Within this project there are three separate projects that all work on completing dynamic simulation activities on a windshield wiper assembly. The project will cover creating and editing joints plus assigning loads. The Output Grapher will also be used in each project to analysis and view key critical data on the performance of the design.

### 1.1 Project 3A – Joint Properties and Defining Loads

In this project you create joints in a wiper assembly to drive the wiper arms. You impose motion on the drive arm and add resistive force to the wiper blades to simulate the friction of the blade on the windshield. Finally you use the Input Grapher to define the resistive force so that it is always opposed to the wiper motion.
1. In this section of the project, you define the joints that connect the drive arm to the cranks that move the wiper subassemblies. Open the WiperAssemblyDEC.iam Autodesk Inventor assembly file.

2. Click Environments tab | Begin panel | Dynamic Simulation.

3. If required, click No to close the message window for viewing the standard tutorial.

4. The model already has some setup items completed and joints created as a starting point for this project. On the ViewCube, click the top-right corner.

5. Review the assembly as shown below. The simulation requires a joint to control the rotation of the Motor_Crank_Asm (1). This subassembly is attached to the wiper motor and drives the Crank_motor2 component (2), which drives the wiper subassemblies (3).
6. On the **Joint** panel, click **Insert Joint** to start the process for creating a new mechanical joint.

7. Select **Cylindrical** from the list. The Cylindrical joint has two (2) DOF one rotation and one translation.

8. For the **Component 1 Z Axis** select the hole of the *Motor_Crank_Asm* as shown.
9. In the **Insert Joint** dialog box, under *Component 2*, click the second selection button.

10. Select the work axis, as shown.

11. Click **OK**. The cylindrical joint enables *the Motor_Crank_Asm* to rotate freely around the work axis and move along the work axis.
12. On the Joint panel, click Insert Joint to start creating another joint.

13. Select Revolution from the list of joint types. This revolution joint will have one rotation DOF around the Z Axis.

14. Select the circular edge as shown below for the Component 1 Z Axis.

15. In the Insert Joint dialog box, under Component 2, click the second selection button.

16. Select the circular edge as shown.
17. Under *Component 2*, click Flip X Axis Direction as shown, to reverse the direction of the X Axis. This will more closely align the two components in place.

![Component 1 and Component 2 with Flip X Axis Direction](image1)

18. Click **Apply**. The *Crank_motor2* moves to the correct position. The **Insert Joint** dialog box remains open.

![Crank Motor 2 in Correct Position](image2)
19. In the **Insert Joint** dialog box, select **Spherical** from the list. This joint will have three (3) DOF all rotational about a point. This type of joint is used to not over constrain the component.

20. Select the circular edge as shown.

![Image of Spherical Joint](image)

21. In the **Insert Joint** dialog box, under **Component 2**, click the second selection button.

22. Select the circular edge as shown.
23. Review the alignment of the coordinate axes. The Z axis (1) of component 1 is pointing in the opposite direction of the Z axis (2) of component 2.

24: Under *Component 2*, click *Flip Z Axis Direction* as shown to reverse the direction of the Z axis.
24. Click **OK** to complete the joint and exit the **Insert Joint** dialog.

25. On the **ViewCube**, click **Home**.
26. **Save** the assembly file.

27. In this section of the project, you impose motion on the *Motor_Crank_Asm* to simulate the wiper motor driving the wipers. Then you add a resistant force to the wipers to simulate the wiper blade rubbing against the windshield as it turns.
   - Zoom in on the assembly as shown.

28. In the **Browser**, right-click *Cylindrical:5 (Motor_Crank_Asm:1, Bearings:1)*. Click **Properties**.
29. On the dof 1 (R) tab, do the following:

- Confirm that **Edit Initial Conditions** is selected. This will specify the starting position of the components.
- For **Position**, enter **-15 deg**
- Click **OK**

30. On the **ViewCube**, click **Front**. Zoom in to the coordinate system axes.

31. Review the axes. The **-15 deg** angle is placed between the X axis of component 1 in the **cylindrical joint** and the X axis of component 2. This is now the initial starting position for the simulation.
32. On the ViewCube, click Home.

33. In the Browser, right-click Cylindrical:5 (Motor_Crank_Asm:1, Bearings:1). Click Properties.

34. On the dof 1 (R) tab, do the following:
   - Click Edit Imposed Motion
   - Select the Enable Imposed Motion check box
   - Under Driving, select Velocity
   - Click the Arrow. Select Constant Value
   - Enter 180 deg/s
   - Click OK

This will set the constraint to rotate about the Z axis 180 degree per each second of the simulation.
35. In the Simulation Player, do the following:

- In the Final Time window, enter 4 s. This simulates two full revolutions.
- Press TAB. In Images, the value should change to 400 to capture 400 frames during the simulation.
- Click Run or Replay the Simulation.

![Simulation Player interface](image)

With the simulation set to four seconds the Motor_Crank_Asm component will make two complete revolutions.

36. In the Simulation Player, click Construction Mode.

37. Zoom in on the assembly as shown.
38. **Save** your Assembly file.

39. On the **Load** panel, click **Force**.

40. Select the work point on the wiper blade, as shown.

41. Select the flat face of the wiper arm, as shown.
42. In the **Force** dialog box, do the following:

- For Magnitude, enter **5 N**.
- Click **Associative Load Direction**. This option causes the force to maintain its relationship to the location point.
- Click the `<< More` button.
- Select the **Display** check box. This displays the force direction when you run a simulation.
- Click **OK**.

43. Review the browser. The **Force** is added beneath the **External Loads** node.
44. **Save** the assembly file.

45. In the Simulation Player, click Run or Replay the Simulation. The force arrow is displayed on the wiper blade during the simulation, as shown.

![Image of wiper blade with force arrow](image)

The resistive force maintains the same direction on the wiper throughout the simulation. The resistive force of the wiper should always be opposite to the motion. You use the Input Grapher to change the direction of the force when the wiper changes direction.

46. In the Simulation Player, click Construction Mode.

47. In the Browser, right-click Force1 (Force on Brush:1). Click Edit.

48. In the Force dialog box, click the Arrow in the Magnitude edit box. Click Input Grapher.
49. In the **Magnitude** dialog box, click **Select Reference**.

50. In the **Select Reference** dialog box, do the following:

- Expand **Standard Joints**
- Expand **Revolution:1 (Bearings:1, Welded group1)**
- Expand **Velocities**
- Select the **V[1]** check box
- Click **OK**
51. In the **Magnitude** dialog box, under **Starting Point**, do the following:

- For \(X1\), enter \(-1\) deg/s.
- For \(Y1\), enter 5 N.
- For \(X2\), enter 1 deg/s.
- For \(Y2\), enter -5 N.

![Starting point diagram](image)

Between -1 deg/s velocity and 1 deg/s velocity, the resistant force reverses itself, avoiding a discontinuity or immediate switch in the resistive force.

52. In the **Magnitude** dialog box, the graph is updated to reflect the values in the **Starting Point** and **Ending Point** areas. The ramp area is shaded in the graph.

![Graph diagram](image)

To create a smooth transition in the resistive force, you can change the ramp from a linear ramp to a cubic ramp.
53. In the **Magnitude** dialog box, under **Property of the Selected Sector**, do the following:

- Expand the list of available laws.
- Select **Cubic Ramp**.
- Click **Replace the Current Law**.

54. In the **Magnitude** dialog box, review the image. The graph is updated to reflect the cubic ramp. Click **OK**.

55. In the **Force** dialog box, click **OK**.
56. In the Simulation Player, click Run or Replay the Simulation. The resistive force arrow direction reverses when the wiper reverses, as shown.

Because both wiper blades rub against the windshield, you need to repeat steps 39 through 56 for the other wiper. This is optional at this time and would be required to have a better simulation.

57. Save and close the file so you can review the simulation later if required.
1.2 Project 3B – Calculate the Drive Torque of the Wiper Assembly

Use Dynamic Simulation on the wiper assembly to calculate the driving torque required to move the wipers so that you can size the wiper motor. You then use the Output Grapher to plot the graph and review the results.
1. Open *WiperAssemblySDP.iam* Autodesk Inventor assembly file as shown below.

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

3. If you are prompted to view the tutorial, click **No** to close the window.

4. On the **Manage** panel, click **Simulation Settings**.
5. In the **Dynamic Simulation Settings** dialog box, select the **Color Mobile Groups** check box. This will assign a color scheme to each of the mobile groups in the assembly. This will provide a visual aid to better understand the mechanism. Click **OK**.

![Dynamic Simulation Settings](image)

6. In the **Browser**, under **Mobile Groups**, expand **Welded group:1**. Select the two assemblies as shown below that are grouped as a welded group.

![Browser](image)
7. Select Blue- Wall Paint- Glossy from the Color Override list.

Next, you impose a velocity on the Motor_Crank_Asm. The mechanism is fully defined and a force of 5 N has been added to each wiper blade assembly to simulate the drag of the blade on the windshield.

8. In the Browser, under Standard Joints, right-click Cylindrical:5 (Motor_Crank_Asm:1, Bearings:1). Click Properties.

9. On the dof 1 (R) tab, do the following:

- Click Edit Imposed Motion
- Select the Enable Imposed Motion check box
- Under Driving, click Velocity
- Click the Arrow next to the edit window. Select Constant Value from the list
- In the edit window, type 30 rpm
- Click OK
10. In the Simulation Player, do the following:

- For **Final Time**, type **2 s**.
- Press **TAB** on the keyboard. The **Images** value changes to **200**.
- Click **Run** or **Replay the Simulation**.

11. On the **Results** panel, click **Output Grapher**.
12. In the **Values** window, right-click in the **U_imposed** column. Click **Search Max.**
13. Review the grapher. From the results you can determine the following:

- In the **Values** window, the maximum driving torque of 4148 N mm is highlighted.
- In the graph, the time bar displays the corresponding time.
- The assembly is synchronized to the grapher, and the mechanism shows the position of the wipers at maximum drive torque.

14. On the **Output Grapher** toolbar, click **Save Simulation**.

15. For file name, type **Simulation_01**. Click **Save**. This will save the current results to an external file that can later be imported back into the **Output Grapher**.
16. On the **Simulation Player**, click **Construction Mode**.

17. In the **Dynamic Simulation** browser, under **Standard Joints**, right-click **Cylindrical:5** 
   (Motor_Crank_Asm:1, Bearings:1). Click **Properties**.

18. For velocity, type **360 deg/s** which is 60 rpm then click **OK**.

19. In the **Simulation Player**, click **Run or Replay the Simulation**.

20. Right-click in the **U_imposed[1]** column. Click **Curve Properties**.

21. Change color to **Red**. Click **OK** twice.
22. The new graph in the Output Grapher should look like the image below.


24. Open Simulation_01.iaa file you saved earlier in the project.

25. In the Output Grapher browser, expand Simulation_01.iaa | Cylindrical:5 (Motor_Crank_Asm:1, Bearings:1).
26. Under **Driving Force**, select the **U_imposed[1]** check box.

27. Review the two graphs.
28. On the Output Grapher toolbar, click Add Trace. You can also start the Trace command from the Results panel of the ribbon menu.

29. Select the Velocity Display Trace check box.

30. Select the point on the end of the wiper as shown then click Apply.
31. Select **Velocity** and then select the point at the opposite end of the wiper as shown then click **OK**.

32. On the **ViewCube**, click **Home**.

33. In the **Simulation Player**, click **Rewind** to the Beginning of the Simulation.
34. Click **Run** or **Replay Simulation**. The traces are displayed.

35. In the **Output Grapher** browser, expand the **Traces** folder.

36. Under **Trace:1 | Velocities**, select the **V** check box.
37. Review the graphs. The velocity of the point selected for the trace is displayed with the previous graphs.

38. Save the file so you can review the results as required.
1.3 Project 3C – Creating a Nonredundant Model

In this project, you create joints in a windshield wiper assembly. Some of the joints that you create contain redundancies. Although you can simulate a redundant model in the Dynamic Simulation environment, it is not advisable to do so. You repair the redundancies and test the joints to confirm the wiper subassembly movement.
1. In this section of the project, you create revolution joints between the wiper subassemblies and the bearings subassembly. Open the WiperAssemblyNRM.iam Autodesk Inventor assembly file as shown below.

2. **Zoom** in to the wiper assembly as shown. The subassemblies Brush_asm_left (1) and the Complete_wiper_left_asm (2) must move as a unit. In the next steps, you weld the two subassemblies together.
3. Click **Environments** tab | **Begin** panel | **Dynamic Simulation**.

4. If you are prompted to view the tutorial, click **No** to close the window.

5. In the **Browser**:
   - Select **Brush_asm_left:1** and **Complete_wiper_left_asm:1**
   - Right-click **Brush_asm_left:1**
   - Click **Weld Parts**

6. Review the **Browser**. A **Welded group:1** is added to the **Grounded** node. These two components will now move and act like one within the dynamic simulation environment.
7. Click on the View tab | Visibility panel | Object Visibility and select User Work Planes to turn on the visibility of user created workplanes in the parts.

8. Click Dynamic Simulation tab | Joint panel | Insert Joint to start creating a new joint. Select Revolution as the joint type.
9. In the **Graphics Window**, select the tubular component (1) to set the rotation axis, and the work plane (2) to set the origin for the coordinate system, as shown.

10. **Orbit** the assembly into a viewing position as shown below.
11. In the Insert Joint dialog box, under Component 2, click the Second Selection button.

![Insert Joint dialog box](image)

12. Click the circular edge as shown to specify the joint origin on the welded assembly.

![Circular edge selection](image)

13. Notice that the two coordinate systems are in general alignment. Click OK to create the joint. Now notice that the two welded group components moved into position as one single item.

14. On the ViewCube, click Home.
15. **Zoom** in to the assembly as shown. The *Complete_wiper_right_asm* (1) and the *Bearings* (2) subassemblies are currently in the proper orientation.

16. Now we will create a new joint between the right wiper and the bearings. Click **Insert Joint** on the Joint panel.

17. Select **Revolution** as the **Joint Type**. This will leave two DOF one rotational and transitional.

18. For **Component 1** select the outer circular edge of the bearing part as shown below.
19. For Component 2 select the cylindrical face as shown below on the right wiper.

20. Select the inner face of the wiper mounting arm as shown below as the Origin for Component 2.

21. To ensure the two coordinate systems are in alignment Flip Z Direction on Component 2.
22. Click **OK** to create the revolution joint. You will notice the wiper part position moved to align the two coordinate systems as shown below.

![Image of windshield wiper assembly]

23. To move the wiper back into the starting position right-click on the *Revolution:*2 standard joint in the **Browser** and select **Properties**.

24. In the **Revolution:**2 dialog complete the following:

- Click the **dof 1 ( R )** tab
- Select **Edit Initial Conditions**
- For the **Position** enter **56.73 deg**
- Click **OK**

25. On the **ViewCube**, click **Home**.
26. In this section of the project, you create revolution joints between the Inter_Crank and the wiper subassemblies. Using only revolution joints to build the four-bar linkage leads to redundancies. You use the **Repair Redundancies** tool to analyze and repair the problem.

- Press `ALT + J` to turn off the user work planes
- Zoom in to the wiper subassemblies, as shown

27. In the **Graphics Window**:

- Place your cursor over the `Complete_wiper_left_asm` as shown.
- Drag in the direction shown. This will move the component using the already created joints and is often a good method to verify your mechanism.
28. Position the wiper blade so that it is close to parallel to the other wiper, as shown below.

29. On the ViewCube, click the TOP-RIGHT-BACK corner.

30. Zoom in to the assembly as shown.
31. Drag the *Inter_Crank* to a new location, as shown. Now you create a revolution joint between the *Inter_Crank* and the *Complete_wiper_left_asm*.

32. On the **Joint** panel, click **Insert Joint**.

33. In the **Insert Joint** dialog box, verify that the **Revolution** joint is active.
34. Select the circular edge to set the coordinate system for the first component and its origin.

35. Review the orientation of the X axis (1) and the Z axis (2).

36. In the Insert Joint dialog box, under Component 2, click the second selection button.
37. Select the circular edge to set the coordinate system and the origin for the coordinate system. Be sure to select the exact circular edge show in the image below.

38. Review the orientation of the X axis (1) and the Z axis (2). The Z axes are pointing in the same direction, so the components maintain their current orientation. The X axes are pointing in nearly opposite directions, which will cause the Inter_Crank part to rotate so that its X axis matches the direction of the X axis on the Complete_wiper_left_asm subassembly. Next, you reverse the direction of the Complete_wiper_left_asm X axis to minimize the rotation.
39. In the **Insert Joint** dialog box, under **Component 2**, click **Flip X Axis**. The X axis for the Complete_wiper_left_asm subassembly reverses direction.

![Insert Joint dialog box](image)

40. Click **OK**. The joint is created. Now you create a revolution joint between the other end of the **Inter_Crank** component and the **Complete_wiper_right_asm** subassembly.

![Assembly view](image)

41. On the **Joint** panel, click **Insert Joint**.
42. Pan to the right as shown.

43. Select the circular edge shown in the image below to set the origin of the coordinate system for the first component. This should be the edge on the opposite side.

44. In the Insert Joint dialog box, under Component 2, click the second selection button.
45. In the Graphics Window, pan over to the end of the *Complete_wiper_right_asm*, as shown.

46. Select the circular edge to set the origin of the coordinate system for the second component.
47. Review the two coordinate system axes. With the Z axes (1) and the X axes (2) for the two components pointing in opposite directions, one of the components tries to flip to match the Z axis of the other component and rotate to match the X axis, causing the joint to fail. Next, you reverse the direction of the X and Z axes of Component 2 to create a successful joint.

48. In the Insert Joint dialog box:

- Under Component 2, click Flip Z Direction. The Z axis reverses direction.
- Under Component 2, click Flip X Direction. The X axis reverses direction.
- Click OK.
- The axes should resemble the image below.
49. A warning box is displayed stating that the mechanism is impossible to assemble.
   - Click OK to close the warning.

50. In the browser, review the Standard Joints group. The Revolution:4 (Inter_Crank:1, Complete_wiper_right_asm:1) joint is shown with an icon indicating that it has been suppressed. Next you will repair this redundancy in the joint of being over constrained.

51. Delete the Revolution:4 joint by right-clicking in the Browser and selecting Delete from the menu.

52. Now we will create a different type of joint. Click Insert Joint from the Joint panel and select Point-Line as the joint type.

53. Select the same geometry for Component 1 and Component 2 as before.
54. Flip the **Z axis** and **X axis** on **Component 2** to align the coordinate systems as shown below. Click **OK** to create the new joint.

55. Now you will notice the mechanism is constrained together as shown below.
56. On the ViewCube, click Home.

57. In the Graphics Window, place the cursor over the either wiper blade. Click and drag to the right as shown in the image below. The wiper subassemblies are linked together through the joints you have created and move together.
58. To review the mechanism and ensure everything is constrained correctly, click **Mechanism Status** in the **Joint** panel. Then click the << expand button. Notice that the **Degree of Redundancy** is now 0. Within this dialog, you can gain insights into issues and repair any constraint issues as needed for future projects.

![Mechanism Status and Redundancies dialog](image)

59. Click **OK** to close the dialog.
60. Notice in the Browser there are three Mobile Groups listed at the first level. These three groups of components move and operate as separate groups in the mechanism based upon the joint relationships.

61. Click **Simulation Settings** in the **Manage** panel. Check the **Color Mobile Groups** checkbox and click OK.
62. Notice how all of the components within each of the mobile groups are displayed as the same color now. This greatly helps understand the relationships and kinematics loops within your mechanism.

63. **Save** the assembly so you can review the items later if required.