Lab 11.1: Joint Creation Lab

Objective

This lab will introduce students to joint creation. The student will become more familiar with both utilizing geometry features and inserting axis systems to define the joints necessary to perform a dynamic analysis on the model shown below.

Lab 11.1 Agenda

1. Setting a body as fixed to ground.
2. Define a revolute joint between the swing2 and bucket bodies.
3. Define a revolute joint between the dumplink and swing2 bodies.
4. Define a spherical joint between the dump_piston and dumplink bodies.
5. Define a spherical joint between the dumplink2 and dumplink bodies.
6. Define a cylindrical joint between the dumplink2 and bucket bodies.
7. Define a translational joint between the dump_cyl and dump_piston bodies.
8. Define a cylindrical joint between the dump_cyl and swing2 bodies.
9. Run a kinematic analysis to animate the system behavior due to a harmonic driver.
10. Run an inverse dynamic analysis to evaluate the force results for the driver.

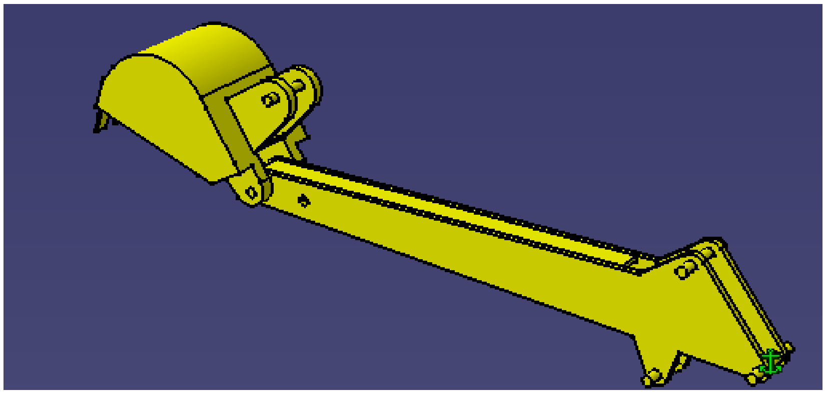
Setting a Body as Fixed to Ground

1. Under the Analysis Model → Bodies branch of the Virtual.Lab Specification Tree, double-click the **swing2** branch.
2. The Body Dialog Box will appear on the screen. Select on the arrow to the right of the *Fixed to Ground* Field, select **`true`** from the list → Click **OK** to close the box.

Define a Revolute Joint Between the swing2 and bucket Bodies

In Virtual.Lab Motion the definition of a revolute joint requires that two axes and two planes perpendicular to the selected axes be specified. By definition the two connecting bodies can only rotate about the specified axes relative to one another. It works best if the perpendicular planes selected are planes that should lie coincident in the model. This first revolute joint will be fully defined utilizing geometry from the Model Display.

An assembled view of the swing2 and bucket bodies is shown here.



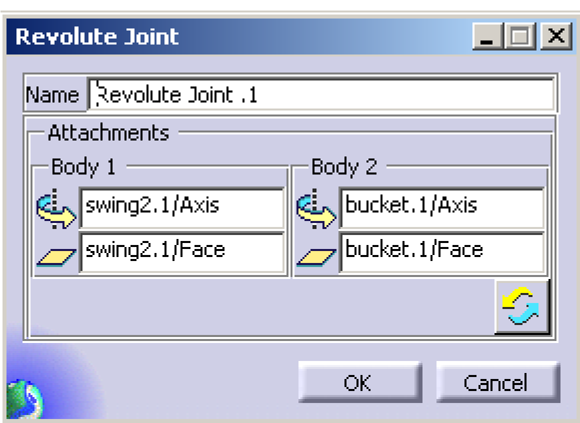
For visualization purposes, hide the four parts associated with the bodies not involved in this section of the lab:

1. Under the Product1_ROOT Product Document, right-click on the dump_cyl (dump_cyl.1) branch of the model Specification Tree, select **Hide/Show** from the contextual menu.
2. Repeat this procedure for the dump_piston (dump_piston.1), dumplink2 (dumplink2.1), and dumplink (dumplink.1) branches.

The body geometry that will be connected by the revolute joint will be a solid cylinder at the back of the bucket body and a solid cylinder at the front of the swing2 body. Both of these cylinders are symmetric about a centerline axis. The centerline axes can be used to define the revolute joint.

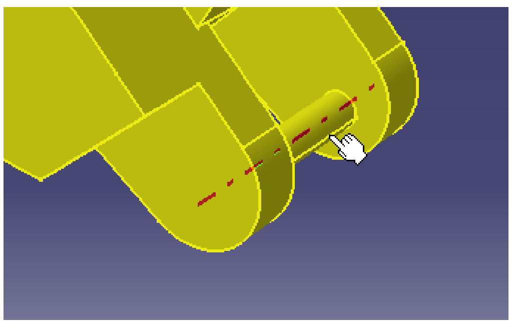
Creating a Revolute Joint Between the swing2 and bucket Bodies

1. Select on the **black arrow** to the right of the **Create Bracket Joint** button.
2. Select the **Revolute Joint** button.
3. The Revolute Joint Definition dialog box will appear on the screen.

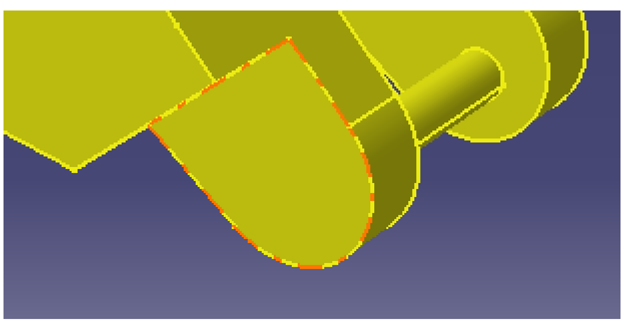


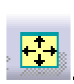
The Attachments section of the Dialog shows that two bodies are required to complete the definition of the revolute joint. There are also pictures next to the Field Entries below each body selection. These pictures are intended as references of the required geometrical entry. In the case of the revolute joint, the pictures are of an axis and a plane. Prior to selecting any geometry from the screen, the field entries below each body in the Attachments portion of the Revolute Joint Dialog shows `No Selection`. Once a pick is made from the Model Display, these field entries will change. The geometry selections must be made in a prescribed order. Highlighting within the Revolute Joint dialog box is interactive to aid in the selection process.

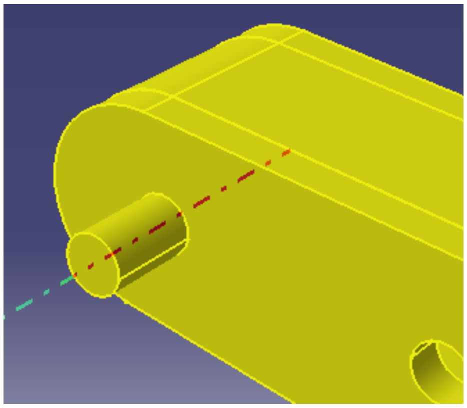
1. Skate the mouse cursor over the solid cylinder at the tail of the bucket body. As the mouse skates over the geometry a centerline axis will show temporarily on the screen. Select once when the centerline axis is highlighted on the screen. This becomes the Body 1 axis selection within the Revolute Joint Dialog.



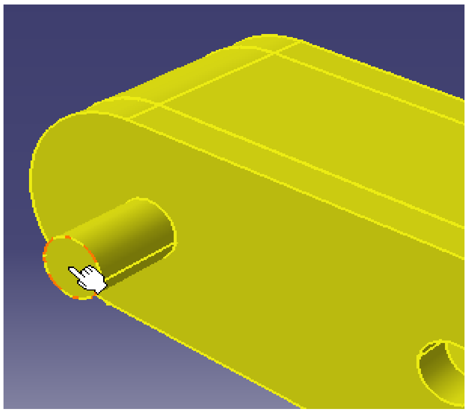
2. Skate the mouse cursor over the plane shown below, select once when this plane highlights on the Model Display. This plane becomes the Body 1 perpendicular plane selection within the Revolute Joint Dialog.



3. Select the **Fit All In** Button .
4. Zoom in on the front section of the swing2 geometry.
5. Skate the mouse over the solid cylindrical shaft on the front section of the swing2 body. Select once when the centerline axis is highlighted on the screen. This becomes the Body 2 axis selection within the Revolute Joint Dialog.



6. Skate the mouse over the plane shown below, select once when the circular plane highlights on the Model Display. This plane becomes the Body 2 perpendicular plane selection within the Revolute Joint Dialog.

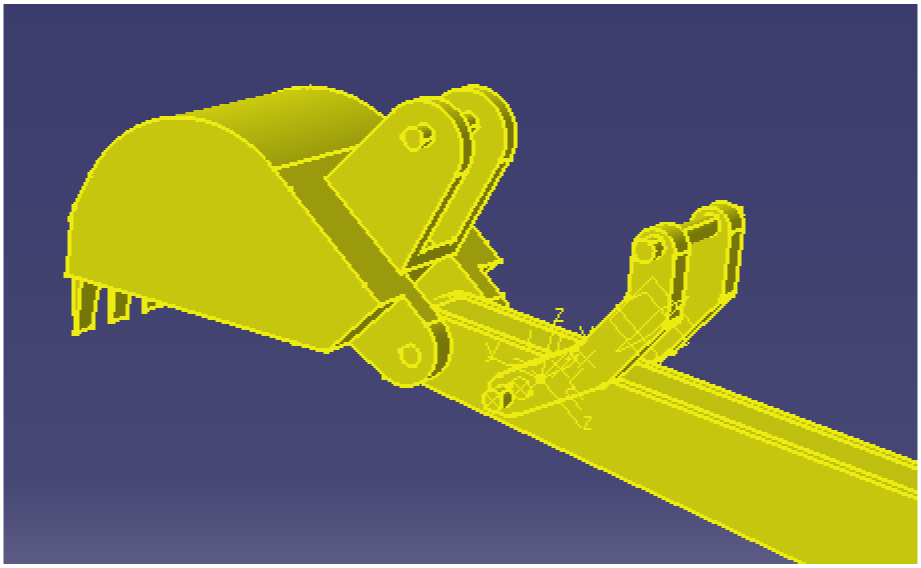


7. The bodies will snap into place immediately following the last plane selection to show the positioning of the bodies due to the joint defintion. Selecting the **Fit All In** Button can help to visualize the assembled position. If the assembled position is incorrect, select Cancel, and try the selection process again.
8. If everything looks fine, select **OK** to close the Dialog box.

Define a Revolute Joint Between the dumplink and swing2 Bodies

As mentioned in the previous section, there is more than one way to make selections for defining a joint connection. In this section we will define the revolute joint connection through the use of an axis system.

An assembled view of the dumplink body added to the previous swing2/bucket assembly is shown here.



For visualization purposes, bring dumplink (dumplink.1) into the visual space with the swing2 and bucket bodies.

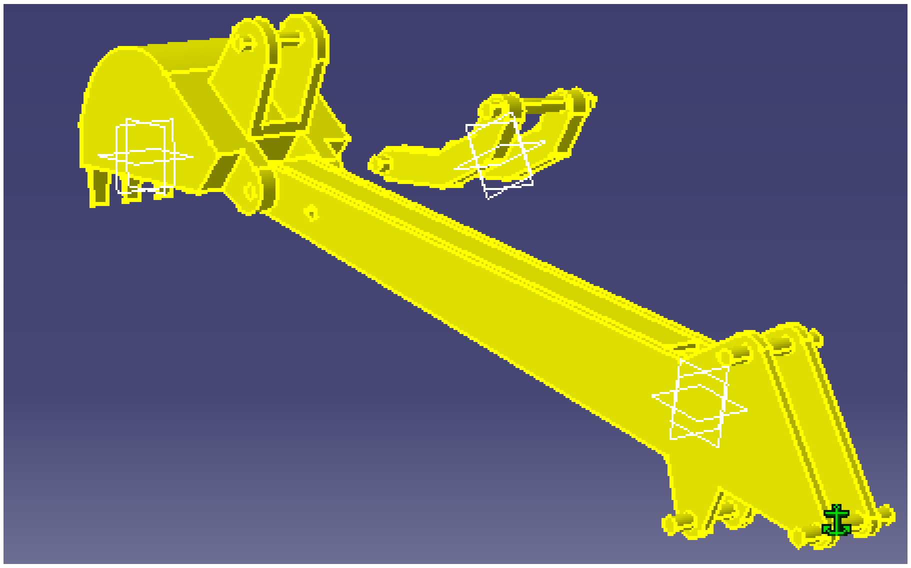
Under the Product1_ROOT branch of the Specification Tree, right-click on the dumplink (dumplink.1) branch, select **Hide/Show** from the menu.

The geometry that will be connected by the revolute joint will be a solid cylinder on the dumplink body and the cylindrical hole (Pocket) on the swing2 body. Both geometry entities are symmetric about a centerline axis. As in the last section these centerline axes can be used to define the revolute joint. However, in this section we will introduce creating a revolute joint by inserting an axis system on each part.

Rotating the dumplink Part Using the Modeling Compass

1. Double-click the **Product1_ROOT** branch of the Specification Tree, this will activate the Assembly Design Workbench.
2. The dumplink part needs to be moved into an approximate assembled position. To do this, first place the cursor over the Compass in the top right hand corner, and right-click. Select **Snap Automatically to Selected Object** from the contextual menu.
3. Under the Product1_ROOT → dumplink (dumplink.1) → dumplink branch of the Specification Tree select the **xy plane** branch.
4. The Compass should move to the dumplink part origin, and change to a green color indicating that it has been assigned.

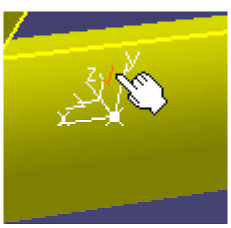
5. Move the dumplink part relative to the swing2/bucket assembly by selecting and grabbing Compass axes individually.



6. Refresh the Compass by placing the cursor over the red box, and then dragging the Compass to an empty space in the Model Display.
7. Once the Compass has been refreshed, place the cursor over the Compass and right-click. Deselect **Snap Automatically to Selected Object** from the menu.

Defining the Revolute Joint

1. Double-click the **Analysis Model** branch of the model Specification Tree to activate the Mechanism Design workbench.
2. When making selections for the Revolute Joint zoom in on each Axis System.
3. Select the **Revolute Joint** button from the workbench.
4. Select the **x-axis** of Axis System.1 on the dumplink body.
5. Select the **yz-plane** of Axis System.1 on the dumplink body.



6. Select the **x-axis** of the Axis System on the swing2 body.
7. Select the **yz-plane** of the Axis System on the swing2 body.
8. Select **OK** to close the Revolute Definition Dialog Box.

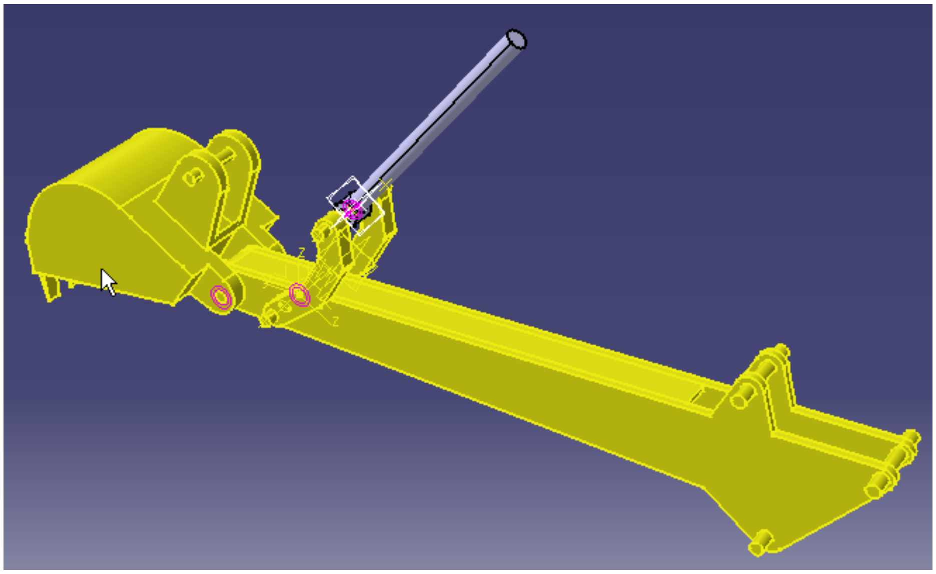
The dumplink body will snap into place in connection with the swing2 body.

Save all documents before proceeding.

Defining a Spherical Joint Between the dump_piston and dumplink Bodies

In Virtual.Lab Motion the definition of a spherical joint requires the selection of two points. These points define the location of the spherical joint. No axis or plane selection is needed, the spherical joint allows for rotation about any axis.

An assembled view of the dumplink and dump_piston bodies is shown below:



For visualization purposes, bring dump_piston(dump_piston.1) into the visual space with the swing2, dumplink and bucket bodies.

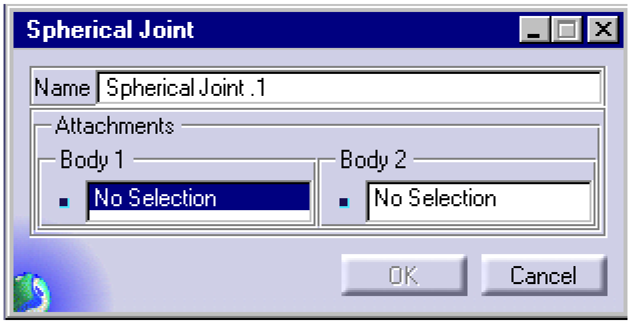
Under the Product1_ROOT branch of the Specification Tree, right-click on the dump_piston (dump_piston.1) branch, select **Hide/Show** from the contextual menu.

Joint connections may be easier if the bodies involved are moved into approximate assembled positions prior to assembly. The figure above shows the dump_piston in its assembled position. Using the Compass as described previously, move the dump_piston to a position slightly above the assembled position shown.

Defining a Spherical Joint

1. Expand the Joint Definition Toolbar by selecting on the black arrow to the right of the Joint Button shown on the Mechanism Design Workbench.

2. Select the **Spherical Joint** Button  from the Joint Definition Toolbar. This will bring up the Spherical Joint definition dialog.



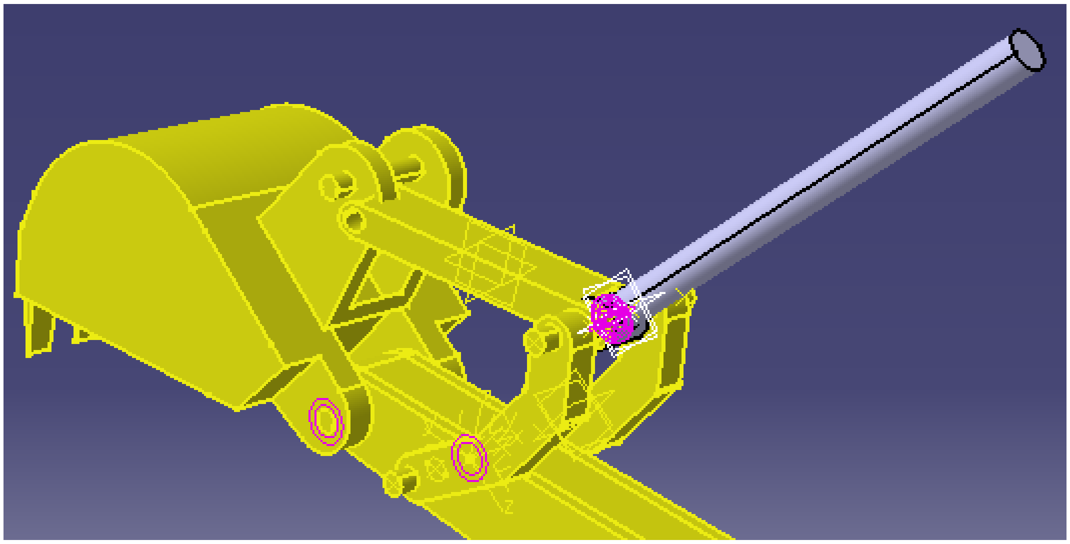
The Attachments section of the dialog shows that two bodies are required to complete the definition of the spherical joint. There are also pictures next to the field entries below each body selection, these pictures are intended as references of the required geometrical entries. In the case of the spherical joint, the pictures are points. To complete the definition of the spherical joint, a point must be selected off of each connecting body, for this spherical joint these points will be Point6 on the dumplink body and Point3 on the dump_piston body.

1. Under the dumplink body select on **Point6**, either from the Model Display or the Point6 branch of the model Specification Tree.

2. Under the dump_piston body select on **Point3**, either from the Model Display or the Point3 branch of the model Specification Tree. Select **OK** to close the dialog box.

Defining a Spherical Joint Between the dumplink2 and dumplink Bodies

An assembled view of the dumplink and dumplink2 bodies is shown here.



For visualization purposes, bring dumplink2 (dumplink2.1) into the visual space with the swing2, dumplink and bucket bodies.

Under the Product1_ROOT branch of the Specification Tree, right-click on the dumplink2 (dumplink2.1). Select **Hide/Show** from the menu.

As with the previous joint connections, the following process works best if the dumplink2 body is moved into the approximate assembled position prior to defining the spherical joint.

Defining a Spherical Joint

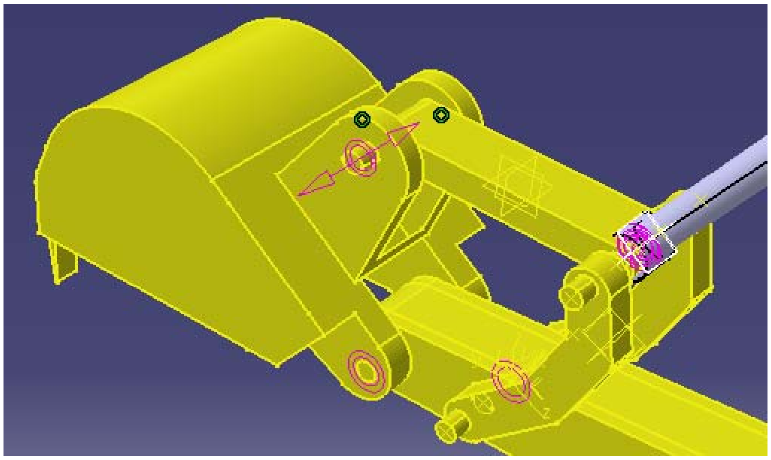
1. Expand the Joint Definition toolbar by selecting on the black arrow to the right of the Joint button shown on the Mechanism Design workbench.
2. Select the **Spherical Joint** button from the Joint Definition toolbar. This will bring up the Spherical Joint Definition dialog.
3. Select **Point3** created under the dumplink2 body either from the Model Display or by selecting the Point3 branch of the Specification Tree. Select **Point6** created under the dumplink body; this selection may be easier if done from the Specification Tree. Select **OK** to close the dialog box.

The dumplink2 body may assemble at an angle to the dumplink body. This is acceptable at this point in the lab session. Further constraints will be added to the model throughout the remainder of the lab that will cause the dumplink2 body to fall into its proper assembled position.

Defining a Cylindrical Joint Between the dumplink2 and bucket Bodies

In Virtual.Lab Motion the definition of a cylindrical joint requires the selection of two axes. These axes define the rotational axis that will exist between the connecting bodies. By definition the two connected bodies can also translate relative to one another along the same axis that was specified for rotation.

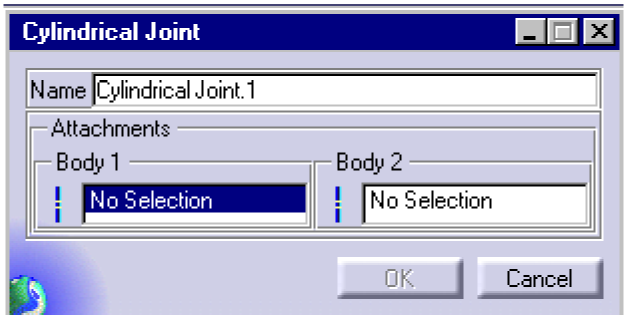
An assembled view of the dumplink2 and bucket bodies is shown below.



Defining a Cylindrical Joint

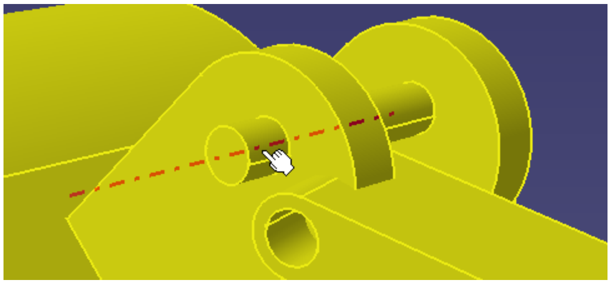
1. Expand the Joint Definition toolbar by selecting on the **black arrow** to the right of the Joint Button shown on the Mechanism Design workbench.

2. Select the **Cylindrical Joint** button from the Joint Definition toolbar. This will bring up the Cylindrical Joint definition dialog.

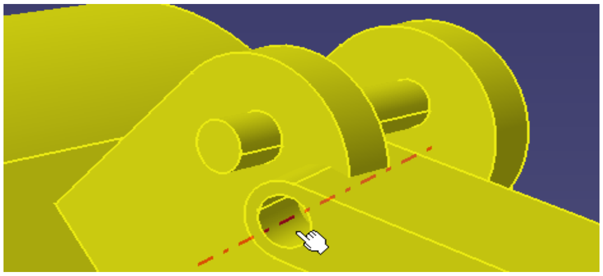


The Attachment section of the dialog shows that two bodies are required to complete the definition of the cylindrical joint. There are also pictures next to the field entries below each body selection. In the case of the cylindrical joint, the picture is that of an axis. Prior to selecting an axis from the screen the field entry below, each body in the Attachment portion of the Cylindrical Joint dialog shows `No Selection`. Once a pick is made from the Model Display, these field entries will change.

1. Skate the mouse cursor over the solid cylinder on the bucket body. As the mouse skates over the geometry an axis will show temporarily on the screen. Select once when the centerline axis is highlighted on the screen. This becomes the Body 1 axis selection within the Cylindrical Joint Definition dialog.



2. Skate the mouse over the cylindrical hole in the dumplink2. Select once when the centerline axis is highlighted on the screen. This becomes the Body 2 axis selection within the Cylindrical Joint Definition Dialog, select **OK** to close the dialog box.

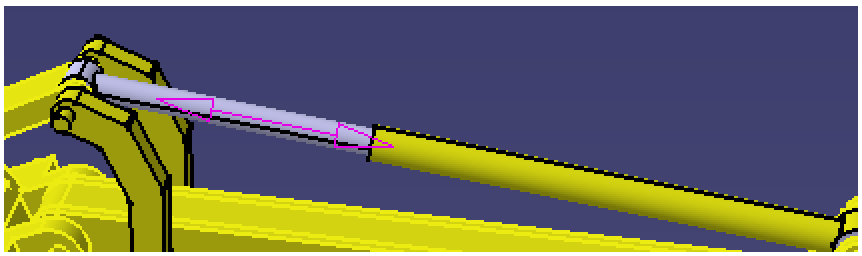


The bucket and dumplink2 bodies should now be assembled. Notice that the dumplink2 body is now in it's proper assembled position.

Defining a Translational Joint Between the dump_piston and dump_cyl Bodies

In Virtual.Lab Motion the definition of a translational joint requires two axes and two planes parallel to the selected axes be specified. By definition the two connecting bodies can only translate along the specified axes relative to one another.

An assembled view of the dump_piston and dump_cyl bodies is shown here.



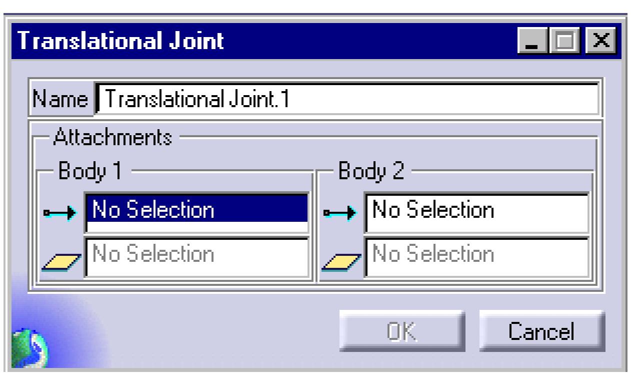
For visualization purposes, bring dump_cyl (dump_cyl.1) into the visual space rest of the assembled bodies.

Under the Product1_ROOT branch of the Specification Tree, right-click on the dump_cyl (dump_cyl.1), select **Hide/Show** from the menu.

For convenience during the model assembly, utilize the manual Compass to move the dump_cyl body into its approximate assembled position before defining the following translational joint.

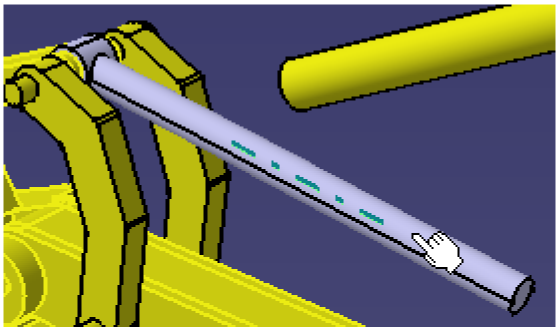
Defining a Translational Joint

1. Double-click the Analysis Model branch of the Specification Tree to activate the Mechanism Design workbench.
2. Expand the Joint Definition Toolbar by selecting on the black arrow to the right of the Joint button shown on the Mechanism Design Workbench.
3. Select the **Translational Joint**  button from the Joint Definition toolbar. This will bring up the Translational Joint Definition dialog.

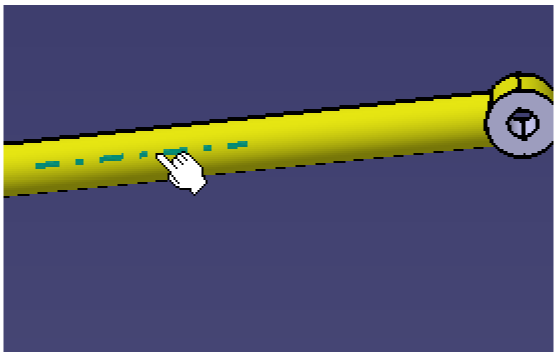


The Attachments section of the Dialog shows that two bodies are required to complete the definition of the translational joint. In the case of the translational joint, the pictures shown under Bodies 1 and 2 are of an axis and a plane. The geometry selections required to define a translational joint must be made in a prescribed order, the highlighting within the Translational Joint Definition dialog box is interactive to aid in the selection process.

1. Skate the mouse cursor over the longer solid cylinder defined under the dump_piston body. As the mouse skates over the geometry an axis will show temporarily on the screen. Select once when the centerline axis is highlighted on the screen. This becomes the Body 1 axis selection within the Translational Joint Definition Dialog.



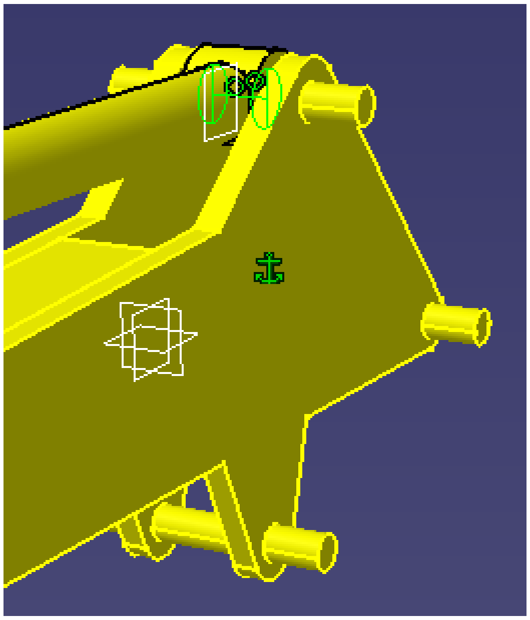
2. Under the Product1_ROOT → dump_piston (dump_piston.1) branch of the Specification Tree expand the dump_piston branch of the tree by selecting on the **plus symbol** to the left of the branch. Select the **yz_plane** branch. This selection becomes the Body 1 plane selection within the Translational Joint Definition dialog.
3. Skate the mouse cursor over the longer cylindrical tube defined under the dump_cyl body. Select once when the centerline axis is highlighted on the screen. This becomes the Body 2 axis selection within the Translational Joint Definition dialog.



4. Under the Product1_ROOT → dump_cyl (dump_cyl.1) branch of the Specification Tree expand the dump_cyl branch of the tree by selecting on the **plus symbol** to the left of the branch. Select the **xy_plane** branch. This selection becomes the Body 2 plane selection within the Translational Joint Definition Dialog.

Defining a Cylindrical Joint Between the dump_cyl and swing2 Bodies

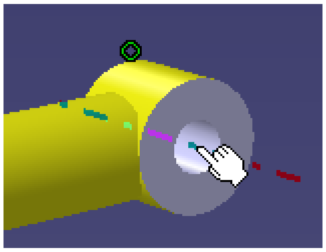
An assembled view of the dump_cyl and swing2 bodies is shown below.



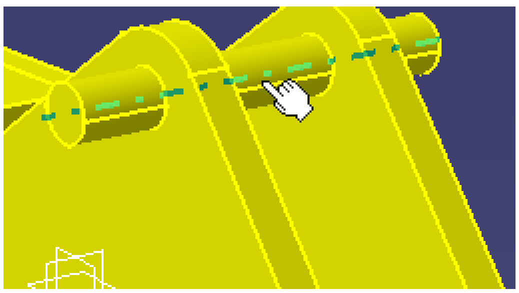
The geometry that will be connected by the cylindrical joint will be the shorter tube on the dump_cyl body and a solid cylinder on the swing2 body. Both geometries are symmetric about a centerline axis. This centerline axis will be used to define the cylindrical joint.

Defining a Cylindrical Joint

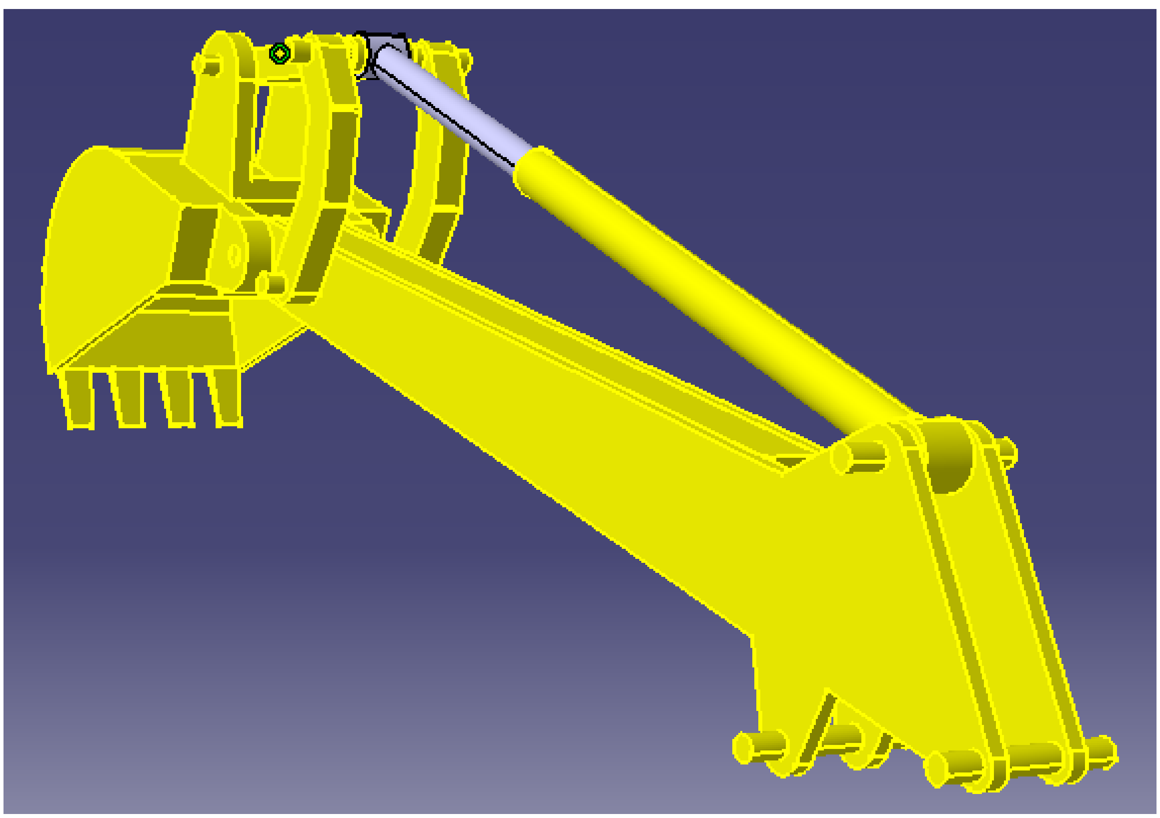
1. Expand the Joint Definition Toolbar by selecting on the **black arrow** to the right of the Joint button shown on the Mechanism Design Workbench.
2. Select the **Cylindrical Joint** button. The Cylindrical Joint Definition dialog will appear on the screen.
3. In the Model Display, zoom in on Pad1 of the dump_cyl body.
4. Skate the mouse cursor over the tube geometry defined as Pad1 of the dump_cyl body. As the mouse skates over the geometry an axis will show temporarily on the screen. Select once when the centerline axis is highlighted on the screen. This becomes the Body 1 axis selection within the Cylindrical Joint Definition dialog.



5. Zoom in on the back portion of the swing2 geometry.
6. Skate the mouse over the top solid cylinder on the back portion of the swing2 body. Select once when the centerline axis is highlighted on the screen. This becomes the Body 2 axis selection within the Cylindrical Joint Definition dialog, select **OK** to close the dialog box.



Congratulations the bodies are now completely assembled!

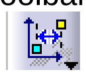


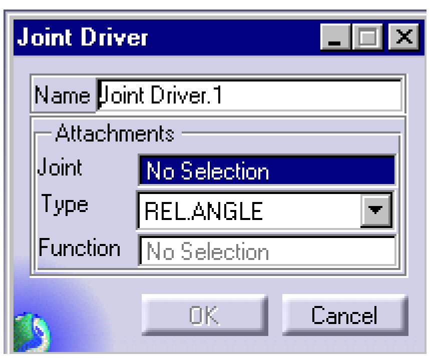
Run a Kinematic Analysis to Evaluate the System Behavior With a Harmonic Driver

Now that the bodies in this model have been fully assembled, it is possible to run a simple analysis to view the motion of the model due to a harmonic driver.

The harmonic driver will be added to the translation joint defined between the dump_cyl and dump_piston bodies. This driver will define the translation motion of the two bodies connected by Translational Joint.1 relative to one another. The harmonic motion will move the bucket up and down.

Defining a Joint Driver

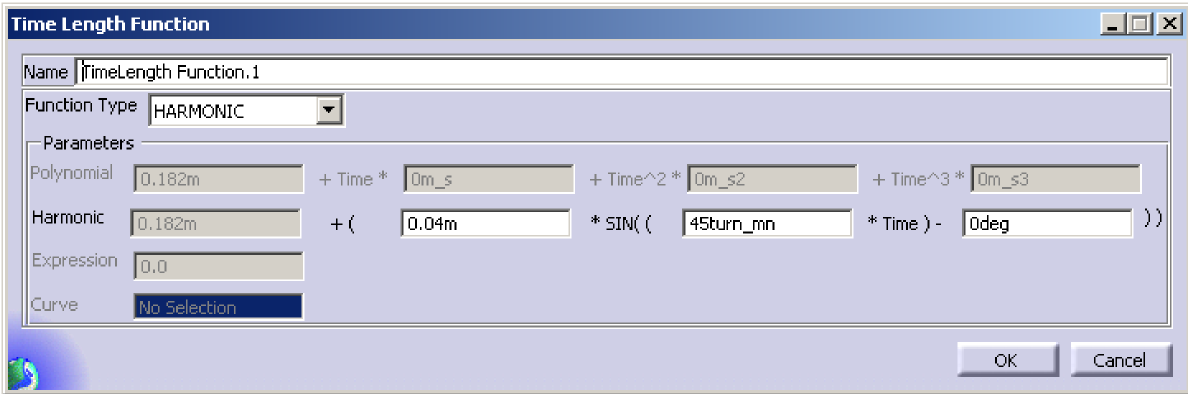
1. Double-click the Analysis Model branch of the Specification Tree to activate the Mechanism Design Workbench.
2. Expand the Constraint Definition Toolbar by selecting once on the black arrow to the right of the **Difference Constraint**  button.
3. Select the **Joint Driver**  button. This will bring up the Joint Driver Definition dialog.



   A Joint Driver requires that a Joint and a Function be specified in the definition dialog. Once a joint has been specified, the driver Types list will update to show only those types applicable to the specified joint.

4. To start defining the Joint Driver, select on the Tranlational Joint.1 icon in the Model Display window, or select on the **Translational Joint.1** branch of the Specification Tree. This will become the *Joint 1* field entry setting in the Joint Definition dialog.

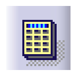
5. In most cases a new function will be defined for a new Joint Driver. To define a new function, right-click in the *Function* field of the Joint Definition dialog. Select **New** from the contextual menu. This will place a new branch in the Specification Tree under the Analysis Model → Data branch. This new branch will be labeled TimeLength Function.1. TimeLength Function.1 becomes the *Function* setting in the Joint Driver Definition Dialog. The TimeLength Function.1 Definition Dialog must be completed before the Joint Driver Definition Dialog can be completed.



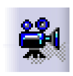
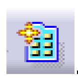
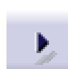
6. Within the Function Definition Dialog select on the arrow to the right of the *Function Type* field entry. Select **HARMONIC** from the list.
7. Once a *Function Type* has been selected, the TimeLength Function.1 Parameters fields will highlight. To complete the definition of the function, four parameters must be defined in the entry blanks. For this model, the first three parameters of the harmonic function will be needed. The first parameter of the function is calculated by default. The first parameter of the harmonic function represents the initial offset of the connecting points on the two bodies specified in Translational Joint.1.
8. In order to specify that the translational joint connection bodies should oscillate an equal distance of .04m in each direction, set parameter two as `.04m`, and parameter three as `45turn_mn`. Parameter two indicates the range of oscillation, and parameter three indicates the rate.
9. Select **OK** to close the dialog box.
10. The type of driver should already be specified as *REL.TRANS.*
11. Select **OK** to close the dialog box.

A Kinematic Analysis requires that the model have no degrees of freedom. Prior to the definition of a harmonic driver on the Translational Joint.1, our model had 1 DOF. A Kinematic Analysis solves a system of zero degrees of freedom to obtain position, velocity, and acceleration for all bodies. A Kinematic Analysis is a useful tool for checking to ensure that all degrees of freedom in the model are accounted for, either by a joint or a driver element. All forces due to mass or imposed by a force element are ignored in a Kinematic Analysis.

Running a Kinematic Analysis

1. Double-click the Analysis Model branch of the Specification Tree to activate the Mechanism Design Workbench.
2. Under the Analysis Model → Analysis Case branch of the model Specification Tree, double-click the **Solution Set** branch. This will open the Solution Parameters window. Parameters applicable to each model solution can be set within each tab of this window.
3. Under the *System* tab, set the *Anaysis Type* to **KINEMATIC**. Select **OK** to close the dialog box.
4. Select the **Compute Solution**  button. A display window containing solution progress information will appear, click on the **Close Window** button once the solution completes successfully.

Animating Analysis Results

1. Select the **Solution Set** branch of the model Specification Tree.
2. Select the **Animate**  button. This will bring up the Animation toolbar.
3. Select the **Parameters** Button .
4. Set the *Sampling Step* field entry to `.05s`. Select the **Parameters** button again to close this window.
5. Select 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.
6. Select the **Play Forward** Button  from the Animation Toolbar.
7. Select the **Animate** Button once to close the Animation Toolbar.

The animation should show the bucket body lowering, raising, and then returning to its initial position.

An Inverse Dynamic Analysis is very similar to a Kinematic Analysis except that the driving joint reaction forces are calculated.

Running a Inverse Dynamic Analysis

1. Under the Analysis Model → AnalysisCase branch of the model Specification Tree, double-click the **Solution Set** branch.
2. Under the *System* tab, set the *Analysis Type* to **INVERSE**.
3. Select **OK** to close the box.

4. Select the **Compute Solution** Button . A display window containing solution progress information will appear, click on the **Close Window** button once the solution is complete.

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 Driver → Joint Driver.1 branch of the Available Results tree, and highlight the fx1 branch to place the setting in the Y-axis selection window.
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. 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.
6. 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_Joint Driver.1:fx1 and then click anywhere in the white space of the Motion Display.

Examine the force plot to determine if the reported force is reasonable. To avoid trouble shooting a large complex model later in the development stages, it is recommended that a model be built and tested gradually. Close the Graphing Display by selecting the X in the top right hand corner of the window. Select **Window → Tile Horizontally** to return the Model Display window to full screen size.

This completes Lab11.1.

All future analysis with this model will be Dynamic.