ME 24-688 – Week 11

Introduction to Dynamic Simulation

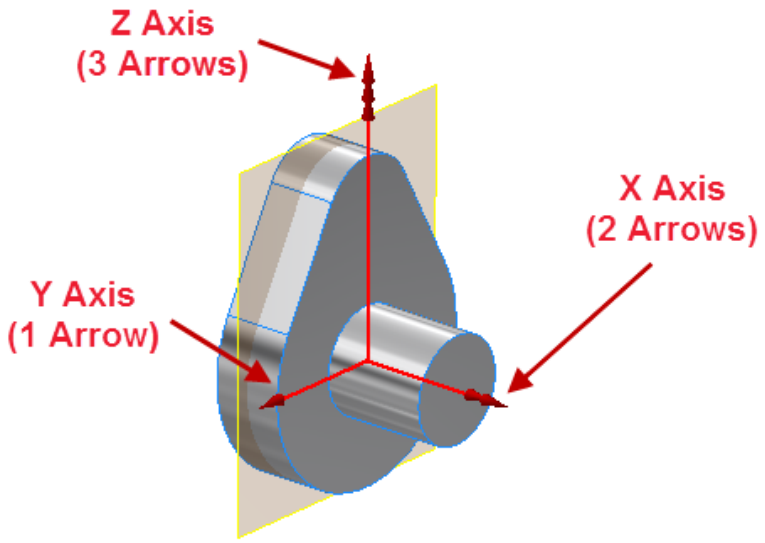
Introduction to Dynamic Simulation

The purpose of this introduction to dynamic simulation project is to explorer the dynamic simulation environment of Autodesk Inventor Professional. This environment allows you to perform rigid body dynamic simulations of designs. This is type of simulation is used on assemblies to simulation time varying behavior of a system.

To assist the learning of the Dynamic Simulation environment of Autodesk Inventor several key points are outlined below and can be used for reference throughout the projects.

Joint Coordinate System

Each joint has coordinate systems that define the joint. Visually they are different from the standard Autodesk Inventor coordinate system icon. The image below shows the axis location for the symbol.

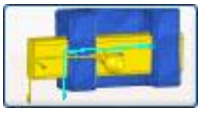
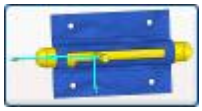
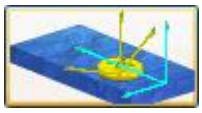
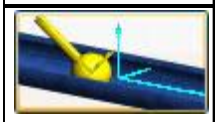
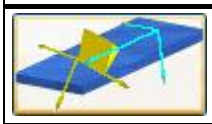
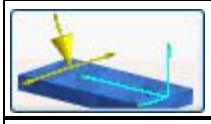
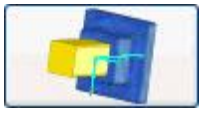


There are several different joint types within Autodesk Inventor Professional. Each of the joint types have different behaviors. The chart below shows each of the available joint types and some general information about them.

ME 24-688 – Week 11

Introduction to Dynamic Simulation

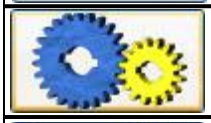
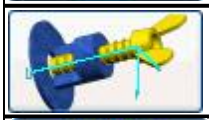
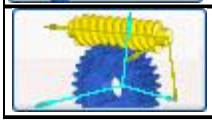
Standard Joints

Dynamic Simulation Joints			D.O.F.
	Revolution	No Translation Rotation around the Z Axis	1
	Prismatic	Translation along the Z Axis No Rotation	1
	Cylindrical	Translation along the Z Axis Rotation around the Z Axis	2
	Spherical	No Translation Rotation along all 3 Axis	3
	Planar	Translation along the X and Z Axis Rotation about the Y Axis	3
	Point - Line	Translation along the Z axis Rotation around all 3 Axis	4
	Line - Plane	Translation along the X and Z Axis Rotation about the Y Axis	3
	Point - Plane	Translation along the X and Z Axis Rotation around all 3 Axis	5
	Spatial	Translation along all 3 Axis Rotation about all 3 Axis	6
	Welding	No Translation No Rotation	0

ME 24-688 – Week 11

Introduction to Dynamic Simulation

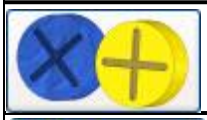
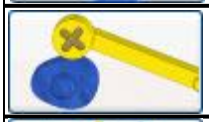
Rolling Joints

Dynamic Simulation Joints			D.O.F.
	RI Cylinder on Plane	This allows motion between a cylinder and plane	N/A
	RI Cylinder on Cylinder	This allows motion between two primitive cylindrical components in opposite directions.	N/A
	RI Cylinder in Cylinder	This allows motion between a rotating cylinder inside another non rotating cylinder.	N/A
	RI Cylinder Curve	This allows motion between a rotating cylinder and a rotating CAM.	N/A
	Belt	This creates motion of two cylinders with the same speed.	N/A
	RI Cone on Plane	This allows motion between a conical face and a planar face.	N/A
	RI Cone in Cone	This allows motion of a rotating conical component within a stationary conical component.	N/A
	Screw	This is the same as a cylindrical component but allows the user to specify pitch.	N/A
	Worm Gear	This allows motion between a worm gear component and a helical gear component.	N/A

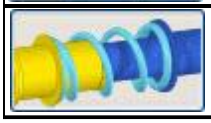
ME 24-688 – Week 11

Introduction to Dynamic Simulation

Sliding Joints

Dynamic Simulation Joints			D.O.F.
	SI Cylinder on Plane	This allows sliding between a nonrotating cylinder and plane.	N/A
	SI Cylinder on Cylinder	This allows sliding between two primitive cylindrical components of which one cylinder is nonrotating.	N/A
	SI Cylinder in Cylinder	This allows sliding between a nonrotating cylinder inside another nonrotating cylinder.	N/A
	SI Cylinder Curve	This allows motion between a nonrotating cylinder and a rotating cam.	N/A
	SI Point Curve	This creates motion of a point on one component to stay on a curve.	N/A

Additional Joints

Dynamic Simulation Joints			D.O.F.
	2D Contact	This allows motion between the curve of a component and the curve of another component.	N/A
	3D Contact	This allows you to create contact between two components.	N/A
	Spring / Damper / Jack	This allows too create springs, dampers, and jacks.	N/A

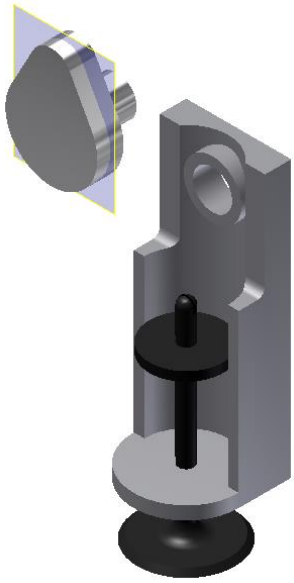
ME 24-688 – Week 11

Introduction to Dynamic Simulation

1.1 Project 1 – Cam and Valve Simulation

In this project, you run a simulation of a cam valve assembly that has already been setup to determine the torque required to rotate the cam to overcome the spring resistance. During the simulation you will compare the results with friction and without friction on the rotating cam.

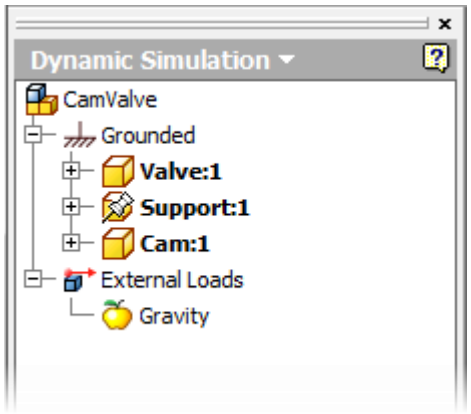
1. Open *CamValve.iam* Autodesk Inventor assembly file in Autodesk Inventor.



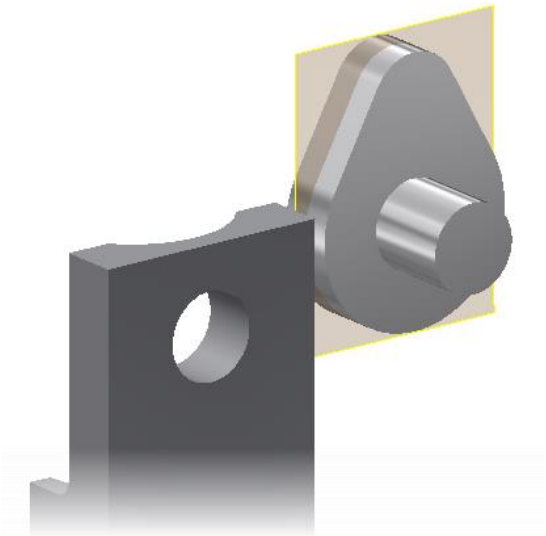
2. Click **Environments tab | Begin panel | Dynamic Simulation**. This will active the Dynamic Simulation environment.

3. If required, click **No** to close the message for beginning a tutorial.

4. Review the **Dynamic Simulation Browser** and see how the three components are listed under the Grounded group node. Also an external load for gravity has been added in the negative Y Axis direction.

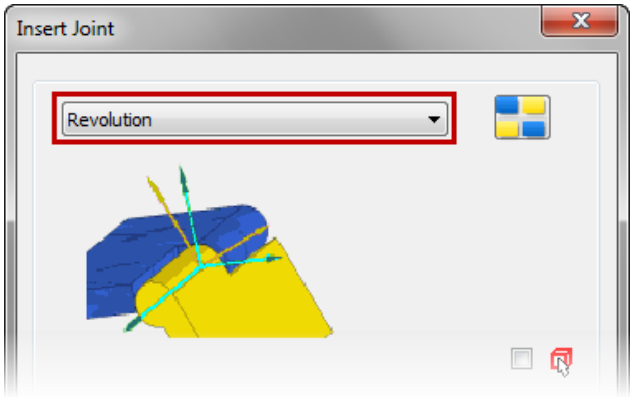


5. **Orbit** the model into a viewing position similar to the image shown below.

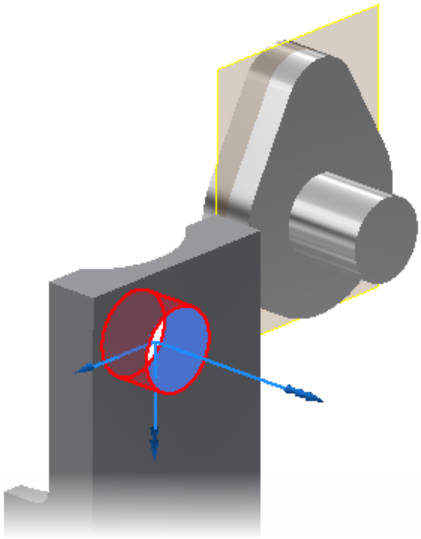


6. Click **Dynamic Simulation tab | Joint panel | Insert Joint** to begin the process of creating a standard mechanical joint.

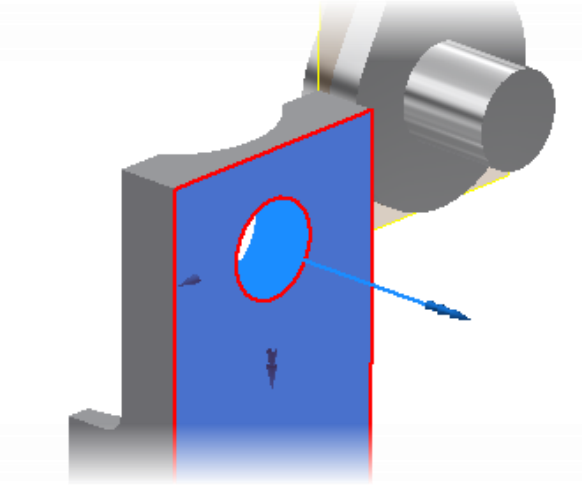
7. Select **Revolution** as the joint type. This will create a joint that has one (1) Degree of Freedom (DOF) allowing the parts to rotate in the Z Axis.



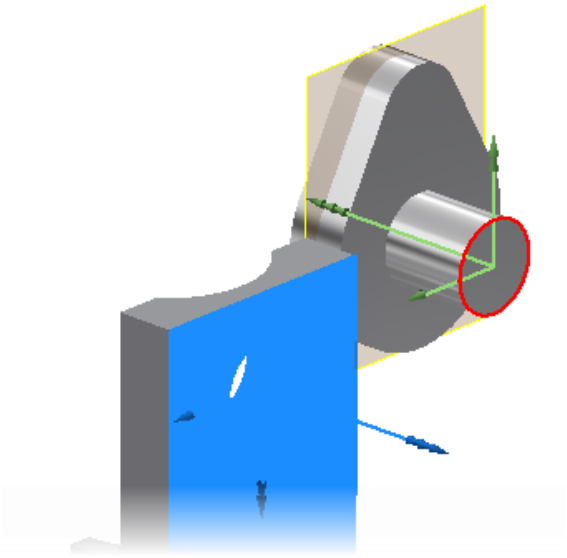
8. For the **Component 1 Z Axis** selection pick the cylindrical surface on the *Support* part as shown below. Notice the origin placement by default is in the center of the cylinder.
9.



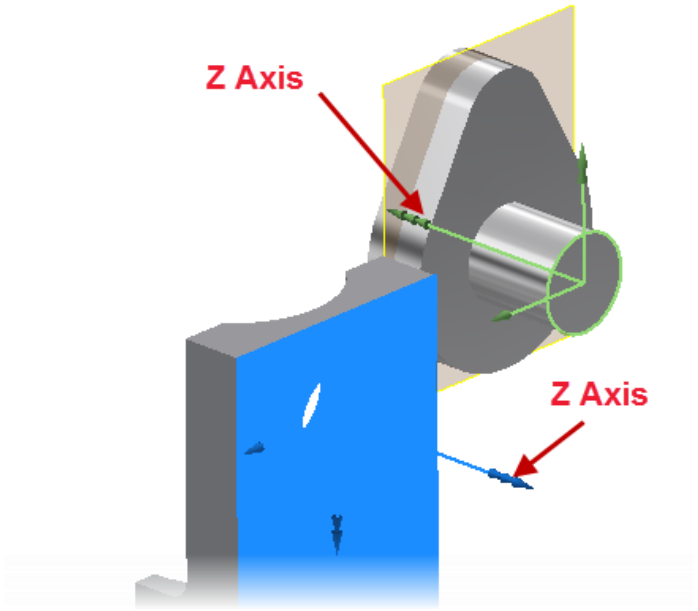
10. Pick the **Component 1 Origin** select button then select the back surface as shown below for the origin plane. This will move the joint coordinate system for the *Support* part back surface while still staying aligned to the center of the hole.



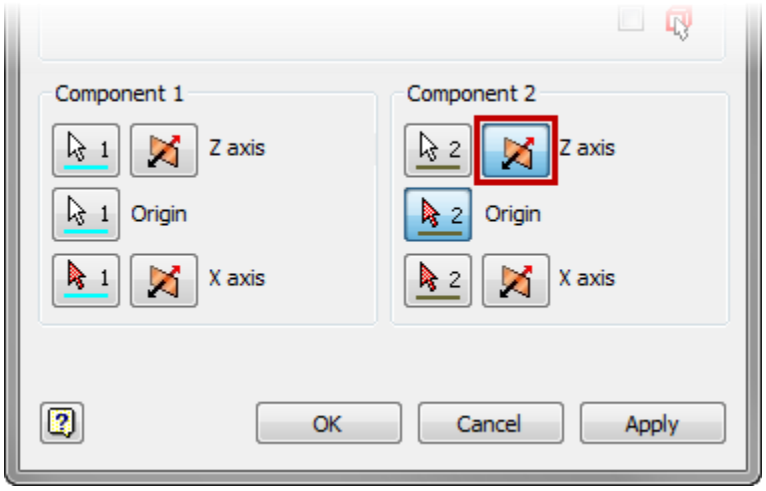
11. Now select the **Component 2 Z Axis** selection button and pick the outer edge of the *Cam* part as shown below.



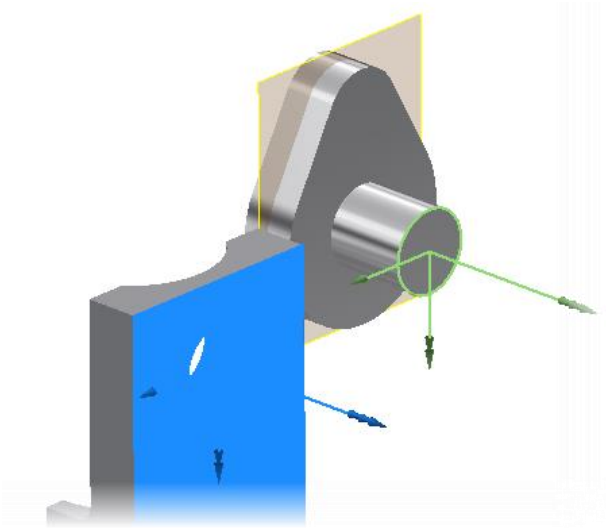
12. Notice the two joint coordinate systems that will define the joint are not aligned. The Z Axis and X Axis are pointing in different directions.



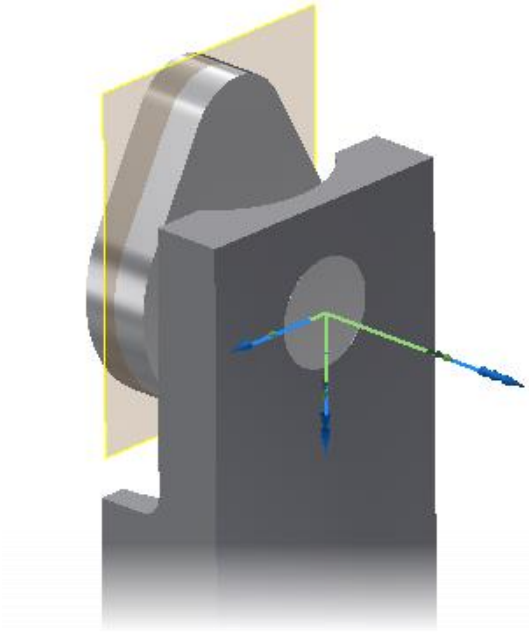
13. To fix the coordinate systems alignments click the **Component 2 Flip Z Axis** button.



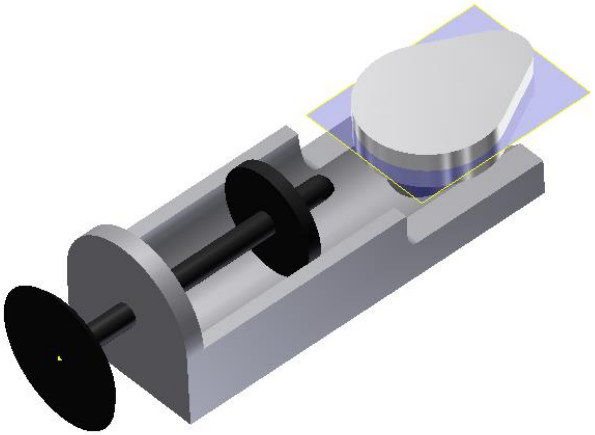
14. Notice now the two joint coordinate systems are aligned. This is a key element to creating joints and one of the most important items to review as you create joints. Click **OK** to complete creating the **Revolution** joint.



15. Now that the joint is created the two parts will snap together by aligning the coordinate systems.

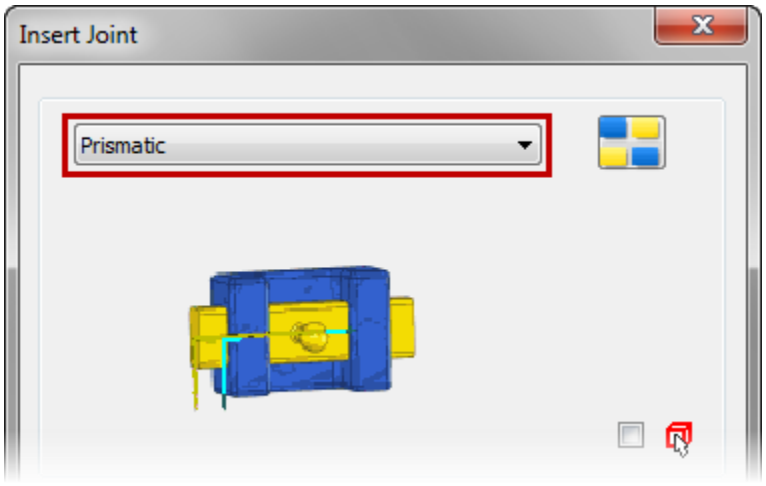


16. **Orbit** the assemble into a viewing position similar to the image shown below.

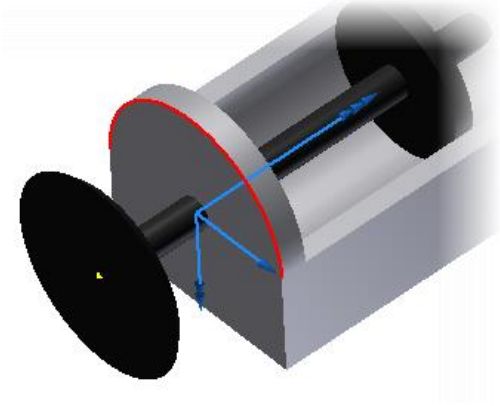


17. Click **Dynamic Simulation tab | Joint panel | Insert Joint** to begin the process of creating a standard mechanical joint.

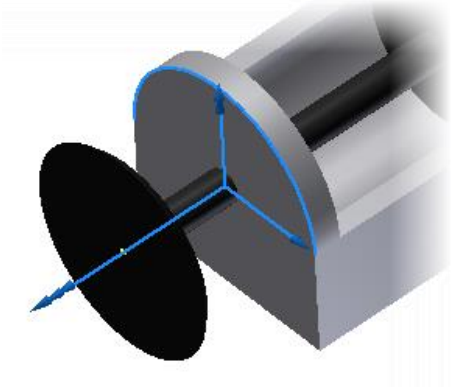
18. Select **Prismatic** as the joint type. This will create a joint that has one (1) Degree of Freedom (DOF) allowing the parts to translation along the Z Axis.



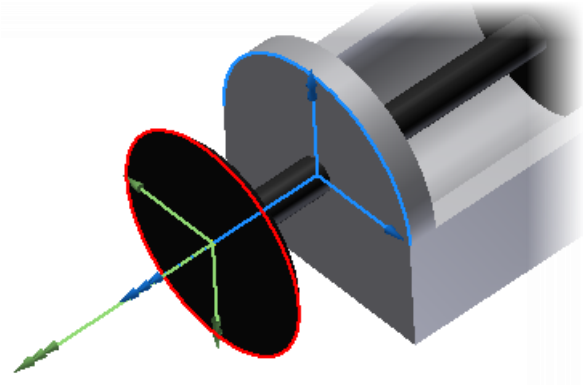
19. Select the **Component 1 Z Axis** select button and pick the bottom circular edge on the *Support* part as shown below.



20. Click the **Flip Z Axis** button for **Component 1** to align the coordinate system like the image below.

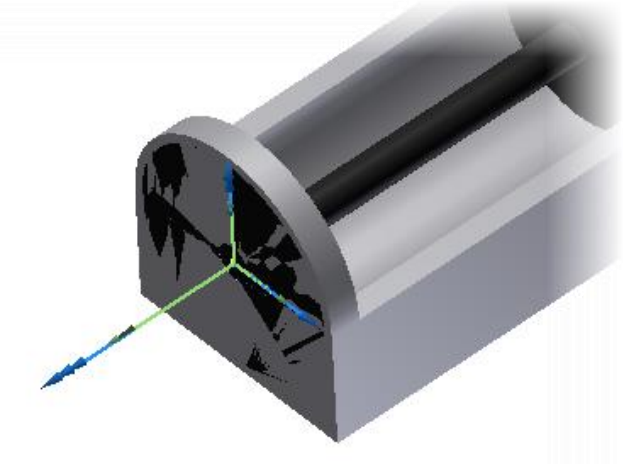


21. Click the select **Z Axis** for **Component 2** and pick the bottom circular edge of the *Value* part as shown below.

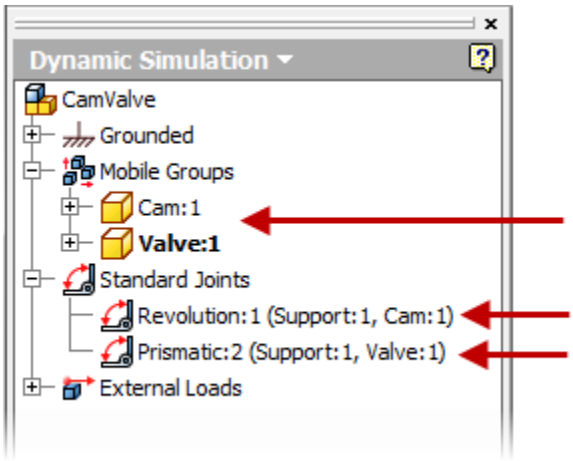


22. The two coordinate systems are aligned so click **OK** to complete creating the joint.

23. The Value part moves into position to align the two coordinate systems as shown below.



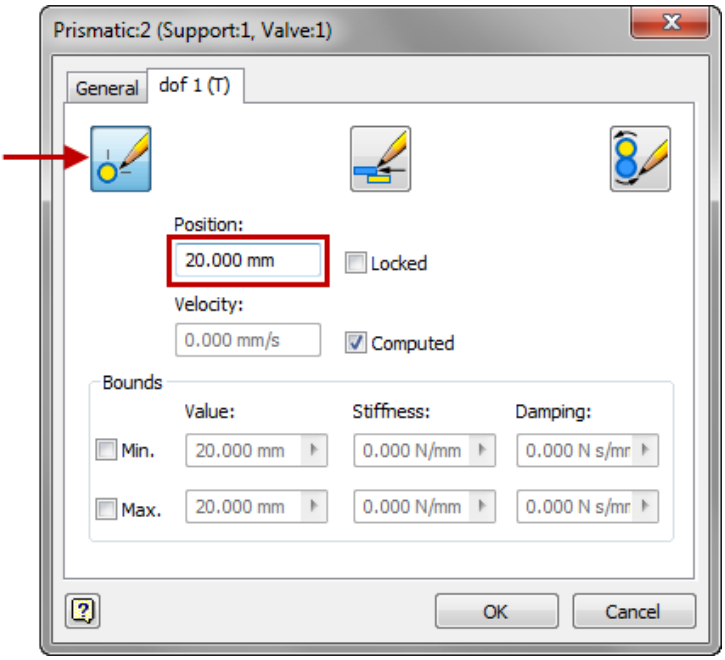
24. Review the **Dynamic Simulation Browser** and notice the two joints that were created are now listed under the **Standards Joints** node. Also notice that the two parts that now are linked to the *Support* part are listed under **Mobile Groups**.



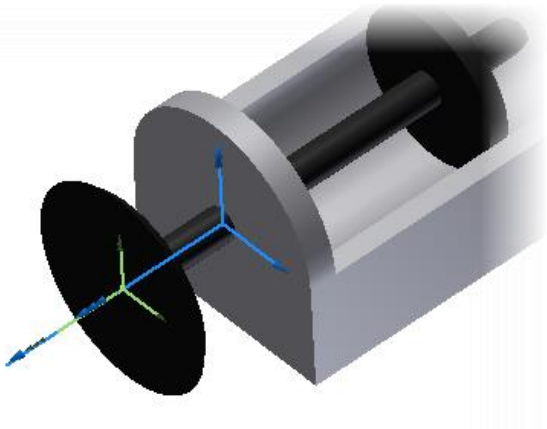
25. Select the *Prismatic:2* joint in the **Browser** and Right-Click and select **Properties**.

26. Within the **Prismatic:2** dialog box complete the following items to change the initial position of the Value part.

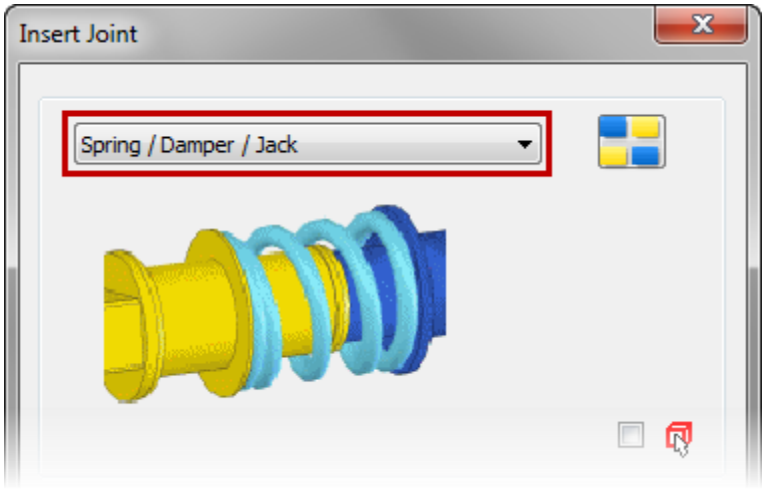
    - Click the **dof 1 (T)** tab. The "T" in the tab name indicates this is translational information.
    - Select the **Edit Initial Conditions** button. This will control the starting position of the component.
    - Enter 20mm as the **Position** value
    - Click **OK**



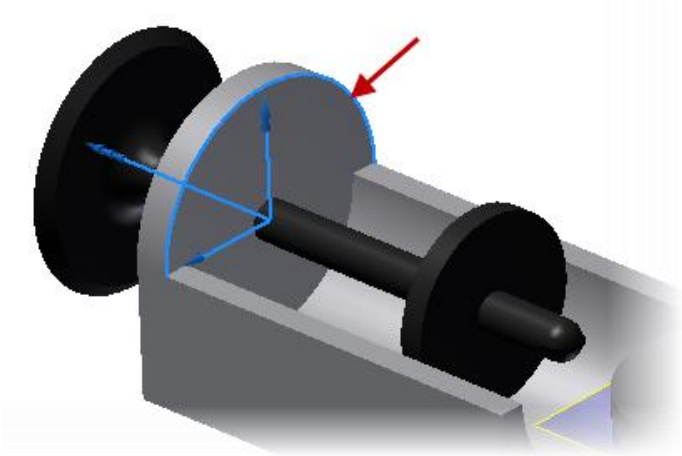
27. Review the assembly and notice the coordinate systems for the **Prismatic** joint now moved apart by 20 mm.



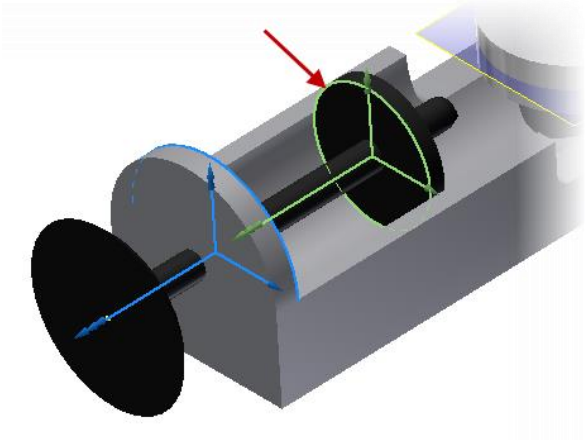
28. Click **Insert Joint** in the **Joint** panel to open the **Insert Joint** dialog. Then select **Spring / Damper / Jack** for the joint type.



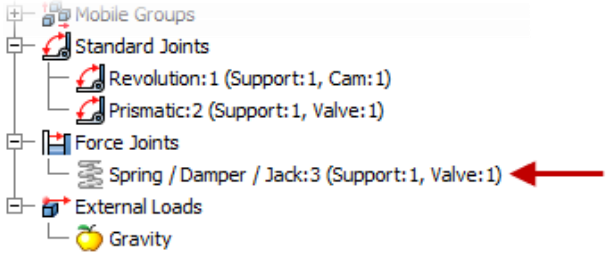
29. This joint will allow us to create a spring object between two components. Select the bottom inner circle edge of the *Support* part as shown below for the **Component 1** selection.



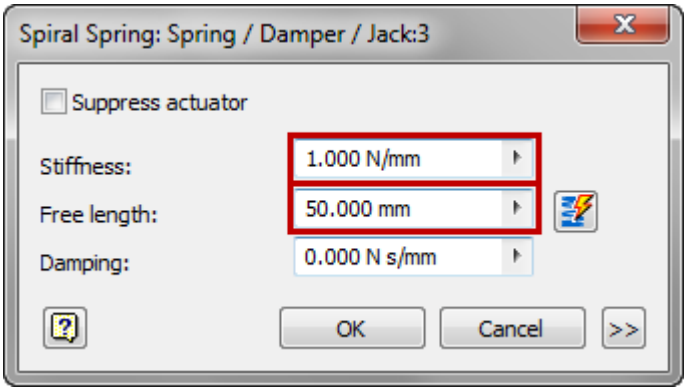
30. Select the bottom edge of the *Value* part as shown below for **Component 2**. Click **OK** to complete creating the joint.



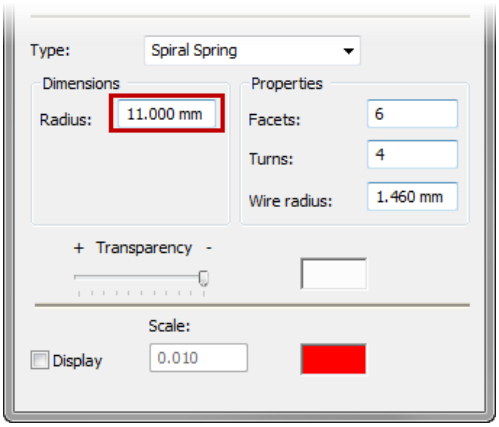
31. Select the **Spring** force in the **Browser** and Right-Click and select **Properties**.



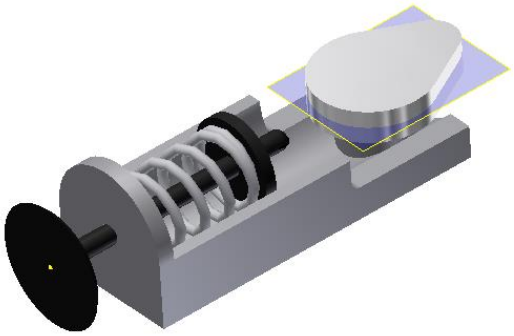
32. Enter a value of 1 N/mm for the **Stiffness** and 50 mm for the **Free Length** of the spring. This will adjust the properties and size of the spring.



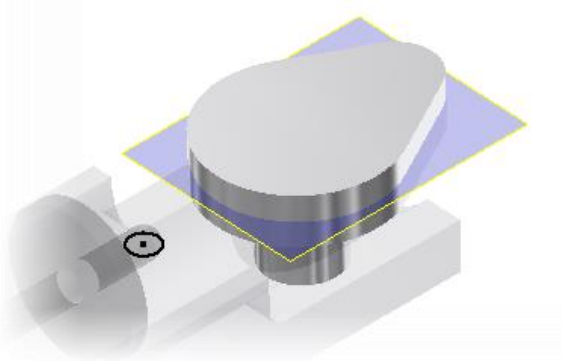
33. Click the **>>** expand button in the **Spring** dialog. Enter 11 mm for the **Radius Dimension** and click **OK** to complete creating the spring.



34. Notice the spring object that is now displayed to represent the joint that was created. This spring will push the *Value* part up into the *Cam* part.

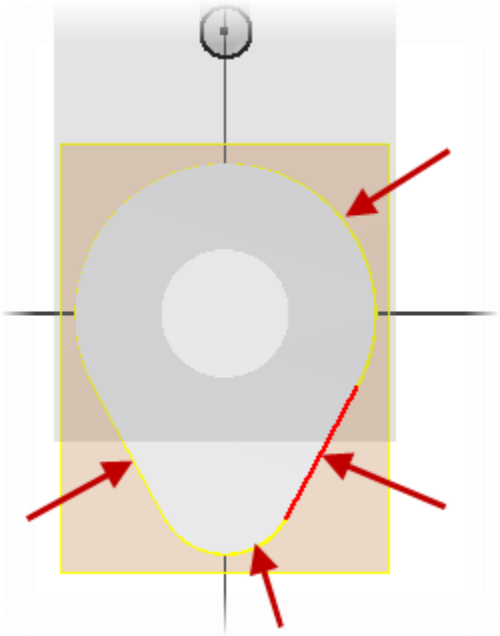


35. Now Double-Click on the *Cam* part to active the part model.



36. To provide a path for the *Value* part to contact the *Cam* part we will create a 2D sketch. Click **Model tab | Sketch panel | Create 2D Sketch** to start creating a new sketch and select the **Work Plane 2** that is already down the middle of the *Cam* part.

37. With the sketch active click **Sketch tab | Draw panel | Project Geometry**. Then select the four (4) outer edges of the *Cam* part as shown below.

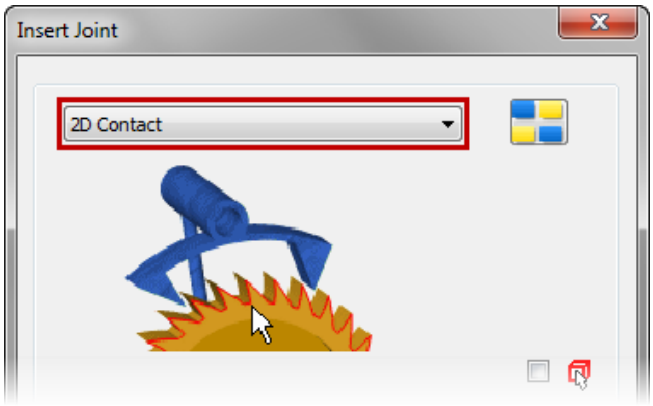


38. Click **Finish Sketch** on the **Exit** panel to complete creating the sketch.

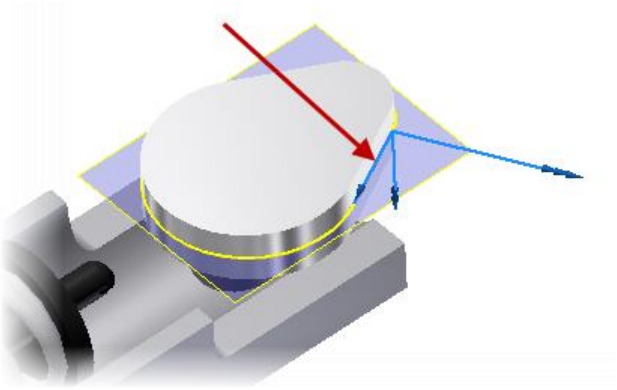
39. Click **Return** on the **Return** panel to exit editing the *Cam* part.

40. Click **Dynamic Simulation tab | Joint panel | Insert Joint** to start creating a new joint.

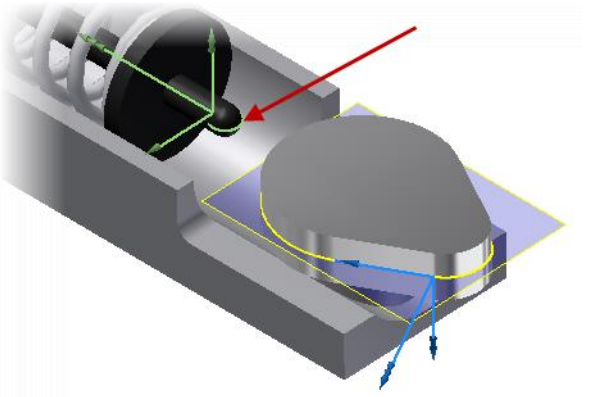
41. Select 2D Contact as the joint type.



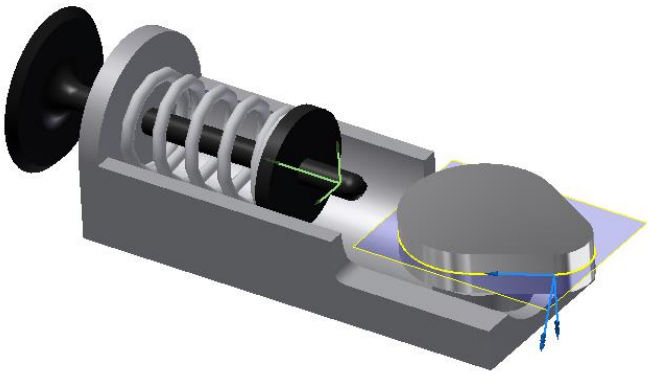
42. Select the new created sketch on the *Cam* part as the Curve for **Loop 1** as shown below.



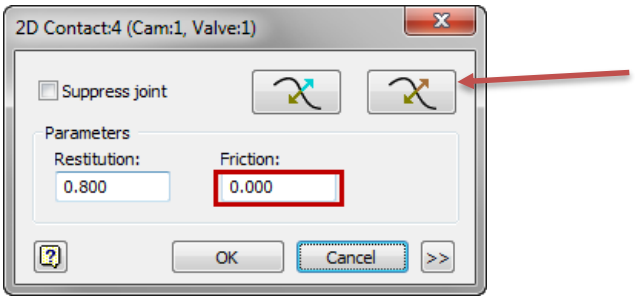
43. Select the already existing sketch that is at the top of the *Value* part as shown below for **Loop 2**.



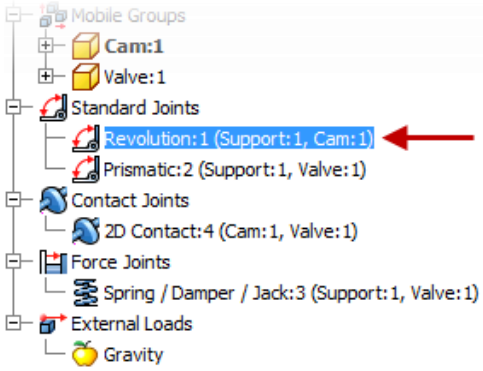
44. Click **OK** to complete the creation of the joint. The model will look like the image below.



45. Now we will adjust the properties of the 2D contact joint. Right Click on the *2D Contact:4* joint in the **Browser** and select **Properties** that is under the **Contact Joints** node. Change the **Friction** value to be 0 and keep the **Restitution** value at 0.8. Keeping the Restitution value at 0.8 will make the *Value* want to bounce off the *Cam* part during contact. Be sure that the arrows are showing the right directions as shown above. Click invert button if necessary. Click **OK** to continue.

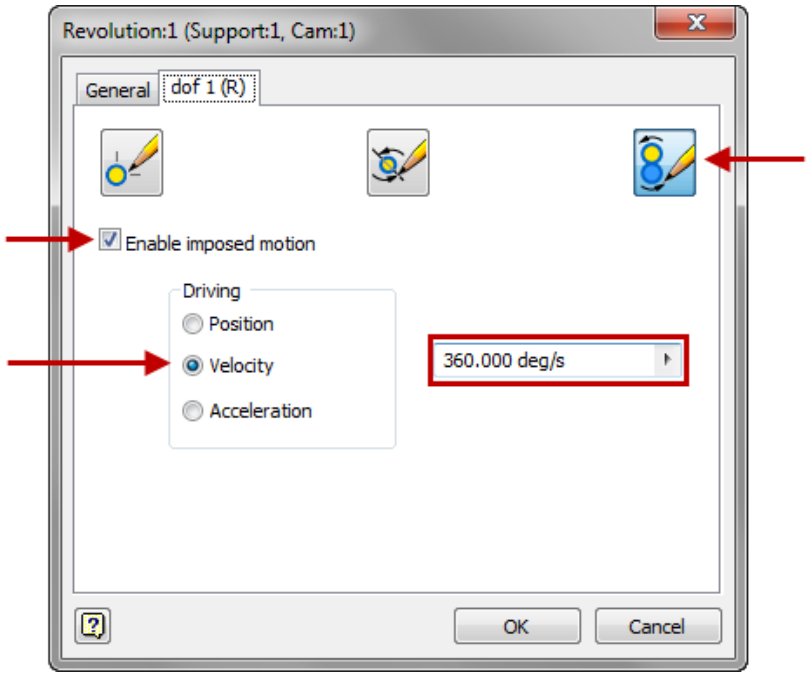


46. Now we will setup the rotation motion on the *Cam* part. Select the *Revolution:1* joint from the **Browser** and right-click to select **Properties**.

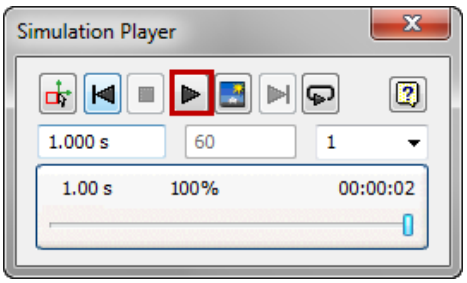

Mobile Groups
Cam:1
Valve:1
Standard Joints
Revolution:1 (Support:1, Cam:1)
Prismatic:2 (Support:1, Valve:1)
Contact Joints
2D Contact:4 (Cam:1, Valve:1)
Force Joints
Spring / Damper / Jack:3 (Support:1, Valve:1)
External Loads
Gravity

47. Within the Revolution:1 dialog complete the following items:

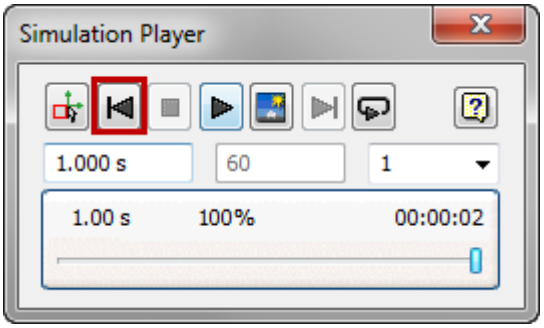
    - Select **Edit Imposed Motion**
    - Check the **Enable Imposed Motion** checkbox
    - Select **Velocity** for the **Driving** option
    - Enter 360 deg/s as the value as a Constant Value. This will make the Cam part rotate one full time for each second of the simulation.
    - Click **OK**



48. On the **Simulation Player,** click **Run** or **Replay the Simulation**. This will cycle the simulation one time and stay at the end position.

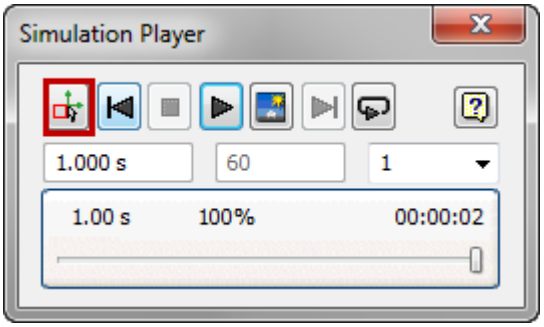


49. Click the **Rewind** button to bring the simulation back to the initial beginning.

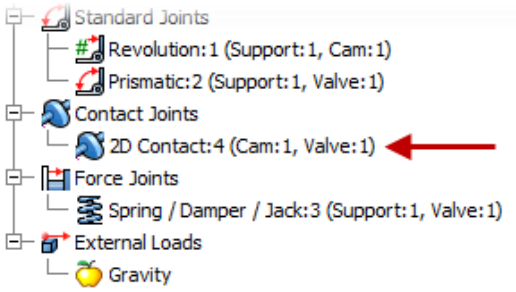


50. Click the **Run** button again to watch the simulation an additional time. You will notice that the beginning that the valve part is bouncing. We will correct this issue later in the project.

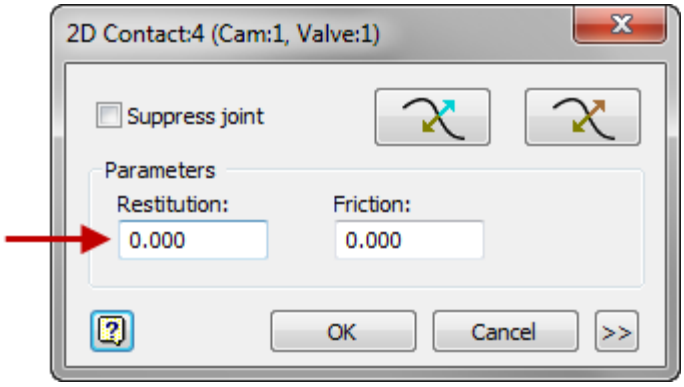
51. On the **Simulation Player**, click **Construction Mode**. This will exit the run mode and allow you to make changes to the simulation.



52. In the **Dynamic Simulation** Browser, expand **Contact Joints**. Right-click *2D Contact:4 (Cam:1, Valve:1).* Click **Properties**.

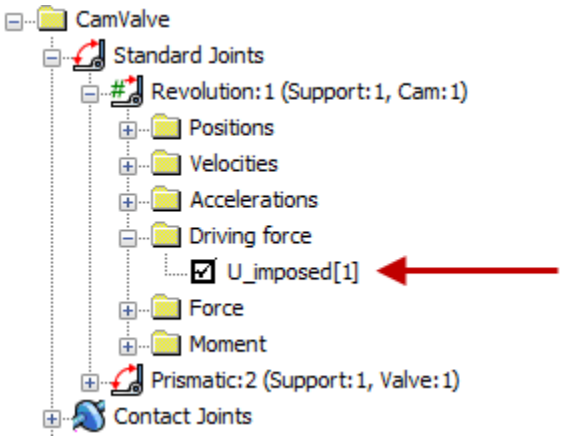


53. In the *2D Contact:4 (Cam:1, Valve:1)* dialog box, for **Restitution**, enter **0**. Setting the Restitution to 0 makes the contact between the two parts inelastic with no normal rebound. This value can be set between 0 and 1 and it is a ratio of the normal velocity between the two components after contact. Having the setting at 1 makes it elastic so it will bounce directly off. Click **OK** to continue.

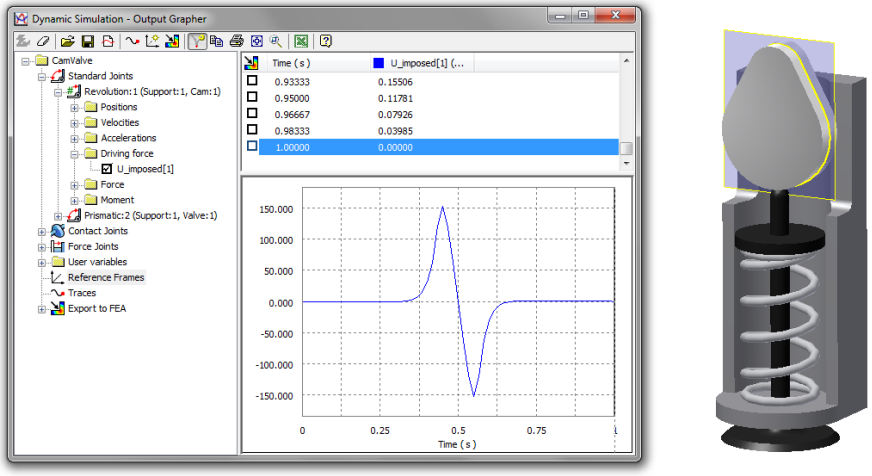


54. On the **Simulation Player**, click **Run** and notice the valve part does not bounce now after contact.

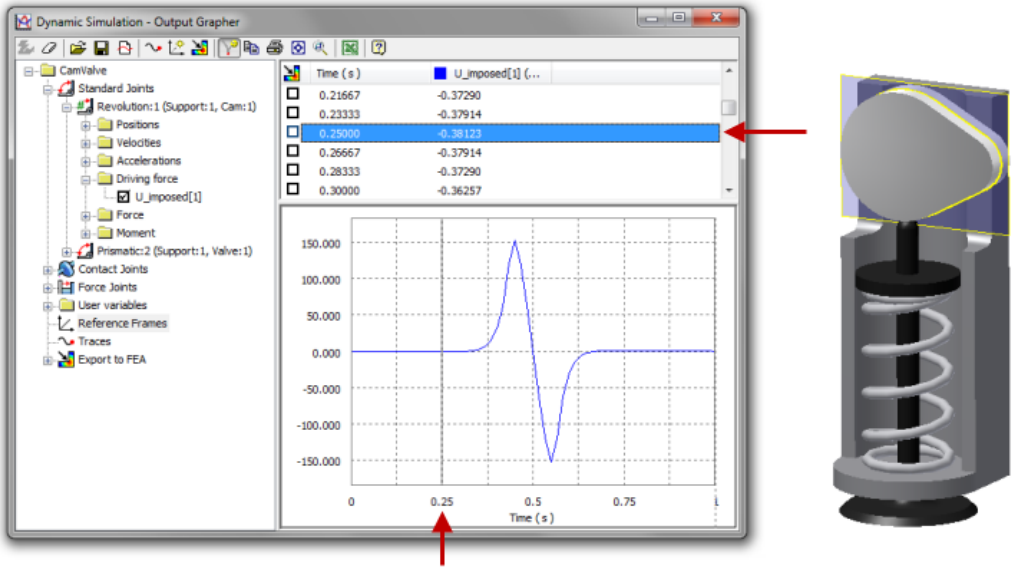
55. On the **Results** panel, click **Output Grapher**. Then under the **Revolution:1** joint expand the **Driving Force** folder and select **U_imposed**. This will display the Torque on the *Cam* rotation joint.



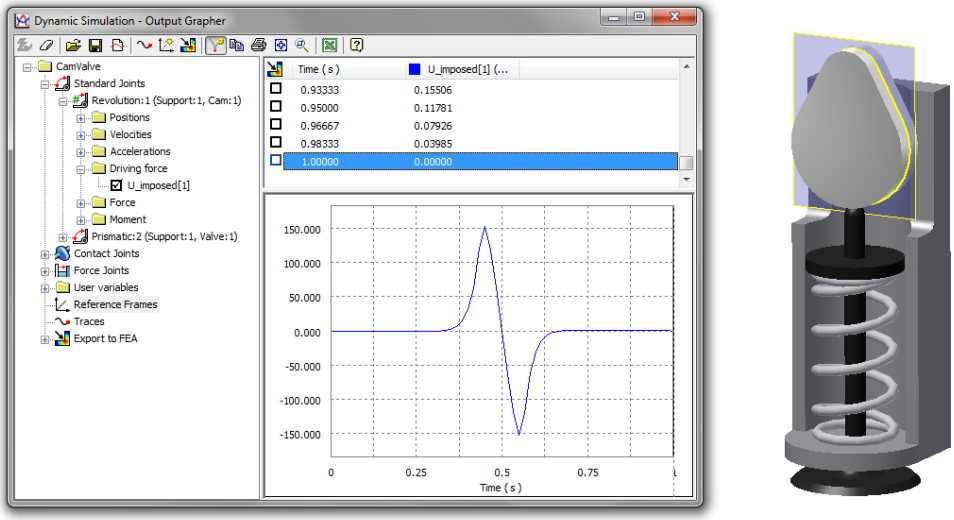
56. Resize the **Output Grapher** and zoom and pan in the *CamValve* assembly model in the graphics window so you can see both.



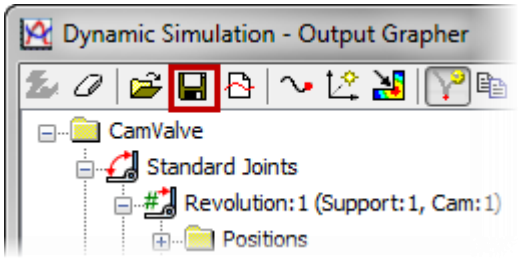
57. In the **Output Grapher**, double-click the dashed line at **0.25** (1). The timeline (2) is displayed, and the cam position (3) is updated to show its position at that point of the simulation.



58. To cycle through the simulation, use the right and left arrow keys on the keyboard to step forward and backward in the simulation. Cycle through the simulation to **1.00** to show the cam at the end of the simulation.



59. In the **Output Grapher**, click **Save Simulation**. Save the file as *CamValve.iaa*. This will save the current simulation run results to a file that can be later imported back into the **Output Grapher**.



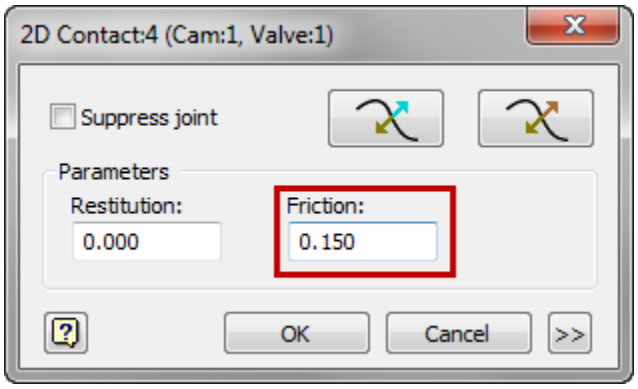
60. On the **Simulation Player**, click **Construction Mode**.

61. The following steps describe how to add a coefficient of friction to calculate the effect on the torque that is required to rotate the cam and overcome the spring force and friction force.

    - In the **Dynamic Simulation** Browser, under **Contact Joints**, right-click *2D Contact:4 (Cam:1, Valve:1)*. Click **Properties**.

    - In the *2D Contact:4 (Cam:1, Valve:1)* dialog box, for **Friction**, enter **0.15**. This enters a coefficient of friction value that will assign a force that resists the motion of the two bodies in relation to another. The most common values range from 0 – 1 with Autodesk Inventor supporting values from 1 – 2 also for unique cases.

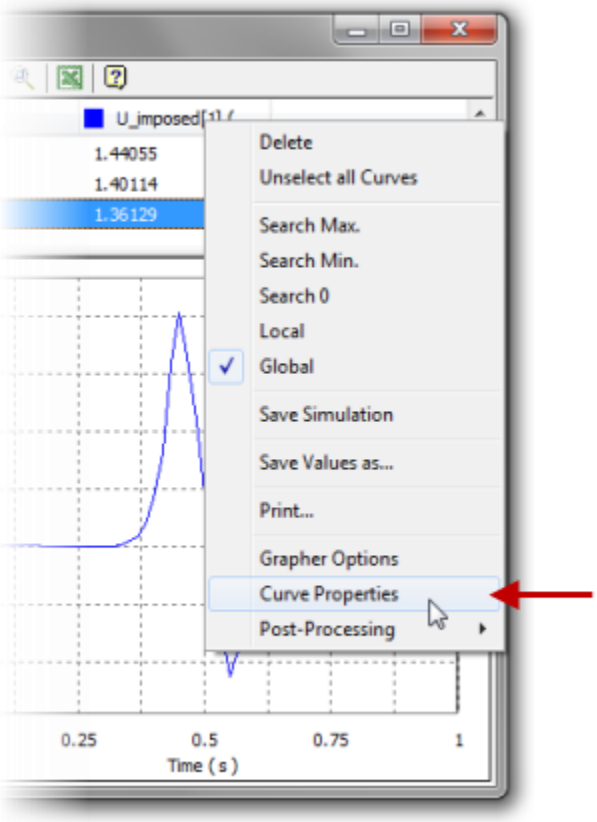
    - Click **OK**.



62. On the **Simulation Player**, click **Run** or **Replay the Simulation**. The **Output Grapher** is still open, so you see the graph being generated as the simulation is running.

    - In the next step, you change the color of the newly generated curve. When you compare the saved graph curve with this new one, you can distinguish between the two of them.

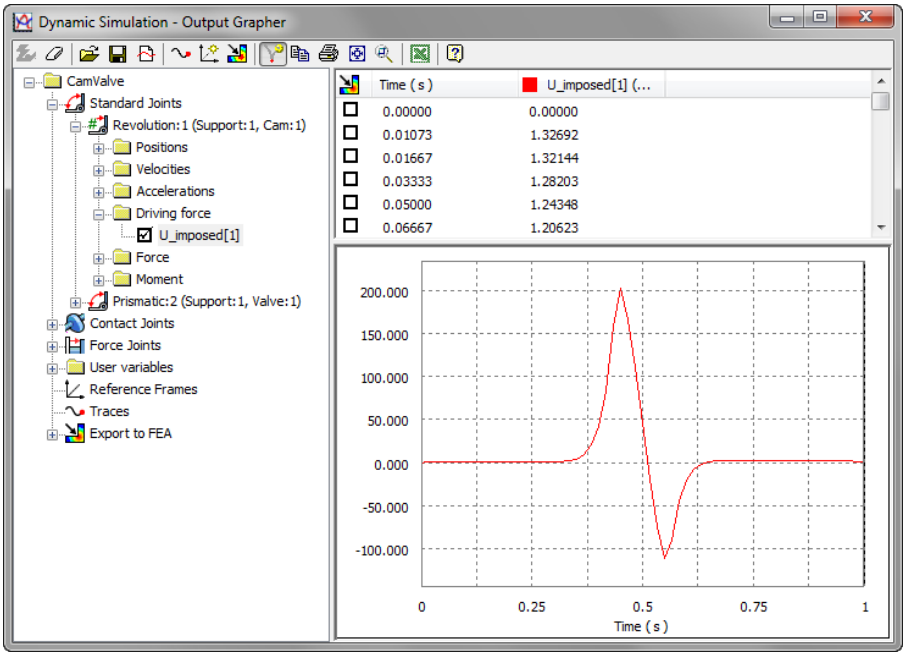
63. In the **Output Grapher**, right-click the **U_imposed[1]/N** mm column heading. Click **Curve Properties**.



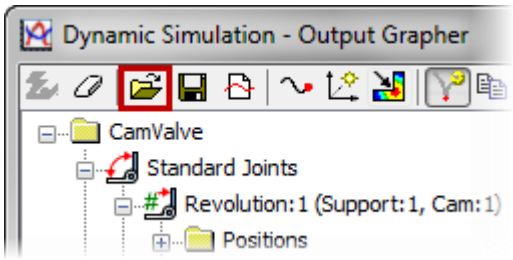
64. In the **Dynamic Simulation - Properties** dialog box, click the color box.

65. In the **Color** dialog box, click the **Red** color swatch. Click **OK**.

66. In the **Dynamic Simulation - Properties** dialog box, click **OK**. The graph changes to red.

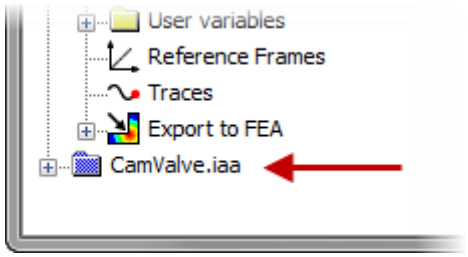


67. On the **Output Grapher** toolbar, click **Import Simulation**. This will allow you to load the past simulation run results we saved so they can be viewed with the current results.

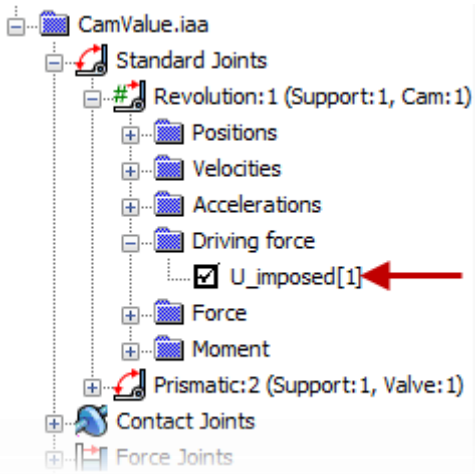


68. In the **Dynamic Simulation - Load** file dialog box, select *CamValve.iaa*. Click **Open**.

69. Review the **Dynamic Simulation** Browser. The *CamValve.iaa* element is added to the **Output Grapher**.

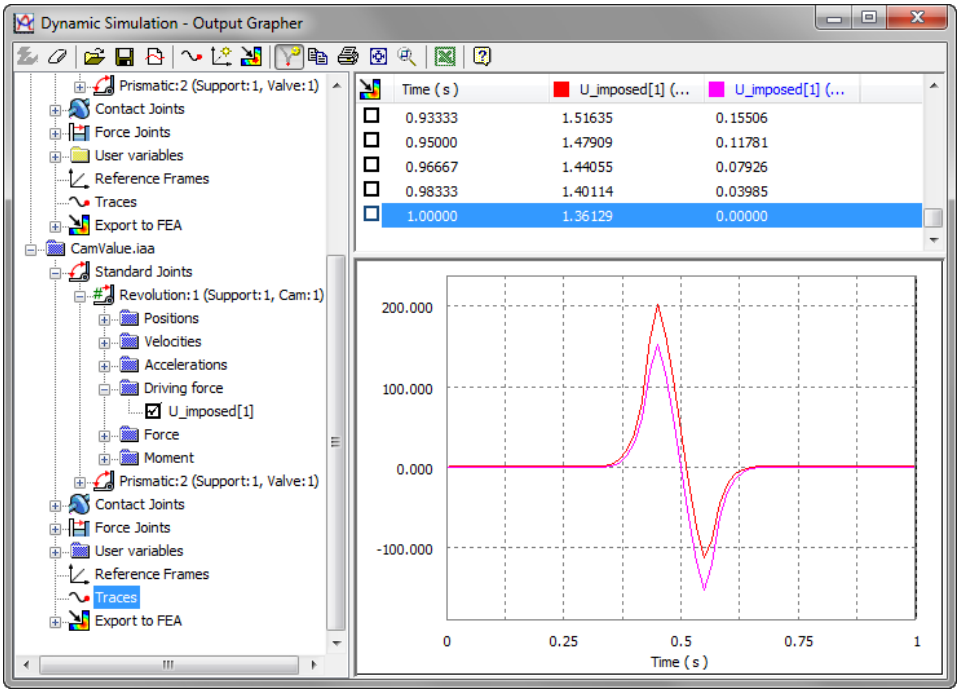


70. In the **Output Grapher**, expand *CamValve.iaa*. Expand *Revolution:1 (Support:1, Cam:1)*. Expand **Driving Force** and select the **U_imposed [1]** check box.



71. The **Output Grapher** displays the graphs of the torque required to rotate the cam without friction (magenta) and with friction (red). Also, a new column is added in the time-steps pane for the numerical values of each force.

    Notice that the maximum torque required with friction is 200 N and without friction it is around 150 N.



72. Close the file. Do not save changes.