Lab 13: Analysis and Reviewing Results

Objective
This lab will focus on running dynamic analyses. The results of these analyses will be reviewed through animating the system and plotting force results through the Virtual.Lab Graphing utility.

Lab 13 Agenda
1. Run a dynamic analysis on the assembled system.
2. Plot the force results reported in Revolute Joint.4.
3. Replace revolute joint Revolute.4 with a bushing force.
4. Replace cylindrical joint Cylindrical.4 with a bushing force.
5. Run a dynamic analysis and review the force results calculated for the bushing force elements.
6. Edit the damping coefficient used in the bushing force elements.
7. Run a dynamic analysis and review the force results in the bushing force elements.

Run a Dynamic Analysis on the Assembled System

Running a Dynamic Analysis
1. Double-click the Analysis Model branch of the Specification Tree to activate the Mechanism Design workbench.
2. Under the Analysis Model → AnalysisCase branch of the model Specification Tree, Double-click the Solution Set branch. This will open the Solution Set dialog box. Parameters applicable to each model solution can be set within each tab of this dialog box.
3. Under the System tab set the Analysis Type to DYNAMIC, set the Ending Time to 12s and the Print Interval to .01s. Click OK to close the dialog box.
4. Click the Compute Solution button. A dialog box containing solution progress information will appear, click the Close Window button once the solution is complete to close this dialog.
Lab Session 13: Analysis and Reviewing Results

Animating Analysis Results
1. Select the Solution Set branch of the model Specification Tree.
2. Click the Animate button. This will bring up the Animation toolbar.
3. Click the Parameters button. Set the Sampling Step field entry to 0.01s. Click the Parameters button again to close this dialog box.
4. Click on the black arrow to the far left of the Animation toolbar. This will change the symbol to that of a looped arrow, indicating that the player will continue to loop through the animation of the model until the player is stopped.
5. Click the Play Forward button from the Animation toolbar.
6. Click the Animate button once to close the Animation toolbar.

Review Force Results in Revolute Joint.4 and Cylindrical Joint.4
To plot the following results, a local axis system will be required on the swing body.

Inserting an Axis System
1. Under the Product1_ROOT Æ swing (swing.1) Æ swing Part Document branch of the Specification Tree, Expand the Open Body.1 branch of the tree by selecting on the plus symbol to the left of the branch. Right-click on the Point.3 branch of the tree, and select Hide/Show from the contextual menu.
2. Double-click the swing part document branch to activate the Wireframe and Surface Design workbench. From the Main Menu Bar, select Insert Æ Axis System. This will bring up the Axis System Definition dialog. Select Point.3 defined under the swing part document from either the Model Display or the Specification Tree.
3. Click OK to close the dialog box.

Plotting Analysis Results
1. From the Mechanism Design Workbench, Click the Open Motion Graph Window button. This will bring up the plot creation window.
2. On the left side of the Motion Graph Window, toggle on the Analysis Case option under the Available Results section. This allows for filtering the results based on an Analysis Case. From the window on the right side of the screen, select the first AnalysisCase branch.
3. The simulation time is automatically plotted on the X-axis. Toggle on the Y-axis selection at the top of the Define Plots section of the Motion Graph Window. Expand
the Revolute → Revolute Joint.4 branch of the Available Results tree. Select the fz1 branch of the Analysis Results Tree. The solution variable Revolute Joint.4:fz1 should now be listed under the Y-Variables half of the dialog box.

4. To finish the definition of the plot, highlight both the X and Y-axis selections one at a time using the left mouse button. Then Click the Create Plots button . A new branch should appear on the right side of the screen below the AnalysisCase branch of the tree.

5. Expand the Cylindrical → Cylindrical Joint.4 branch of the Available Results tree. Select the fz1 branch of the Analysis Results Tree. The solution variable Cylindrical Joint.4:fz1 should now be listed under the Y-Variables half of the dialog box.

6. To finish the definition of the plot, highlight both the X and Y-axis selections one at a time using the left mouse button. Then Click the Create Plots button . A new branch should appear on the right side of the screen below the AnalysisCase branch of the tree.

7. The forces we are interested in must be plotted in the local reference frame of Axis System.1 on the swing body. To specify this setting, double-click the SYSTEM:Time__Revolute Joint.4:fz1 branch of the Specification Tree.

8. Mark the check box to the left of the Reference Frame field. The selection window to the right of the field entry should now highlight, this allows you to pick a local reference frame from either the Model Display or Specification Tree. Under the swing body, select on the Axis System.4 branch of the Specification Tree. Click OK to close the dialog.

9. Repeat this procedure for the System:Time__Cylindrical Joint.4:fz1 branch of the Specification Tree.

10. To open the graphing window, Click the New Display button . This will bring up the New Function Display wizard. Accept the setting of Motion Display by clicking the Next button. Accept the XY Plot setting by clicking the Finish button in the wizard window.

11. To place the previously defined plot in the Motion Display window, Right-Click and then pick the Select Data option. Within the Default Data Selection window highlight the ***AnalysisCase*** → SYSTEM:Time_ Revolute Joint.4:fz1 option and then click anywhere in the white space of the Motion Display. The plot should appear as shown below:
As noted in the graph Legend, the red line represents the calculated force in the lateral direction, along the rotational axis of the revolute joint. The green line represents the calculated force along the rotational axis of the cylindrical joint. By definition there is no constraint applied along the rotational axis of a cylindrical joint. Therefore, the revolute joint must absorb all the lateral force that results from the system swinging side to side. In the next section of the lab, we will see the resulting forces when both the Cylindrical Joint.4 and Revolute Joint.4 are replaced by bushings.

**Replace Cylindrical Joint Cylindrical.4 with a Bushing Force**

**Deleting an Element**

1. Under the Analysis Model \(\rightarrow\) Joints branch of the Specification Tree select the **Cylindrical.4 branch**. Right-click, and Select **Disable** from the contextual menu.

**Showing Pre-defined Axis Systems**

1. Under the Product1_ROOT \(\rightarrow\) boom_cyl (boom_cyl.1) \(\rightarrow\) boom_cyl branch of the Specification Tree, expand the Axis System branch by selecting on the plus symbol
Lab Session 13: Analysis and Reviewing Results

Defining a Standard Bushing Element

The Standard Bushing element requires that an Axis System on each of the connecting bodies be selected. Further requirements include providing values for stiffness and damping coefficients.

1. Click the **Bushing** button. This will bring up the Standard Bushing definition dialog box.

2. Under the Product1_ROOT → swing (swing.1) → swing branch of the Specification Tree, select the **Axis System.2** branch.

3. Under the Product1_ROOT → boom_cyl (boom_cyl.1) → boom_cyl branch of the Specification Tree, select the **Axis System.1** branch.

4. There are several stiffness and damping values that must be defined for the Standard Bushing Element. Set the **Spring Radial x**, **Spring Radial y**, and **Spring Axial** field entries to 1e7N_m and the **Spring Conical x**, **Spring Conical y** field entries to 1e7Nxm_rad.

5. Set the **Damping Radial x**, **Damping Radial y**, and **Damping Axial** field entries to 10000kg_s, and the **Damping Conical x**, **Damping Conical y** field entries to 10000kg_s_rad.

6. Click **OK** to close the box.

Replace Revolute Joint Revolute.4 with a Bushing Force

Deleting an element

Under the Analysis Model → Joints branch of the Specification Tree select the **Revolute.4 branch**. Right-click, and select **Disable** from the contextual menu.

A Bushing Force Element is a six degree of freedom force. A force element prescribes compliant motion between the connecting bodies. The compliance of the element is defined based on three stiffness and three damping values in all six degrees of freedom. The Virtual.Lab Reference Manual provides a detailed description of each force coefficient. A Standard Bushing Force Element requires an axis system be
defined on each of the connecting bodies. These axis systems will have to be inserted on the boom2, boom_cyl, and swing bodies.

Showing a Pre-defined Axis System
1. Under the Product1_ROOT ➔ boom2 (boom2.1) ➔ boom2 branch of the Specification Tree, expand the Axis System branch by selecting on the plus symbol to the left of the branch. If Axis System.1 is hidden, right-click and select Hide/Show from the contextual menu.

Defining a Standard Bushing Element
The Standard Bushing element requires that an Axis System on each of the connecting bodies be selected. Further requirements include providing values for stiffness and damping coefficients.

1. Select the Bushing button. This will bring up the Standard Bushing dialog.
2. Under the Product1_ROOT ➔ swing (swing.1) ➔ swing branch of the Specification Tree, select the Axis System.3 branch.
3. Under the Product1_ROOT ➔ boom2 (boom2.1) ➔ boom2 branch of the Specification Tree, select the Axis System.1 branch.
7. There are several stiffness and damping values that must be defined for the Standard Bushing Element. Set the Spring Radial x, Spring Radial y, and Spring Axial field entries to 1e7N_m and the Spring Conical x, and Spring Conical y field entries to 1e7Nxm_rad.
8. Set the Damping Radial x, Damping Radial y, and Damping Axial field entries to 10000kg_s, and the Damping Conical x, and Damping Conical y field entries to 10000kg_s_rad.
4. Click OK to close the box.

Run a Dynamic Analysis and Review the Force Results Calculated for the Bushing Force Elements
1. Under the Virtual.Lab Main Menu, select Insert ➔ New Analysis Case.
2. Under the Analysis Model branch of the Specification Tree expand the new Analysis Case.1 branch by selecting on the plus symbol to the left of the branch. Double-click on the Solution Set branch. This will open the Solution Set dialog box.
3. Under the System Tab set the Ending Time to 12s and the Print Interval to 0.01s.
4. Select on the Dynamic Tab of the Solution Parameters Dialog box. Set the Reaction Forces field to true and the Method Integration field to BDF. The BDF method solves quicker and is more appropriate for use with Bushing force elements.

5. Click OK to close the dialog box.

6. Click the Compute Solution button. A display dialog box containing solution progress information will appear, select the Close Window button once the solution is complete.

### Animating Analysis Results

1. Under the Analysis Case.1 branch of the Specification Tree, Select the Solution Set branch.

2. Click the Animate button. This will bring up the Animation toolbar.

3. Click the Parameters button. Set the Sampling Step field entry to 0.01s. Click on the Parameters button again to close this dialog box.

4. Click on the black arrow to the far left of the Animation toolbar. This will change the symbol to that of a looped arrow, indicating that the player will continue to loop through the animation of the model until the player is stopped.

5. Click the Play Forward button on the Animation toolbar.

6. Click the Animate button once to close the Animation toolbar.

### Plotting Analysis Results
Lab Session 13: Analysis and Reviewing Results

1. From the Mechanism Design Workbench, Click the Open Motion Graph Window button. This will bring up the plot creation window.

2. The two plot branches previously defined for Cylindrical and Revolute Joint.4 are no longer valid as these elements have been disabled in the solution. These branches of the tree can be deleted by first highlighting the branch, Right-Click, and select Delete from the resulting menu.

3. On the left side of the Motion Graph Window, toggle on the Analysis Case option under the Available Results section. This allows for filtering the results based on an Analysis Case. From the window on the right side of the screen, select the first AnalysisCase branch.

4. The simulation time is automatically plotted on the X-axis. Toggle on the Y-axis selection at the top of the Define Plots section of the Motion Graph Window. Expand the Bushing → Standard Bushing.1 branch of the Available Results tree. Select the fz1 branch of the Available Results Tree. The solution variable Standard Bushing.1:fz1 should now be listed under the Y-Variables half of the dialog box.

5. To finish the definition of the plot, highlight both the X and Y-axis selections one at a time using the left mouse button. Then Click the Create Plots button. A new branch should appear on the right side of the screen below the AnalysisCase branch of the tree.

6. Expand the Bushing → Standard Bushing.2 branch of the Available Results tree. Select the fz1 branch of the Available Results Tree. The solution variable Standard Bushing.2:fz1 should now be listed under the Y-Variables half of the dialog box.

7. To finish the definition of the plot, highlight both the X and Y-axis selections one at a time using the left mouse button. Then Click the Create Plots button. A new branch should appear on the right side of the screen below the AnalysisCase branch of the tree.

8. To place the previously defined plot in the Motion Display window, Right-Click and then pick the Select Data option. Within the Default Data Selection window highlight the ***AnalysisCase*** → SYSTEM:Time_ Standard Bushing.1:fz1 option and then click anywhere in the white space of the Motion Display. Within the Default Data Selection window highlight the ***AnalysisCase*** → SYSTEM:Time_ Standard Bushing.2:fz1 option and then click anywhere in the white space of the Motion Display.

Shown in the figure below is an image of the bushing force (N) vs simulation time (s) plots:
The legend indicates that the red plot is the Force (N) calculated in the lateral direction for Standard Bushing.1, and the green plot is the Force (N) calculated in the lateral direction for Standard Bushing.2.

As indicated by the plots the force that was previously being exerted entirely within the Revolute Joint.4, has now been evenly distributed between two equal bushing forces. The force calculated for each bushing is now equal but opposite in sign. This is one method of distributing force within a system.