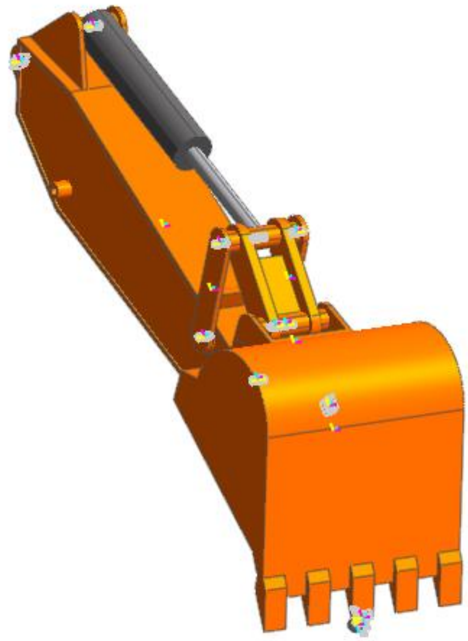
Dipper Stick with Bucket Tutorial (Professional)



FunctionBay

Edition Note

This document describes the release information of **RecurDyn V9R4**.

Table of Contents

Chapter

1

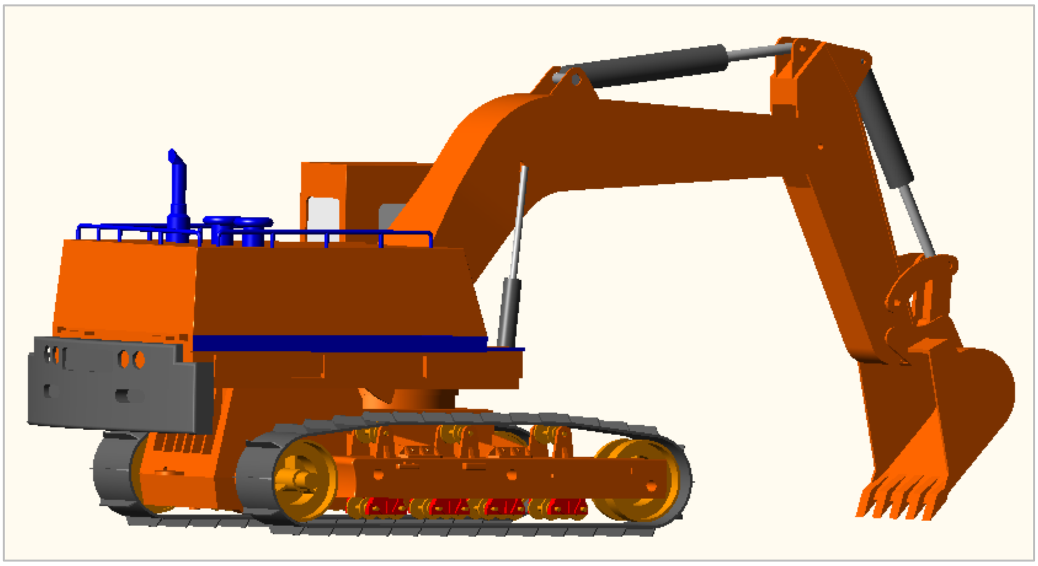
Getting Started

Objective

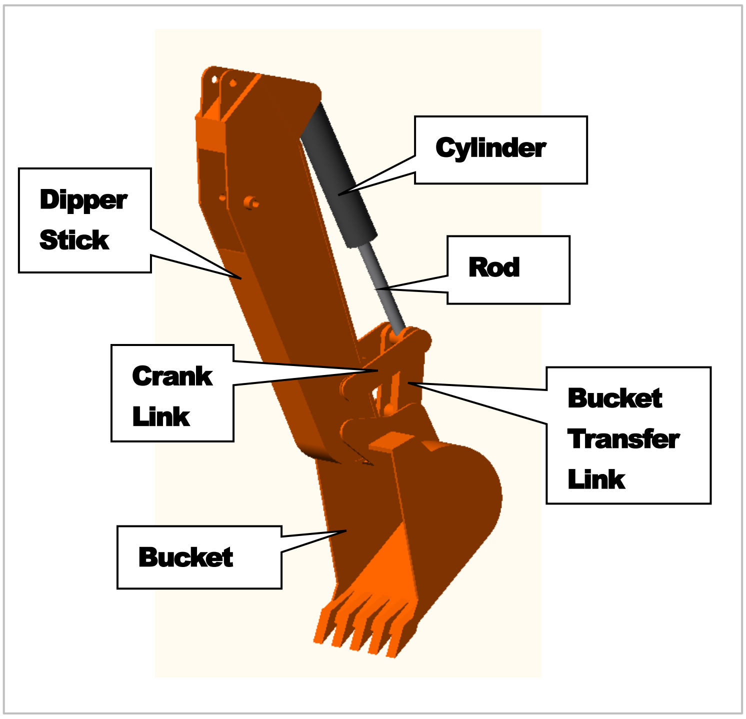
This tutorial familiarizes you with the methods used to design dynamic systems. This includes parametric modeling using parametric points and values at both the main and subsystem levels, parametric bodies, and expressions. It also includes setting up design variables, performance indexes, and a design study. This tutorial concludes with instructions for simulating the best candidate designs in batch mode and plotting their results simultaneously.

Model Used

The model used in this tutorial comes from an excavator. You will look specifically at the design of the four-bar linkage connecting the hydraulic cylinder and the bucket to the dipper stick as shown in the following figure.



For ease of working with the model and to decrease the simulation time, we only include the dipper stick and the bodies of the bucket linkage in this tutorial. The next figure shows the simplified model and the names of the major bodies and links.



The objective of the design study is to decrease the amount of power required to drive the bucket through a dig and return motion, while at the same time maintaining a high range of motion for the bucket. The design variables are the length of the crank link and the position of the bucket joint where it connects to the bucket transfer link. While varying these two design variables the initial length of the hydraulic cylinder should be unaltered so that the same hydraulic cylinder can be used in the modified design. This makes the comparison between designs more objective because the same actuator is used consistently.

The equations to enforce this constraint are beyond the scope of this tutorial, so they are provided for you in the base model. In fact, almost all of the parametric points and values are provided in the base model, and only a few were left out so that this tutorial could be completed in a reasonable amount of time while still showing the methods used in creating this model. If you are interested, you should take some time after completing this tutorial to go back through the model and see what was provided for you. This will be easier to understand after having seen the steps used to create the additional components of the model as presented in this tutorial.

Audience

This tutorial is intended for experienced users of RecurDyn. All new tasks are explained carefully.

Prerequisites

Users should first work through the 3D Crank-Slider, Engine with Propeller, and Pinball (2D contact) tutorials, or the equivalent. We assume that you have a basic knowledge of physics.

- Procedures

The tutorial is comprised of the following procedures. The estimated time to complete each procedure is shown in the table below.

If you do not want to go through the model construction portions of the tutorial, we have provided a completed model so that you can begin working in Chapter 6, Creating an Expression. Simply load **Excavator_final.rdyn** from the tutorial directory and start with the instructions in Chapter 6.

Procedures	Time(minutes)
Creating the link	15
Creating the hydraulic cylinder	15
Adding motion to the hydraulic cylinder	15
Adding a load to the tip of the bucket	15
Calculating power consumption	15
Calculating the bucket's range of motion	15
Running and analyzing a design study	15
Running a simulation in batch mode	15
Total:	120

Estimated Time to Complete

This tutorial takes approximately two hours to complete if you perform all the tasks, and one hour if you begin at Chapter 6 when the model construction is complete.

Chapter

2

Creating the Link

Task Objective

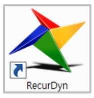
Learn how to create a parametric point and create and modify the right-side link. You will also attach this link to the subsystem using revolute joints.

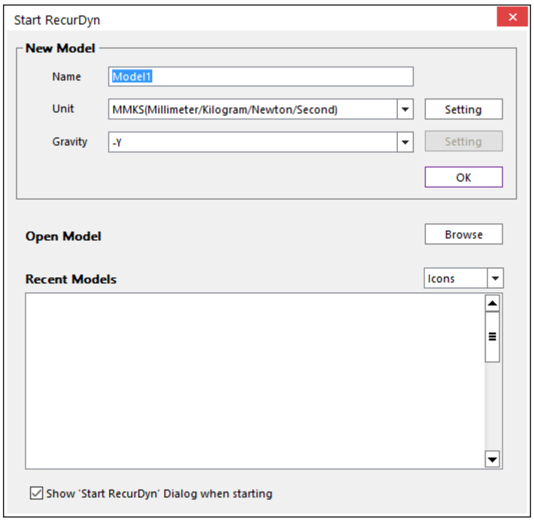
Estimated Time to Complete

15 minutes

Starting RecurDyn

To start RecurDyn and open the base model:





1. On your Desktop, double-click the **RecurDyn** icon.

   RecurDyn starts and the Start RecurDyn dialog box appears.

2. Close Start RecurDyn dialog box. You will use an existing model.

3. In the **Quick Access Toolbar**, click the **Open** and select **Excavator.rdyn** (**The file location**: <Install Dir> /Help /Tutorial /Professional /DipperStickWithBucket, ask your instructor for the location of the directory if you cannot find it).

   The excavator appears in the modeling window. Note that the hydraulic cylinder is missing. You will import it in the next chapter.

Setting the Working Plane

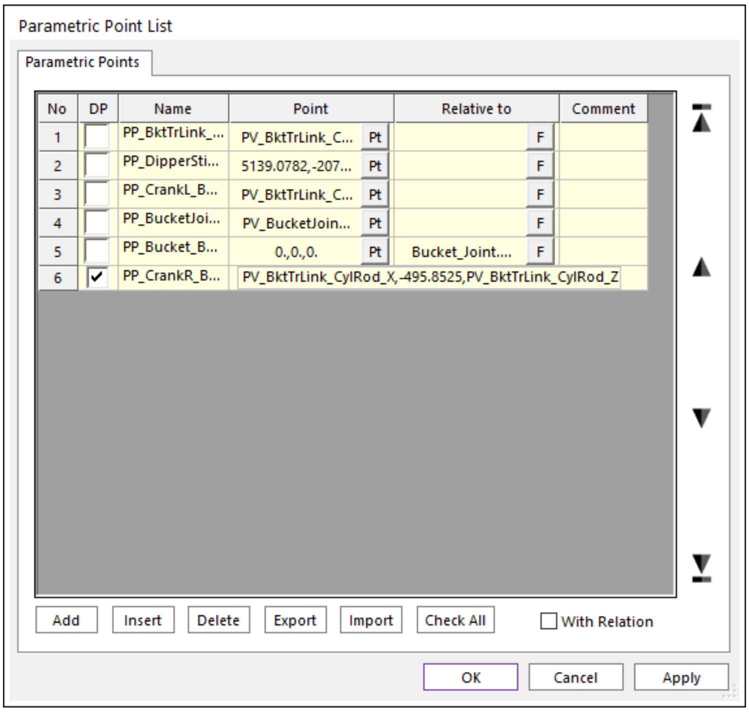
After opening the model, you will notice that the working plane is the XZ plane, as indicated by the grid. This is a good working plane orientation because all the motions you are concerned with occur in this plane.

To ensure that all the components you create in this tutorial will be in the correct orientation, always make sure that the model's working plane is set to the XZ plane. If it is not, you can reset it to the correct orientation by clicking the **Working Plane** tool and selecting the XZ plane.

Creating a Parametric Point

To create a parametric point:

1. From the **Parameter** group in the **SubEntity** tab, click **Parametric Point**.
2. Click **Add** to create a new parametric point.
3. Change the name of the point to **PP_CrankR_BktTrLink**.
4. Double-click the cell shown in grey in the picture on the right, and enter the following in cell to define the location of the point:



PV_BktTrLink_CylRod_X,-495.8525,PV_BktTrLink_CylRod_Z

This means that the x-location of this parametric point is determined by the parametric value **PV_BktTrLink_CylRod_X** and the z-location is determined by **PV_BktTrLink_CylRod_Z**. The y-location is set to be the center of the link you are going to create.

5. Click **OK**.

Parametric points allow you to parametrically change the configuration or design of the model in an automated way. The equations used to calculate the parametric values for the x and z locations of the point you created are functions of two other parametric values, the crank length and the bucket joint angle. For the crank length you are creating here, if you want the end point of the link to move when you change the crank length, you need to define the end point using a parametric point.

Creating the Right-Side Crank Link

To create the right-side crank link:

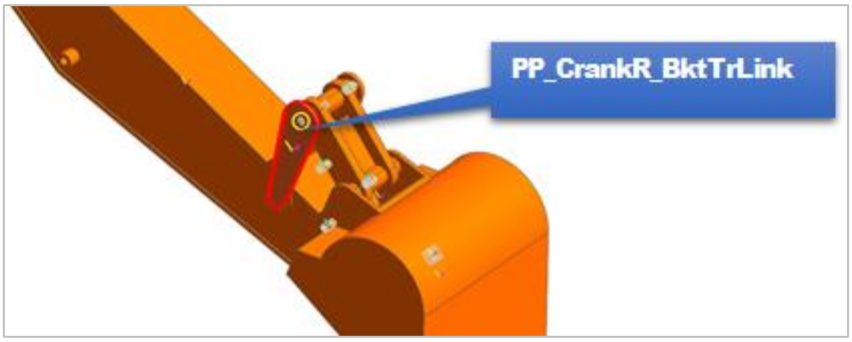
1. From the **Marker and Body** group in the **Professional** tab, click **Link**.
2. Set the Creation Method toolbar to **Point, Point, HalfDepth**.
3. creation method, enter the following coordinates into the Input Window toolbar for the first link point:

   5506.1017, -495.8525, 2231.9958

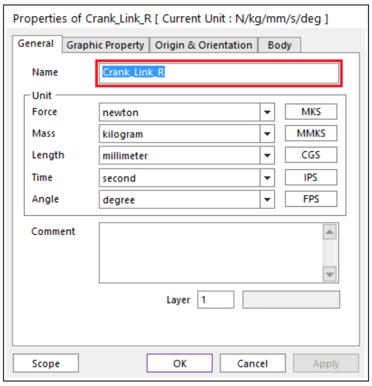
4. Click the newly created parametric point (**PP_CrankR_BktTrLink**) to select the second link point.

5. Set the thickness of the link by entering **17.5** for the **HalfDepth** in the **Modeling Input** toolbar.

   The link should look like the following figure.



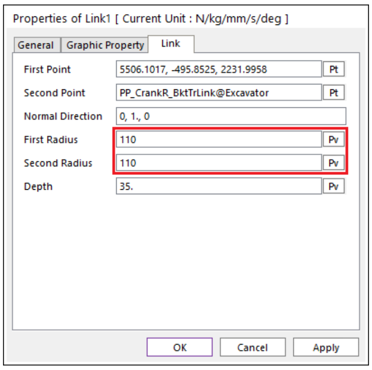
6. Right-click the link and click **Properties**.
7. On the **General** page, change the name from **Body1** to **Crank_Link_R**.
8. Click **OK**.



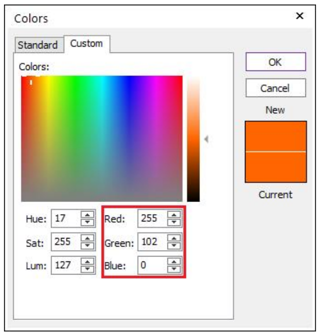
Modifying the Link Shape and Color

Now, you will modify the link shape and color to make it match the rest of the model.

To modify the link shape and color:

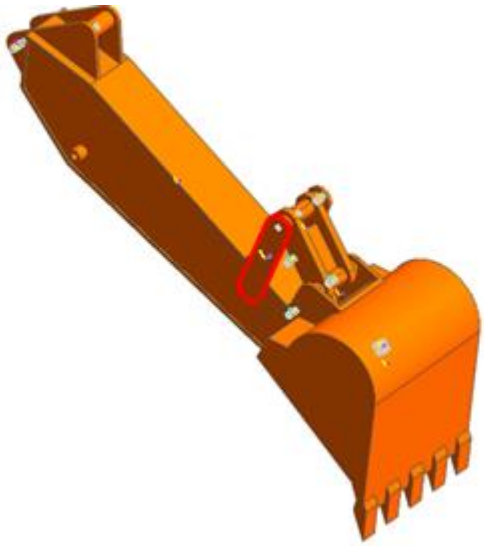


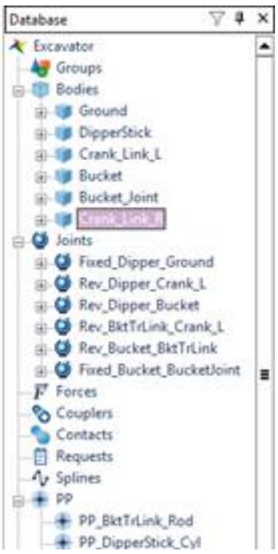
1. Double-click the link to enter **Body Edit** mode.
2. Right-click the link and click **Properties**.
3. First, modify the end radii to match the other crank link. Click the **Link** tab to modify the dimensions.
4. Enter **110** for **First Radius** and **Second Radius**.
5. Click the **Graphic Property** tab to modify the color.
6. Set **Color** to **More Colors**.
7. In the dialog box that appears for changing the color, click the **Custom** tab and enter the following:
   - Red: 255
   - Green: 102
   - Blue: 0



Tip: You do not need to modify the Hue, Saturation (Sat), or Luminosity (Lum) because they update automatically when you change the RGB values.

8. Click **OK**.
9. Click **OK** again until back in **Body Edit** mode and all the dialog boxes are closed.
10. Click the **Exit** arrow to exit Body Edit mode.
11. Your model should look like the figure on the right.





Attaching the Link

You now attach the link to the rest of the subsystem using revolute joints.

To attach the link:

1. From the **Joint** group in the **Professional** tab, click **Revolute**.
2. Set the Creation Method toolbar to **Body, Body, Point**.
3. Create a joint at the bottom of the link by selecting the link (**Crank_Link_R**), then the **DipperStick** and entering the following point information:

   5506.1017, -477.85255, 2231.9959

Tip: As long as the working grid is in the XZ plane (which it should be by default), the orientation of the joint will be correct.

Now you will create the second joint at the top of the link in the same way, but because the bucket transfer link (BktTrLink) is part of a subsystem, you must hold down the Shift key while selecting BktTrLink as the second body for the joint.

4. From the **Joint** group in the **Professional** tab, click **Revolute**.
5. Choose **Crank_Link_R** as the first body, the shaft at the top of the **BktTrLink_CylRod_Cylinder@BktTrLink** subsystem as the second (while holding down the **Shift** key), and then click on the newly created point **PP_CrankR_BktTrLink** as the location for the joint.

You must choose the parametric point as the location of the joint so the location of the joint will update when the parametric point is moved.

6. Save your model very under Desktop directory named Excavator. (C:\Users\PC-Name\Desktop\Excavator)

Chapter
3

Creating the Hydraulic Cylinder

Task Objective

In this chapter, you will import the hydraulic cylinder and link it to the model using parametric points and test the model.

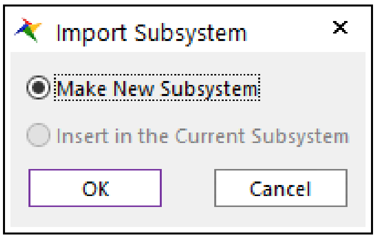
Estimated Time to Complete

15 minutes

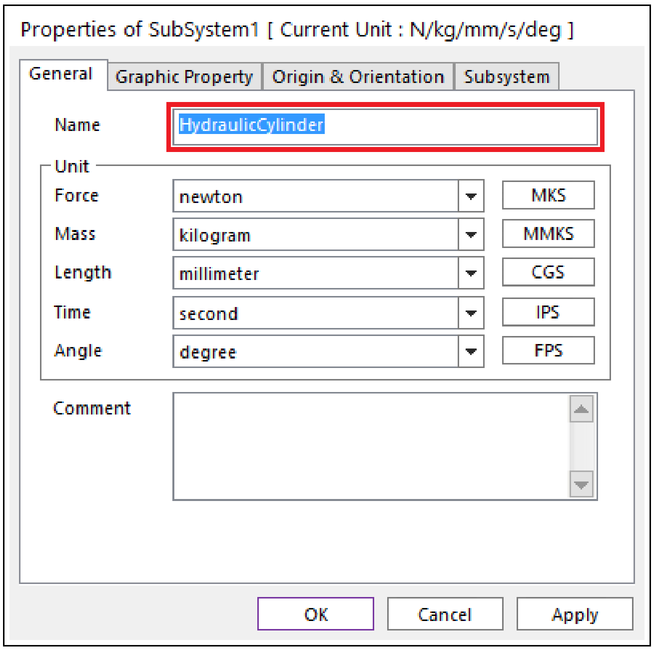
Importing the Generic Hydraulic Cylinder Subsystem

To import the hydraulic cylinder subsystem:

1. From the **File** menu, click **Import**.
2. From the list of files, choose **Hydraulic_Cylinder.rdsb**. (**The file location**: <Install Dir>\Help\Tutorial\Professional\DipperStickWithBucket, ask your instructor for the location of the directory if you cannot find it).
3. Click **Open**.
4. In the dialog box that appears, click **OK** to create a new subsystem.



5. Now, you will rename the imported subsystem:
   - In the Database window, right-click **SubSystem1** and select **Properties**.
   - Click the **General** tab and enter **HydraulicCylinder** as its name as shown in the figure on the right.
   - Click **OK**.



The hydraulic cylinder you just imported already has a number of parametric points, parametric values, and expressions in it. They were created to make this hydraulic cylinder totally parametric so it can be used in any model. All you need to do is to create parametric points at the model level for the end points of the hydraulic cylinder and then connect them to the subsystem using parametric point connectors, which you will do next.

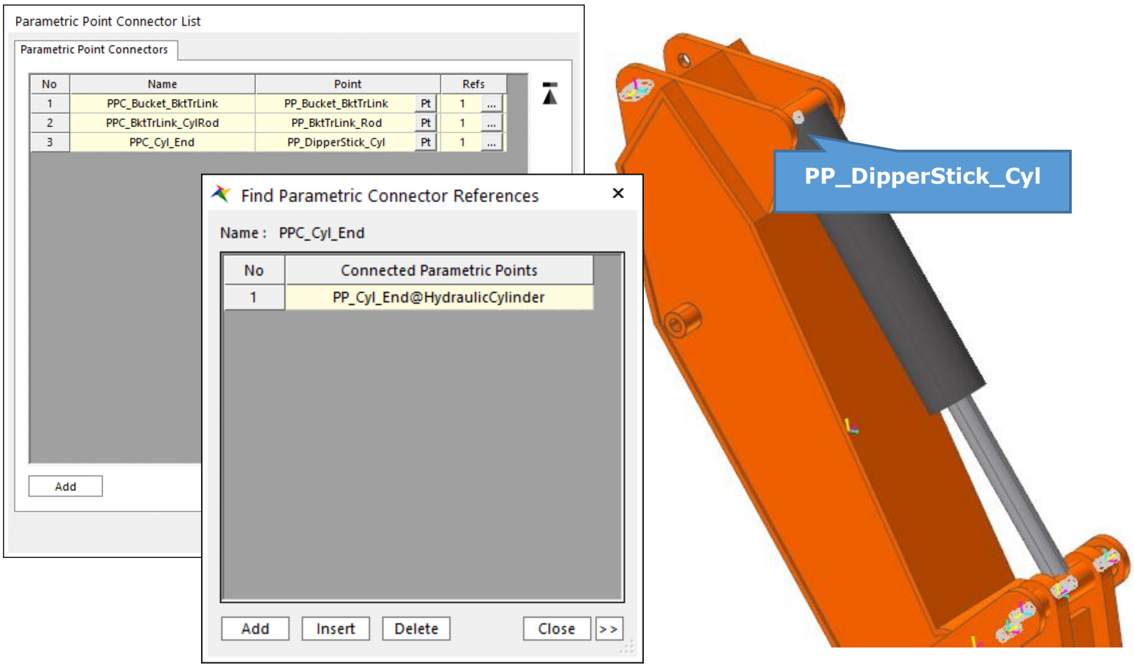
Setting the Location of the Hydraulic Cylinder Subsystem

In this section, you set the location of the hydraulic cylinder subsystem by linking the parametric points at the main model level to parametric points in the hydraulic cylinder subsystem, using parametric point connectors (PPC). The parametric points at the main model level are the first two on the list of parametric points, **PP_BktTrLink_Rod** and **PP_DipperStick_Cyl**, and their names indicate what they were made to connect.

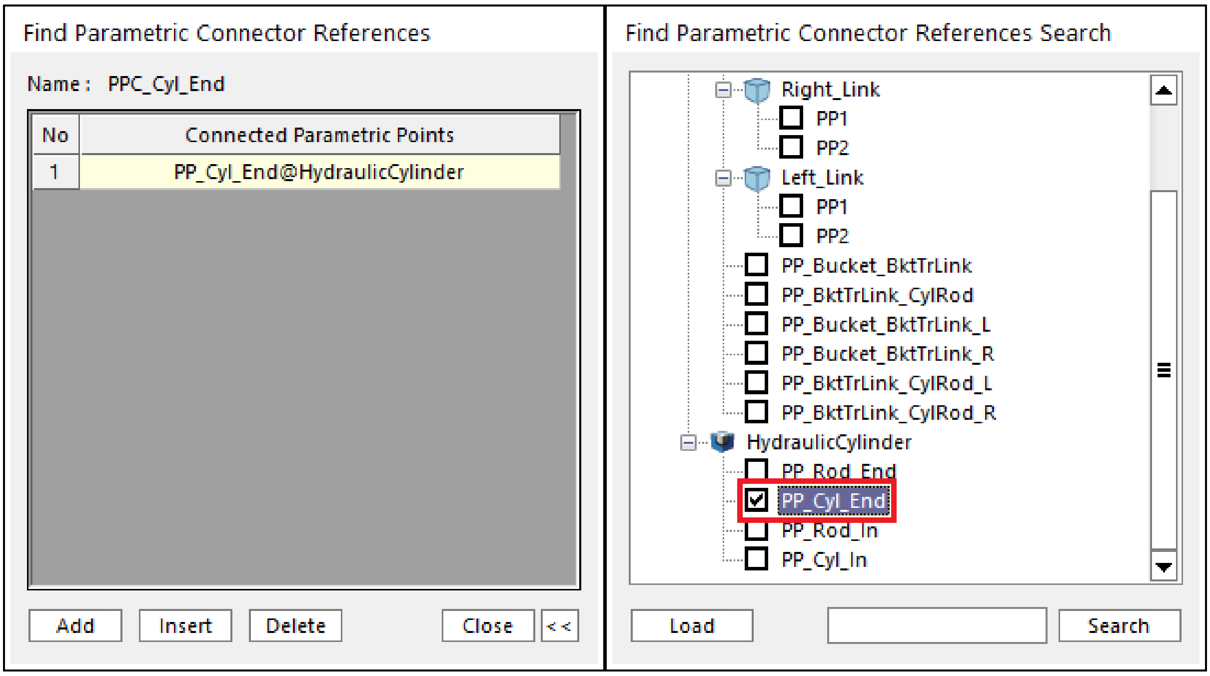
To link the hydraulic cylinder subsystem:

1. From the **Parameter** group in the **SubEntity** tab, click the **Parametric Point Connector**.
2. Click **Add** to create a new PPC.
3. Double-click **PPC1** and rename the PPC to **PPC_Cyl_End**.

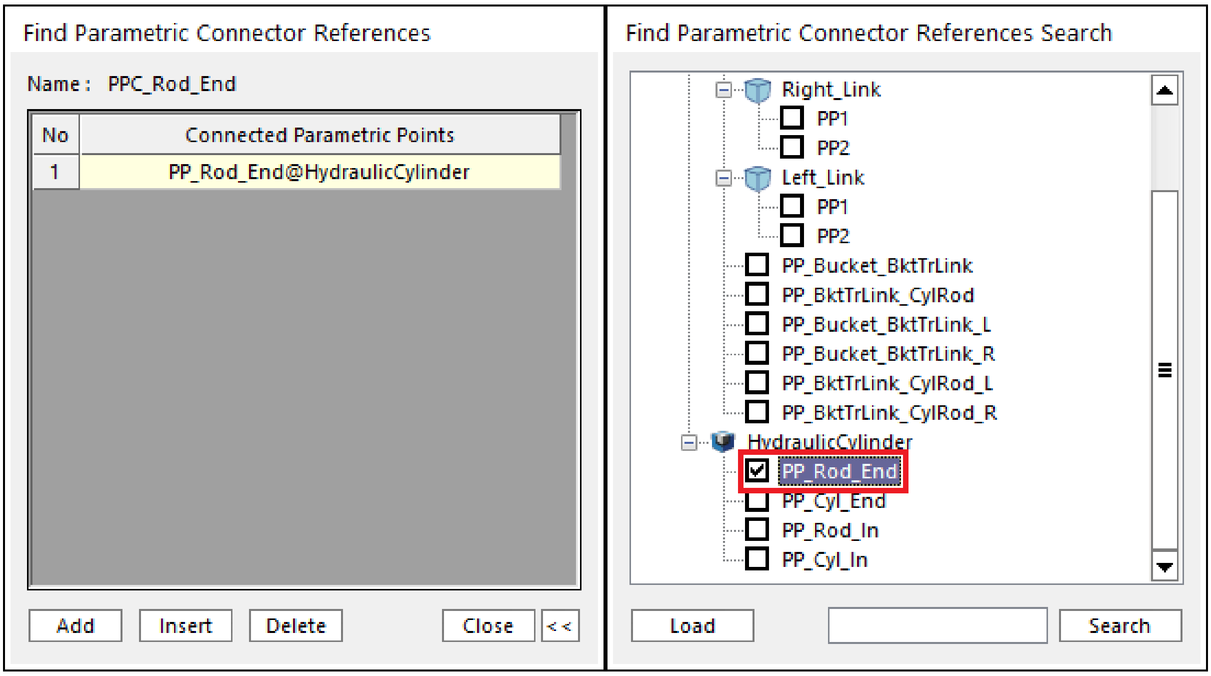
4. In the **Point** text box, select **PP_DipperStick_Cyl**, by doing one of the following:
   - Double-click the text box and enter the name of the parametric point.
   - Click **Pt** and then click the parametric point in the Working window as shown in the figure as above.
   - Click **Pt** and then click & drag the parametric point named **PP_DipperStick_Cyl** in the Database window to Input Window toolbar.



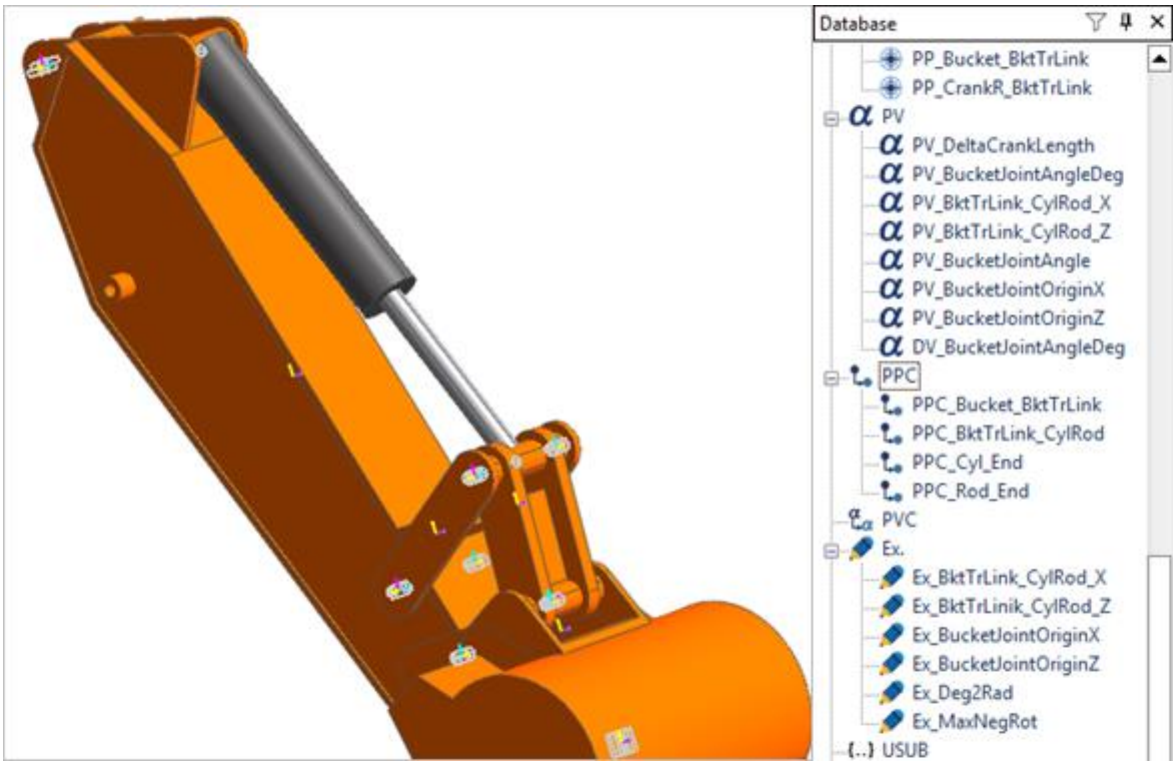
5. Click the **More** button (**...**) in the **Refs** column.
6. Click the **Continue** button (**>>**) in the bottom right corner, as shown in the second figure.
7. In the dialog box that appears, scroll down and click the box next to **PP_Cyl_End** under **HydraulicCylinder** as shown in the second figure on the right.
8. Click **Load** to enter the appropriate data in the dialog box on the left and click the **Close** button to exit the dialog boxes. See the figure below.



9. Perform the same set of operations for the other end of the hydraulic cylinder, but this time rename the PPC to **PPC_Rod_End** and select the parametric point **PP_BktTrLink_Rod** as the desired point. This PPC should reference **PP_Rod_End** from the **HydraulicCylinder** subsystem. This setup is summarized in the following figures.



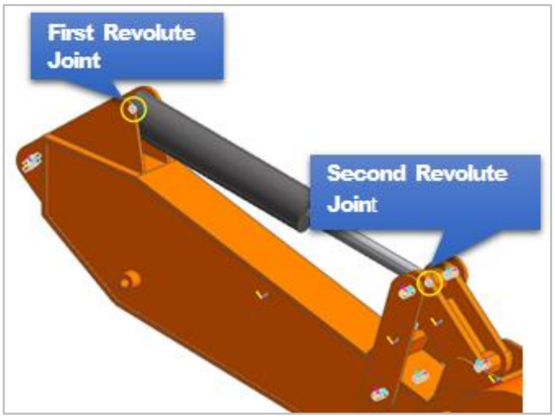
When you click **OK** to accept your changes in the Parametric Point Connector List dialog box, the hydraulic cylinder automatically moves itself into the correct position as shown in the following figure.


Database
PP_Bucket_BktTrLink
PP_CrankR_BktTrLink
PV
PV_DeltaCrankLength
PV_BucketJointAngleDeg
PV_BktTrLink_CylRod_X
PV_BktTrLink_CylRod_Z
PV_BucketJointAngle
PV_BucketJointOriginX
PV_BucketJointOriginZ
DV_BucketJointAngleDeg
PPC
PPC_Bucket_BktTrLink
PPC_BktTrLink_CylRod
PPC_Cyl_End
PPC_Rod_End
PVC
Ex.
Ex_BktTrLink_CylRod_X
Ex_BktTrLinik_CylRod_Z
Ex_BucketJointOriginX
Ex_BucketJointOriginZ
Ex_Deg2Rad
Ex_MaxNegRot
{..} USUB

Attaching the Hydraulic Cylinder to the Model

You will now attach the hydraulic cylinder subsystem to the model with two revolute joints:

- The first revolute joint connects the dipper and the cylinder
- The second revolute joint connects the top shaft of the bucket transfer link and the rod.

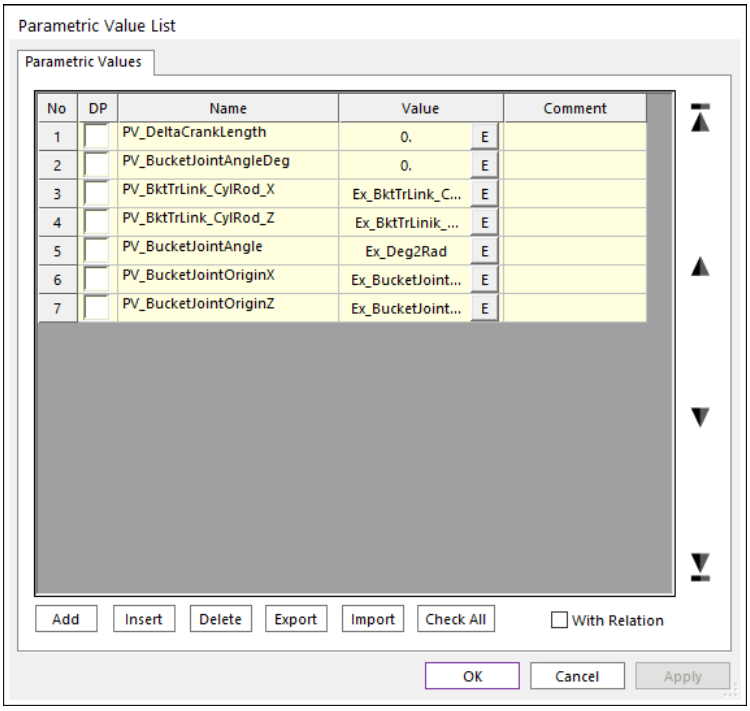


To attach the hydraulic cylinder:

1. From the **Joint** group in the **Professional** tab, click **Revolute**.
2. Set the Creation Method toolbar to **Body, Body, Point**.
3. Click **DipperStick** and while holding down the **Shift** key, click the **Cylinder** from the **HydraulicCylinder** subsystem. Then, click the parametric point at the end of the cylinder (**PP_Dipperstick_Cyl**) as thet location of the joint.
4. Follow the same procedure to create the revolute joint at the other end of the hydraulic cylinder. This time, while holding down the **Shift** key, click **Rod** from the **HydraulicCylinder** subsystem and then the **BktTrLink_CylRod_Cylinder** from the **BktTrLink** subsystem. Finally, release the Shift key and click the parametric point located at the end of the rod (**PP_BktTrLink_Rod**).
   - Make sure to release the Shift key before selecting the joint location because the parametric point you want is located at the system level.

Exercising the Model

Having completed the previous steps, your model should now update the geometry automatically when the values for **PV_DeltaCrankLength** and **PV_BucketJointAngleDeg** are changed. Try this to ensure that it is working (you can choose not to do this step without losing continuity).



To exercise the model:

1. In the **Database** window, double-click any of the parametric values (PVs) to display the Parametric Value List dialog box shown in the figure on the right.

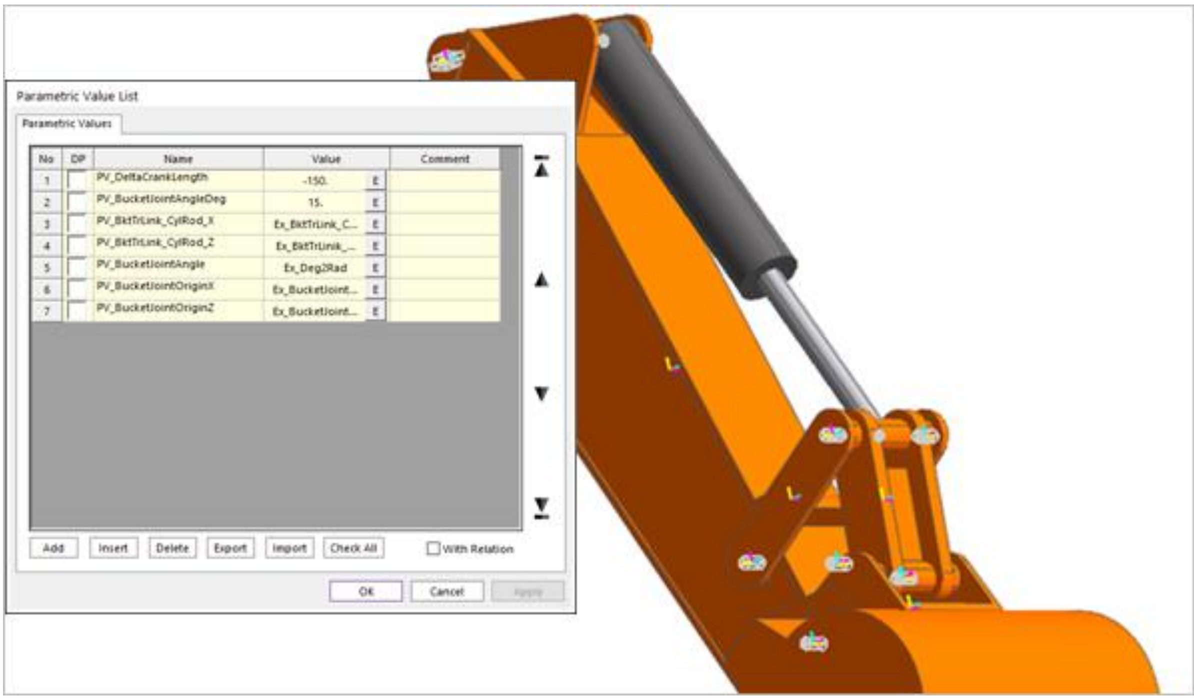
2. Try changing the value for **PV_DeltaCrankLength** to a value between -150 and 150 and then click **Apply** to see the model update.
3. Similarly, try changing the value for **PV_BucketJointAngleDeg** from 0. to a value between -15 and 15 and then click **Apply** to see the model update.

Tip: Why is the fixed joint between the bucket joint and the bucket located out in space, rather than within the intersection of these two bodies?
While exercising the model, you might notice that this joint location is not where you would expect the joint forces to actually occur. This location was chosen to facilitate being able to specify the bucket joint position easily with only an angle. This location is also acceptable because in this model, you are not concerned with the reaction forces at this joint, and the location of a fixed joint does not influence the behavior of the two joined rigid bodies with respect to each other.

Tip: Other values could be chosen for these PVs but the model will not necessarily update appropriately because the expressions and parametric entities were not created to go beyond these limits.

For comparison, the figure below shows what the model should look like when the values are set to -150 and 15, respectively.

4. When you are finished experimenting, be sure to change the values back to zero for both PVs and click **OK**.



5. Save your model.

Chapter 4

Add Motion to the Hydraulic Cylinder

Task Objective

In this chapter, you will add translational motion to the hydraulic cylinder to drive the bucket rotation. You will do this at the HydraulicCylinder subsystem level where the translational joint is located. You will define a parametric value in the subsystem and connect its value, using a parametric value connector, with a parametric amplitude that is defined at the system level.

Estimated Time to Complete

15 minutes

Creating a Parametric Value

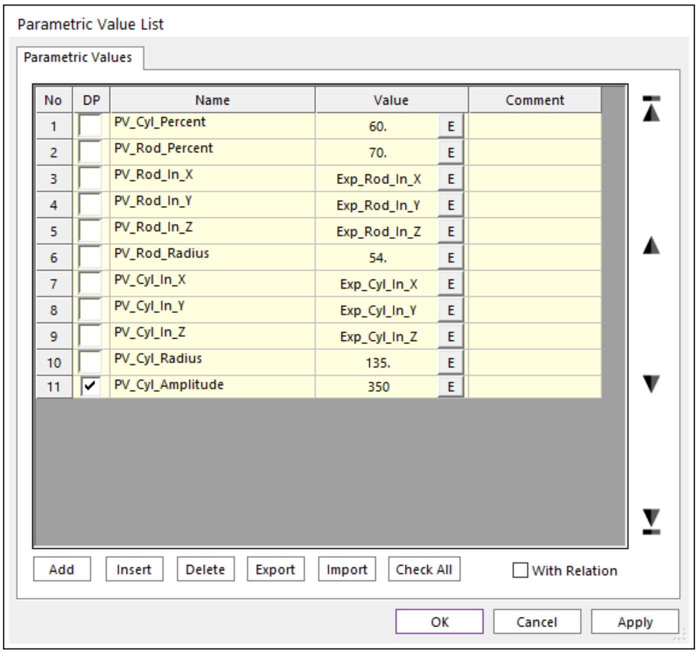
You will first create a parametric value (PV) in the Hydraulic cylinder subsystem.

To create a PV:

1. Enter the **Subsystem edit** mode for the hydraulic cylinder. Either:
   - In the Working window, double-click the **hydraulic cylinder**.
   - In the Database window, right-click **HydraulicCylinder** and click **Edit**.

   Only the hydraulic cylinder appears.

2. Open the Parametric Value List as described previously. (**Tip: SubEntity → Parametric Value.**)
3. Click **Add** to create a new PV and rename it **PV_Cyl_Amplitude** as shown in the figure on the right.
4. Set the value to **350** (this means that the hydraulic cylinder will oscillate 350 mm in each direction during the simulation).
5. Click **OK**.

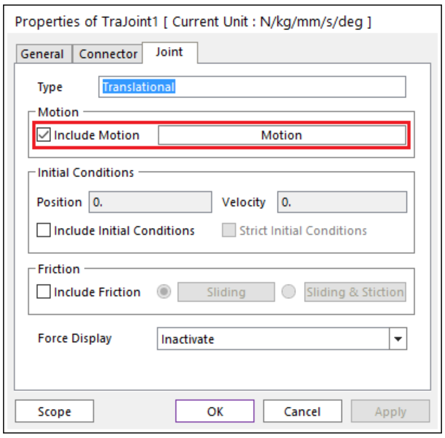


Adding Motion to the Translation Joint

You could define the needed motion in many different ways, ranging from something as simple as a sinusoidal displacement to something as complex as a cycloidal or modified trapezoidal displacement profile that limits the magnitude of the derivative of the acceleration (jerk) of the cylinder. Rather than use one of these motion definitions, you will use step changes in the joint velocity. This minimizes the jerk while still being a very simple profile (the Step function implemented in RecurDyn uses cubic functions at the corners of the step profile to minimize jerk).

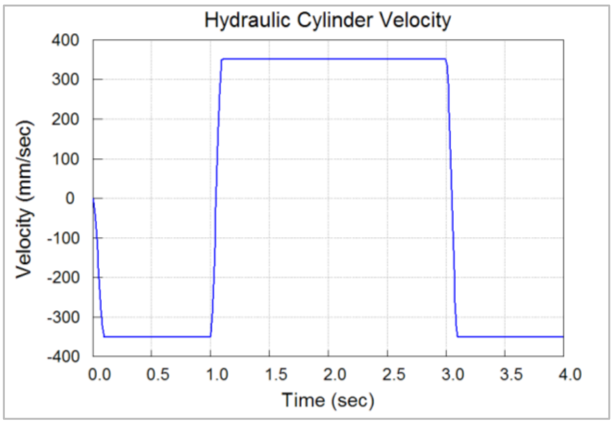
To add motion:

1. In the Database window, right-click **TraJoint1** and select **Properties**.
2. Click **Include Motion** and then click **Motion** as shown in the figure on the right.
3. In the Motion dialog box that appears, set the motion to **Velocity** (the second drop-down box) and leave the initial position at 0.0 (the default) as shown in the figure on the right.
4. Click **EL** to view the expression list from which you will define the velocity profile.



A number of expressions already exist, but you will create a new one

5. Click **Create**.
6. Rename the expression **Ex_StepCylVel** and enter the following equation:
   - PV_Cyl_Amplitude*(STEP(TIME,0,0,0.1,-1) + STEP(TIME,1,0,1.1,2)
   - +STEP(TIME,3,0,3.1,-2))



This equation will drive the joint at a speed of negative **PV_Cyl_Amplitude** per second for one second, then switch to driving at the same speed in the positive direction for two seconds, and then switch back to negative for one more second, returning it to its original position. The plot on the right shows the resulting function when **PV_Cyl_Amplitude** has a value of 350.

The Expression dialog box should now look like the figure on the right.

7. Click **OK** to accept the expression definition and then click **OK** three more times to accept all of the steps in creating the joint motion.

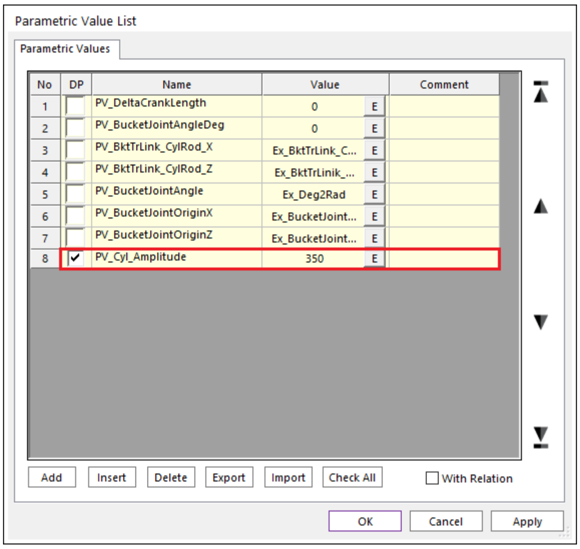
8. Click the **Exit** arrow to exit **Subsystem Edit** mode.

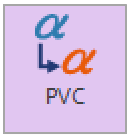
Creating a Parametric Value Connector

You will create a parametric value connector (PVC) to pass the value of PV_Cyl_Amplitude from the Assembly mode to the Hydraulic Cylinder subsystem.

To create PVC (parametric value connector):

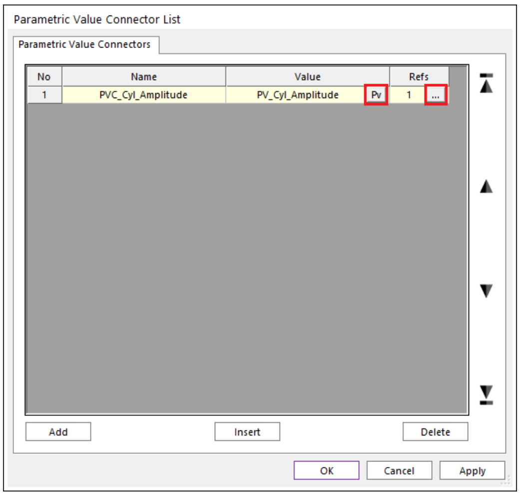
1. Create a parametric value called **PV_Cyl_Amplitude** at the Assembly mode just as you did in the subsystem mode with a value of **350**.



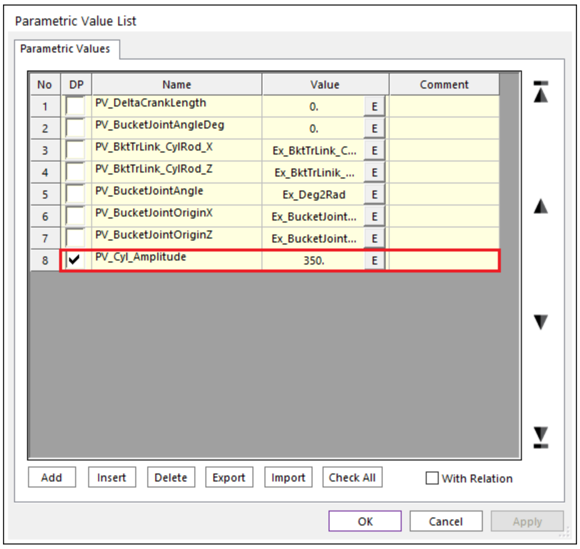


2. From the **Parameter** group in the **SubEntity** tab, click **Parametric Value Connector**.
3. In the Parametric Value Connector dialog box, click **Add** and rename the PVC to **PVC_Cyl_Amplitude**.

   The dialog box should look like the second figure on the right.
4. Click **PV** in the **Value** column to specify the appropriate PV.

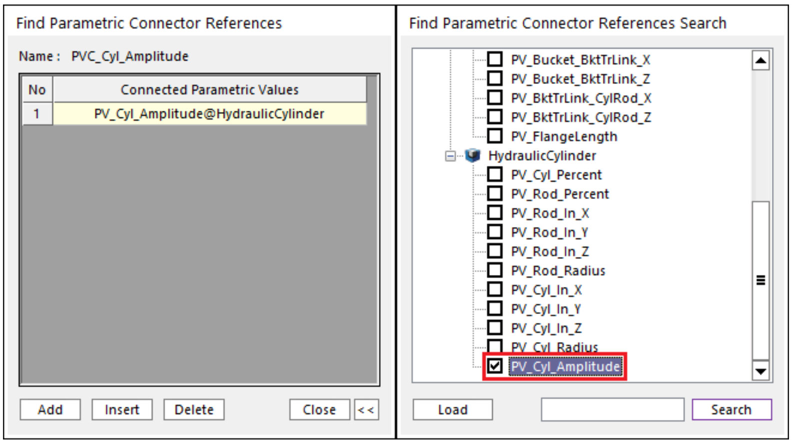


5. Select **PV_Cyl_Amplitude** as shown below and then click **OK** to go return to the Parametric Value Connector List dialog box.



6. Click the **More** button (**...**) in the **Refs** column to display the dialog box where you will select the appropriate PV from the **HydraulicCylinder** subsystem. (This is similar to what you did for PPs and PPCs i n Chapter 3 of this tutorial.)
7. Click the **Continue** button (**>>**) next to the Close button and then scroll down to select the box next to **PV_Cyl_Amplitude** in the HydraulicCylinder subsystem.
8. Click **Load** to enter the appropriate values in the dialog box.

   Your dialog boxes should now look like the following figure.
9. Finish the creation process by clicking **Close** and then **OK**.
10. Save your model.



Chapter 5

Adding a Bucket Tip Load

Task Objective

The load that you will add in this chapter represents the force at the tip of the bucket when digging. Therefore, you want the load to remain in a fixed orientation with respect to the bucket. You will accomplish this by creating an extra body (dummy body) that is fixed to the bucket (to maintain correct orientation) and applying an axial force between them that only applies force on the bucket with no reaction load on the dummy body. You will then run your first simulation of the model.

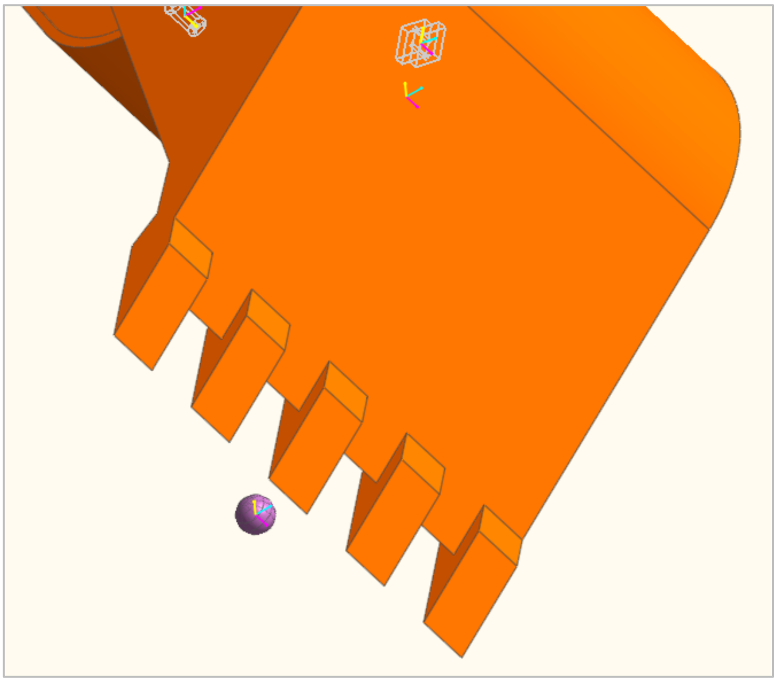
Estimated Time to Complete

15 minutes

Creating the Dummy Body

To create the dummy body:

1. From the **Marker and Body** group in the **Professional** tab, click **Ellipsoid**.
2. Set the creation method toolbar to **Point, Distance**.
3. Enter the following coordinates for the center of the sphere:
   5579.2685, -207.8525, 62.560441
4. Enter **50** for the radius of the sphere (distance).
5. Zoom in on the tip of the bucket to see the sphere you created.



6. Right-click the sphere and click **Properties**.
7. In the **General** tab, change the name of the body to **BucketTip**.
8. Click the **Graphics** tab and choose **Gray-50%** as the color.
9. Click **OK**.

Attaching the Dummy Body to the Bucket

In this step, you will attach the dummy body to the bucket using a fixed joint.

To attach the dummy body to the bucket:

1. From the **Joint** group in the **Professional** tab, click **Fixed**.
2. Set the Creation Method toolbar to **Body, Body, Point**.
3. Click the **BucketTip** and then the **Bucket**, and enter the following coordinates into the Modeling Input toolbar:

   5679.2685, -207.8525, 62.560441

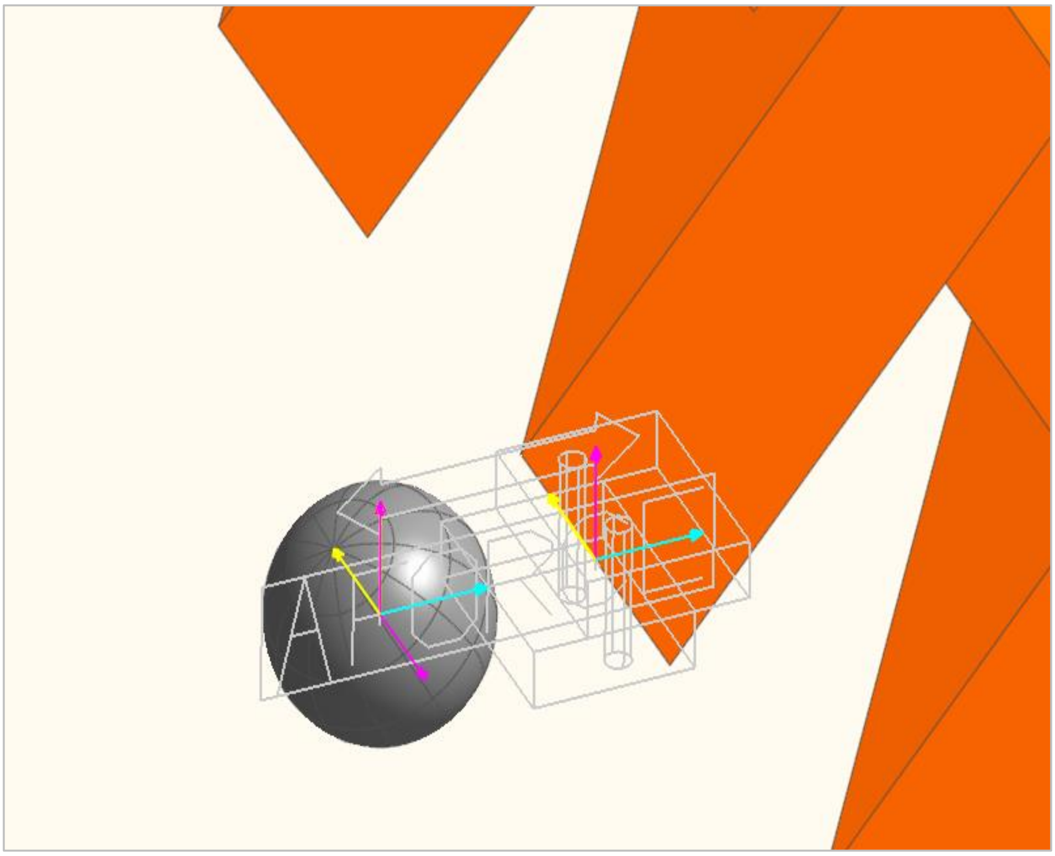
This creates the fixed joint with the sphere (BucketTip body) as the Base body and the Bucket as the Action body with the joint located at the tip of the bucket where you want the load applied.

Applying an Axial Force between the Sphere and Bucket

To apply an axial force:

1. From the **Force** group in the **Professional** tab, click Axial.
2. Set the Creation Method toolbar to **Point, Point**.
3. Click the center of the **BucketTip** and then the center of the fixed joint you just created.

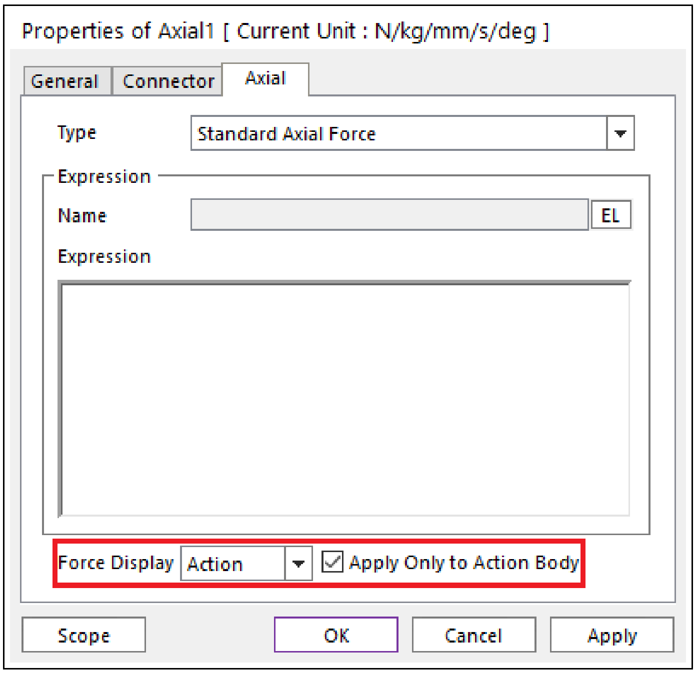
   A close-up view of the **Bucket Tip** should now look like the following:

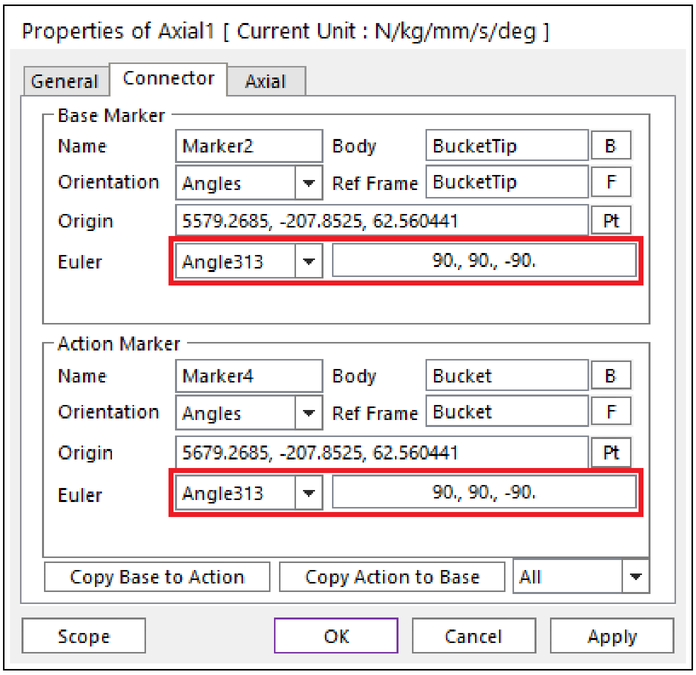


A characteristic of the axial force is that it transmits force in the z-direction of the markers at its two points. If you look closely at the model, you will see that the z-axis (yellow arrows) needs to be rotated to point in the correct direction. You also need to turn on the display of the force and make sure the force is only applied to the Action (Bucket) body. You will make these changes now.

4. Right-click the **AFORCE** icon and click **Properties**.
5. On the **Axial** page:
   - Click Apply Only to Action Body.
   - Set **Force Display** to **Action** as seen in the figure on the right.

On the Connector page of this dialog box, you will rotate the markers to set their z-axes in the correct direction.





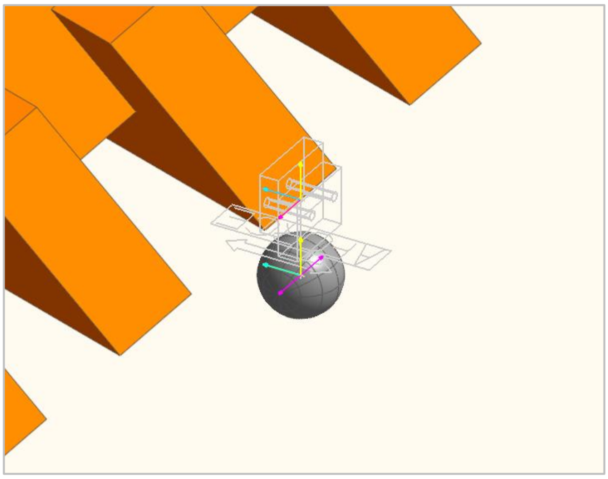
6. Click the **Connector** tab and change the orientations of both markers to **90., 90., -90.** as shown in the second figure on the right.

   This rotates the marker 90 degrees about its +z axis, then 90 degrees about the newly positioned +x axis, then -90 degrees about the newly positioned +z axis. The result is to point the +z axis of the marker in the direction of the +x axis of the global coordinate system.

7. Click the **General** tab and change the name to **BucketTipLoad**.

8. Click **OK**.

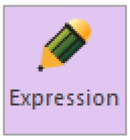
   The model should now look like the following:



Defining the Expression for the Axial Force

You will do this step a little differently than you would normally do. You will first define the expression in the **Expression List** and then return to the **Axial Force Properties** dialog box and choose the expression that you created.

To define the expression:

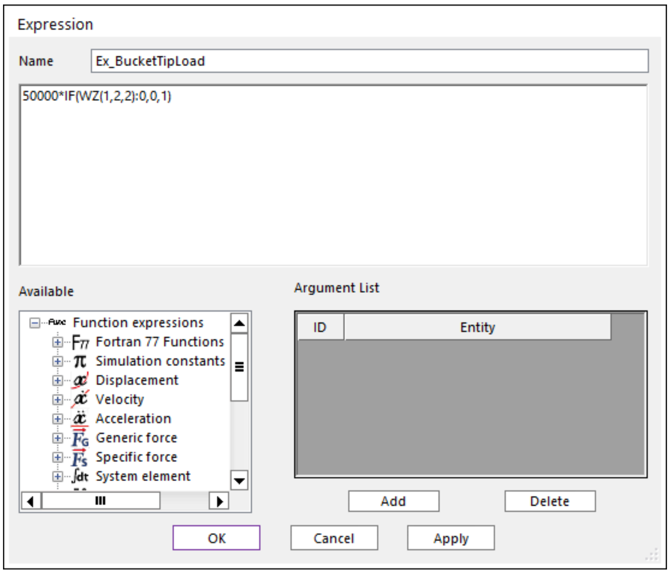


1. From the **Expression** group in the **SubEntity** tab, click **Expression** to open the **Expression List** dialog box.

2. Click **Create**.

3. Name the expression **Ex_BucketTipLoad** and enter the following function:

   50000*IF(WZ(1,2,2):0,0,1)



   This will provide a 50 kN load on the bucket tip when the rotational velocity (WZ) of the bucket is greater than zero and zero otherwise. For more information on syntax for the IF function, search for "IF Statement" in the RecurDyn Help.

   The WZ function requires a marker as its argument. You write the function as WZ(1) and then define the marker **Bucket.Marker1** as the first entry in the argument list as shown in the figure on the right.

4. To add an entity to the argument list, click **Add** and then either double-click the cell and type in the name **Bucket.Marker1** directly or drag the marker name from the Database window and drop it in the cell (expand **Bodies →Bucket → Markers → Marker1**).

5. Follow the same procedure to add second entity named **DipperStick.Marker2**

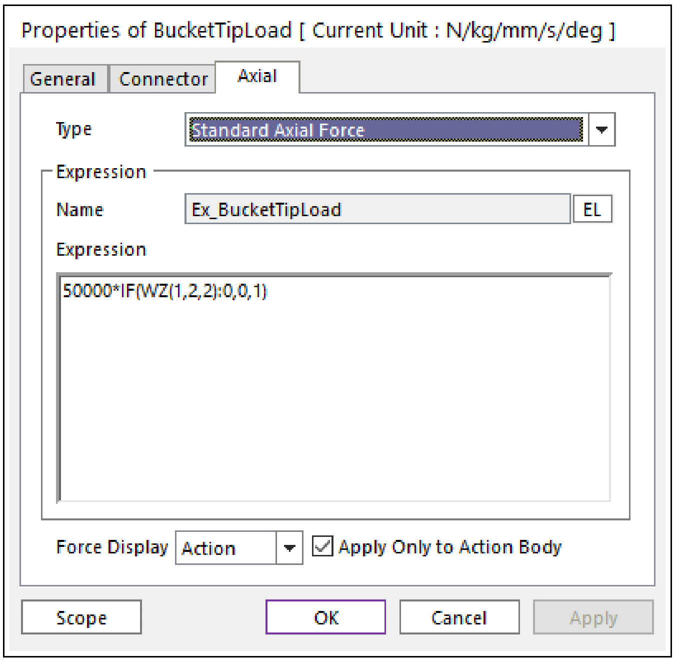
6. Click **OK** to accept the expression and **OK** again to exit the **Expression List** dialog box.

7. Return to the Properties dialog box for the **BucketTipLoad** force.

8. In the **Axial** page, click **EL** and then select **Ex_BucketTipLoad** from the list.

9. Click **OK**.

   The BucketTipLoad Properties dialog box should now look like the figure on the right.



10. Click **OK**.

Running a Simulation

Now you will run a simulation of the model to verify that the force is in the correct direction.

To run a simulation:

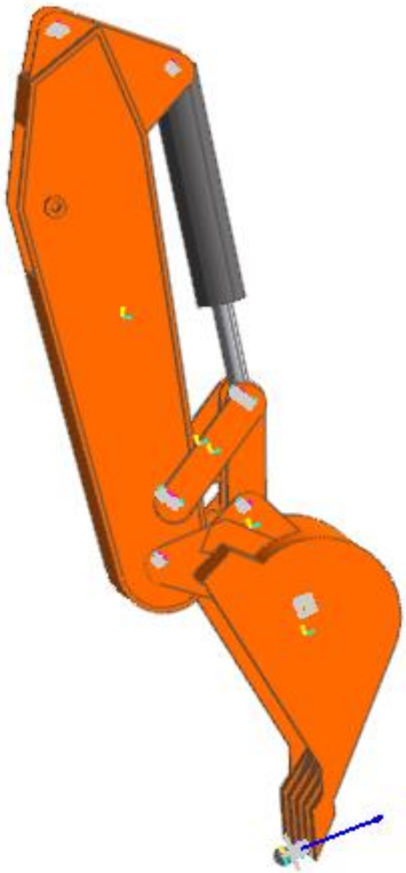
1. From the **Simulation Type** group in the **Analysis** tab click **Dyn/Kin**.
2. Set the following:
   - **End time**: 4.0
   - **Step**: 400

3. Click **Simulate**.

4. Click **Play**.

The following figure shows what the model should look like 1.5 seconds into the simulation.

Note in particular that the blue force vector at the tip of the bucket shows that the direction of the force is into the bucket as it should be.



Chapter
6

Calculting Power Consumption

Task Objective

Tip: You can start the tutorial here using **Excavator_final.rdyn**, which has all the model construction completed. It is in the tutorial directory.

The design study will use the hydraulic cylinder power as a performance index (or measure). In this chapter, you will learn how to access the necessary data and how to create an expression to calculate the power used by the hydraulic cylinder. Power is the product of force and velocity.

To calculate the hydraulic cylinder power, you need to access the driving force of the translational joint in the **HydraulicCylinder** subsystem and pass it through to the system level. You can almost achieve this by using the reaction force of the revolute joint you just created between the dipper and the cylinder, but this value includes forces from the mass and motion of the hydraulic cylinder and so is not exact.

You will use a special technique, instead, to obtain the exact force value. You will create a dummy body in the **HydraulicCylinder** subsystem and apply a force to it equal in magnitude to the driving force of the translational joint. Then, you will fix this body to ground at the system level and use the reaction force in this fixed joint as the measure of the desired force.

The joint velocity will be measured from the relative velocity of the two markers defining the translational joint at the subsystem level.

Estimated Time to Complete

15 minutes

Creating the Dummy Body

First, you will create the dummy body in the **HydraulicCylinder** subsystem.

To create the dummy body:

1. Double-click the **HydraulicCylinder** subsystem in the Working model window to enter **Subsystem Edit** mode.

2. From the **Marker and Body** group in the **Professional** tab, click **Ellipsoid**.
3. Set the Creation Method toolbar to **Point, Distance**.
4. Create a sphere at the following location:

   6100, -207.8525, 4200
5. Enter **50** in the **Input Window toolbar** as the radius.

   The sphere appears as shown below.

6. Right-click the newly created body and select **Properties** to change the name and color of the body.
   - In the **General** page, change the name to **DrivingForceBody**.
   - In the **Graphics Properties** page, change the color to Gray-50%.

Creating an Axial Force to Act on the DrivingForceBody

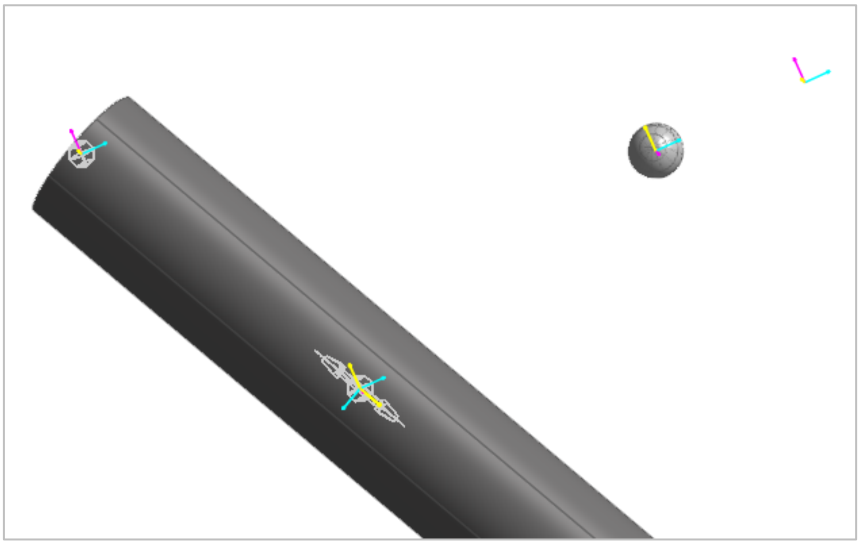
The first thing you need to do is define a marker for the reaction force on ground.

To create a marker:

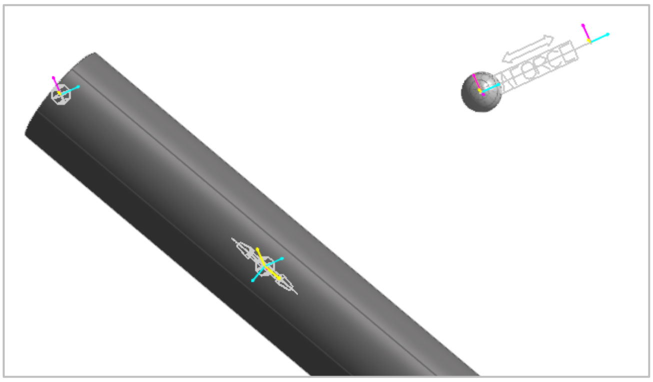
1. From the **Marker and Body** group in the **Professional** tab, click **Marker**.
2. Set the **Creation Method toolbar** to **Body, Point**.
3. Click **MotherBody** (anywhere on the screen other than on the cylinder or rod) and then enter the following coordinates for the marker location:

   6400, -207.8525, 4200

   The cylinder, marker and sphere appear as shown below.



4. From the **Force** group in the **Professional** tab, click **Axial**.
5. Set the **Creation Method toolbar** to **Point, Point**.
6. Click the marker at the center of **DrivingForceBody** and then the newly created marker.

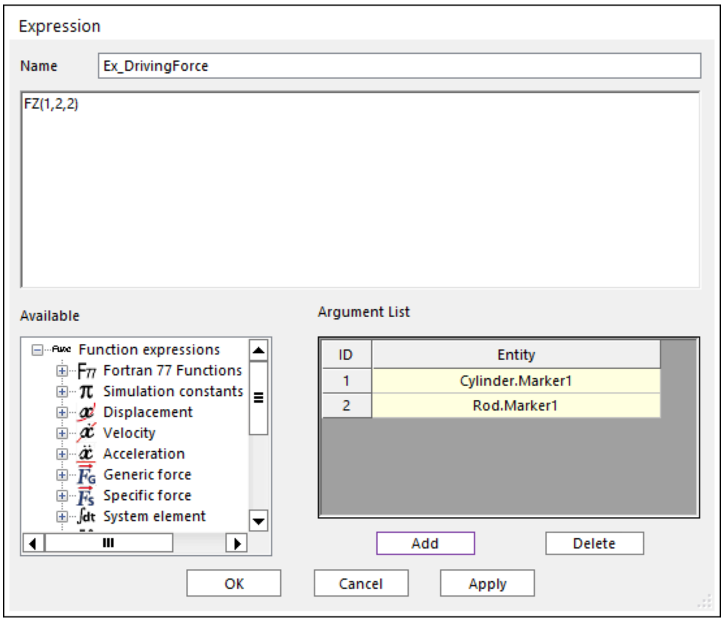


7. Right-click the axial force, and click **Properties**
8. In the **Axial** tab of the Properties dialog box (near the bottom), set **Force Display** to **Base**.

Tip: Because you clicked the **DrivingForceBody** first, it is the base body.

9. To define the expression for the force, click **EL**.
10. In the **Expression List** dialog box that appears, click **Create** and then:
    - Change the name of the expression to **Ex_DrivingForce**.
    - Enter the following expression:
    - FZ(1,2,2)
    - The arguments to FZ are the markers defined in the translation joint:
11. Click **Add** under the **Argument List** and enter the arguments.

- Cylinder.Marker1
- Rod.Marker1



Tip: You can expand the list of markers under the translational joint and drag the marker names into the Argument List.

The Expression dialog box should look like the figure.

12. Click **OK** three times to accept the changes.

13. Click the **Exit** arrow to return to the **Assembly mode**.

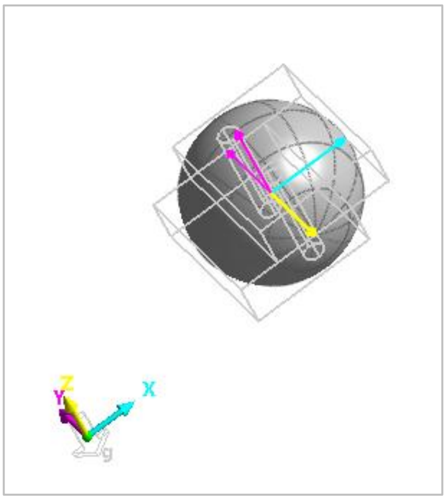
Fixing DrivingForceBody to Ground

To fix DrivingForceBody to ground:

1. From the **Joint** group in the **Professional** tab, click **Fixed**.
2. Set the Creation Method toolbar to **Body, Body, Point**.
3. Click ground and then, while holding down the **Shift** key, click **DrivingForceBody** from the **HydraulicCylinder** subsystem.
4. While still holding down the **Shift** key, click **DrivingForceBody** again to locate the fixed joint at the center of the body.

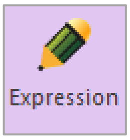
The orientation of the markers is important here because their orientation will be used to calculate the force in the next step.

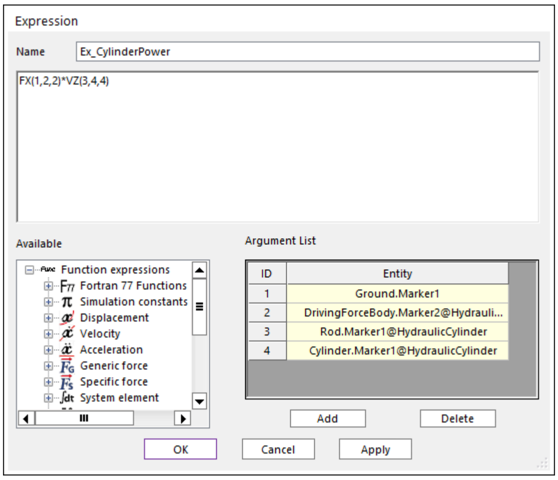
The **DrivingForceBody** should look like the figure. Note that the marker's x-axis is pointing in the global x-direction (the direction that the cylinder driving force is in).



Creating the Expression for Calculating Power

To create the expression:





1. From the **Expression** group in the **SubEntity** tab, click **Expression** to open the **Expression List** dialog box.

   Click **Create** in the **Expression List** and then the **Expression window** should look like the right figure.

2. Rename the expression **Ex_CylinderPower**.

3. Enter the following expression:

   FX(1,2,2)*VZ(3,4,4)

4. Enter the four entities from the list below in that order into the **Argument List**:

   - **Ground.Marker1**
   - **DrivingForceBody.Marker2@HydraulicCylinder**
   - **Rod.Marker1@HydraulicCylinder**
   - **Cylinder.Marker1@HydraulicCylinder**

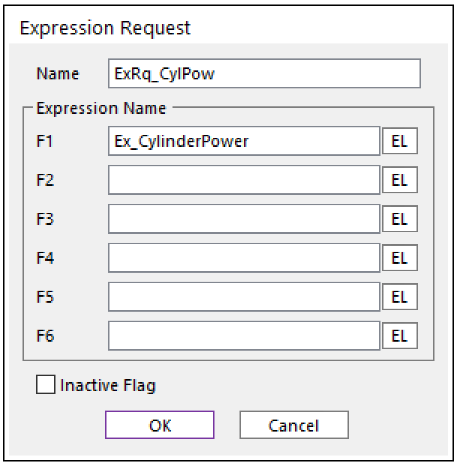
5. Click **OK** twice to accept the changes and close out of the **Expression List**.

Tip: @HydraulicCylinder in the entities names means that the specified markers are located in the HydraulicCylinder subsystem. The first two entities can be dragged and dropped from the Database window because they are the markers used in the fixed joint between DrivingForceBody and ground, but the names of the other two entities must be typed in directly because they are not accessible at assembly mode.

Creating an Output Request

In this section, you will create an expression output request so you can plot **Ex_CylinderPower** in the Plotting environment.

To create an output request:



1. From the **Expression** group in the **SubEntity** tab, click **Request**.
2. Click the **Expression** tab, and then click **Create**.
3. Change the name of the expression request to **ExRq_CylPow**.
4. Click **EL** in the **F1** row and select **Ex_CylinderPower** from the Expression List.
5. Click **OK**.

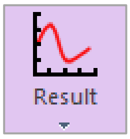
   The **Expression Request** dialog box should now look like the one shown on the right.
6. Click **OK** twice to accept changes and close the dialog boxes.

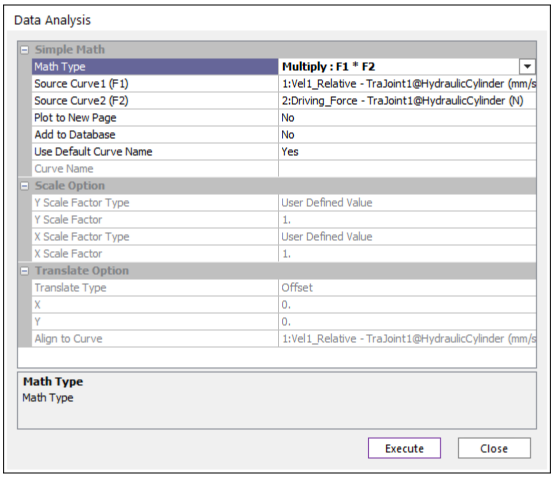
Running a Simulation and Plotting the Results

In this step, you will verify that the expression is calculating the cylinder power correctly. If the expression is correct, there will be relatively no difference between the calculated power and the power calculated from the translational joint's relative velocity multiplied by its driving force.

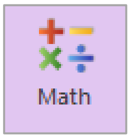
To run a simulation and plot the results:

1. Run the simulation as described previously.
2. Click the **Plot Results**.
3. In the **Database** window, expand **Joints→TraJoint1@HydraulicCylinder**.





4. Double-click **Vel1_Relative** to plot the joint velocity.
5. In the Database window, double-click **Driving_Force**.



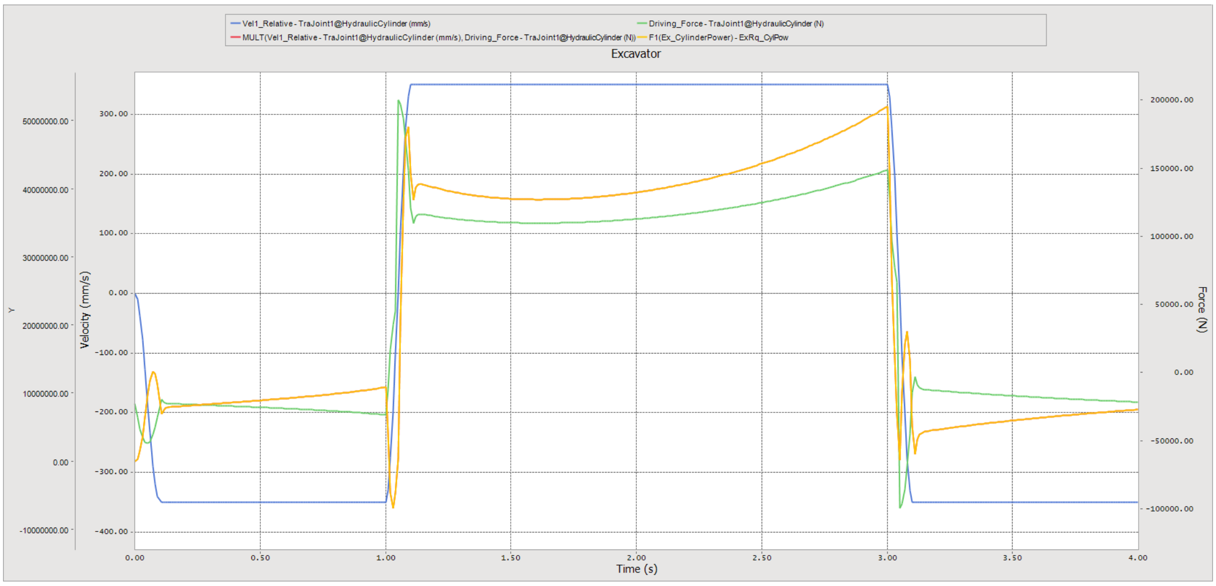
6. Multiply these two curves together using the **Simple Math** tool:
   - From the **Analysis** Group in the **Tool** Tab, click **Simple Math**.
   - Select the **Multiply : F1 * F2** option in Operation Type.

- Leave **Source Curve1 (F1)** as **Vel1_Relative** but change **Souce Curve2 (F2)** to **Driving_Force** by selecting it from the drop-down list, as shown in the figure on the right.
- Click **Execute** to create the new curve.

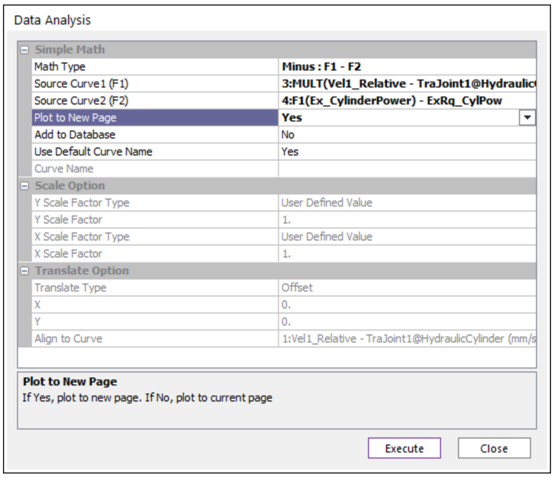
Now plot the expression for cylinder power.

7. In the Database window, expand **Request→Expressions→ExRq_CylPow**.
8. Double-click **F1(Ex_CylinderPower)**.

The two curves are practically identical as shown in the following figure.



To verify that the two curves really are identical, you will calculate their difference using the Simple Math tool again.

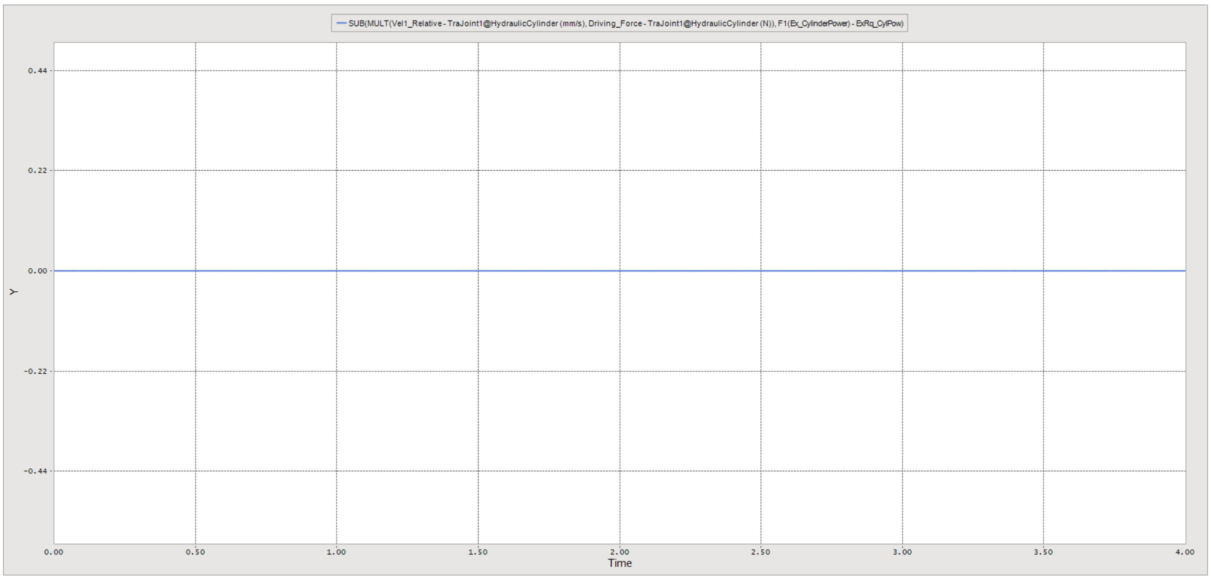


9. Open the Simple Math tool as before but this time:

   - Change the operation to Minus: **F1 – F2**.
   - Set **Source Curve1 (F1)** as **MULT(Vel1_Relative-TraJoint1@...)**,and **Souce Curve2 (F2)** to **F1(Ex_CylinderPower)** by selecting it from the drop-down list, as shown in the figure on the right.
   - Set **Plot to New Page** to **Yes** and **Execute**, so you will see an uncluttered curve for the error.

   The setup of the Simple Math dialog box should look like the figure on the right.

Tip: You can change the title of the plot by right clicking to show Edit title and then changing the title text in the dialog box that appears.



Chapter

7

Calculating the Range of Motion

Task Objective

In this chapter, you will create an expression that will calculate the range of motion of the bucket. This range of motion will change when you adjust the crank link length and the bucket joint angle, because these two parametric values change the geometry of the four-bar transmission at the end of the dipper. This happens even though the hydraulic cylinder length and range of travel do not change.

When you perform the design study in the next chapter, you want to include this range of motion as a performance index, so you need an expression to calculate it. The expression for finding the largest rotation in the negative direction has already been created for you. Therefore, in this chapter, you will create a similar expression for finding the largest rotation in the positive direction and then put them together to calculate the total range of motion.

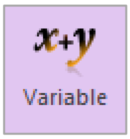
Estimated Time to Complete

15 minutes

Calculating the Maximum Positive Rotation of the Bucket

In this section, you will create a new variable equation to calculate the maximum positive rotation of the bucket.

To calculate the rotation:



1. From the **Equation** group in the **SubEntity** tab, click **Variable Equation**.
2. In the **Variable Equation List** dialog box, click **Create**.
3. Change the name to **VE_MaxPosRot**.
4. In the **Expression** portion of the **Variable Equation** dialog box, click **EL**.
5. In the **Expression List** dialog box, click **Create**.

   This expression needs to be able to reference itself so it can keep track of whether the current rotation is greater than previous values at each step of the simulation. Therefore, you need to enter a placeholder expression that you will then modify once you have finished creating the variable equation.
6. Change the name of the expression to **Ex_MaxPosRot** and enter the number **0** as the expression itself.
7. Click **OK** four times.
8. Return to the list of expressions in the **Database** window, and modify **Ex_MaxPosRot**:
   - Click **E** to change the expression to the following:
   - IF(VARVAL(3)-AZ(1,2):AZ(1,2),VARVAL(3),VARVAL(3))

Tip: **VARVAL** evaluates the variable expression in its argument, which you will set to **VE_MaxPosRot** in the next step. This VE, in turn, calls the expression **EX_MaxPosRot**, which you are currently editing. This loop means that VARVAL(3) will provide the current value of this expression. The IF statement above should be read as follows:

If the current value of this expression minus the current rotation of the bucket (VARVAL(3)-AZ(1,2)) is less than zero, then this expression equals the current bucket rotation (AZ(1,2)). Otherwise, the expression should remain unchanged.

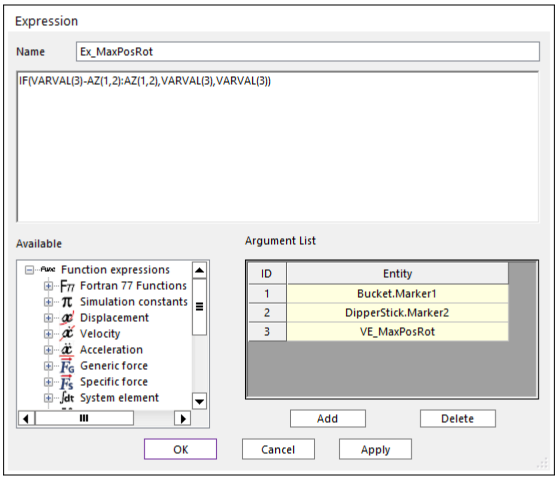
For more information on the VARVAL and AZ functions, see the **RecurDyn Help**.

   - Now add the following two entities to the Argument List in this order:
     - **Bucket.Marker1**
     - **DipperStick.Marker2**

- **VE_MaxPosRot**

After making these modifications the Expression dialog box should appear as follows.

9. Click **OK**.



Calculating the Range of Motion of the Bucket

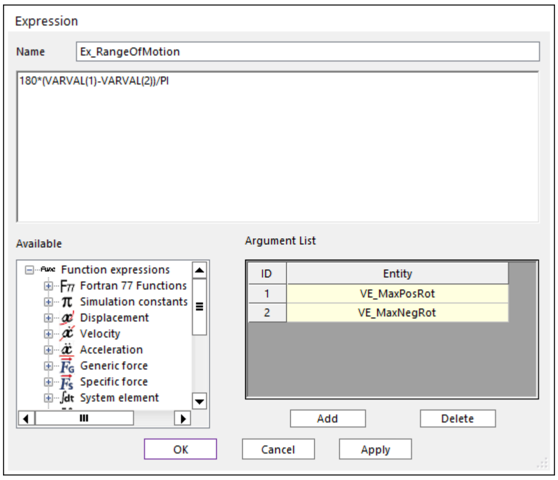
You will calculate the range of motion of the bucket by subtracting the maximum negative rotation from the maximum positive rotation. This is the same as adding their magnitudes because the negative rotation was created so it retains its negative sign. The rotation is also converted from radians to degrees.

To calculate the range of motion:

1. In the still open Expression List dialog box, click **Create** and set up the expression with the following characteristics:
   - Name: **Ex_RangeOfMotion**
   - Expression: **180*(VARVAL(1)-VARVAL(2))/PI**
   - Entities: **VE_MaxPosRot and VE_MaxNegRot**

   The Expression dialog box should appear as shown on the right.

2. Click **OK** twice to exit.

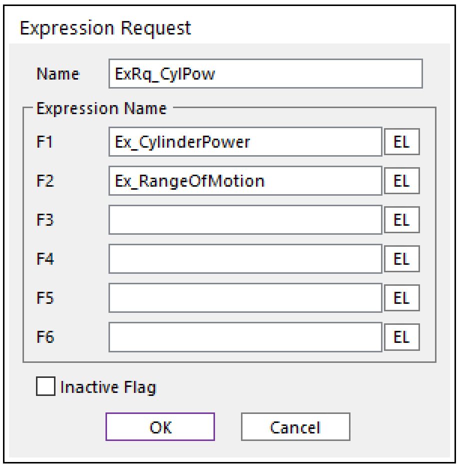


Adding the New Expression to the Request

To add the new expression:

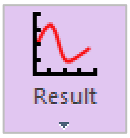
1. In the Database window, double-click the request **ExRq_CylPow**.
2. In the Request List dialog box, click **Rq** to modify the request.
3. Click **EL** in the **F2** row.
4. Select **Ex_RangeOfMotion** from the **Expression List** dialog box.
5. Click **OK** three times to exit.

The **Expression Request** dialog box should look like the figure shown on the right.



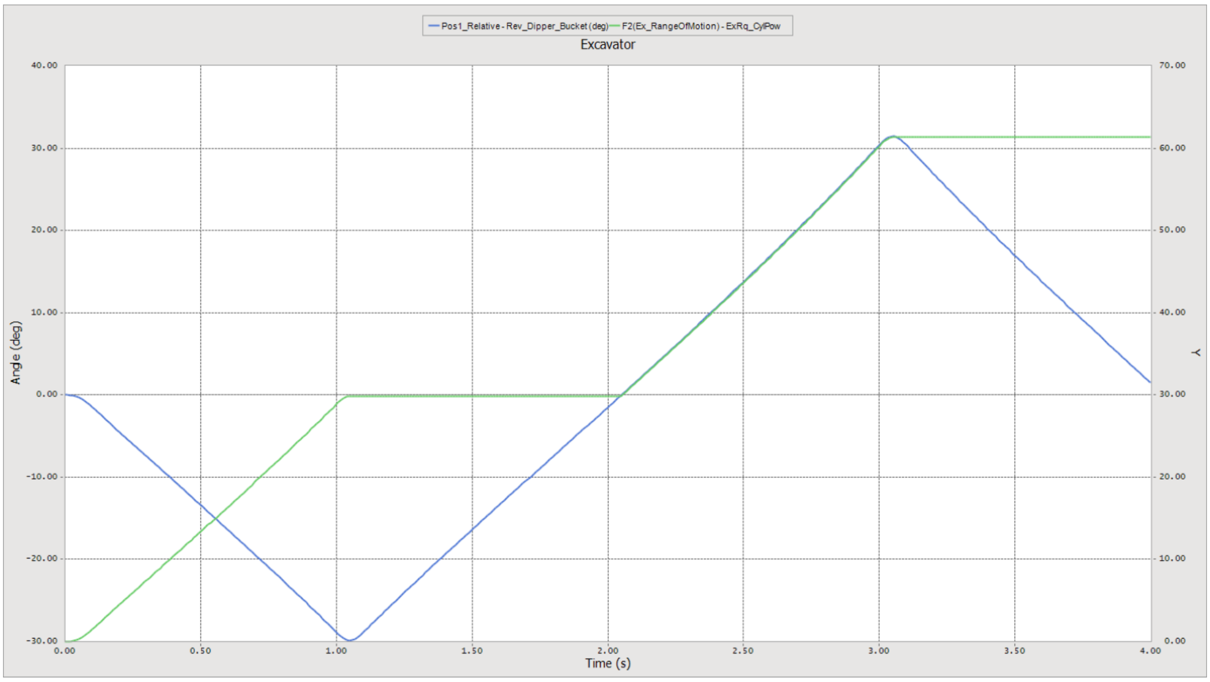
Plotting the Expression to Verify Results

To plot the results:



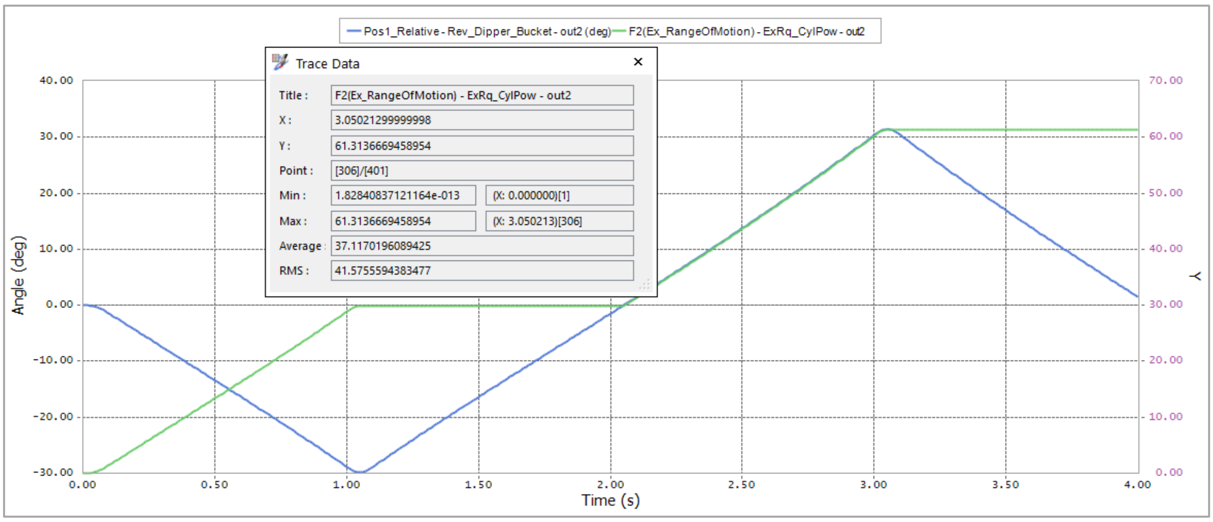
1. Run the simulation as before. (**Tip: End time: 4, Step: 400**)
2. Click the **Plot Result** to open the plotting environment.
3. To plot the rotation of the bucket joint, in the Database window, expand **Joints→ Rev_Dipper_Bucket**, double-click **Pos1_Relative**.
4. To plot Ex_RangeOfMotion, in the Database window, expand **Request→Expressions→ ExRq_CylPow** and double-click **F2(Ex_RangeOfMotion)**.

   The next figure shows the desired plot.



A visual inspection of the curves indicates that the function is working correctly, but to view a numerical comparison, uses the **Trace Data** tool.

5. Click the **TraceData** of the **Tools** group in the **Home** tab and scroll around the two curves. Placing the cursor on the expression's curve shows that the maximum value is 61.34 degrees. Placing the cursor on the red curve shows the following.



The maximum rotation is 31.47 and the minimum is -29.84 so the range of motion should be 31.47 deg - (-29.84 deg) = 61.31 deg. The small difference is explainable because the expression evaluates every single point in the entire simulation while the plot only provides the requested number of data points (400 in this case). The bucket must have rotated just a little bit farther than what is reported in the plot data.

6. Return to the modeling window. (**Tip: File → Close**)

7. Save your model.

Chapter

8

Running and Analyzing a Design Study

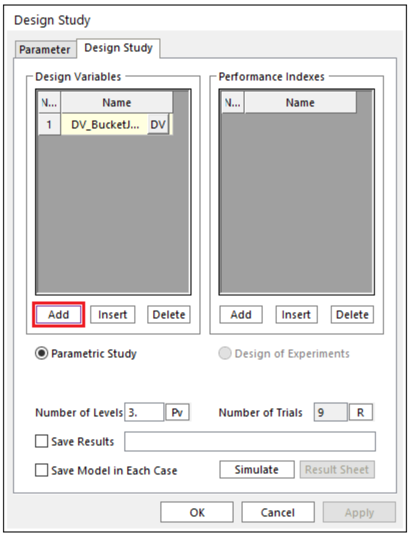
Task Objective

In this chapter, you will set up and run a design study to look at the effects of the crank length and bucket joint angle on the range of motion and cylinder power as defined previously. You will run a design of experiments with three levels for each variable and then look at the results.

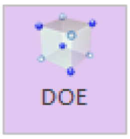
Estimated Time to Complete

15 minutes

Setting Up the Design Variables

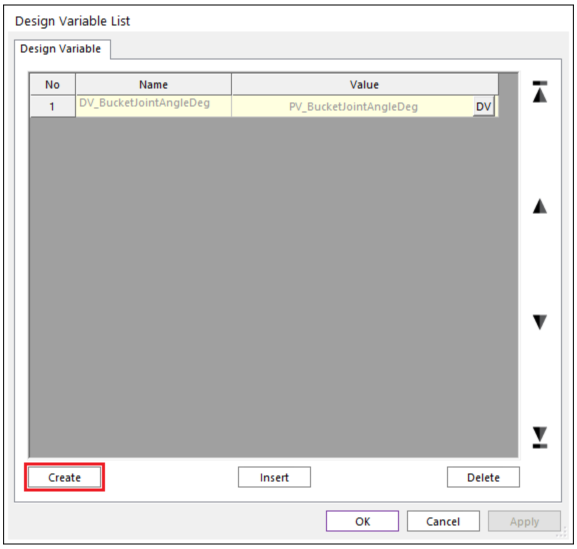


To set up the design variables:

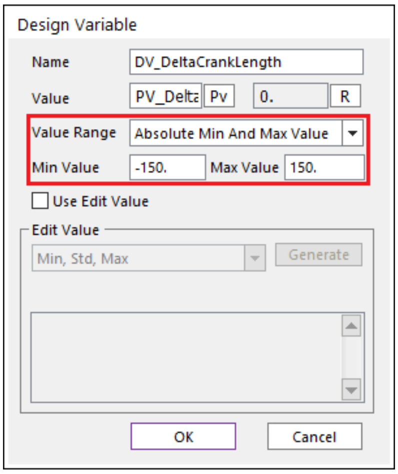


1. From the **Simulation Type** group in the **Analysis** tab, click **DOE**.

   The Design Study dialog box appears as shown on the right. Note that the bucket joint angle is already defined as a design variable.
2. Click **Add** in the **Design Variables** area.
3. In the **Design Variable List** dialog box that appears, click **Create**.
4. In the **Design Variable** dialog box that appears, change the name of the design variable to **DV_DeltaCrankLength**.
5. Click **PV** next to the **Value** text box and then select **PV_DeltaCrankLength** from the Parametric Value List dialog box (second dialog box on the right).
6. Click **OK**.

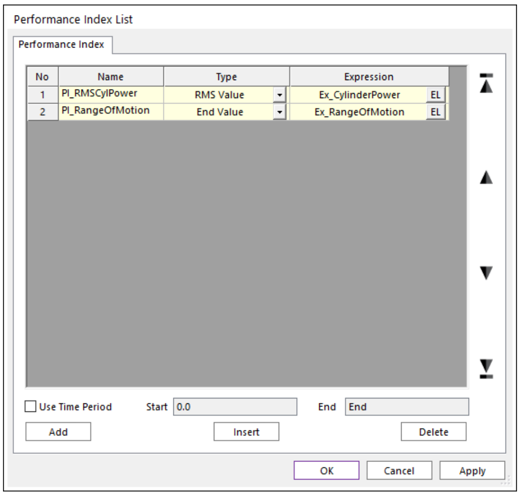


7. In the **Design Variable** dialog box, set **Value Range** to **Absolute Min and Max Value**.
8. Set **Min Value** to **-150.0** and **Max Value** to **150.0** as shown in the following dialog box.
9. Click **OK** twice to accept settings and close the dialog boxes, leaving the Design Study dialog box open.

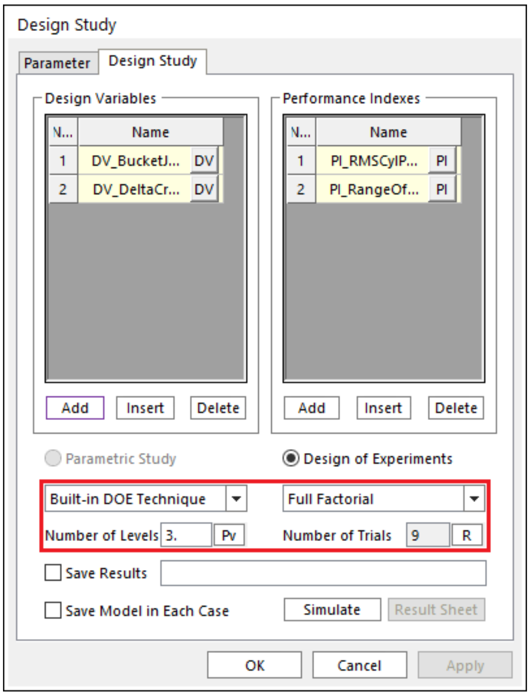


Defining the Performance Indexes

To define the performance indexes:



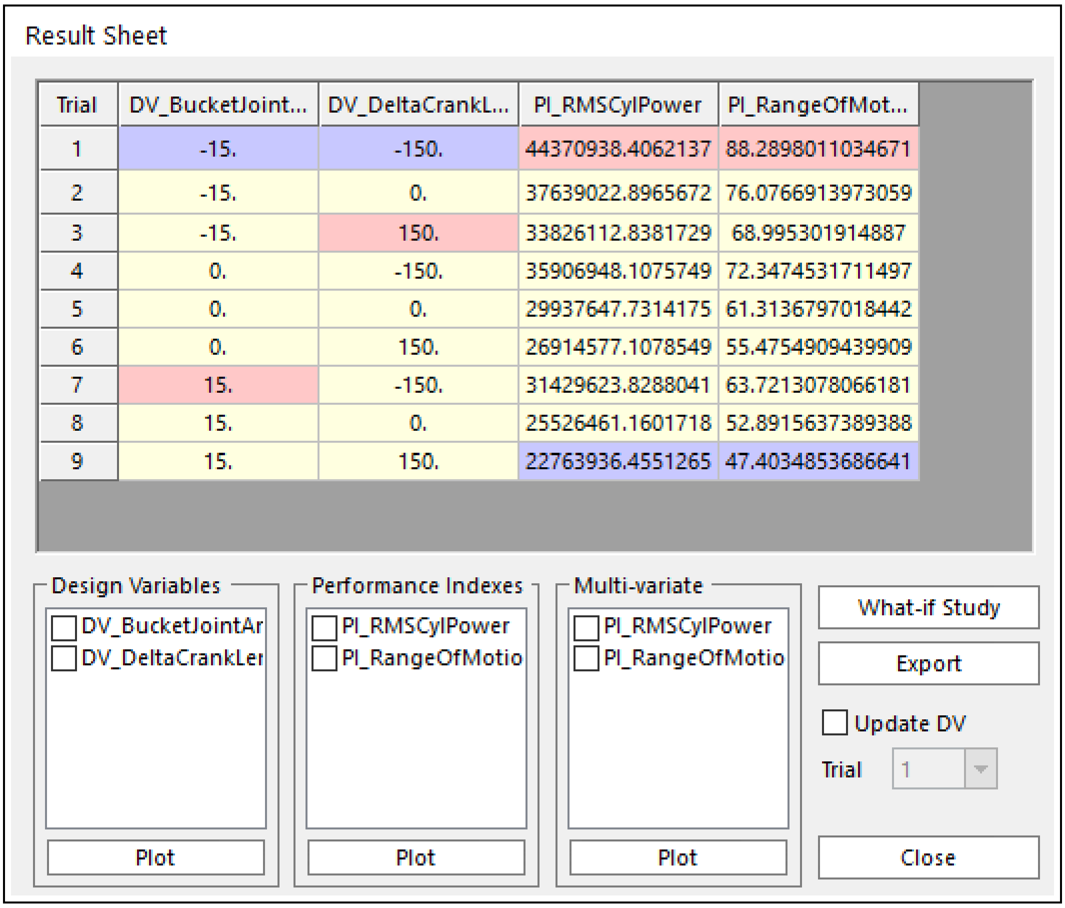
1. In the Design Study dialog box, in the Performance Indexes area, click **Add**.
2. In the Performance Index List dialog box:
   - Click **Add** and change its name to **PI_RMSCylPower**.
   - Set **Type** to **RMS Value** and choose **Ex_CylinderPower** as the expression using the same methods as before (**Tip**: Click **EL**, select the expression, click **OK**).
   - The **Performance Index List** now appears shown on the right.
   - Click **OK**.
3. In the Design Study dialog box, click **Add** to add another Performance Index:
   - Click **Add** again in the Performance Index List.
   - Change the name to **PI_RangeOfMotion**.
   - Leave Type as End Value.
   - Choose **Ex_RangeOfMotion** as the expression as shown in the figure on the right.
4. Click **OK** and keep the **Design Study** dialog box open.



Running the Design Study

To run the design study:

1. In the Design Study dialog box:
   - Leave the settings at Built-in DOE Technique and Full Factorial.
   - Set **Number of Levels** to **3** (see the figure).
2. Click **Simulate** to run the design study.
3. When the study is finished, click **Result Sheet** to see the results.



Trial	DV_BucketJoint...	DV_DeltaCrankL...	PI_RMSCylPower	PI_RangeOfMot...
1	-15.	-150.	44370938.4062137	88.2898011034671
2	-15.	0.	37639022.8965672	76.0766913973059
3	-15.	150.	33826112.8381729	68.995301914887
4	0.	-150.	35906948.1075749	72.3474531711497
5	0.	0.	29937647.7314175	61.3136797018442
6	0.	150.	26914577.1078549	55.4754909439909
7	15.	-150.	31429623.8288041	63.7213078066181
8	15.	0.	25526461.1601718	52.8915637389388
9	15.	150.	22763936.4551265	47.4034853686641

In the **Result Sheet**, the red and blue boxes indicate the minimum and maximum values, respectively, of each variable or result. The results for this study show that to obtain a reduction in the RMS cylinder power, you need to forfeit range of motion. This is quite reasonable when considered in terms of mechanical and geometric advantage. If you change the four-bar linkage so the mechanical advantage increases, you will be able to decrease the amount of force required from the actuator to resist a constant load at the bucket tip.

But an increase in mechanical advantage means that the geometric advantage will decrease. That is, the same actuator displacement will result in a smaller output displacement, and the result is a smaller range of motion. There is a fundamental trade-off between the range of motion and the actuator force (which affects the actuator power). To explore this trade-off further, you will perform a **What-if Study**.

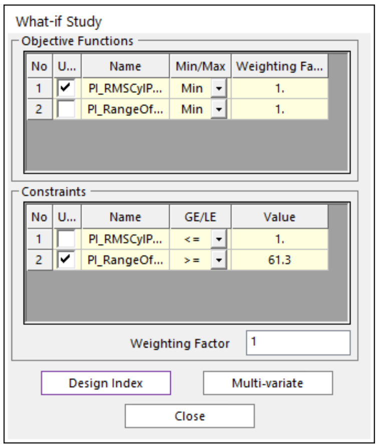
Performing a What-if Study

To perform a What-if Study:

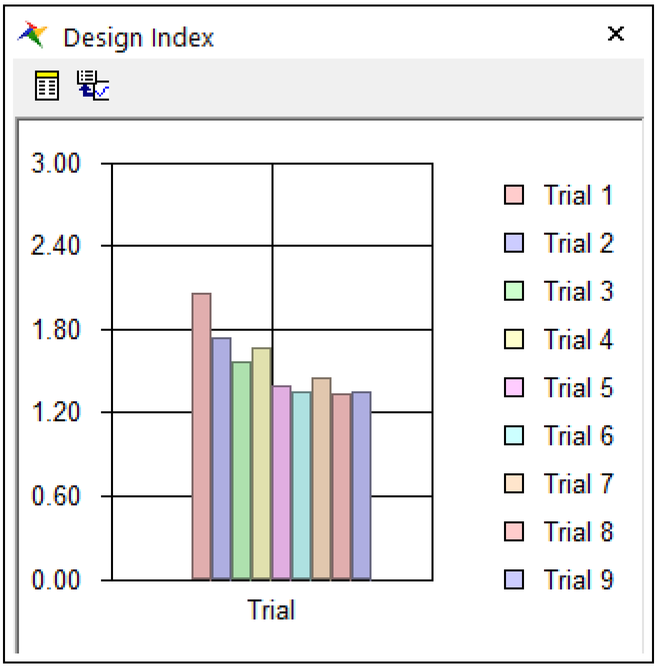
1. In the Result Sheet window, click **What-if Study**.

2. Set the Objective Function to minimize the power with a constraint that the range of motion be greater than or equal to 61.3 (the value for the baseline configuration). In the What-if Study dialog box:

   - Under **Objective Functions**, click Use next to **PI_RMSCylPower**.
   - Under **Constraints**, click **Use** next to **PI_RangeOfMotion**, change the **GE/LE** operator to **>=,** and enter **61.3** as the value.
   - The dialog box should now appear as shown on the right.



3. Click **Design Index** to see which configuration the best is based on this criterion.



   The design index is calculated so that the lowest value indicates the optimal design. More detailed information on the calculation of the design index can be found in the RecurDyn Help.

   Scrolling your cursor over the bars shows that Trial 8 is the lowest, followed by Trials 8 and then 9. All of these trials have lower powers, but they all have lower ranges of motion, which is in violation of the constraint. Increase the Weighting Factor for the constraints and see how that affects the optimal configuration.

4. Double the Weighting Factor for the constraints in the What-if Study dialog box.
5. Click **Design Index** again.

   Now the results show that Trial 5 is the lowest, followed by Trials 6 and 7. The **Result Sheet** shows that Trial 5 does lower the RMS power but has a small decrease in range of motion while Trial 6 has only a small increase in power while achieving an increased range of motion. Trial 9 is no longer viewed as one of the best designs due to its large violation of the range of motion constraint.

It appears that configurations 5, 6, and 7 are the ones most worth pursuing for further analysis, so they will be used in the next section where you will perform a batch run and then plot the results for all three configurations at the same time.

6. Close the **Design Index** window, **What-if Study** dialog box, and **Result Sheet** windows.
7. Click **OK** to close the **Design Study** dialog box.

Chapter 9

Running the Simulation in Batch Mode

Task Objective

In this chapter, you will set up three simulations that will run in batch mode. The simulations you will set up correspond to trials 5, 6, and 7 that were performed in the Design Study. You will then plot the hydraulic cylinder power and bucket motion.

Note that the results will be the same as the results from the design study, but there is an important reason to learn how to run batch simulations. The design study automates the running of a set of trials; however, the limitation is that only one design study can be run at a time. In batch mode, however, there is no limit to the number of simulations that can be run. For this reason, running RecurDyn in the batch mode is a good way to take advantage of nights and weekends.

Estimated Time to Complete

15 minutes

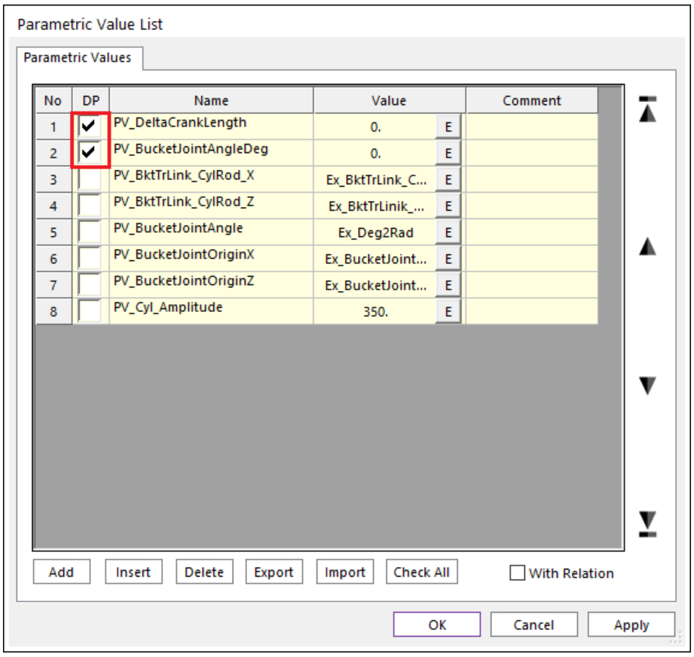
Setting up and Exporting the RecurDyn Design Parameter File

First, you need to tell RecurDyn which points, and values will be in the design parameters used in the batch simulations. This model only uses two parametric values (PVs) to make changes to the model, so you will use them here. Parametric points could just as easily be used in batch simulations, however, and the procedure is the same.

The one limitation to keep in mind is that only parametric points and parametric values at the main system level can be used. If parametric points or values are located in subsystems, they must first be set up at the main level and synchronized with parametric point connectors (PPCs) and parametric value connectors (PVCs) as discussed previously in this tutorial.

To set up and export the file:

1. In the **Database** window, double-click any PV to open the **Parametric Value List**.
2. Check the boxes in the **DP** column for **PV_DeltaCrankLength** and **PV_BucketJointAngleDeg** as shown in the figure on the right.
3. Click **OK**.
4. From the **File** menu, click **Export**.
5. Set the **Files of type** to **RecurDyn Design Parameter File (*.rdp)**.
6. Set the export directory where RycurDyn Model file is located.
7. Name the file **Excavator.rdp** and click **Save**.



This creates a number of files in your working directory. The RDP file is the database that tells RecurDyn where to find PVs and PPs during the batch simulations. You should also notice that. rpp and .rpv files were created for the main system and each subsystem that contains PPs and PVs. Because you have only defined two PVs at the system level, the only file with any information in it should be Excavator.rpv. The contents appear as follows when opened in Notepad:

!========================RecurDyn Parametric Value===================|

PV_DeltaCrankLength = 0.

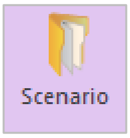
PV_BucketJointAngleDeg = 0.

This means that the current values for these design parameters are zero. When you write the batch file in the following steps, you will tell it to change these values and RecurDyn will update them automatically.

Setting Up and Exporting the RecurDyn Scenario File

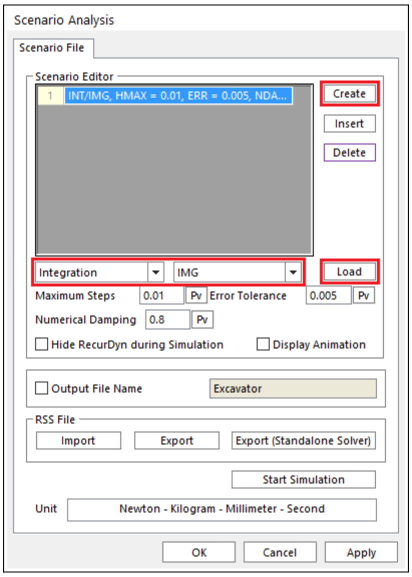
The RecurDyn scenario file (.RSS) tells RecurDyn how to run the simulation in batch mode. First, you will specify which integrator to use and then specify the simulation settings, including the simulation time and number of steps.

To set up and export the scenario file:

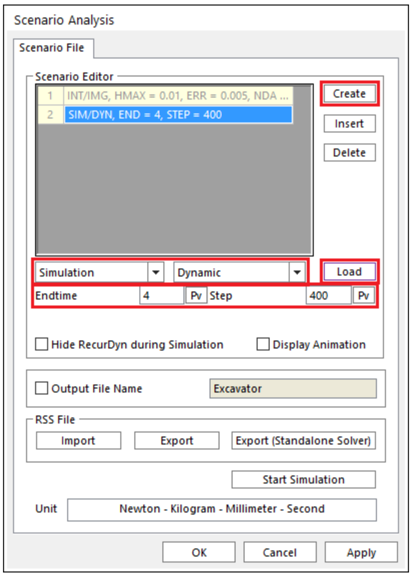


1. From the **Simulation Type** group in the **Analysis tab**, click **Scenario**.
2. In the Scenario Analysis dialog box, click **Insert**.
3. Set the integration settings by changing the drop-down boxes under the Scenario Editor:
   - Set the first box to **Integration**.
   - Set the second to **IMG**. (This indicates the integrator to be used.)
   - Use the default values for all other options.
4. Click **Load** to load these settings into the empty row in the Scenario Editor.

   Your window should look like the first one on the right.



5. Set up the simulation settings:
   - Click **Create** again to create a new row in the Scenario Editor.
   - Change the left-most drop-down box to **Simulation**.
   - Change the **Endtime** to **4** and the **Steps** to **400**.
   - Click **Load**.
   - Your window should now look like the second one on the right.



With the settings set, you will now test the scenario.

To test the scenario:

1. Click **Start Simulation**.
2. It runs the same simulation as in the last section.
3. In the **Scenario Analysis** dialog box, click **Export**.
4. Save the file as **Excavator.rss** (where RecurDyn Model file is located).

   Open the contents of the file, in a text editor, such as Notepad. It looks like the following:

   INT/IMG, HMAX = 0.01, ERR = 0.005, NDA = 0.8

   SIM/DYN, END = 4, STEP = 400

   STOP

Creating the Batch File and Running the Simulation

Now you will create the batch file.

To create the batch file:

1. Open a text editor, such as Notepad, and enter the following text:
   - mkdir out
   - "<Install Dir>\Help\DP_Study\ConvertDP.exe" /clean /convert Excavator.rdp PV_DeltaCrankLength=0 PV_BucketJointAngleDeg=0
   - "<Install Dir>\Bin\Recurdyn.exe" "Excavator.rdyn" /rdp Excavator.rdp /rss Excavator.rss /out out\out1
   - "<Install Dir>\Help\DP_Study\ConvertDP.exe" /clean /convert Excavator.rdp PV_DeltaCrankLength=150 PV_BucketJointAngleDeg=0
   - "<Install Dir>\Bin\Recurdyn.exe" "Excavator.rdyn" /rdp Excavator.rdp /rss Excavator.rss /out out\out2
   - "<Install Dir>\Help\DP_Study\ConvertDP.exe" /clean /convert Excavator.rdp PV_DeltaCrankLength=-150 PV_BucketJointAngleDeg=15
   - "<Install Dir>\Bin\Recurdyn.exe" "Excavator.rdyn" /rdp Excavator.rdp /rss Excavator.rss /out out\out3

   Note that you can directly copy the above text in the batch file if you have access to the digital form of this tutorial document.
2. Now save the file as **Excavator.bat** and close it.

   The following is a brief description of what each command means and does:
   - mkdir out
   - Creates a directory named **out** in your working directory. This is where RecurDyn will store the results of the simulations.

   "<Install Dir>\Help\DP_Study\ConvertDP.exe"

- Runs the **ConvertDP** executable file, which is in the standard RecurDyn installation in the **Help>DP_Study** folder. If your installation directory is different than that specified above, you will need to change this line appropriately.
- **/clean /convert Excavator.rdp PV_DeltaCrankLength=0 PV_BucketJointAngleDeg=0**
  These are the arguments to the **ConvertDP** executable. They tell RecurDyn to use the **Excavator.rdp** file and set the value of PV_DeltaCrankLength to 0 and PV_BucketJointAngleDeg to 0. Because these are already the default values, they are not really necessary but are included to show the pattern.
- **"<Install Dir>\Bin\Recurdyn.exe"**
  This provides the full path to the RecurDyn program. This should be the default for the version 8.1 installation, but if you are running a different version or specified a different location for the program, you should update your text appropriately.
- **"Excavator.rdyn"**
  Tells RecurDyn which model (.rdyn) file to use for the simulations. If you have renamed your file when saving, you should update this text accordingly. Moreover the batch file you just created should be located as same folder as Excavator.rdyn file saved, or else you must set full directory as following.
- **/rdp Excavator.rdp /rss Excavator.rss**
  Defines the file names for the .RDP and .RSS files you created earlier.
- **/out out\out1**
  Tells RecurDyn to store the output files in the out folder you created in the first line of the batch file and with the filename out1. In this case, uncheck **the Create Output Folder** of the **Simulation Model Setting** group in the **Home** Tab.

The remainder of the batch (.bat) file contains three repetitions of the same code except the PVs are set to the values from configurations 6 and 7 in the design study you performed and the output files are named out2 and out3, respectively.

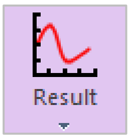
3. To run the simulation, first close your model in RecurDyn. You do not need to exit RecurDyn, but the batch simulation cannot run if the model it is trying to update and simulate is already open.
4. Double-click the **Excavator.bat** file you just created.

   After double-clicking the file, a DOS command prompt opens and the first five lines (three commands: mkdir, ConvertDP.exe, and recurdyn.exe) of code appear. RecurDyn opens, runs the simulation, and then closes.

   The next ConvertDP.exe and recurdyn.exe appear, and RecurDyn runs and closes again, and then once again.

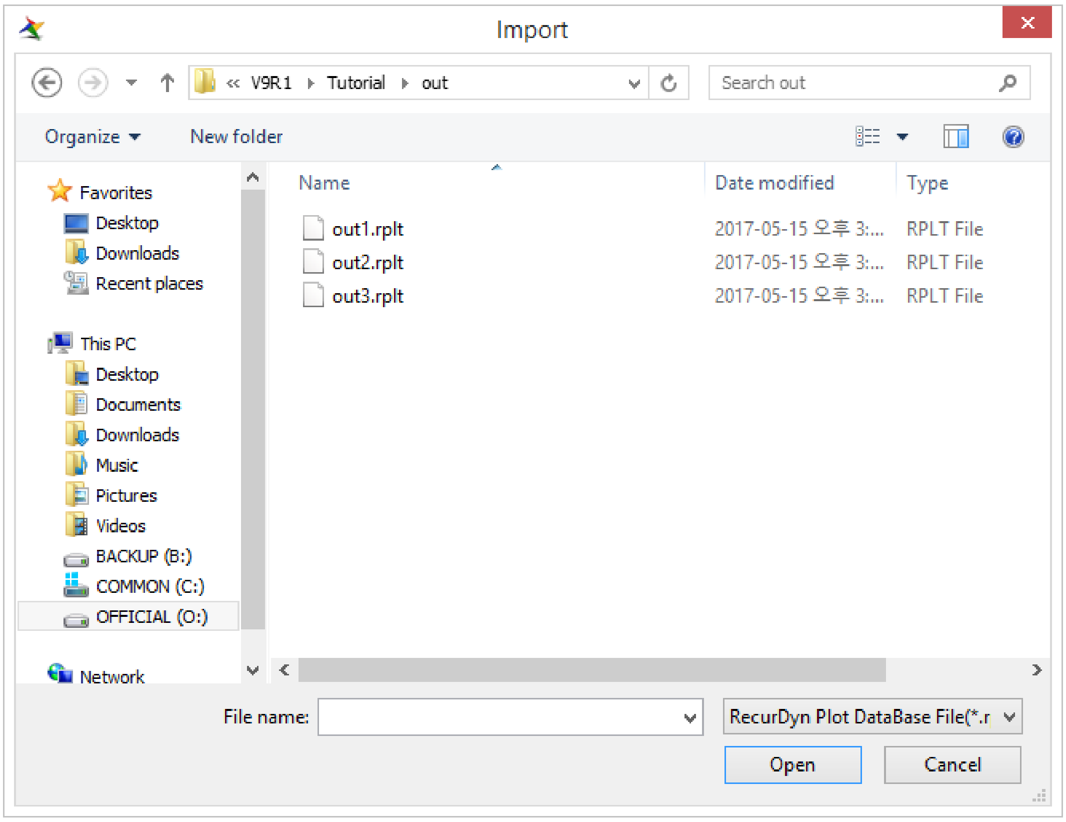
Plotting the Results

To plot the results:



1. Open your RecurDyn model again and click on the **Plot Results** tool to enter the plotting environment.
2. From the **File** menu, click **Import** and then click **Import File**.
3. Double-click the **out** folder and then select all three output files.

Tip: You can do this easily by first clicking **out3.rplt** and then, while holding down **Shift** key, clicking **out1.rplt**.



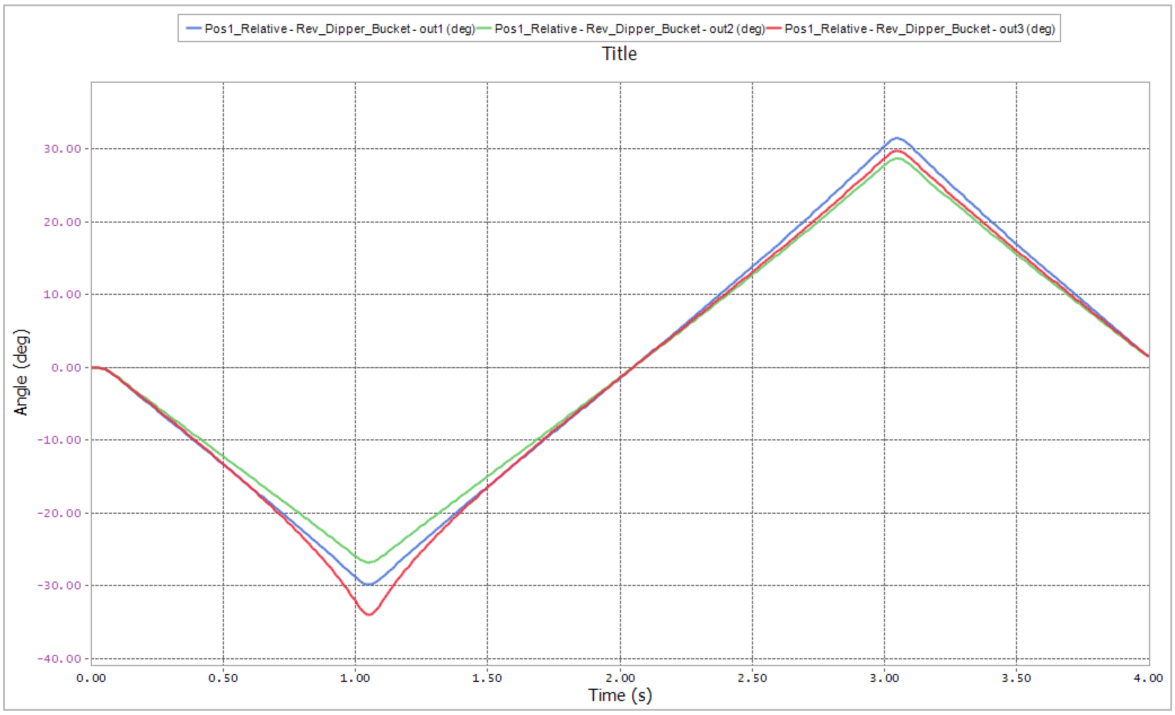
4. Click **Open**.

   Now you will plot the bucket rotation and hydraulic cylinder power for all three configurations.

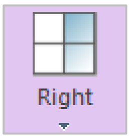
5. In the **Database** window, expand **Joints→Rev_Dipper_Bucket** right-click **Pos1_Relative**, and select **Multidraw**.

   Three curves are added to the plot. Note that, depending on how you selected the RecurDyn Plot database files in Step 3, **out1, out2**, and **out3** may be in a different order in the Database window.

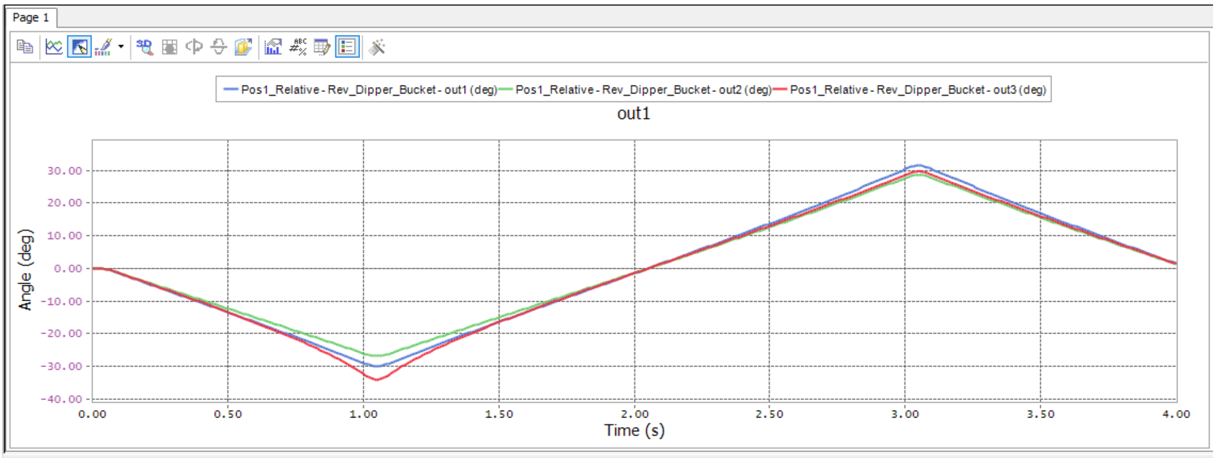
   Your plot window appears as shown next.

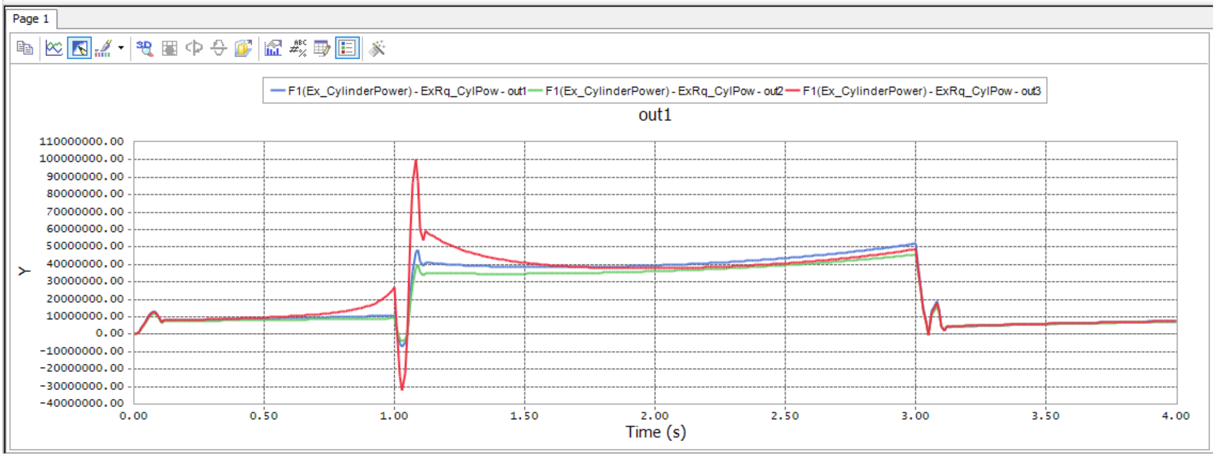


This plot shows that the first (default) configuration has the largest positive rotation, but the third one has the largest negative rotation, giving it the largest range of motion (as you discovered earlier in the Design Study). The second configuration has about the same positive rotation as the third, but it has significantly smaller negative rotation giving it the smallest range of motion.



6. Click **Show Right Windows** of the View group to open a new plot window.
7. Click the lower plot anywhere in the window to activate it.
8. In the Database window, expand **Request→Expressions→ExRq_CylPow** and right-click **F1**, and select **Multidraw**.





From this plot, you see that the power for the third configuration is very large at the point of maximum negative rotation. This is indicative of a poor mechanical advantage, meaning that a large input (hydraulic cylinder) force is required to sustain a significantly smaller output force (bucket tip load). Because a poor mechanical advantage corresponds to a good geometric advantage, it makes sense that at that point in the simulation the third configuration has a larger rotation of the bucket than the other configurations for the same input displacement.

If you would like, you can change the titles, axis labels, and legend entries for the plots so they look like the final versions shown on the next page and on the first page of the tutorial.

To change a title or axis label:

1. Double-click the title or axis label.
2. Enter new text.

To change the legend:

1. Double–click the legend.
2. Click Change Name.

You can reposition any of the above items by simply clicking and dragging

Thanks for participating in this tutorial!