Create Joints Project

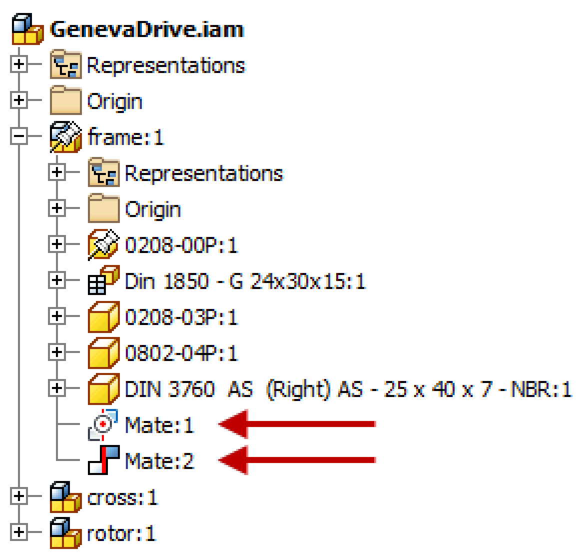
One of the most important elements of dynamic simulation is setting up and verifying that proper joints are created. Joints are links between two rigid components that applies force from the first component on the second component. Each joint has a coordinate system that is used to define motions and efforts in the joint. During the various projects you will explorer and create several different types of joints within an assembly.

1.1 Project 2 – Creating Joints

In this project, you create joints to add degrees of freedom to components in an assembly. First, you create two revolution joints by converting assembly constraints and by using the **Insert Joint** tool. Next, you create two 2D contact joints to control the relationship between the two revolving subassemblies. Finally, you impose motion on the joints to see the effect in the assembly.

1. **Open** *GenevaDrive.iam* Autodesk Inventor assembly file.

2. In the **Browser**, expand the *frame:1* part node to view the two Mate constraints.

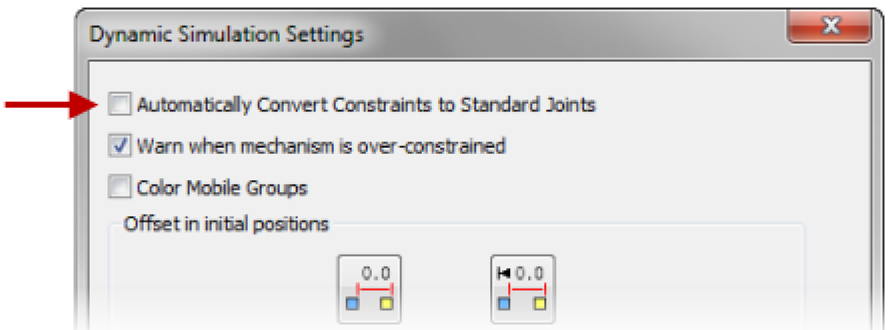


3. Click **Environments tab | Begin panel | Dynamic Simulation**.

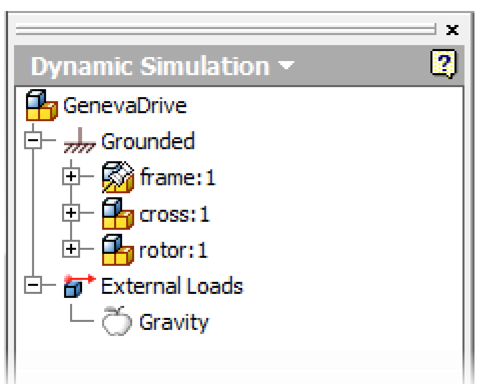
4. If required, click **No** to close the message window about viewing the dynamic simulation tutorial.

5. Depending on your Dynamic Simulation Settings at the time of activation the standard assembly constrains may be converted into Standard joints. Click on the **Manage** panel and select **Simulation Settings**.

6. Ensure that the **Automatically Convert Constraints to Standard Joints** option is unchecked. If you need to uncheck the option select **No** to the message to maintain standard joints that have already been created.

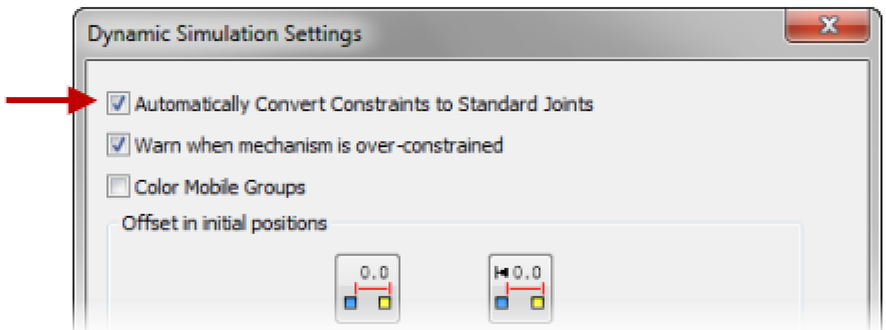


7. Review the **Dynamic Simulation Browser**. The subassemblies are listed under **Grounded** group because no joints have been applied to these components at this time.



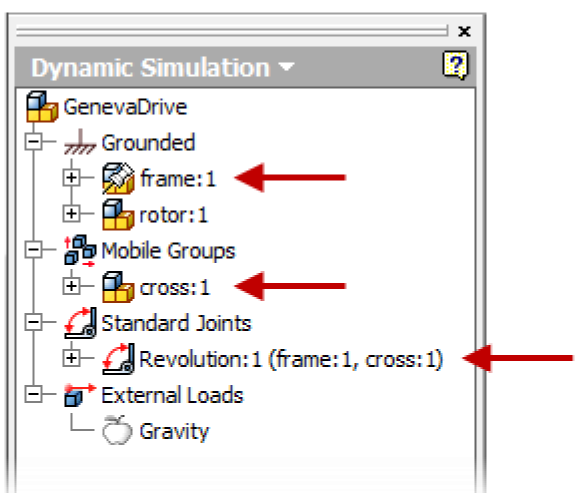
8. On the **Manage** panel, click **Dynamic Simulation Settings**.

9. In the **Dynamic Simulation Settings** dialog box, select the **Automatically Convert Constraints to Standard Joints** check box. This will now take each of the standard assembly model constraints and convert them into dynamic simulation joints. Click **OK** to complete the process.



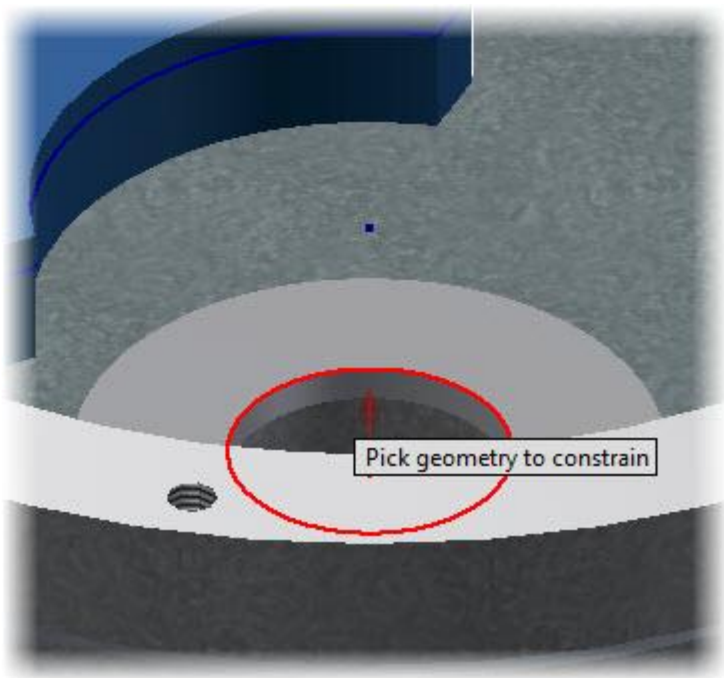
10. Review the **Dynamic Simulation Browser**. Look at the following:

- The frame assembly is grounded in the assembly environment as it remains in the **Grounded** group.
- A **Mobile Groups** node is added and the cross subassembly is placed in that group.
- A **Standard Joints** node is added and the newly created Revolution joint is placed there as a result of the existing assembly mate constraint.

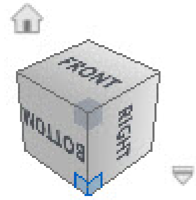


11. Now we will create a new dynamic revolution simulation joint manually. This is often the most common method as the way you assemble items with standard assembly constrains is often not the ideal way for setting up simulation joints. Right-click in the graphics window and select **Constraint**. This will start a standard assembly constraint.

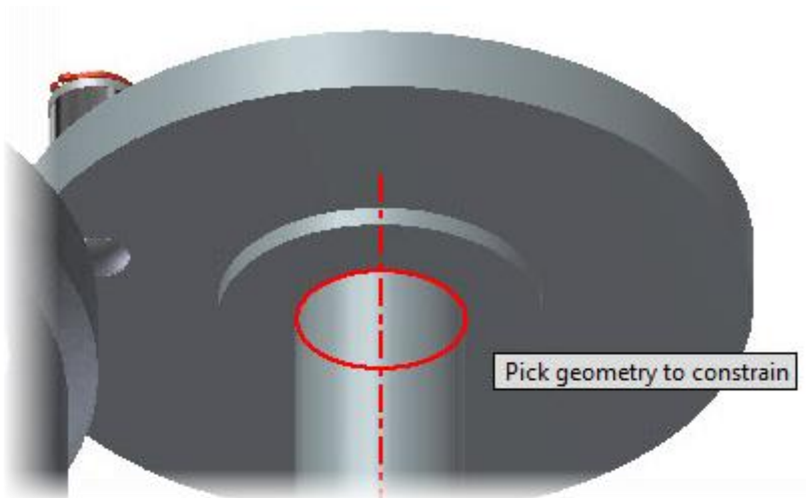
12. Under **Type**, click **Insert Constraint**. Select the edge as shown below.



13. On the **ViewCube**, click the bottom corner.



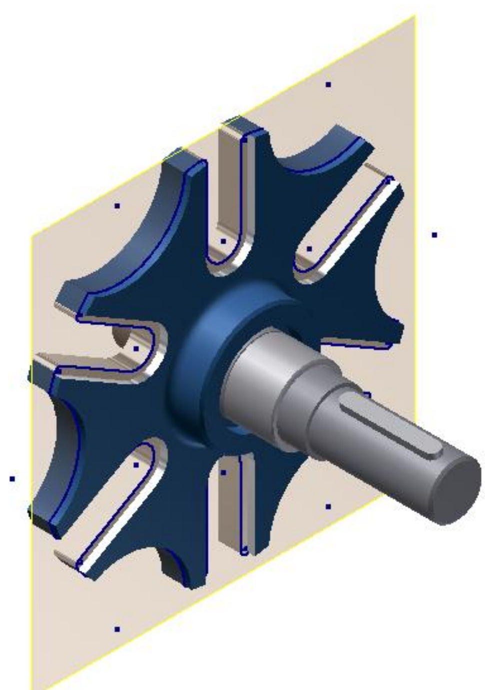
14. Select the edge as shown. Click **OK**.



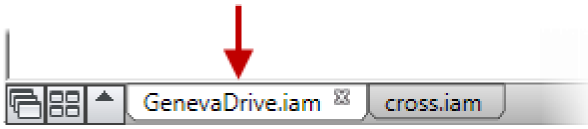
15. On the **ViewCube**, click **Home** symbol to return to the view shown below.

16. In this section of the project, you create 2D contact joints between the cross and rotor subassemblies and set the initial position of the rotor subassembly. In the **Dynamic Simulation Browser**, under **Mobile Groups**, right-click *cross:1*. Click **Open**. This will open that assembly file as a new document in Autodesk Inventor.

17. Review the visible geometry on the *cross.iam* file. You use the work plane and the projected edges to create the **2D contact** joint.

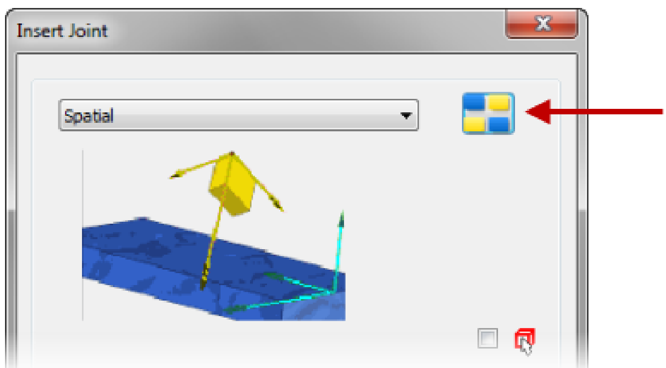


18. Switch to the *GenevaDrive.iam* file window by click on the tab at the bottom of the graphics window.

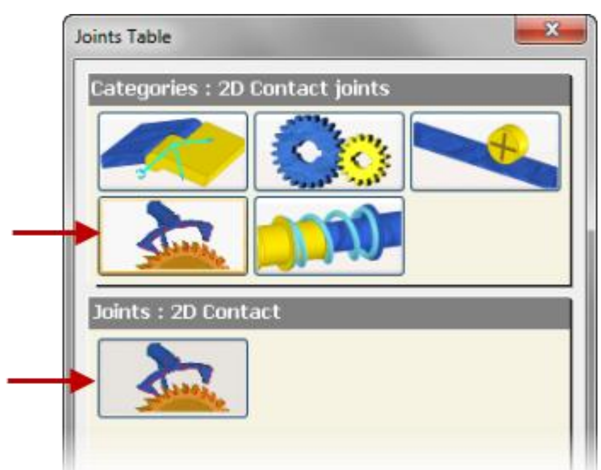


19. Click on **Dynamic Simulation tab | Joint panel | Insert Joint**. This will start the most common used feature of dynamic simulation to start the process of creating a new joint.

20. In the **Insert Joint** dialog box, click **Display Joints Table**.

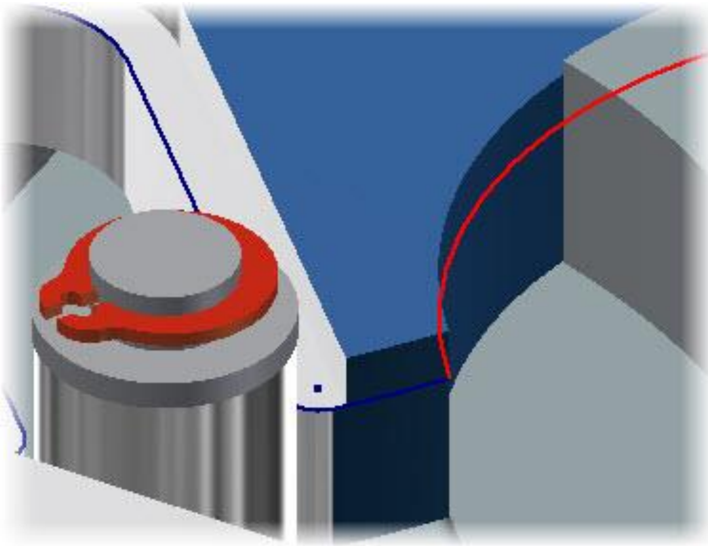


21. In the **Joints Table** dialog box, do the following:
    - Click 2D Contact Joints.
    - Click 2D Contact Joint.

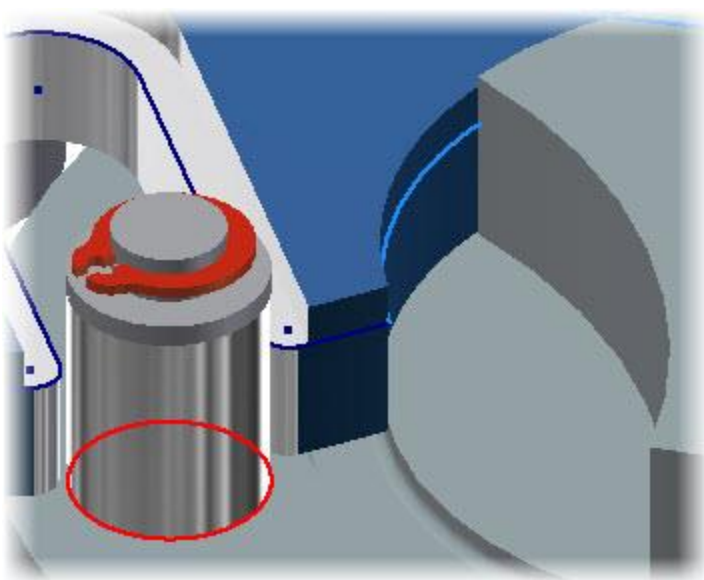


22. Click **OK** to close the **Display Joints Table** dialog.

23. On the *cross* assembly, select the projected loop, as shown in red as the first joint curve loop.

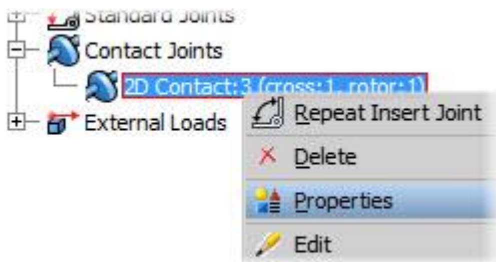


24. On the *rotor* assembly, select the edge on the cylindrical face, as shown. Click **OK** to complete the joint.



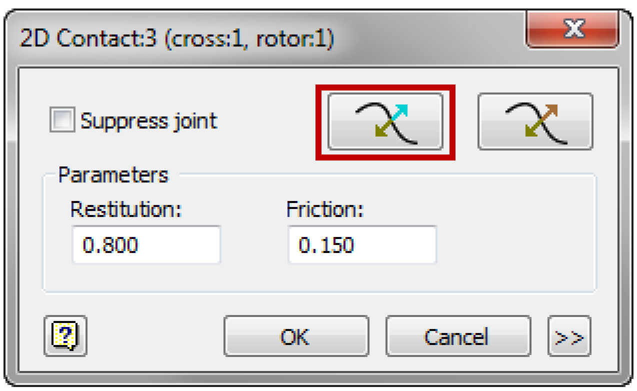
25. The Z axis for the coordinate system of the projected loop must be inverted for the joint to function properly. In its present orientation, it functions as a hole and not a perimeter edge.

26. In the Browser, under **Contact Joints**, right-click *2D Contact:3 (cross:1, rotor:1)*. Click **Properties**.



27. In the *2D Contact:3 (cross:1, rotor:1)* dialog box, do the following:

- Click **Invert Normal**.
- Review the Z axis. It should be pointing away from the cross assembly.
- Click **OK**.

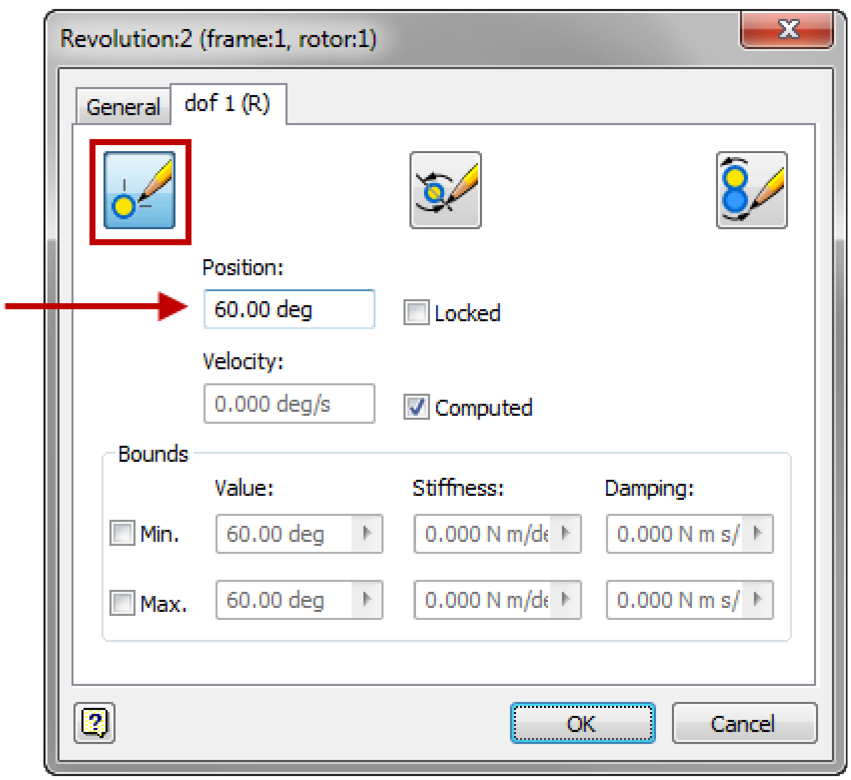


28. Next, you set up the initial state of the rotor relative to the cross. This is to make sure that the two components are not interfering at the start of the simulation.

29. In the **Browser**, under **Standard Joints**, right-click *Revolution:2 (frame:1, rotor:1)*. Click **Properties**.

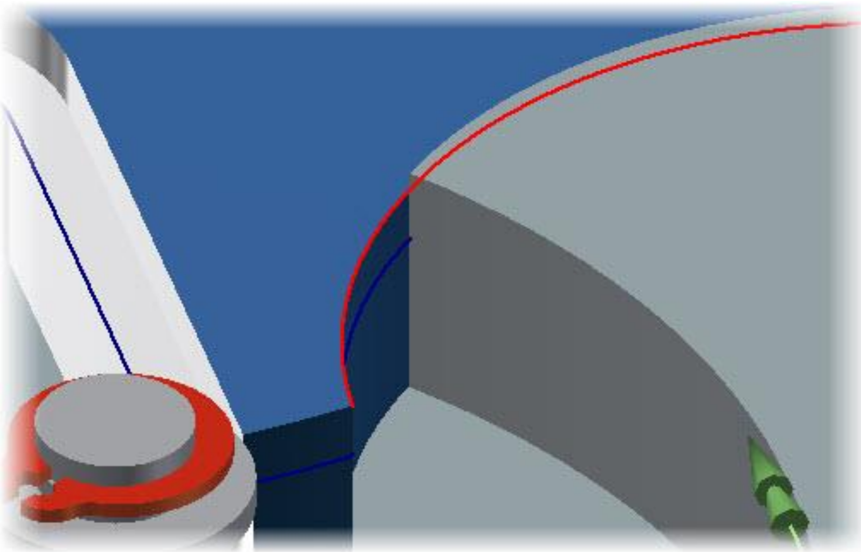
30. In the *Revolution:2 (frame:1, rotor:1)* dialog box, do the following:

- Click the **dof 1 (R) tab**
- Confirm that **Edit Initial Conditions** is selected
- For **Position**, enter **60 deg**
- Click **OK**

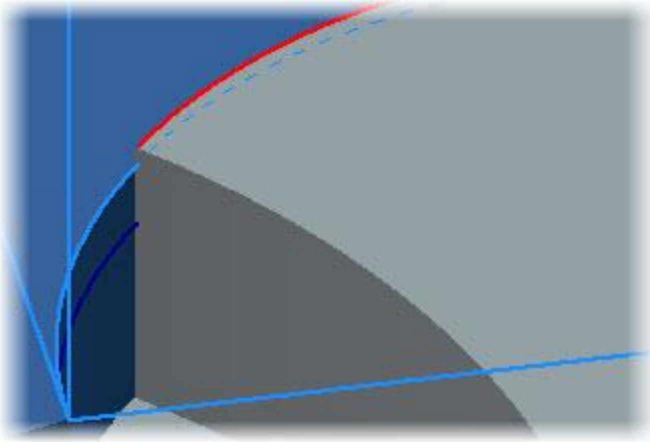


31. On the **Joint** panel, click **Insert Joint**. In the **Insert Joint** dialog box, select **2D Contact** from the **Joint Type** list, if not already selected.

32. For the first loop, on the cross subassembly, select the radius edge as shown in red.



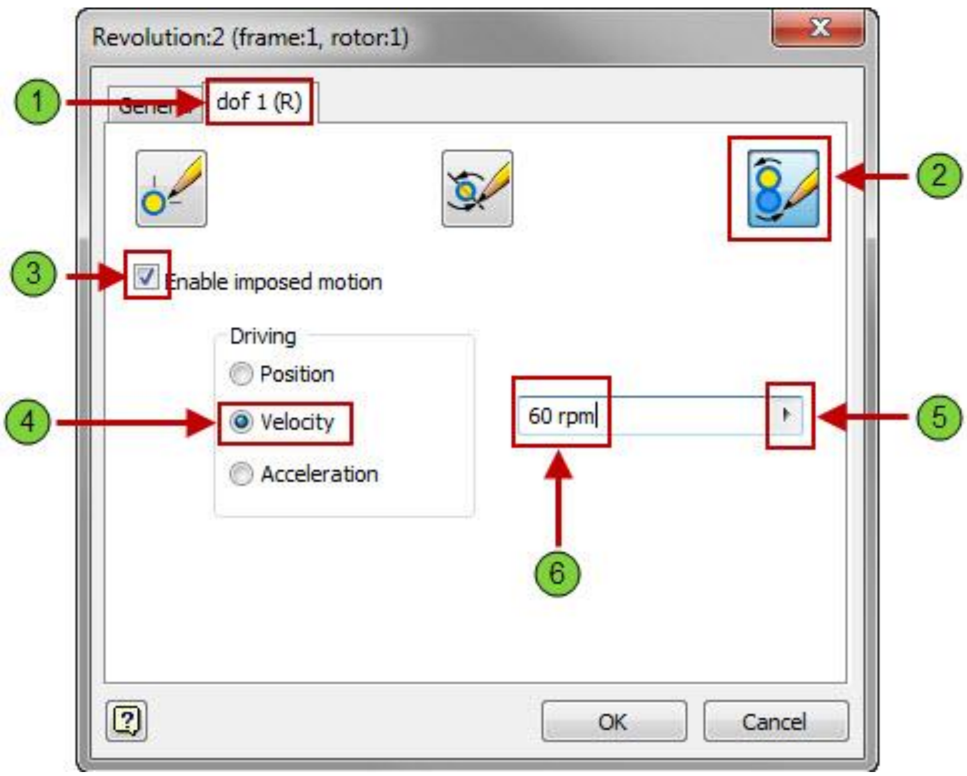
33. For the second loop, on the rotor subassembly, select the radius edge as show in red. Click **OK**.



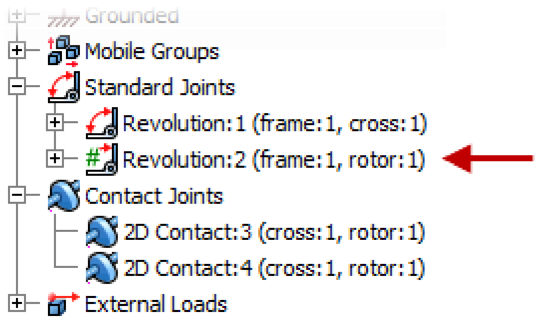
34. Before you run the simulation, you specify the motion of the rotor. In the **Browser**, under **Standard Joints**, right-click *Revolution:2 (frame:1, rotor:1)*. Click **Properties**.

35. In the *Revolution:2* dialog box, do the following. Be sure that **Lock dofs** box is unchecked under **General** Panel.

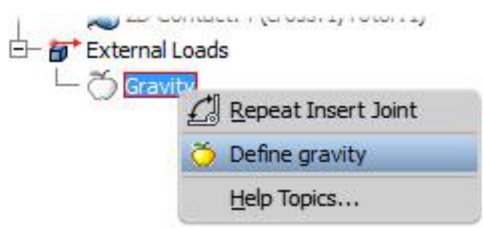
- Click the **dof 1 (R) tab**
- Click **Edit Imposed Motion**
- Select the **Enable Imposed Motion checkbox**
- Click **Velocity**
- Click the arrow. **Select Constant Value**
- Enter **60 rpm**
- Click **OK**



36. In the Browser, review the icon for *Revolution:2 (frame:1, rotor:1)*. A # symbol was added to signify that motion was applied to the joint.



37. In the Browser, under **External Loads**, right-click **Gravity**. Click **Define Gravity**.

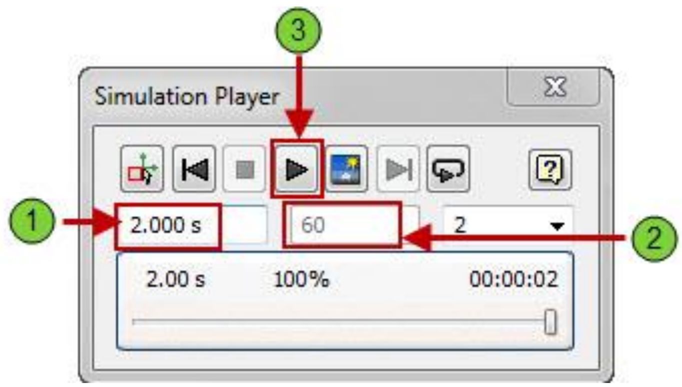


38. In the **Gravity** dialog box, do the following:

- Clear the **Suppress** check box
- Click **Vector Components**
- For g[Y], enter **-9.81**
- For g[Z], enter **0**
- Click **OK**

39. On the Simulation Player, do the following:

- For Final Time, enter 2 s
- For Images, enter 60
- Click **Run** or **Replay the Simulation**



40. View the simulation. The rotor makes two revolutions and drives the cross subassembly.

41. Close all files. Do not save changes.