add a multiplier of 360 to represent the total angular displacement (deg). Therefore you should enter `*45/(2*pi)*360` immediately after `sim_time()` in the "`X Expression`" box.
2. In the "`Y Expression`" box, change from `deg` to `rad` in the same manner, by appending `*(pi/180)` to the string beginning with `MEASURE(...Z_component)`.
3. Click "`Apply`" at lower right to make the changes. The axis numbers should change, and the "blue" curve will shrink noticeably.
4. Select `curve_2` by either clicking on the blue curve on the plot or in the legend, and perform the same operations on it. Click "`Apply`" at lower right to make the changes. Both curves should now look as they did originally.
5. In the left side window, click "`haxis`", then in the lower pane select "`labels`" and change the contents from "`Time (sec)`" to (perhaps) "`Link 2 Angle (deg)`" ...or something like that. Repeat for the "`vaxis`" and change "`Angular Velocity (deg/sec)`" to the same thing but with "`rad`" instead.

6. At this time you may want to change other features of the plot; particularly the "length" of the horizontal axis. Select "`haxis`", then below select "`Format`", and *UNCHECK* the "`Auto Scale`" checkbox. The "`Limits`" boxes will then appear, and you should enter "`0`" and "`360`" (then press RETURN).
7. You may also use the small right-arrow to go over to "`Tics`", *UNCHECK* "`Auto Divisions`" then select "`Divisions`" or "`Increments`" and enter what you want. It's best to experiment with these to see how they work. You can also change the font sizes for labels, numbers; add a title, *etc.* You can change virtually *everything* on a plot. See my lower plot on the previous page for an example.

6 Creating Measures in ADAMS/View

Measures are intended for producing explicit scalar measurements of a kinematic displacement. There are two kinds of measures:

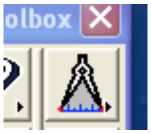
1. Point-to-Point Measures
2. Angle Measures

As an example of a Point-to-Point Measure, recall the HW problem in which two airplanes were flying in different directions—we wanted the distance between them. I created a Point-to-Point Measure to produce that distance.

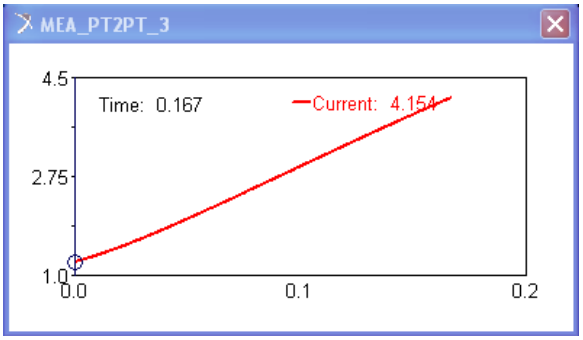
Angle Measures are useful anywhere you wish to measure the angle of a body. In the Freudenstein's Equation linkage synthesis problem, two angle measures are necessary to produce the link angles θ_2 and θ_4 for subsequent MATLAB processing.

6.1 Point-to-Point Measures

To create a Point-to-Point Measure, click on the "divider-like" icon in the Main Toolbox:



You will be prompted to select locations (markers) to measure **From** and to measure **To**. Select these markers in the standard manner. After selection—if a simulation has previously been done—a small "strip chart" window will appear:



Note that the **name** of this Measure is `MEA_PT2PT_3` or something equally arcane. I suggest you rename the Measure something meaningful using the **Rename** option from **Tools/Database Navigator** (you can rename both the measure and its display, for example `theta_2` and `theta_2display`.

When you animate the simulation the circle in the Measure Display will show you the progress of the simulation. I usually delete the "strip chart" window and use the Measure from within ADAMS/Postprocessor when making plots.

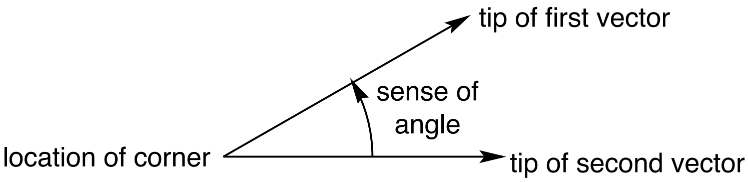
6.2 Angle Measures

To create an Angle Measure, right-click the "divider" icon and select the "protractor" icon:

You will be prompted to:

1. Pick the tip of the first vector (must be an existing marker)
2. Pick the location of the corner (must be an existing marker)
3. Pick the tip of the second vector (must be an existing marker)

These three points (markrs) define a **plane** within which the angle will be measured. The sense of the angle is as shown below:



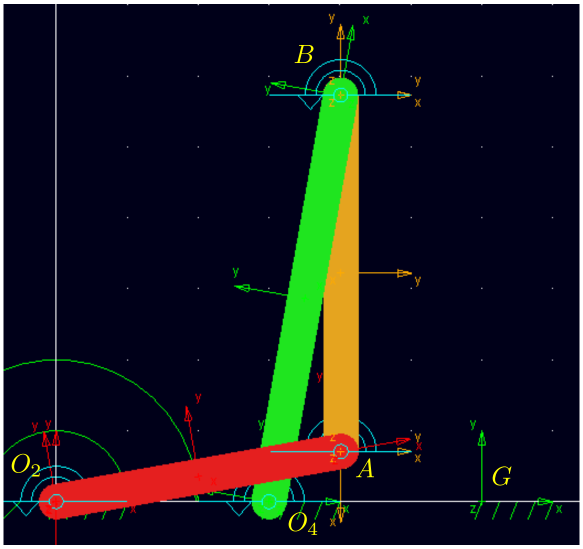
Make sure the three markers are in a plane parallel to the screen; otherwise your Measure will have a slight error. I suppose sometimes you might want an out-of-parallel plane for the Measures, but not in ME 314.

You will get a "strip chart" window just like with Point-to-Point Measures; delete it if you wish. Again, I suggest you rename the Measure something meaningful.

Note that Measures are **not** visible in the ADAMS/View window; the only way to see them is *via* the **Tools/Database Navigator** window.

6.3 Using Measures in Plotting

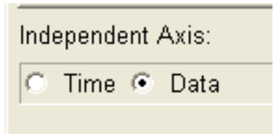
Frequently the reason you create Measures is to subsequently plot those variables. Consider the four-bar linkage of my Freudenstein Example:



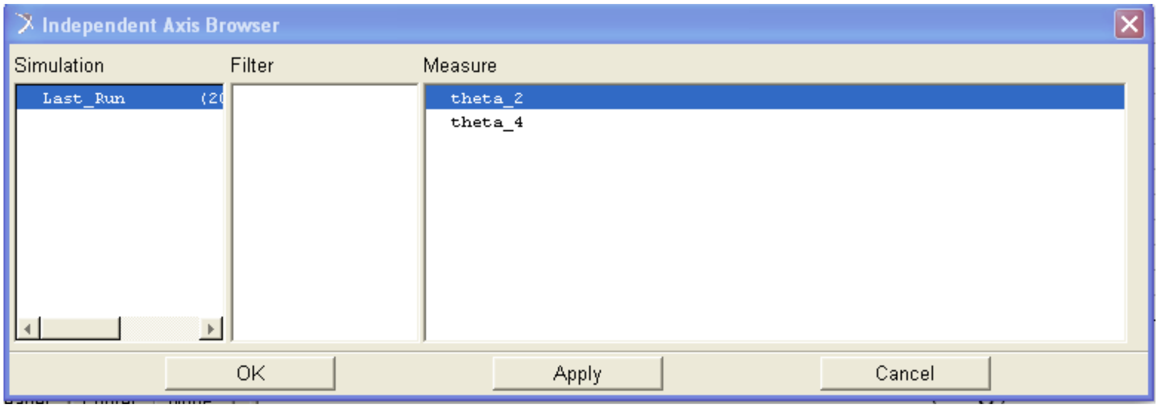
I created two angle measures which I renamed `theta_2` and `theta_4`, each with the following **Tip 1**, **Corner**, and **Tip 2** markers:

Angle Measure	Tip 1	Corner	Tip 2
`theta_2`	A	O_2	O_4
`theta_4`	B	O_4	G

Simulate the mechanism however you wish, then when you go to the ADAMS/Postprocessor window, the Measures are available for use in plotting. For example, to plot θ_4 *vs* θ_2 you would select a "`Data`" plot (lower right of window):

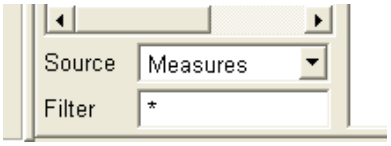


The **Independent Axis Browser** window would then appear:

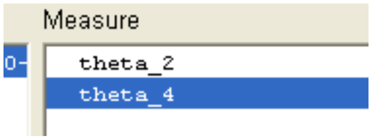


in which I would select *dependent* variable `theta_2` (as shown) and click **OK**.

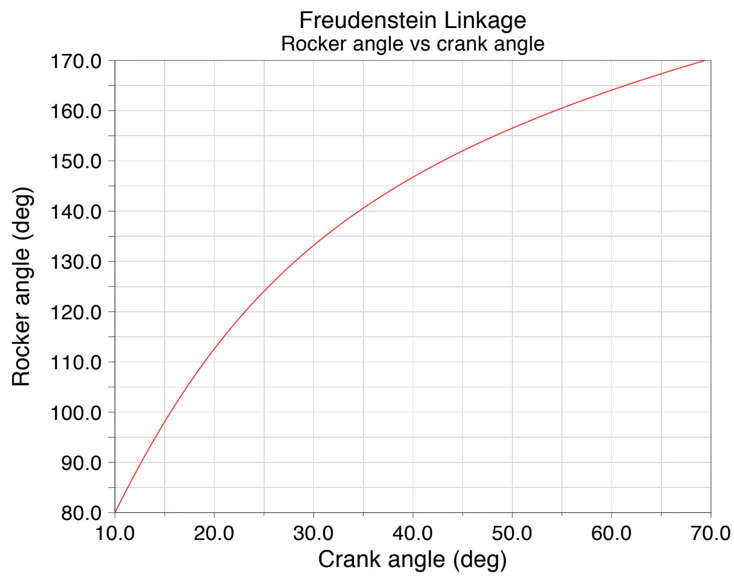
Then from the ADAMS/Postprocessor, select **Measures** in the **Source** pane, as shown below:



Now you can select *independent* variable θ_4 (well, its spelling), like this:



Finally, you're set to make the beautiful plot shown on the next page...



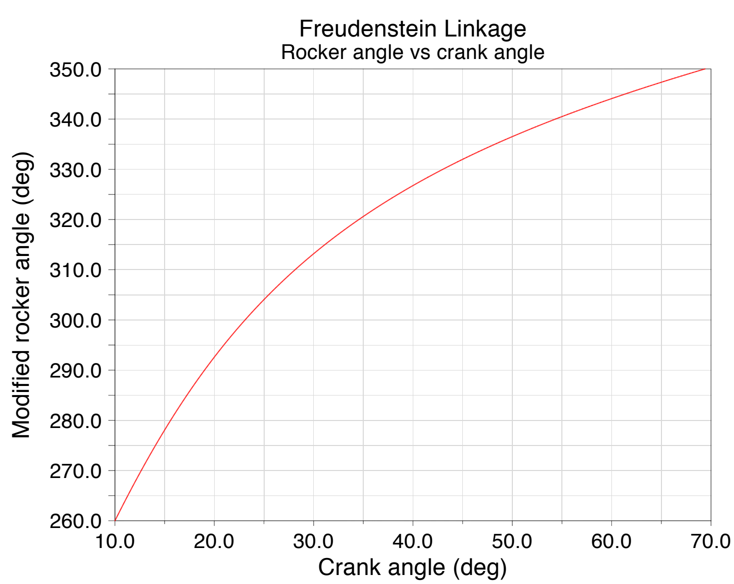
You may notice that $80° \leq \theta_4 \leq 170°$ in the plot, whereas in the actual Freudenstein linkage $260° \leq \theta_4 \leq 350°$. This can be corrected by modifying the data, as shown below.

6.3.1 Modifying Data in ADAMS/Postprocessor

It may happen that you need to modify the result of a simulation—for example, the measure `theta_4` used here **does not** measure the θ_4 shown in the Freudenstein analysis of text Figure 11.25. To correct this, we must add 180° to the data generated by the measure. Here's how:

1. Select the "`Math`" tab in the lower pane...
2. Click anywhere on the curve; the fields will be populated...
3. In this case we want to modify the "`Y Expression`" (`X` would be the same); we wish to add 180...
4. Simply append "`+180`" to the desired field, then click "`Apply`"...

Note that you now may have to modify the "`vaxis`" limits, but the plot will now reflect the modified measure.

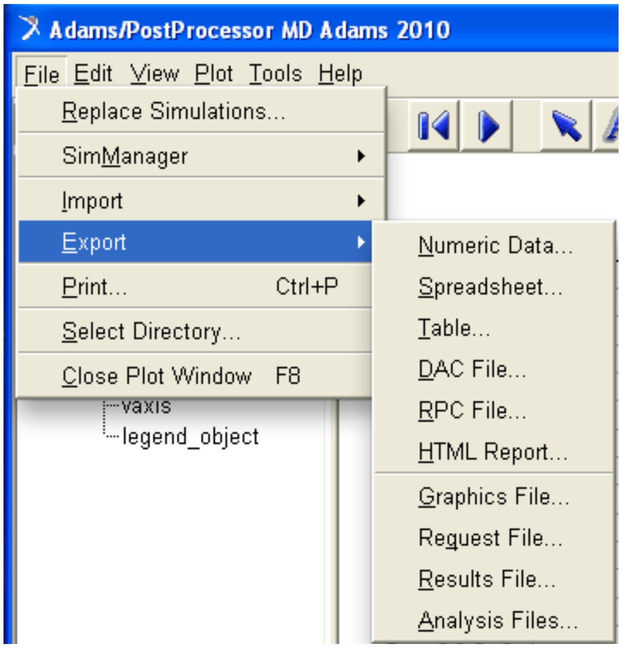


7 Exporting Numeric Data from ADAMS

Sometimes simply plotting data within ADAMS/Postprocessor is not enough—you need to export it for further analysis by other software.

7.1 Export Formats

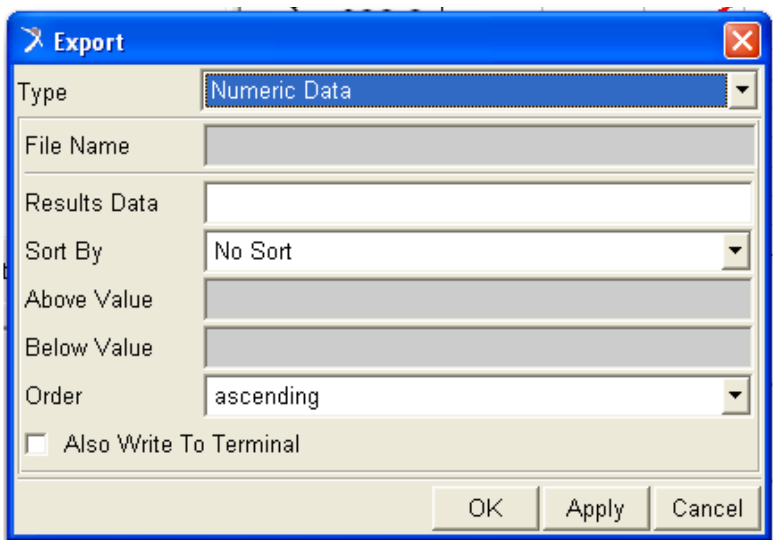
From the ADAMS/Postprocessor, data may exported in several formats, as shown below under the "File" menu:



In ME 314, you are usually going to be processing the ADAMS results with MATLAB, so I'll focus on exporting to a MATLAB-compatible format. This is the first format, "Numeric Data..."

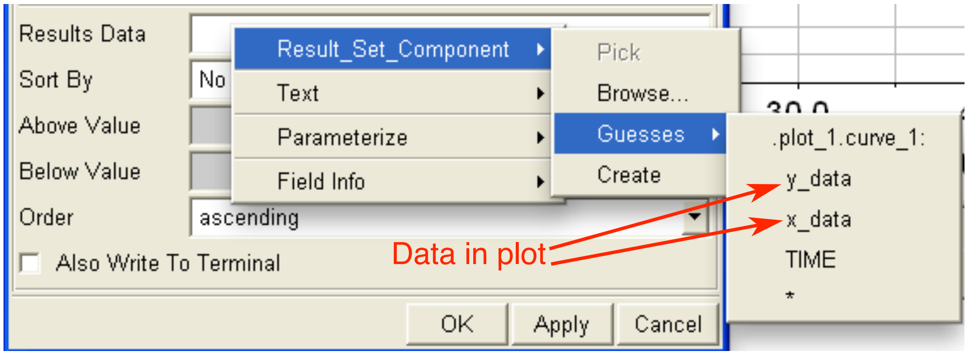
7.2 Exporting Numeric Data...

When you select the Numeric Data... item, the Export window opens:

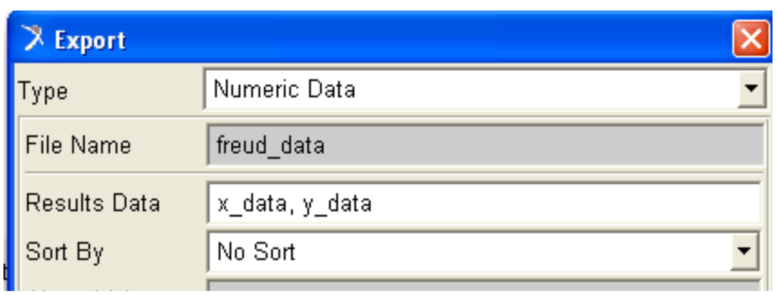


You should set the fields to the following:

- `Type`—leave this on `Numeric Data`
- `File Name`—your choice, but use a meaningful name
- `Results Data`—since you have already created a plot, RIGHT-CLICK here, go to `Result_Set_Component/Guesses` where you will see "`y_data` and `x_data`"



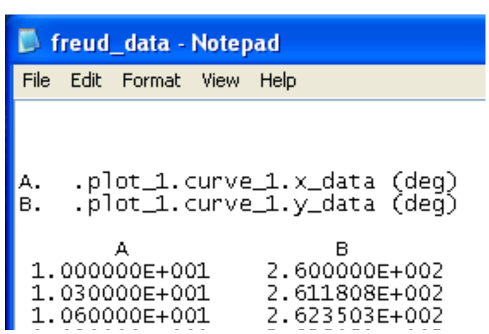
- Select `y_data` or `x_data`
- Finish selecting variables (they are ordered by column as they appear in this list)
- `Sort by`—leave at `No Sort`
- `Above Value, Below Value, Order`—leave blank, `Order` doesn't apply...
- `Also Write To Terminal`—never tried this; no reason to check it...
- My choices for the relevant portion of the `Export` window are here:



- Click `OK` and the data will be exported as `Numeric Data`...

7.2.1 Removing Text from Header

If you open the datafile in a text editor (`Notepad`, for example), you will see that there is some alphanumerica content at the beginning:



Since these data will be opened from MATLAB, this alphanumeric header must be removed. Do that (all the way down to—and including—the `A` and `B` column titles) and resave the file.

7.2.2 Opening Exported File within MATLAB

Although this is an ADAMS Guide, I'll show how to open the datafile we just exported from within MATLAB. Once the file is in your working MATLABdirectory, simply load it:

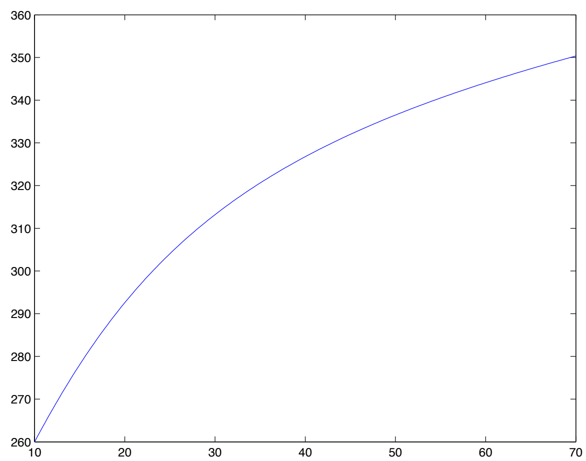
```
>> load freud_data
```

Probably you would like to extract column 1 to a variable called **theta2** and column 2 to variable **theta4**. Here's how:

```
>> theta2 = freud_data(:,1);  % The ":" operator means "all values", so these mean
>> theta4 = freud_data(:,2);  % "column 1, all rows" and "column 2, all rows."
```

Now it may be comforting to plot θ_4 *vs* θ_2 and see the same thing as the ADAMS plot:

```
>> plot(theta2,theta4)
```



This is the same as the plot in Section 6.3.1. Of course, you can add labels, change the linestyle, fontsize, *etc.* But the ADAMS data is now in MATLAB so you can do whatever you want with it.

8 Creating General Constraints (GCON) in ADAMS/View

ADAMS/View "GCON" motion constraints are frequently necessary for enforcing conditions like

- one body sliding on another (*e.g.* pin-in-slot)
- one body rolling on another without slip (*e.g.* disk rolling on plane)

8.1 Mechanism Joints and DOF

All "joints" in ADAMS/View are connections of bodies that remove degrees of freedom (DOF). For example, a "hinge" (revolute) joint removes all three translational DOF (x, y, z) and the two rotational DOF with axes perpendicular to the hinge axis. Likewise a "translational" joint removes the two translational DOF perpendicular to the motion axis, and removes all three rotational DOF.

8.2 GCON—General Constraint

A GCON is simply another means of removing or linking the DOF of mechanism components. Its general formulation is:

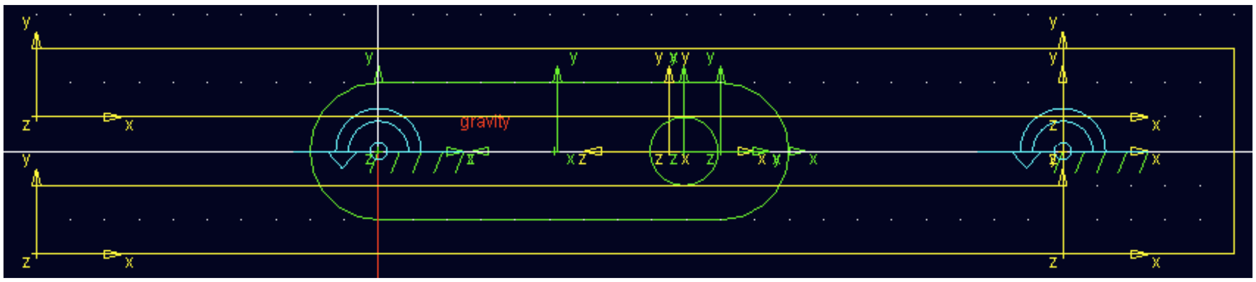
$$f(q) = 0 \tag{3}$$

where q are one (or more) generalized (linear or angular) displacements or velocities.

In this section I'll develop the GCON for keeping the "pin" in the "slot" for the Geneva Stop mechanism.

8.2.1 Geneva Stop Model

From the ADAMS/View Command File, you have the following ADAMS/Viewwindow:

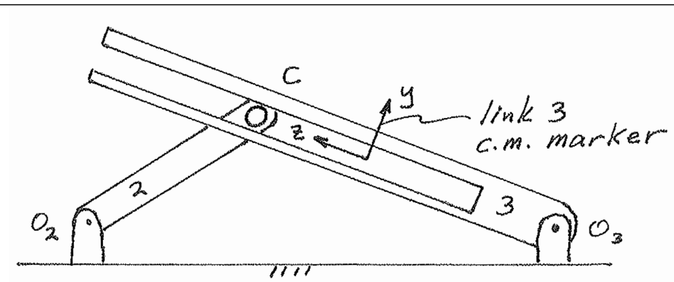


Notice that there are several markers (frames) near the pin—these will be important in creating the GCON that keeps the pin in the slot.

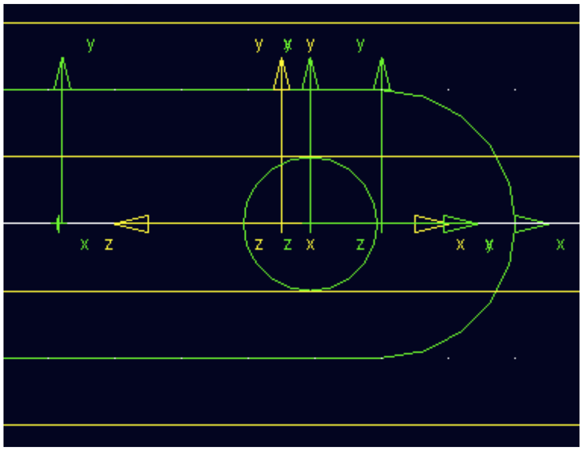
8.2.2 Decription of our Constraint

You must first have an analytical understanding of the constraint under consideration. Consider the sketch of the Geneva Stop below, where the link 3 c.m. marker is fixed to link 3 (of course). For the pin to remain in the slot:

The displacement of point C in the y direction of the c.m. marker MUST BE ZERO



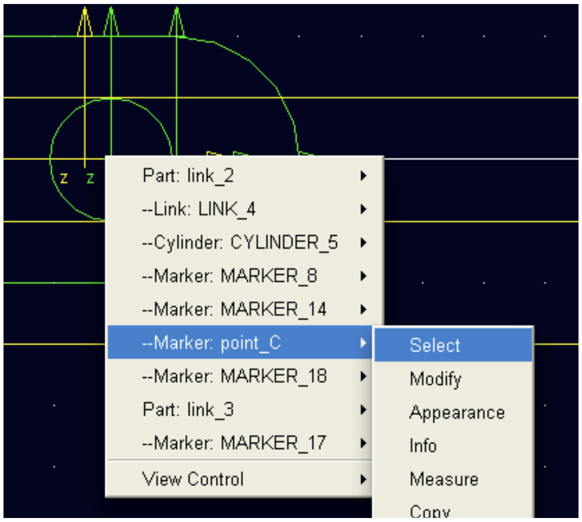
Now we need to identify "point C" and the "c.m. marker of link 3" on the ADAMS/View model. Consider the "zoomed in" area near the pin (left figure below):



The crank (link 2) is green, while the follower (slotted link 3) is yellow. The circular outline of the pin is clear in the figure.

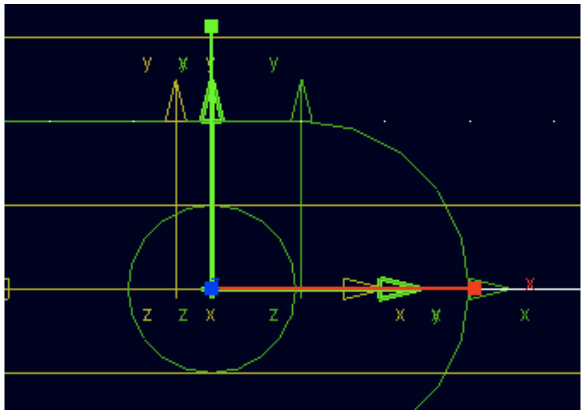
8.2.3 Markers of Interest

There are two markers of interest: (1) the marker at the center of the pin (point C), and a marker fixed to the slotted link (the marker shown in the sketch above). To "examine" a marker, right-click near the origin of the marker, select the marker, and select "Select." I renamed the marker at the center of the pin "point_C", so we have



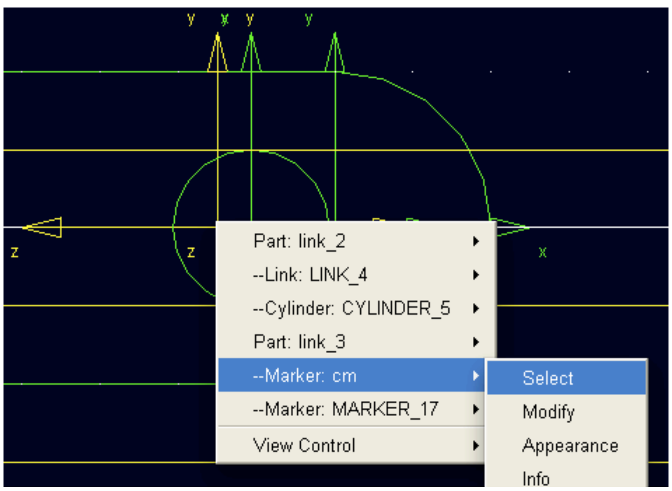
This will "highlight" the marker in ADAMS/View so you can see it (xyz axes are RGB respectively). See next page.

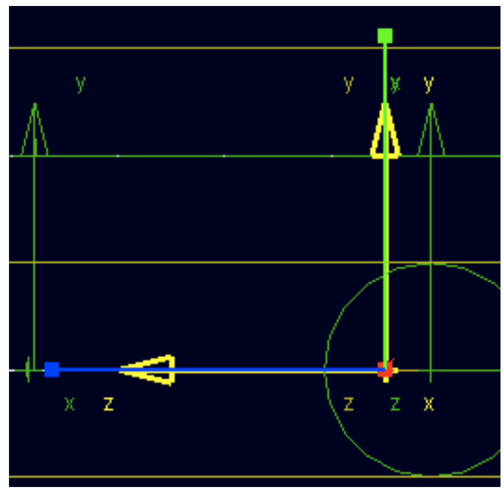
I don't know why, but seems like you have to zoom the ADAMS/View window in or out to make the marker show up. Once you do, you'll see it as below (remember, xyz=RGB):



So this is the marker at "point C." Click anywhere in the ADAMS/View window to deselect the marker.

We now need to find a marker attached to link 3 (the slotted link); the link 3 center-of-mass (c.m.) marker is a convenient one (it's actually just to the left of point C). In the same manner, select it (left figure below):





The highlighted link 3 c.m. marker is shown in the right figure above. So the link 3 c.m. marker has its y axis perpendicular to the slot (like my drawing on the previous page). That's the direction in which point C can have *NO* motion.

Now we have the two necessary markers located. It is *VERY IMPORTANT* to identify these markers.

8.2.4 Creation of the GCON

Remember, the following statement describes the desired constraint:

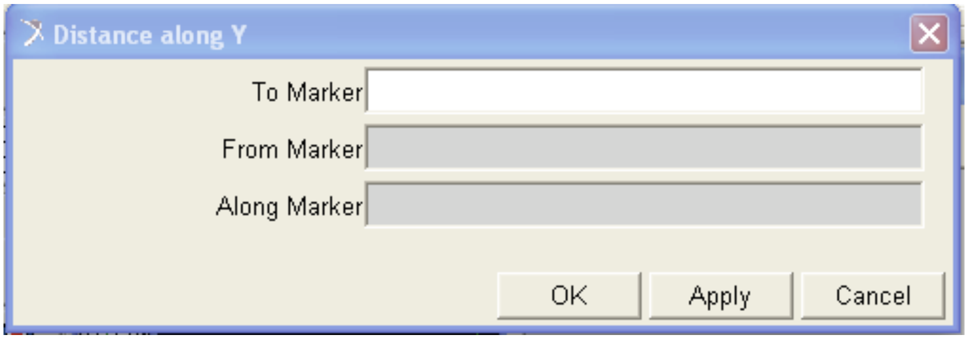
The displacement of point C in the y direction of the c.m. marker MUST BE ZERO

The procedure will be the following:

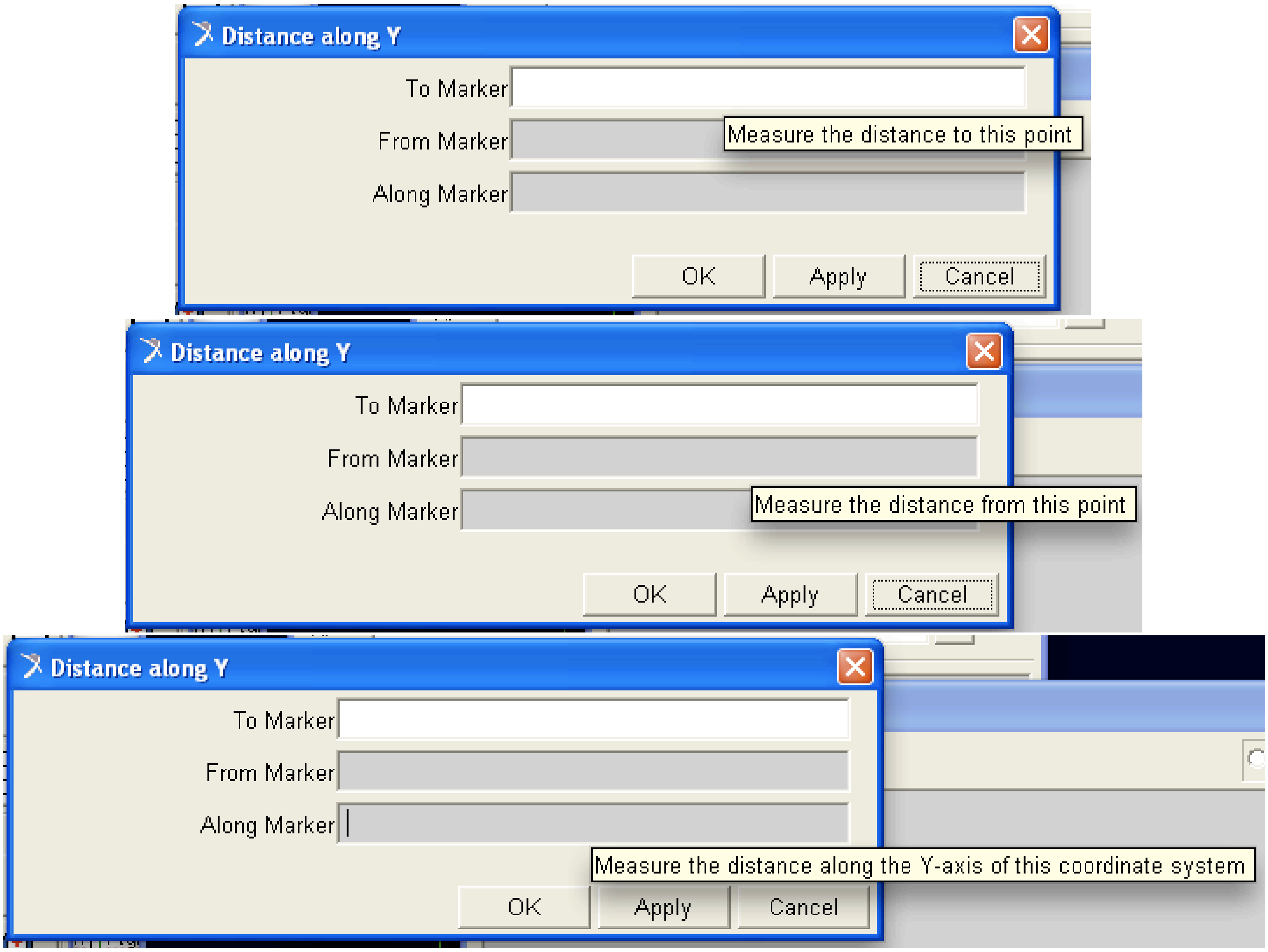
1. Select the GCON (f(q)=0) "joint" from the Main Toolbox palette
2. From the "Create General Constraint" dialog box, select the "..." button to activate the Function Builder
3. Select "Displacement" from among the Functions
4. Select "Distance along Y"
5. At the bottom of the Function Builder window, the format "DY(To_Marker, From_Marker, Along_Marker" appears

6. Click the “Assist” button to well, assist...

You will get the following box:



Selection of Markers. If you "hover" the cursor over the three panes of the box above, descriptions of the markers will appear:



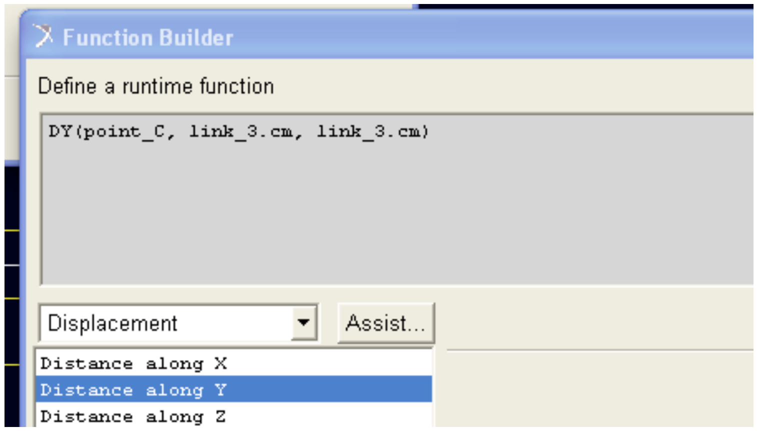
Recall the statement of our constraint:

The displacement of point C in the y direction of the c.m. marker MUST BE ZERO

From the "hints" of the previous page, we can reason that:

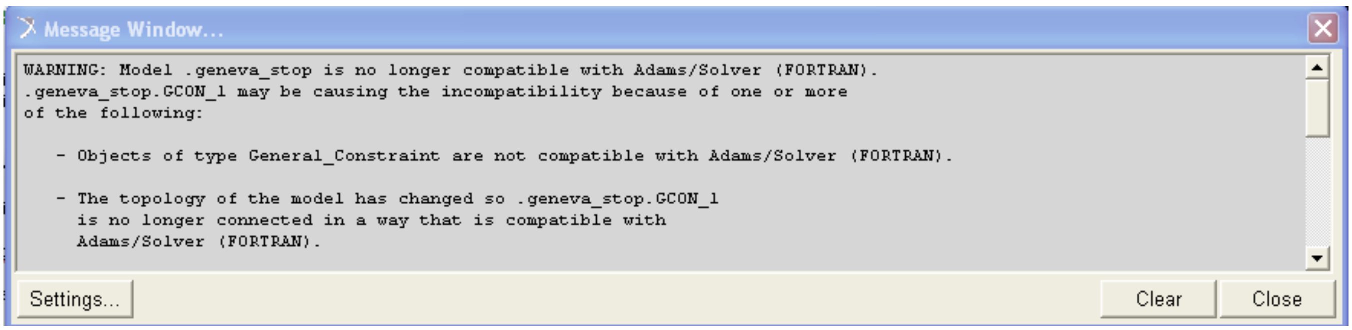
1. *Measure the distance to this point:* This will be the marker at point C on link 2, which I renamed "`point_C`"
2. *Measure the distance from this point:* This will be the marker at the c.m. of link 3
3. *Measure the distance along the Y-axis of this coordinate system:* To guarantee no displacement in the y direction of the c.m. marker, this marker will be the same as the previous: the c.m. marker of link 3.

I find it easiest to "`Browse...`" for the markers, so right-click in each pane and browse for the markers above. When you finish, the "`Function Builder`" window will look like this (the upper left portion):



If there is a “zero” at the end of the string (a zero always occupies the window initially) be sure to delete it.

Click “`OK`” twice (both windows) and you’re DONE!! **OOPS!** This error may pop up...



The *STUPID* ADAMS/Solver always seems to be “stuck in Fortran” which isn’t compatible with GCONs. You have to change the ADAMS/Solver to “`C++`” mode.

In the ADAMS/View window, select “`Settings/Solver/Executable ...`” and you will see this window:

Click the “`C++`” button and *now* you’re done. Resimulate the model and the pin should stay in the slot.

9 Disk Cam with Reciprocating Flat-Face Follower

This is most complicated example we've done. That is, it requires creating several ADAMS/View data structures that you haven't seen before. The process may seem confusing, but I will try to explain each step as clearly as possible. We will use the cam example that I did in class; the one with a SHM rise and return with $\beta_1 = 120°$ and $\beta_2 = 240°$.

9.1 Overview

The steps in building the model are shown below:

1. Create file of (x, y) points on cam profile using MATLAB.
2. Import this file into ADAMS/View and create "splines."
3. Create an ADAMS/View "matrix" from these splines.
4. Create an ADAMS/View "curve" from the matrix.
5. Create ADAMS/View "geometry" from the curve.
6. Assign mass and c.m. to the cam, add motion, and create the follower.
7. Create the Curve-to-Edge constraint between cam and follower.

Each of these steps will dealt with in sequence.

9.2 Create file of (x, y, z) points on cam profile using MATLAB

Recall the script file `cam.m` that I wrote to design the cam example in class. That script plotted y, y', y'' for both the rise and return, computed the required followed radius (0.75 in), found the minimum base circle radius (0.125 inc) but let the user enter a radius (I used $R_o = 0.5$ in), and finally generated θ (both RAD and DEG), follower lift y, and follower first kinematic coefficient y' for the entire motion.

The next step is to use (θ, y, y') to generate the cam profile.

9.2.1 Use of MATLAB function `camprofile.m`

I wrote this simple function, which is:

```
>> help camprofile
  FUNCTION [xc,yc] = camprofile(Ro,y,yp,theta,dir_flag)

  This function finds the cam profile for a disk cam with an reciprocating
  flat-face follower.  It accepts the base circle radius Ro, vectors of y
  and yp (all in displacement units), and cam angle theta (RAD).  These
  vectors should all correspond to a full cycle of cam rotation from 0 to
  2*PI.  Obviously the smaller the stepsize the more accurate and smoother
  the final plot of cam profile.

  Vectors xc and yc containing the coordinates of the cam profile are
  returned.  The cam profile points are generated so the follower axis is
  vertical (i.e. pi/2 has been added to follower angle).

  NOTE: The "dir" flag is either 'CW' or 'CCW' to denote the direction of
  cam rotation.  The algorithm is written for 'CW' rotation, but if the
  direction is 'CCW' all that must be done is to change the sign of the x
  coordinates of the cam profile.
```

Note that *YOUR* version of `camprofile.m` does *NOT* have the "direction flag." I'm still waiting for someone to ask me what changes to make to enable CCW rotation (`camprofile.m` was written for CW rotation).

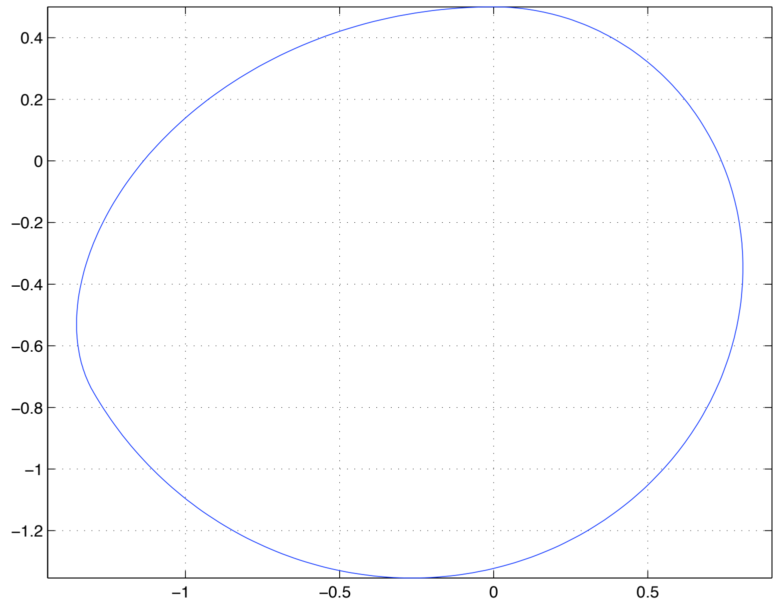
The parameters needed by `camprofile.m` are: `Ro,y,yp,theta,dir_flag`, all of which we have. So to get the cam profile, we do the following:

```
>> [xc,yc] = camprofile(Ro,y,yp,thetar,'CW');
```

where I've used `thetar` which is θ (RAD). One can now plot the cam profile by doing the following:

```
>> plot(xc,yc); axis equal; grid;
```

which plots, sets the axis scaling equal (to get the real "shape" of the cam), and draws a grid. We get the figure below:



The base circle radius of 0.50 in was larger than the minimum of 0.125, so the profile is quite smooth.

9.2.2 Saving (x, y) MATLAB Cam Profile Data in ADAMS/View format

We need to save the cam profile data into a file with four columns: (1) `theta (DEG)`, (2) `xc`, (3) `yc`, (4) `zc`. Even though `zc` is zero, we need it. Also, θ is just used as in index, so it can be in `DEG`.

So make up a dummy `zc` of the same length:

```
>> N = length(xc);   % Find length of coordinate vector xc
>> zc = zeros(N,1);  % Create a zc of zeros of same length
```

Next create a matrix of these four columns:

```
CAM = [thetad xc yc zc];
```

Finally create a textfile called `‘‘cam.dat’’` with this matrix.

```
>> save cam.dat CAM -ascii
```

Don't forget the "`-ascii`" flag; otherwise you will create a MATLAB binary file. It is the textfile "`cam.dat`" which ADAMS/View will import to create the cam.

That is the end of this section.

9.3 Import this file into ADAMS/View and create "splines."

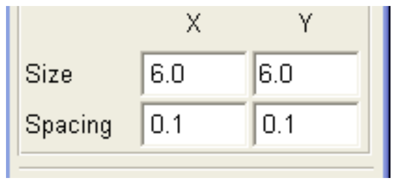
9.3.1 Launching ADAMS/View

Now we have to launch ADAMS/View.

Select your directory, "Create a new model", "No Gravity", and "IPS" units (appropriate for this example). Call the model something like "cam_example"...

9.3.2 Setting up the Working Grid

Under `Settings/Working Grid` select `Size` of 6 and `Spacing` of 0.1 (these are in units of inches):



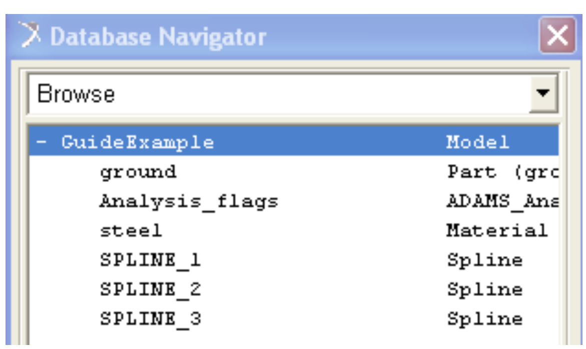
Then zoom (press/release "Z" and move cursor up and down) until you're satisfied.

9.3.3 Importing the Cam Data to Splines

You should have the file "`cam.dat`" in the ADAMS/View directory you've selected.

- Select "File/Import" from the ADAMS/View menu
- Set "File Type" to `Test Data (*.)`
- Select "Create Splines"
- Right-click "File to Read", "Browse," and select `cam`
- Set "Independent Column Index" to 1
- Uncheck "Names in File"
- Click OK.

To verify the splines have really been created, from the ADAMS/View window select `Tools/Database Navigator`, make sure the upper pane is set to `Browse`, then double-click on the model name:



Note that three splines were created: each spline is of the form $[\theta(t)\ \ xc(t)]$.

If you wish, you can RENAME the splines by selecting "`Rename`" in the `Database Navigator`, double-clicking the model name, selecting the SPLINEs, entering new names, and clicking `Apply`.

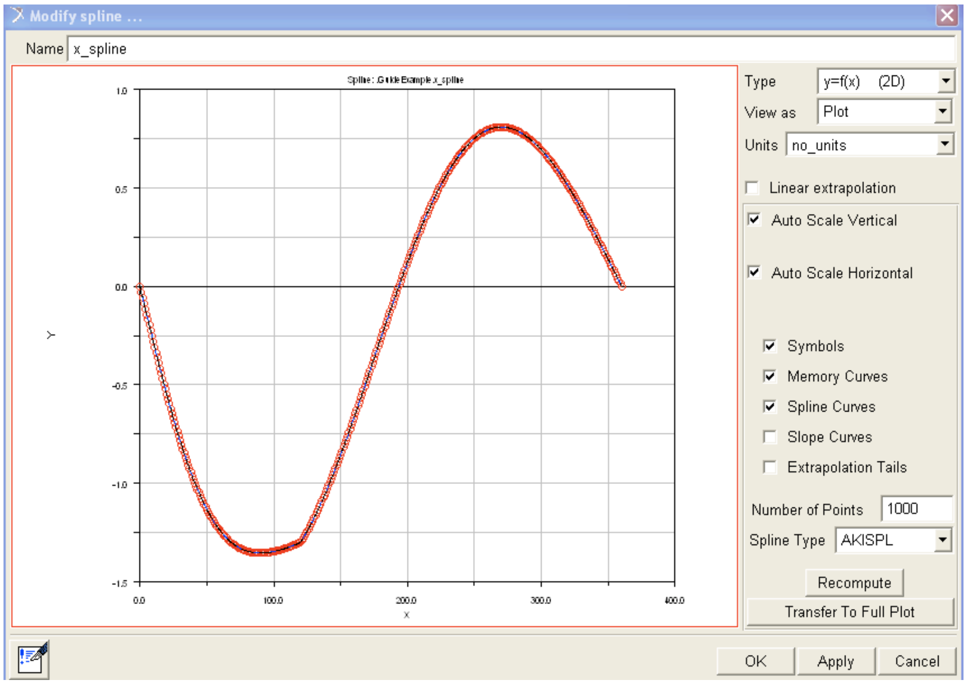
I named my splines `x_spline, y_spline, z_spline`.

9.3.4 Modifying (and Examining) the Splines

You need to modify (and you can examine) the splines you just created. Probably the easiest way is by selecting `Edit/Modify...` (or CTRL-E): this brings up the `Database Navigator` window again. Double-click the model, the double-click one of the splines. You will get a "`Modify Spline`" window, which will present the data in a matrix form.

- Under `Units`, select `length`...this *MUST* be done!

Near the upper right is a "`View as`" pane which is set to `Tabular Data`. Change it to `Plot` to see the plot shown below. This is `xc` *vs* θ (`DEG`). The `yc` plot would be similar, while the `zc` plot would be zero.



Click `OK` to close the `Modify Spline` window (and save the units) when you're done.

9.4 Create an ADAMS/View "matrix" from these Splines

From ADAMS/View, got to `Build/Data Elements/Matrix/New...`

- Change the name from `MATRIX_1` to something reasonable (I'll use `cam_matrix`)
- `Units` should be set to `length`
- Select `Full Matrix`
- Select `Enter input ordered by columns`
- Select `User Entered Numbers`
- Set `Row Count` to the number of rows (361 in my case)
- Set `Column Count` to 3

Now Right-click the `Values` pane and select `Parameterize/Expression Builder...`

- Under `Getting Object Data` select `All Objects`
- In the grey pane to the right, `Right-click/All/Browse`...the `Database Navigator` will appear...double-click the Model name, then double-click the "`x_spline`" term; the term "`x_spline`" will appear next to the `All Objects` field.

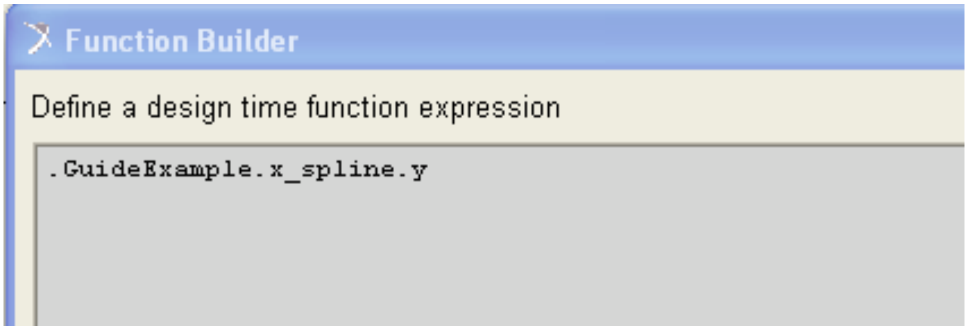
Now is when you have to really pay attention...on to the next page.

- Just under the grey box there is a button `Get Data Owned By Object`...click it. This brings up a `Selection` window containing *ALL* the data "owned" by that spline.

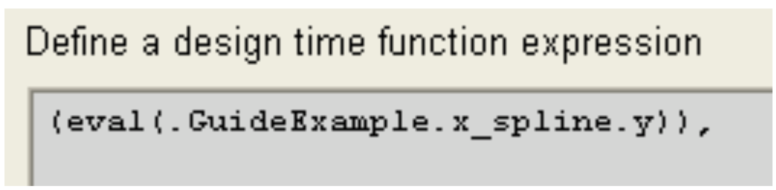
  The spline is a 2-column entity, and you want to select the *SECOND* column, which is the y column. This contains the coordinate data.

- Scroll the `Selection` window down until you see the `x y z` rows, then select the `y` row, and click `OK` (or double-click the `y` row)

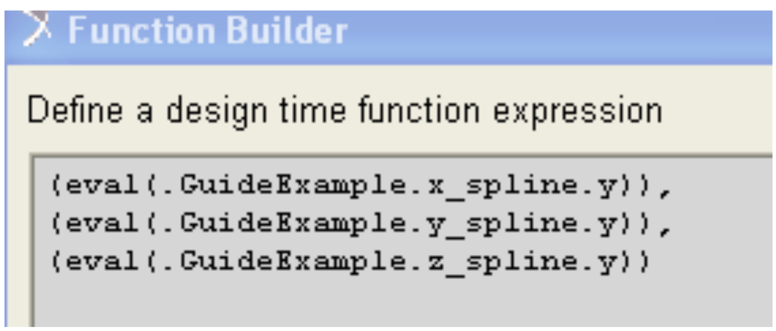
The `Function Builder` window will now contain the string:



We need to add the cam profile y and z data. But first edit this string by typing "`eval`" before the string, and adding bounding parentheses (and a comma at the end) as shown:



Repeat the above steps for the `y_spline` and `z_spline`—or, you can simply copy and paste into the `Function Builder` window to get the entry below:



From the `Function Builder`, click `OK`.

From the `Create Matrix` window, click `OK`.

If you wish, open the `Database Navigator`, `Browse`, open the `ModelName`, and you will see your matrix is now present.

The above steps are the most complicated in the entire process. If you get this far, congratulations! The rest is EASIER!

9.5 Create an ADAMS/View "curve" from the matrix

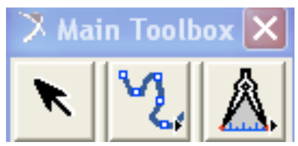
From ADAMS/View, select `Build/Data Elements/Curve/New...`a `Data Element Create Curve` window will open.

- Curve Name: Your choice...I'll use `cam_curve`
- Closed: Set this to `yes`
- Select `Define Using Matrix`
- Matrix Name: Right-click/ADAMS/View `Matrix/Browse/your matrix`
- Interpolation Order: 4 (cubic b-spline)
- Click `OK`

We're almost there...

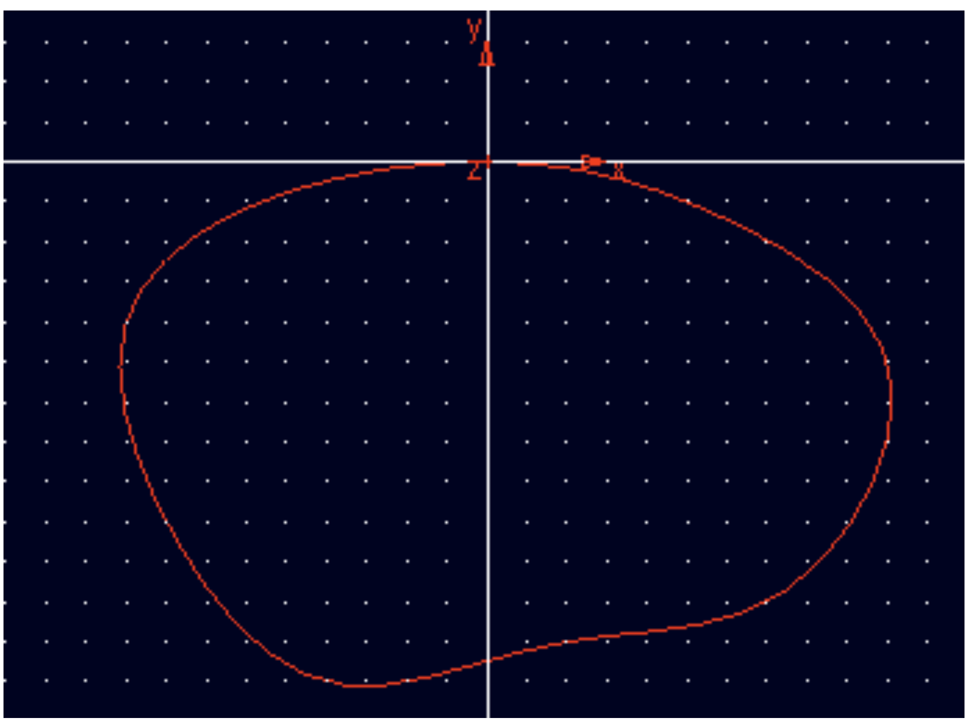
9.6 Create ADAMS/View "Geometry" from the Curve

In ADAMS/View, "geometry" is what gives "shape" to a part. So here—when we create the geometry, we really create the part. You will finally see something in the ADAMS/View window.

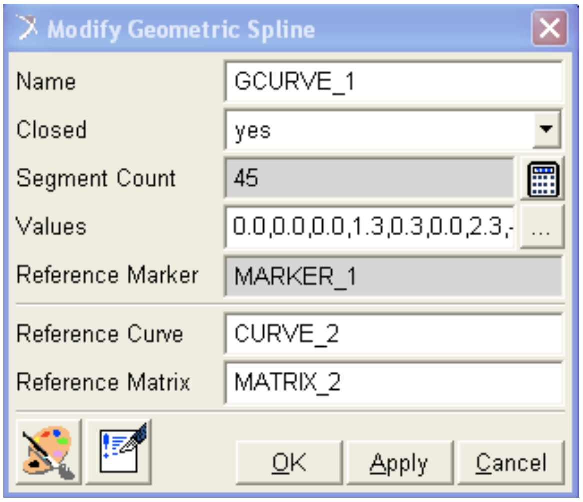


- From the `Main Toolbox`, right-slick the "bodies" button and select the "spline" tool, as shown at center above
- Select `New Curve`, `Closed`, and Create by Picking `Points`
- Left-click to place the points (pick them in an oval...at least 8 points), then right-click to finish. **IMPORTANT: Pick the first point on the "crosshairs"; and (0,0). This will put the cam axis at (0,0)**
- The curve will appear, along with an error message about having `NO MASS`—close the error window

Your ADAMS/Viewscreen will look something like this; the marker was where I first clicked (0,0).



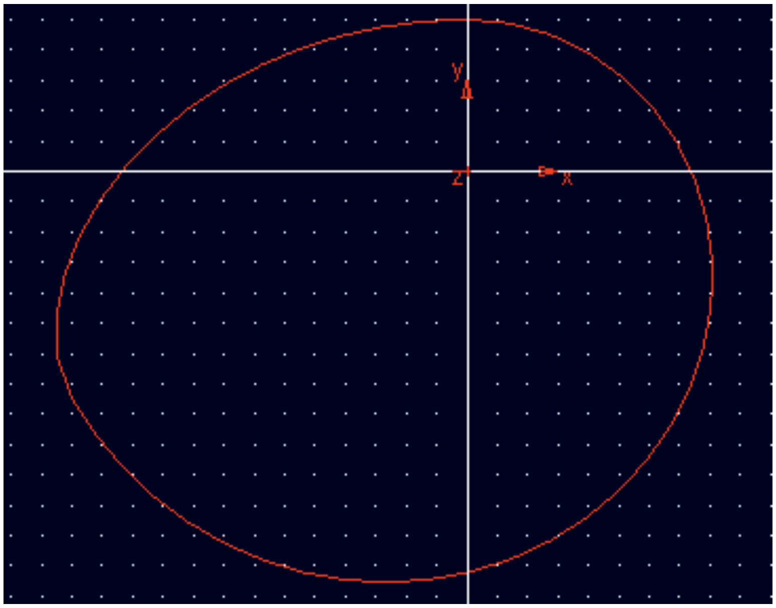
Right-click on the curve, select `BSpline:GCURVE_1/Modify`...you will get this window:



Change the following:

- Segment Count: Make this much larger (2000 is fine, for example)
- Reference Curve: Right-click /ADAMS/View/ Curve/ and browse for your cam curve (or use "Guesses")...when selected, the Reference Matrix should change to your matrix; the numbers in the `Values` field will also change...
- Click `OK`

Now the cam profile *SHOULD* show up. Sometimes it doesn't—try double-clicking the "arrow" in the Main Toolbox, try right-clicking on the cam in the ADAMS/View window, then "Select"...keep trying *something*...when it does, you'll have something like below:



Note that at the grid spacing of 0.1 in, the distance from (0,0) to the top of the cam is exactly 0.5 in—this is minimum radius $R_o = 0.5$ in, which we specified in the first place.

9.7 Assign Mass and c.m. to the Cam, add Motion, and Create the Follower

When the cam is created like this, it is a thin disk with zero volume, and hence zero mass. With zero mass, it cannot be simulated.

There must be a way to give the cam some "depth," but I don't know how to do it. So we'll just give the cam some mass the direct way.

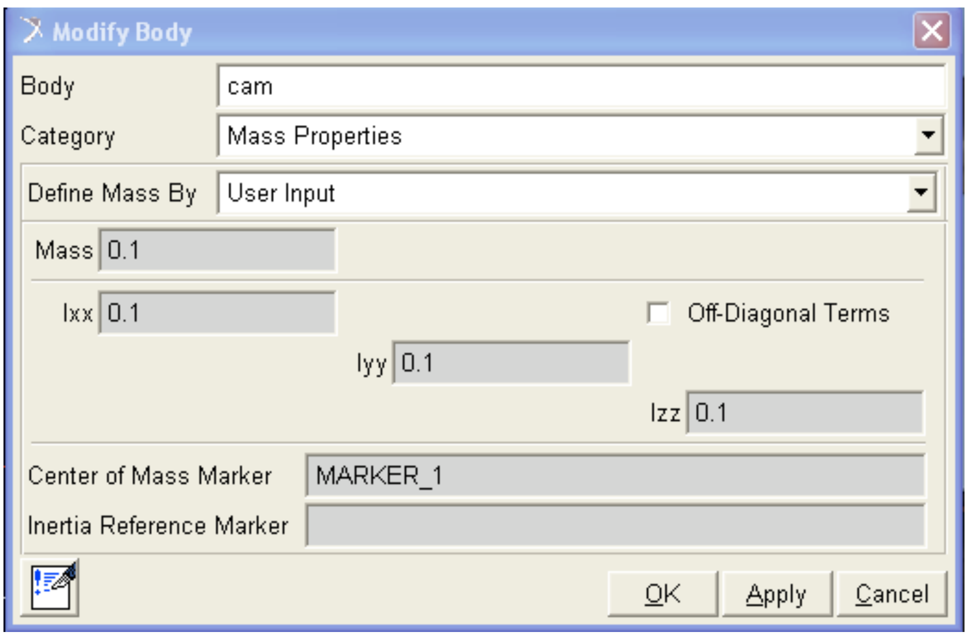
9.7.1 Specifying the Cam Mass and c.m.

First—in the usual way—rename the cam something like "`cam`."

To specify the mass, moments of inertia, and center of mass, do the following:

- Right-click on the cam, and select `Modify`
- The "Category" is probably already set to `Mass Properties`
- Set "Define Mass By" to `User Input`. You will see an entry for Mass, and entries for the three Principal Moments of Inertia
- Enter values for these. I set them all to 0.1 (lbm) and (lbm-in**2). These numbers will have no effect on the results we are interested in for this simulation.
- Set the "Center of Mass Marker" to the marker at the cam axis (it was `MARKER_1` in my model)

After this the `Modify Body` window looks like:



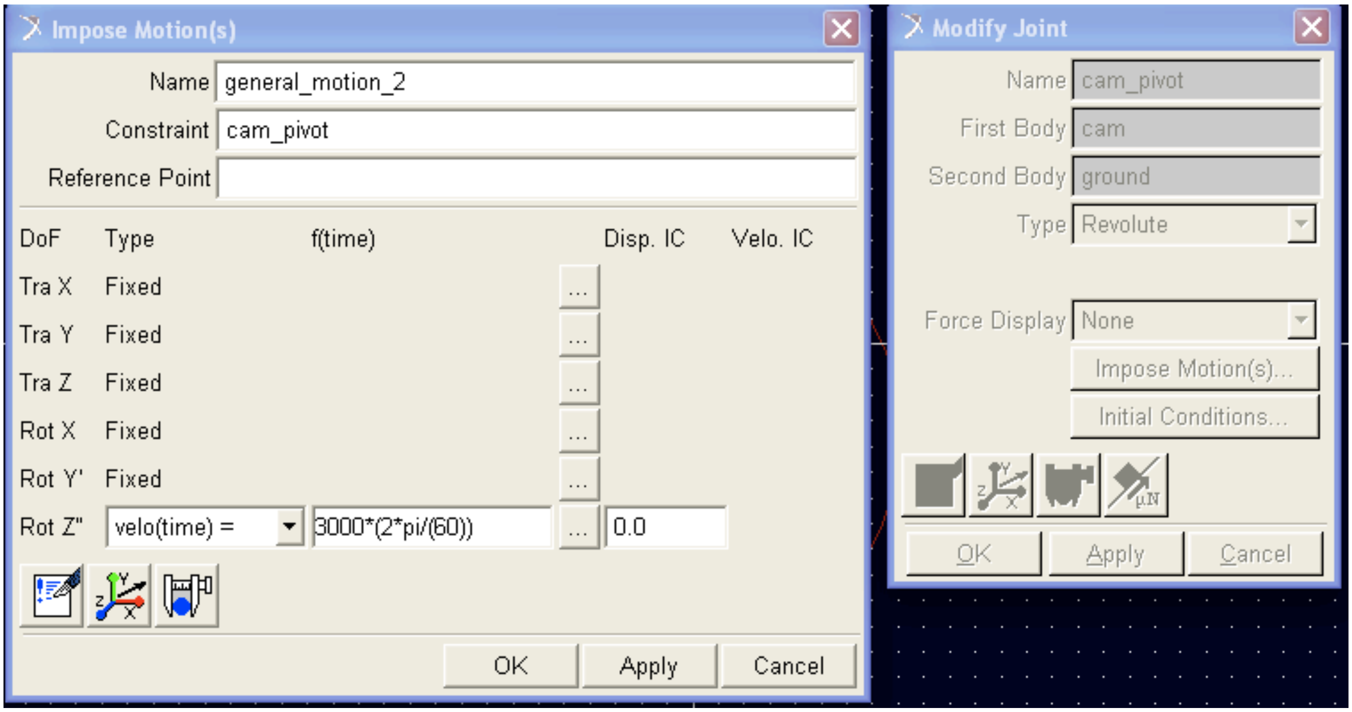
Click `OK` to finish.

9.7.2 Adding Cam Motion

I don't think we specified at angular velocity for this cam, so let's try 3,000 rpm. That should make the follower move right along! In the "motion" window in ADAMS/View, the units of angular velocity are `RAD/SEC`. So we will enter $3000 * ((2\pi)/60))$.

Oops, first we have to put a revolute joint between the cam and ground. So do that, and call it `cam_pivot` or something similar.

Then right-click the joint, select `Modify`, then `Impose Motion(s)`...something like below:



Click `OK` to both windows to finish.

Simulation. You may want to run a simulation of the cam spinning around just to verify that it spins. Set the simulation duration to 60/3000 (the number of seconds to rotate one revolution), the time step (0.0002) or number of points (90 or 180) accordingly, and try it.

If the cam goes CCW, change the sign of the angular velocity.

9.7.3 Creating the Follower

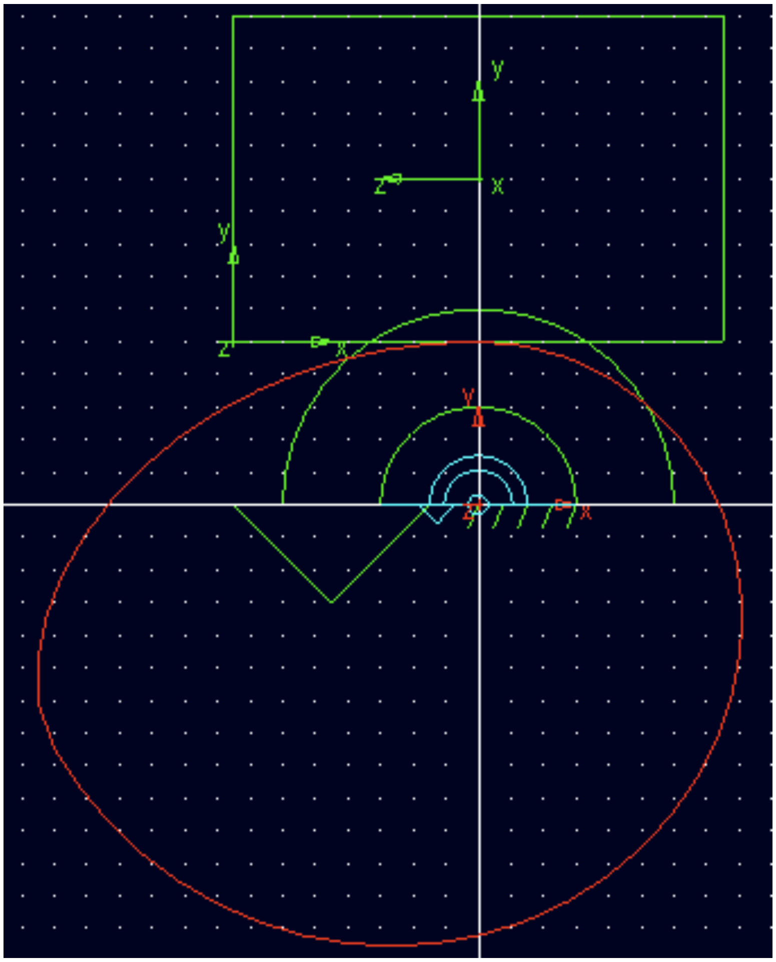
The minimum follower radius is 0.75 in, so the overall width of the follower face will be 1.50 in. Even though the follower will be usually be cylindrical, here we will use a "block" for the follower.

In the Main Toolbox, select the "block" body, with a Length = 1.5, Height = 1 in, and Depth = 0.5 in (Heigh and Depth not critical; those are just what I used).

Place the follower on top of the cam, then use the "linear placement" tool to move it until it is perfectly centered (some multiple of 0.05 in).

Oh yeah, probably should rename the "block" something like "follower"...

When you finish, you'll have something that looks like this:

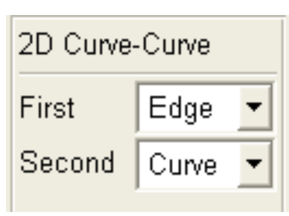


9.8 Create the Curve-to-Edge Constraint Between Cam and Follower

This constraint is what prevents the cam from penetrating the follower. This constraint will not allow the follower to "float" away from the cam; one must examine force to determine that.

This constraint *IS* present in the Main Toolbox, and is the "cam" style figure:

We are going to use an "Edge-Curve" constraint here, so in the constraint window, use:



When you move the cursor to the ADAMS/View window, you are prompted to:

- Select the `edge` on the first part—click on the bottom edge of the follower

You are then prompted to

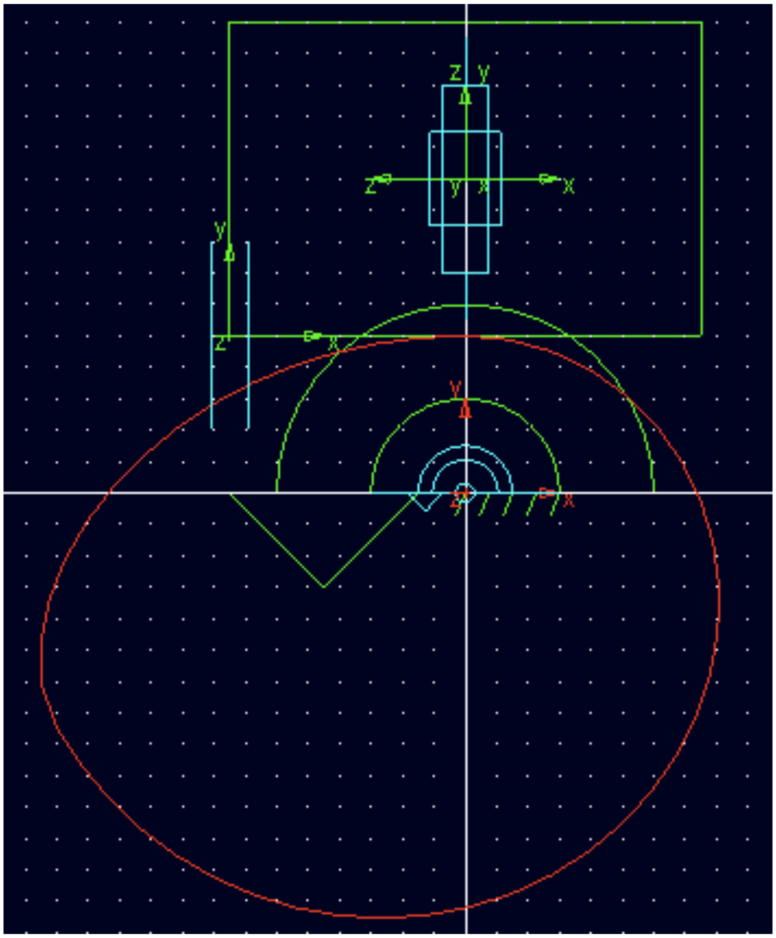
- Select the `cuve` on the second part—click on the cam curve

Probably two vertical lines will appear somewhere near the cam follower: this is the constraint.

9.8.1 Create the Translational Joint Between Follower and Ground

Finally, don't forget the `Translational Joint` between the follower and ground. This is done in the standard way, using a vertical axis for the joint.

A screenshot of the complete cam/follower mechanism is below:



9.9 Mechanism Simulation

Now you can simulate the cam-follower mechanism: the follower should move vertically as the cam rotates beneath it.

Examine the contact point between cam and follower; it should move right to the edge of the follower (at the greatest follower velocity). The follower must be wide enough to accommodate that point.

9.10 High-Speed Motion and the Cam/Follower Force

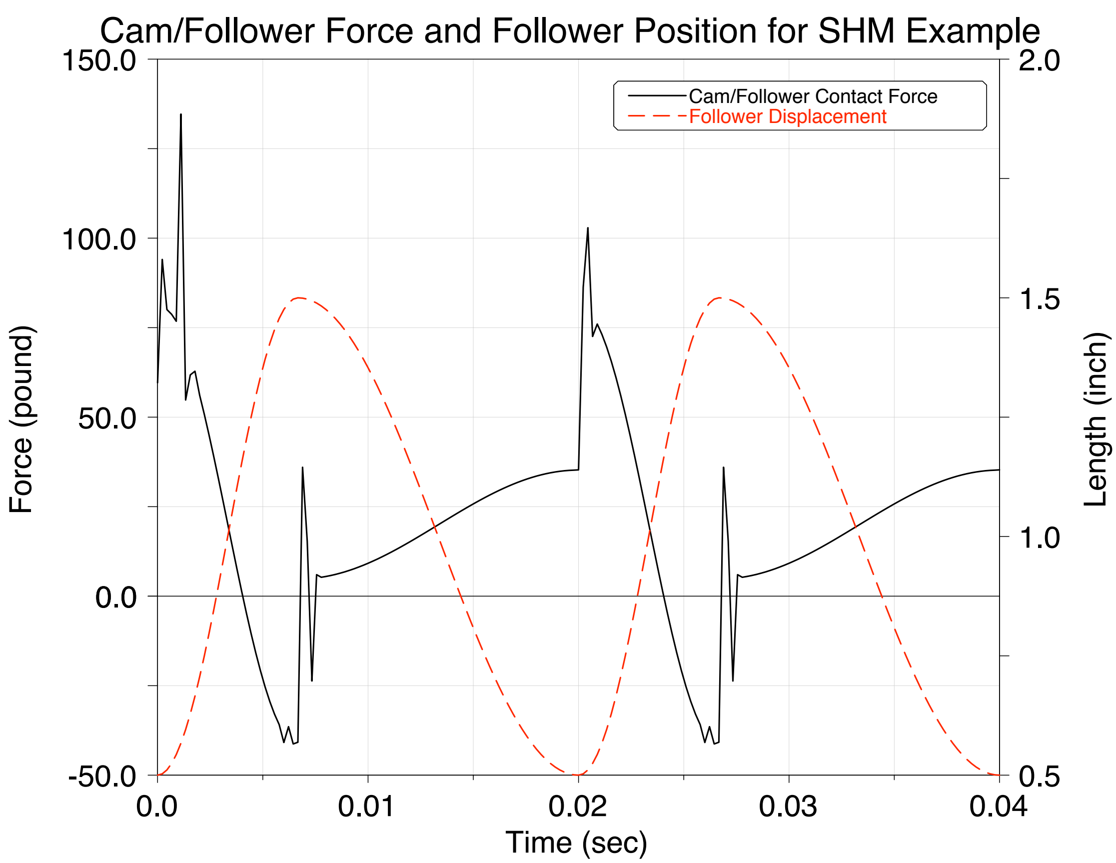
This cam is rotating at 3,000 rpm, which qualifies as "high speed." I specified the duration of the simulation to allow two revolutions.

9.10.1 Plotting the Cam/Follower Force

After simulation, go to the ADAMS/Postprocessor (click the "plot" button).

- Make sure "Source" is `Objects`
- Under "Filter" select `Constraint`
- Under "Object" select the `Curve-Curve constraint`, which was `CVCV_1` in my case
- Under "Characteristic" select `Element_Force`
- Under "Component" select `Y`
- Click `Add Curves`

I obtained the plot below (for two revolutions, remember):



Acceleration Discontinuities. Remember the **DISCONTINUITIES** in the acceleration profile? The infinite jerk? Those occurred at 0° and 120°. That would be FOUR in two rotations. Look at the "force spikes" in the plot above...I would conclude that *Simple Harmonic Motion* is a poor choice for high-speed cam use.

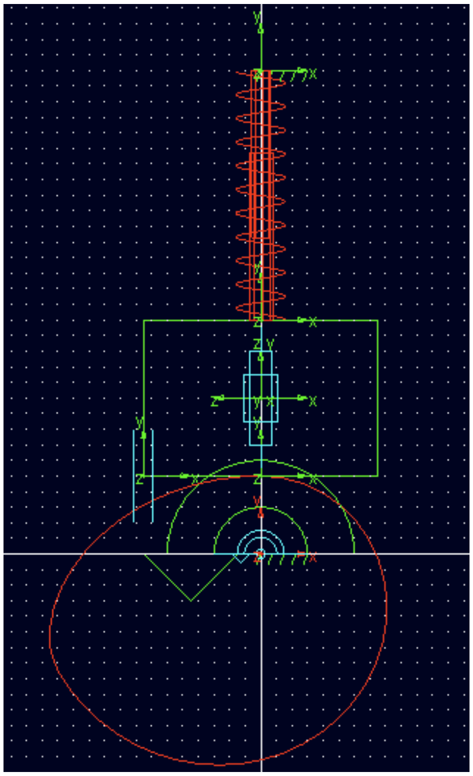
Follower Float. The force above goes negative; this cannot actually happen. What this indicates is the cam is "pulling" the follower back down. So we need a *SPRING* to maintain contact between cam and follower. This is very simple to add to the ADAMS/View model.

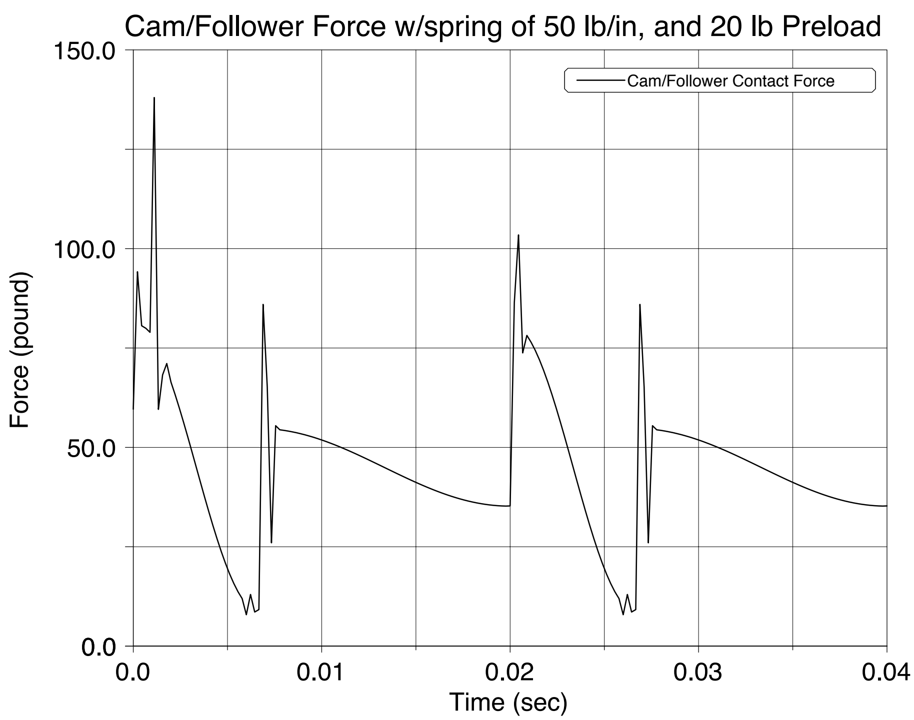
9.11 Adding a "Valve Spring"

Select from the "spring" icon in the Mail Toolbox and "draw" a spring from the top of the follower to some point above. A marker attached to ground will be placed at the top of the spring.

Note that if you look at a "side" view (type r or l from within ADAMS/View) the top of the spring is probably back "into" to screen. You can modify the upper marker to move it out.

Since the follower "pulling force" was nearly 50 lb when the displacement was 1.0 inch, I used a spring rate of $k = 50$ lb/in. This was close. I then increased the "preload" to 20 lb, and that did it. Screenshot and plot below.





10 Mass Distribution in ADAMS/View

When you create a part in ADAMS/View, the `material` of the part is (I think) set to `steel`, and the mass and mass moment of inertia is automatically calculated based on the density of the material (steel), and the volume and shape of the part.

A marker at the mass center (the "`c.m.`" marker is created for the part, and the inertia properties are expressed relative to this marker.

Typically this is what you want—you can proceed with the design and subsequent simulation. You may wish to change the material, and ADAMS/View has a variety of materials you can select.

However, sometimes—perhaps when you need to match the parameters given in a textbook homework problem—you may need to modify the mass and inertia parameters.

10.1 Modification of Mass Parameters

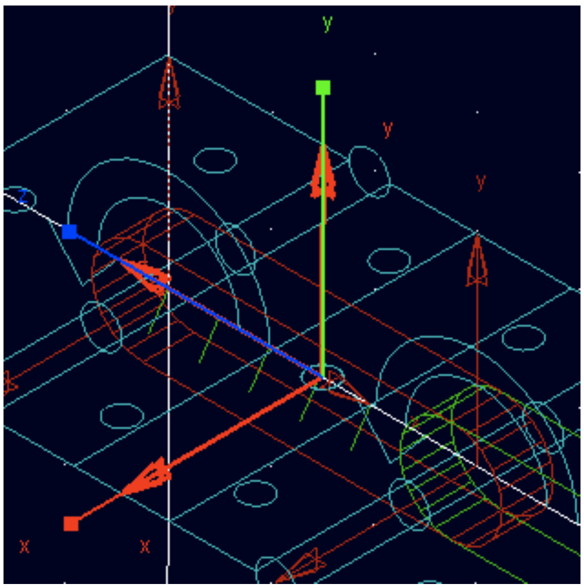
Modification of the mass and inertia properties of a part is simply a matter of (1) right-clicking the part, (2) selecting `Modify`, and (3) typing in the desired values.

However, one must be careful to select the correct `AXIS` in entering the moments of inertia.

10.1.1 Which Axis do I Use?

The Moment of Inertia of a part is *ALWAYS* expressed relative to the `c.m.` marker for the part. So you need to see which axis of this `c.m.` marker is the one you want.

All of our mechanisms are planar, so the axis you want is the axis that is perpendicular to the plane of motion. Consider the crank (link 2) of Problem 6, which is shown in the closeup below. It's the `RED` part; a little difficult to see because of the `hinge` joint.



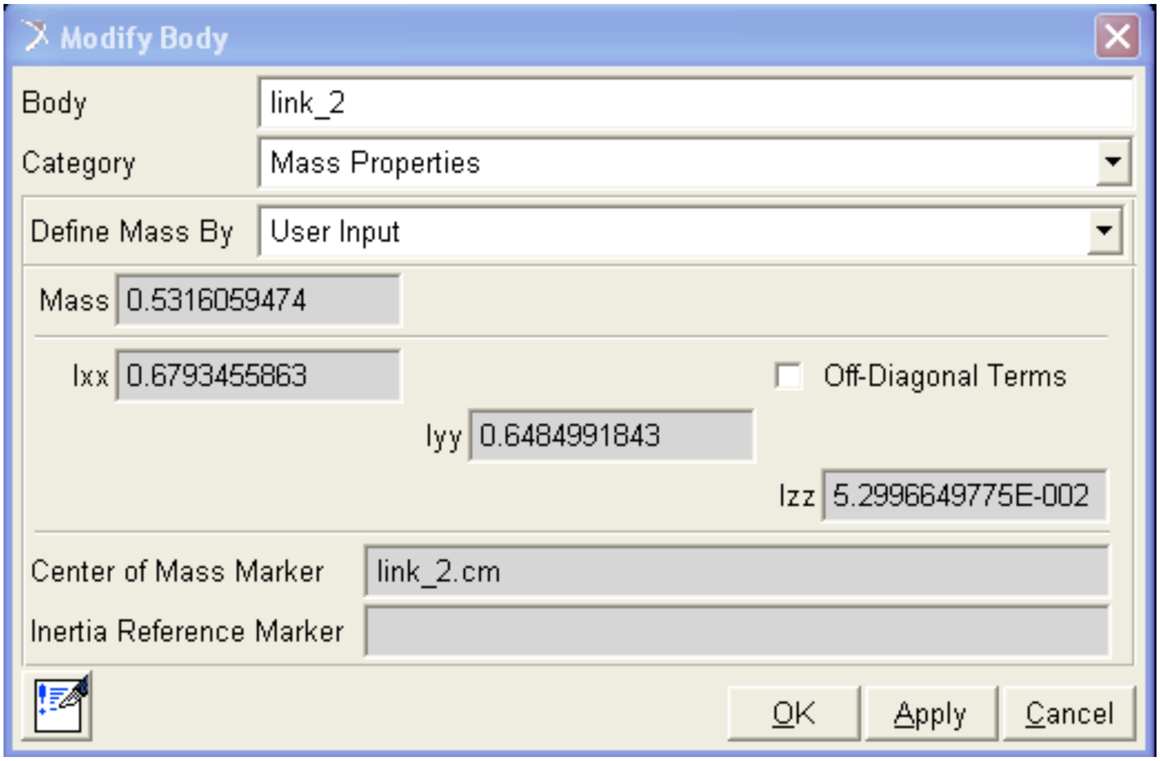
There will be a marker at the center of mass (the `c.m.` marker); it you right-click on it and `select` it, it will become highlighted and you can see the axes. The color system in ADAMS/View is always:

X axis: RED $\quad$ Y axis: GREEN $\quad$ Z axis: BLUE

Therefore from the figure above the axis perpendicular to the plane of motion is the RED (X) axis. Now we know to modify the I_{xx} element in the inertia matrix.

10.1.2 The "Modify Body" Window

When you right-click the part and select `Modify`, you will get the following window:



Make sure the `Category` value is set to `Mass Properties`, and the `Define Mass By` value is set to `User Input`—then you can simply type in the desired values.

Make sure to get the correct units, as I discussed in the Chapter 15 HW Hints.

Although there is no need—in this case—to change the I_{yy} or I_{zz} fields, when you perform the simulation you may get a warning, something like:

> $I_{yy} + I_{zz}$ must be larger than I_{xx}!!

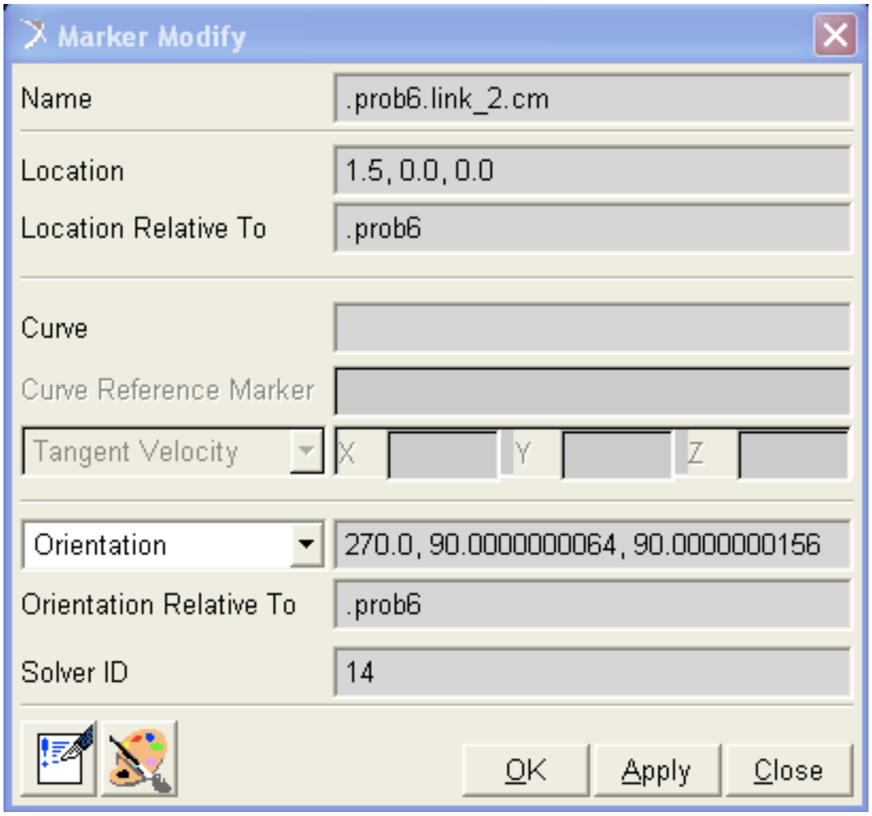
If you want to fix this, simply enter an I_{yy} value the same as I_{xx}.

Of course, if you're designing a *REAL* part, ADAMS/View just computes the inertia automatically from the material and the geometry and everthing is good. But with these stupid textbook HW problems...

10.2 Placing the c.m. Marker

There is one more thing you may need to do: reposition the `c.m.` marker to correspond to the textbook location.

To do this, right-click the `c.m.` marker, then select `Modify`. You will see the window shown on the next page.



The location of the origin of the `c.m.` marker is given in the `Location` item, with *xyz* coordinates. To change the location of the `c.m.` marker simply type in the new location in that field.

That's it!