Getting the Most Out of Inventor Professional

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MD6091

Learning Objectives

At the end of this class, you will be able to:

- Learn how to use Inventor Simulation software to validate your design.

- Explore Routed Systems to incorporate piping and electrical harnesses in your design.

- Understand plastic part design functionality in Inventor.

- Learn how to make the most out of Product Design Suite Professional and Ultimate

This handout serves as a supplemental hands-on exercise to explore one of the primary topics in the course, leveraging Dynamic Simulation, and Model Based Design Automation with iLogic in Inventor Professional. Demo dataset is available for download via the course materials.

Hands On Topic #1: Simulation

Exercise: Dynamic Simulation

In this exercise, you utilize Dynamic Simulation to analyze the mechanical linkage. Specifically, you analyze the hydraulic cylinder and the force required to move the linkage. Based on this information, you can determine the proper size of the cylinder.

Please find video based instruction for this exercise here: Instruction Video

Preliminary Setup

For this exercise, make sure that *PrDS Test Drive.ipj* is set as the active project in Inventor prior to starting the steps. Please use the following steps to set the project file.

- Open Autodesk Inventor.
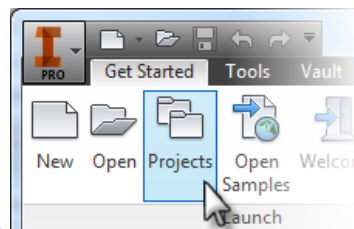
- On the Getting Started ribbon, select **Projects**.
- At the bottom of the Projects dialog, click Browse and navigate to **C:\PrDS Test Drive**.

- Double click on the file called **PrDS Test Drive.ipj**.
- Confirm that the PrDS Test Drive project appears in the Projects dialog and is marked with a check.
- Click **Done**.

Open PrDS Test Drive / Inventor Simulation / _**Assy, Chassis, Front DS.iam**
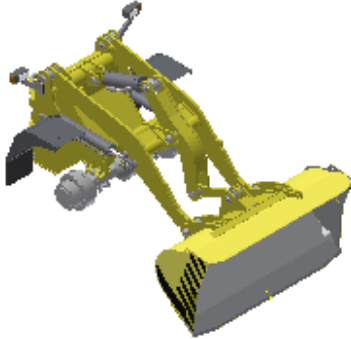
In the browser, under Representations > Level of Detail, right-click **Dynamic Simulation**. Click **Activate**.
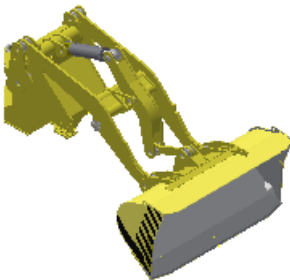
Press the **Shift** button and the **Scroll** wheel to Orbit and review the components that are now active in the assembly.
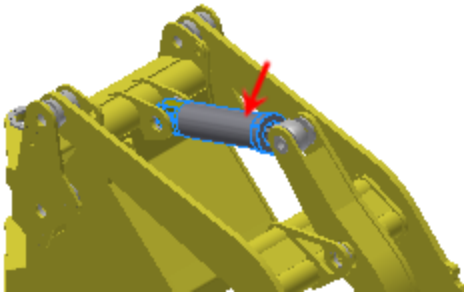
On the Environments tab, Begin panel, click **Dynamic Simulation**.

If the Dynamic Simulation dialog box displays asking if you want to start the Dynamic Simulation tutorial now, click **No**.

In the Dynamic Simulation browser, collapse the Grounded level so it displays as shown.
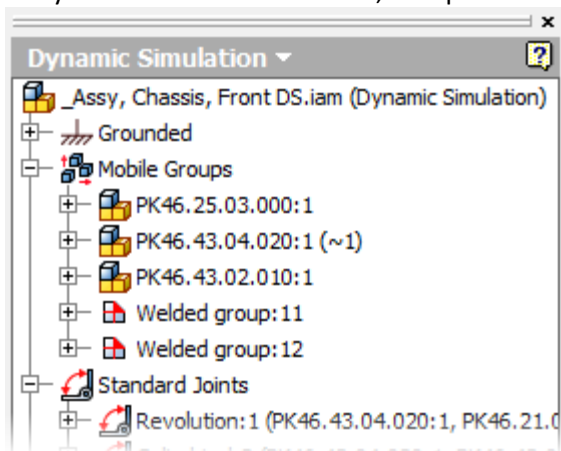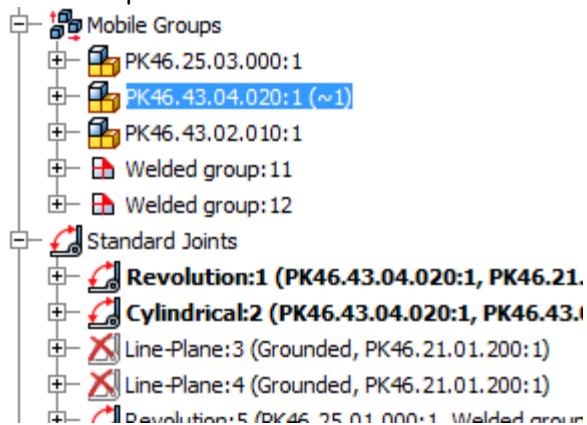

In canvas, select the identified hydraulic cylinder.

In the browser:

- Under Mobile Groups, notice which assembly appears in bold text to identify it as being selected.
- Under Standard Joints, notice which joints appear in bold text to indicate the joints that correspond to what is selected.
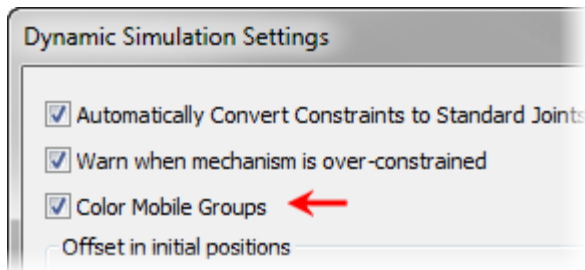

In the browser, under Standard Joints, select Cylindrical:2. Notice the two mobile groups that correspond to this joint.
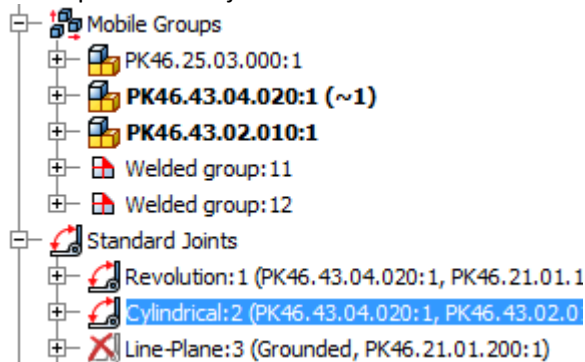
On the Dynamic Simulation tab, Manage panel, click **Simulation Settings**.
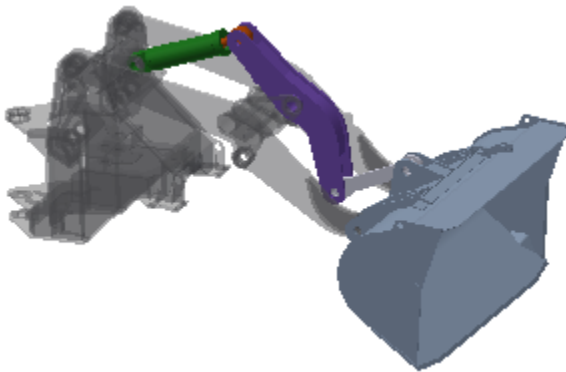
In the Dynamic Simulation Settings dialog box:

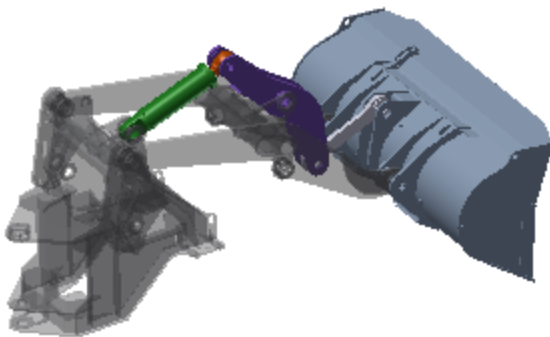- Select the **Color Mobile Groups** check box.
- Click **OK**.

In the Dynamic Simulation warning dialog box asking if you want to overwrite the existing view representation, click **Yes**.

Review the change to the display. Only the components that are in the mobile groups appear in color. The other components appear translucent.
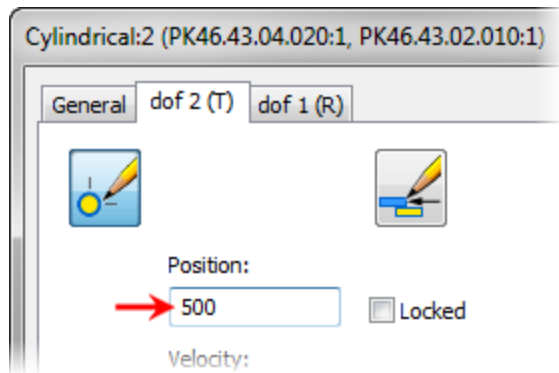

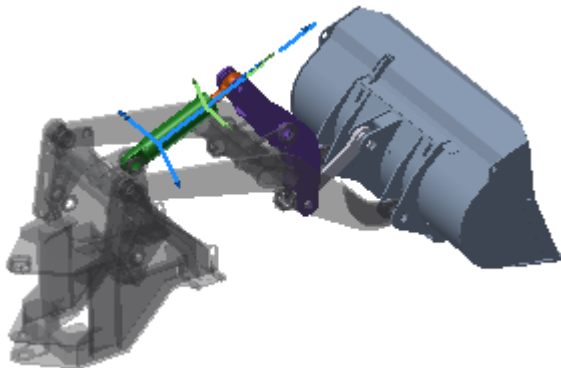Orbit the display to a view approximately as shown.

In the browser, under Standard Joints, right-click **Cylindrical:2**. Click **Properties**.

In the Cylindrical:2 dialog box:

- On the **dof 2 (T)** tab, in the Position field, enter **500**.
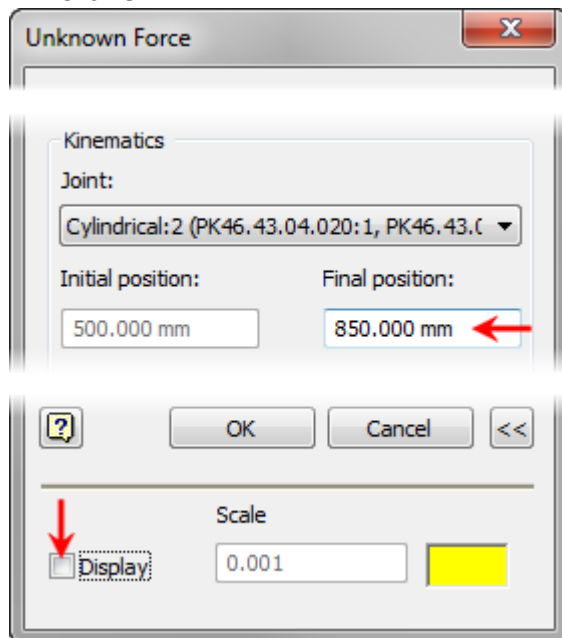- Click **OK**.

In canvas, review the change to the stroke of the hydraulic cylinder.

On the Dynamic Simulation tab, Results panel, click **Unknown Force**.
In the Unknown Force dialog box:
- In the Final Position field, enter **850**.
- In the expanded area of the dialog box, clear the Display check box.
- Click **OK**.

As the results are being calculated, reposition the Dynamic Simulation – Output Grapher so you can see the assembly in canvas and the resulting graph.
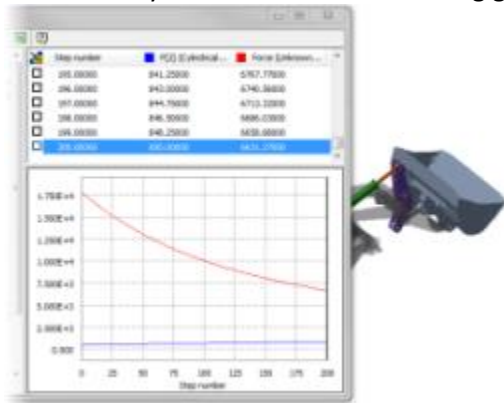
To identify the step with the largest force, in the Output Grapher, right-click in the Force results column. Click **Search Max**.
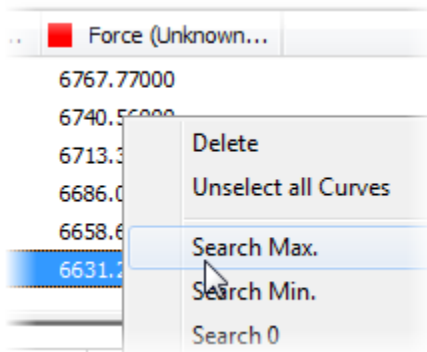

Review the results.

- In the Output Grapher, notice that step 0 is now selected and it has a force greater than **17,800**. This is the step with the highest required force because it is the position with the least mechanical advantage.
- In canvas, notice that the position updated to match that distance setting.

To review more information about the force curve:

- Right-click in the Force results column. Click **Curve Properties**.
- After reviewing the values in the Dynamic Simulation – Properties dialog box, click **Cancel**.
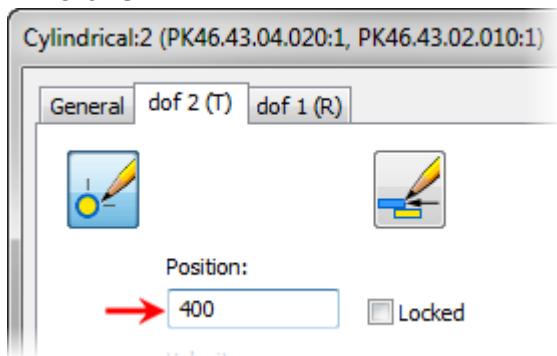
Close the Output Grapher.

In the Simulation Player dialog box, click **Construction Mode**.
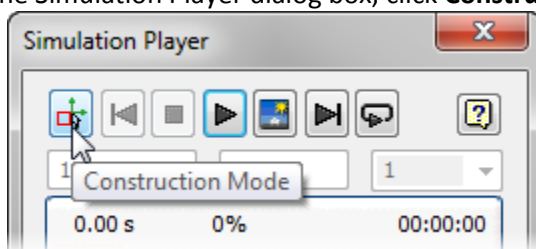
To change the starting position of the simulation:

- In the browser, under Standard Joints, right-click **Cylindrical:2**. Click **Properties**.
- In the Cylindrical:2 dialog box, dof 2 (T) tab, in the Position field, enter **400**.
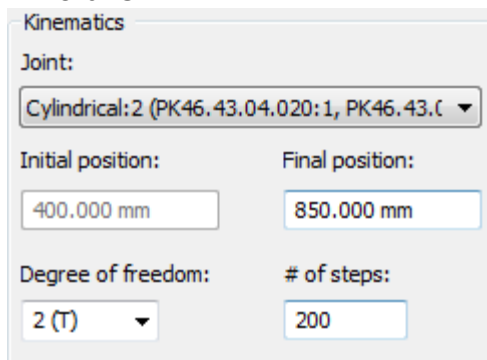
- Click **OK**.

On the Dynamic Simulation tab, Results panel, click **Unknown Force**.
In the Unknown Force dialog box:

- Review the value in the Initial Position field.
- Ensure that the value in the Final Position field is still **850**.
- Click **OK**.

As the results are being calculated, reposition the Dynamic Simulation – Output Grapher so you can see the assembly in canvas and the resulting graph.
To identify the step with the largest force, in the Output Grapher, right-click in the Force results column.
Click Search Max.
Review the results.

- In the Output Grapher, notice that step 0 is now selected and it has a force greater than **25,700**. The required force is now greater than before because this position has even less mechanical advantage to push the linkage than when the hydraulic cylinder is extended 100 mm farther out.
- In canvas, notice that the position updated to match that distance setting.
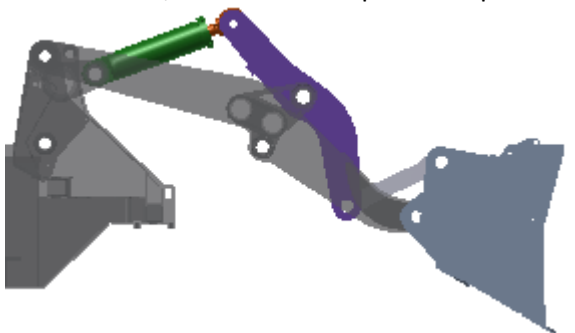
To review more information about the force curve:

- Right-click in the Force results column. Click **Curve Properties**.
- After reviewing the values in the Dynamic Simulation – Properties dialog box, click **Cancel**.
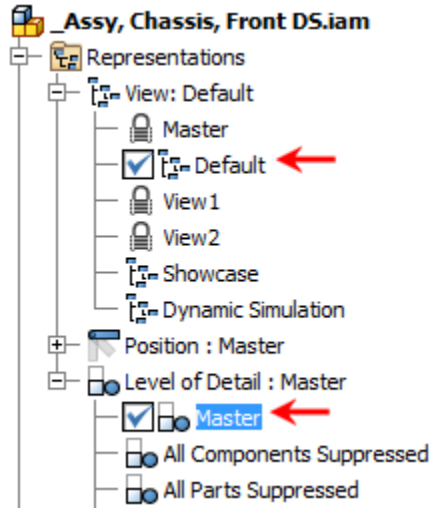
Close the Output Grapher.

In the Simulation Player dialog box, click **Construction Mode**.

On the Dynamic Simulation tab, Exit panel, click **Finish Dynamic Simulation**.

To see all of the components in the design, in the browser, under Representations:

- Under View, right-click **Default**. Click **Activate**.
- Under Level of Detail, double-click Master.

Close all files. Do not save changes.

Hands On Topic #2: Model Based Automation with iLogic in Inventor Professional.

ILogic

### Exercise1: Model Based Automation

In this exercise, you develop iLogic code to configure a model by modifying several features that capture a broader design concept.

Please find video based instruction for this exercise here: Instruction Video

Preliminary Setup

For this exercise, make sure that *PrDS Test Drive.ipj* is set as the active project in Inventor prior to starting the steps. Please use the following steps to set the project file.

- Open Autodesk Inventor.
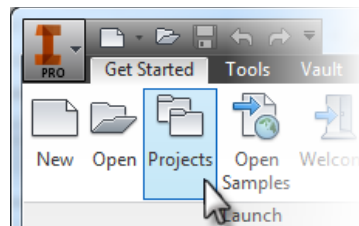
- On the Getting Started ribbon, select **Projects**.

- At the bottom of the Projects dialog, click Browse and navigate to **C:\PrDS Test Drive**.
- Double click on the file called **PrDS Test Drive.ipj**.
- Confirm that the PrDS Test Drive project appears in the Projects dialog and is marked with a check.
- Click **Done**.

In Autodesk Inventor, Navigate to: \Inventor iLogic\Exercise_1 and open **iLogic_Part_START.ipt**
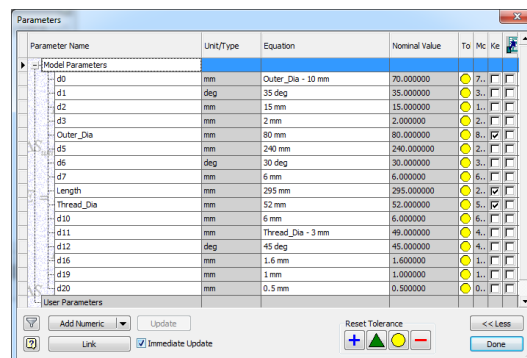

On the Manage Tab select **Parameters,** to view the model parameters dialog box.
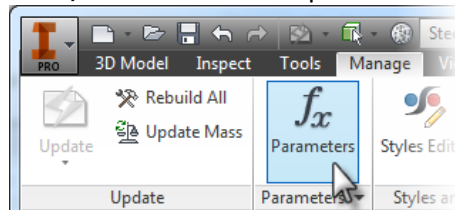
The values listed parametrically control the size and shape of the model. Some of the values are functions, mathematical formulas that reflect design intent for that feature. You can easily change the name of the parameters to make them easier to work with. Notice that several parameters have been renamed.

- *Outer_Dia* – The outer diameter of the shaft.
- *Length* – The overall length of the shaft.
- *Thread_Dia* – The diameter of the threaded shaft segment.
- *Thread_Length* – The Length of the threaded section of the shaft

Parameter Name	Unit/Type	Equation	Nominal Value	To	Mc	Ke
Model Parameters						
d0	mm	Outer_Dia - 10 mm	70.000000	7..		
d1	deg	35 deg	35.000000	3..		
d2	mm	15 mm	15.000000	1..		
d3	mm	2 mm	2.000000	2..		
Outer_Dia	mm	80 mm	80.000000	8..	☑	
d5	mm	240 mm	240.000000	2..		
d6	deg	30 deg	30.000000	3..		
d7	mm	6 mm	6.000000	6..		
Length	mm	295 mm	295.000000	2..	☑	
Thread_Dia	mm	52 mm	52.000000	5..	☑	
d10	mm	6 mm	6.000000	6..		
d11	mm	Thread_Dia - 3 mm	49.000000	4..		
d12	deg	45 deg	45.000000	4..		
d16	mm	1.6 mm	1.600000	1..		
d19	mm	1 mm	1.000000	1..		
d20	mm	0.5 mm	0.500000	0..		
User Parameters						

The renamed parameters are important to our shaft configuration. Notice how they have been marked as "key" parameters via the checkmarks in the *"Key"* column. A complex model could have many parameters to sort through. This designation allows us to filter the parameter list to make is easier to work with.

Filter the parameters dialog box to show only the key parameters.
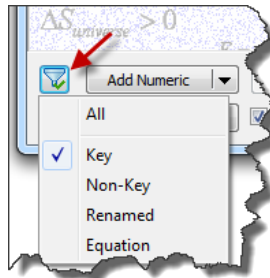
- Most of the parameters are now filtered from the list.
- You can use this filter to help organize your parameter list in several other ways as required.

In addition to the numeric parameters you will also create two additional "User Parameters"

- Click the small down-arrow and select *"Add True/False"*.
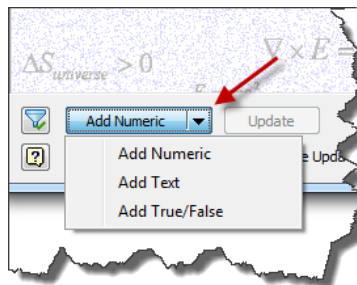
- Name the new parameter "**Fillet**" and mark it as "**Key**"
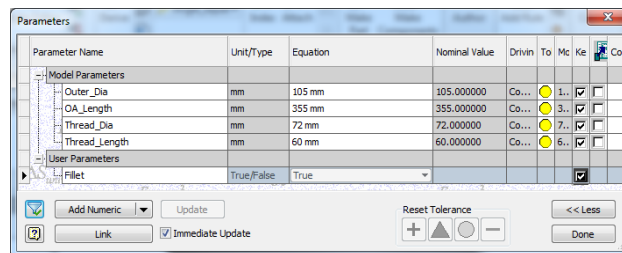
- The new parameter is created with an option to select "True" or "False" in the equation column. This parameter currently does not have a related rule associated with it, so modifying the value will not have any effect on the model.

Create a new Text parameter by selecting *"Add Text"*. Name the parameter *"Classification"* and mark it as *"Key"*
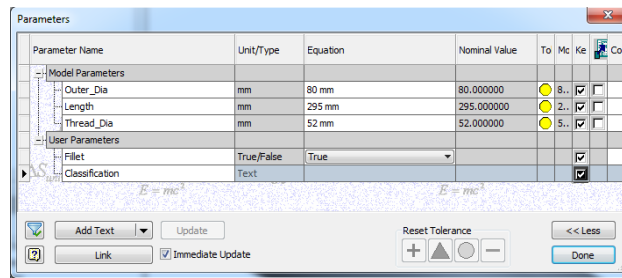
- The new parameter is created, but the equation is not yet defined
- Numeric and text parameters can be single or multi-value equations

Click your right mouse button in the equation cell of the "Classification" parameter and select "**Make Multi-Value**"
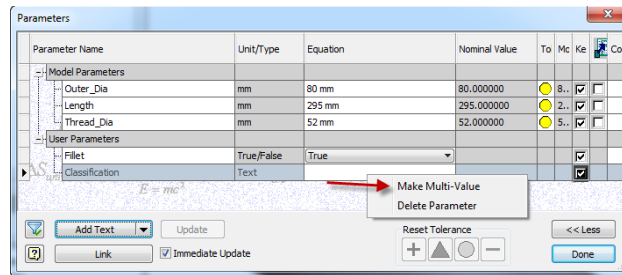
- The Value List Editor is displayed.
- In the "Add New Items" window enter: *"Heavy_Duty"* on the first line.
- On the second line enter *"Normal_Duty"*
- On the third line enter *"Light_Duty"*
- Click the "**Add**" button to add the list to the "Value" section of the list editor.
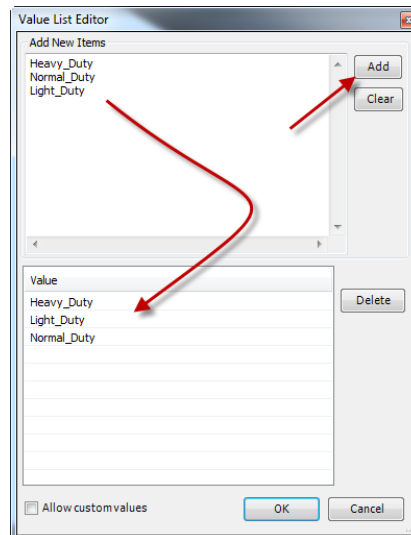
- When you have finished click "**OK**" to dismiss the Value List Editor.

- The values are now equations in the "Classification" parameter. This parameter currently does not have a related action associated with it. You will define that later.
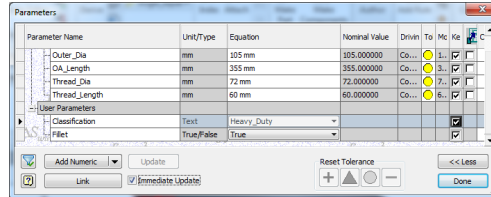
- Select "**Done**" to dismiss the Parameters dialog box.

Now that you have defined some parameters, you will use iLogic to define exactly what you want the parameter to do:

- On the Manage Tab select the "**iLogic Browser**" icon from the iLogic panel.

- The iLogic Browser will appear
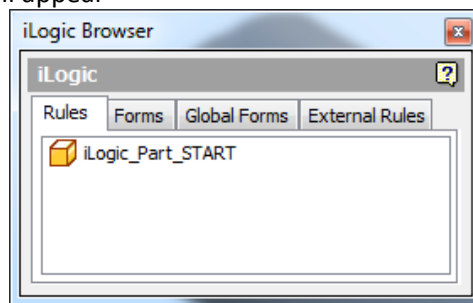
- You can position the browser anywhere on the screen or dock it along the left or right side of the Inventor window.

You will now create a rule that defines exactly what you want the previously created parameters to do.

- On the Manage Tab click "**Add Rule**" on the iLogic tile
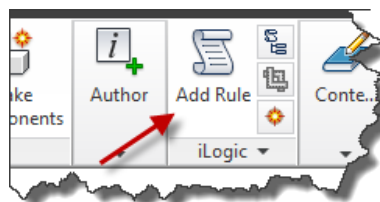
- Name the new rule "**Config_Rules**" and select **OK** on the Rule Name dialog box
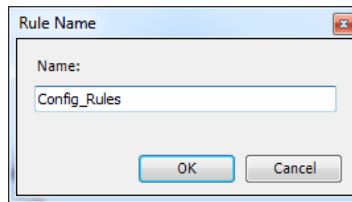
- The new rule will appear in the iLogic Browser and the iLogic Rule Editor Dialog box will appear.
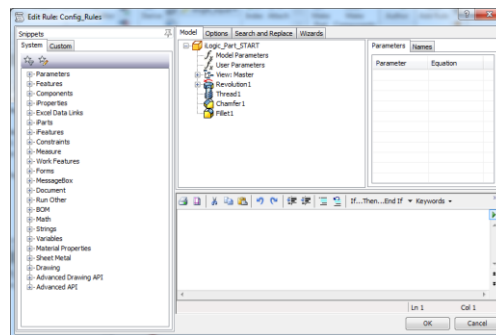
- The features and functionality of the iLogic Rule Editor are extensive and assist you in many ways while developing the code for very complex objectives.
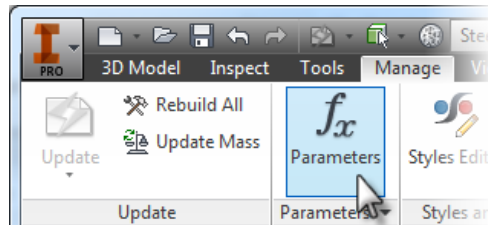- **Type or copy** the following code in the code editor window:

```
If Classification = "Light_Duty" Then
    OA_Length = 295 mm
ElseIf Classification = "Normal_Duty" Then
    OA_Length = 325 mm
ElseIf Classification = "Heavy_Duty" Then
    OA_Length = 355 mm
End If
```
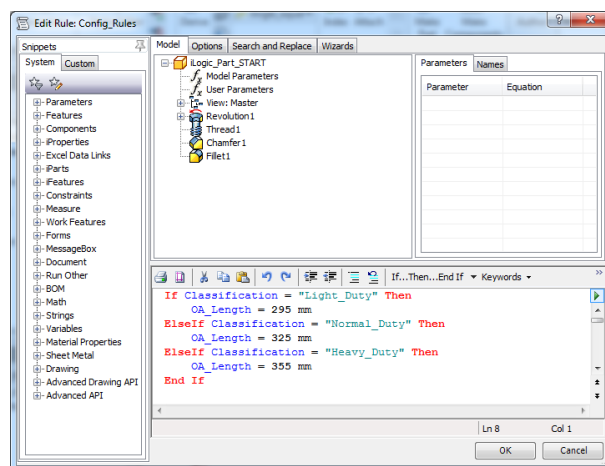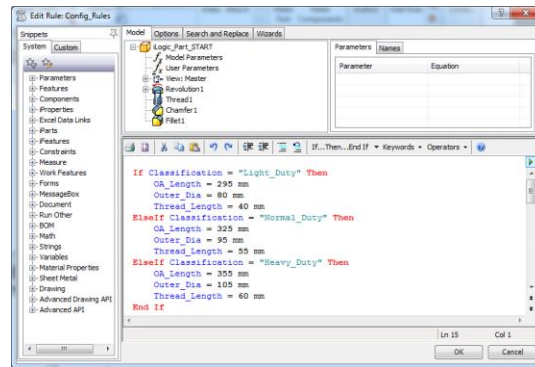

- The code will look for the selected value of the "Classification" parameter and adjust the "OA_Length" parameter accordingly
- Click "**OK**" to dismiss the iLogic Rule Editor

Test the rule

- Launch the Parameter Editor.

- Change the value of the "Classification" parameter.
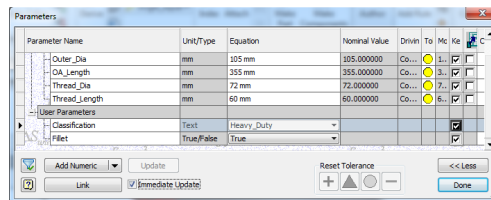
- Notice how the length of the shaft changes in response to the parameter change according to the rule.

Add additional conditions

- Right-Mouse-Click on the Config_Rule in the iLogic Browser and select "**Edit Rule**".
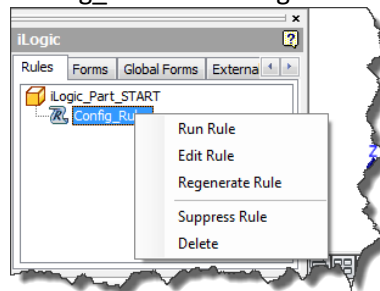
- Replace the old code by typing or copying the following code in the code editor window:

```
If Classification = "Light_Duty" Then
    OA_Length = 295 mm
    Outer_Dia = 80 mm
    Thread_Length = 40 mm
ElseIf Classification = "Normal_Duty" Then
    OA_Length = 325 mm
    Outer_Dia = 95 mm
    Thread_Length = 55 mm
ElseIf Classification = "Heavy_Duty" Then
    OA_Length = 355 mm
    Outer_Dia = 105 mm
```

Thread_Length **= 60** mm
End If

- The new code will evaluate the "Classification" parameter and adjust the value of "OA_Length", "Outer_Dia" and "Thread_Length" accordingly.

- Click "**OK**" to dismiss the iLogic Rule Editor.

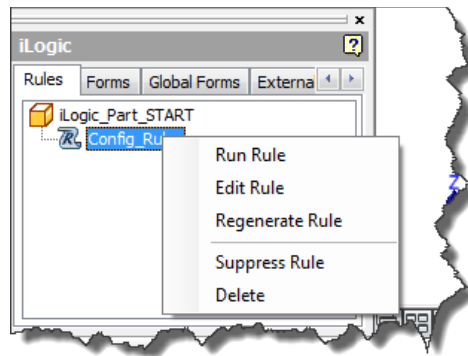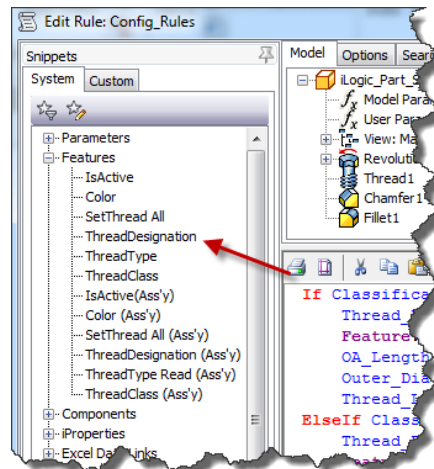Test the rule
- Launch the Parameter Editor.
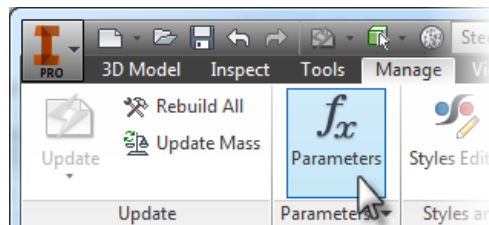
- Change the value of the "Classification" parameter.
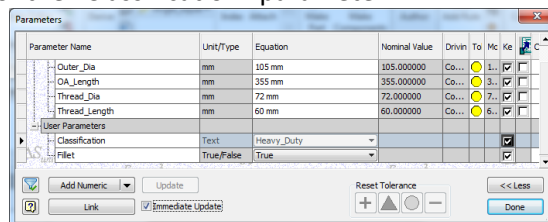
- Notice how the shaft changes in response to the parameter change according to the rule.

Add additional conditions to adjust the thread diameter and the thread size.
- Right-Mouse-Click on the Config_Rule in the iLogic Browser and select "**Edit Rule**".

- Replace the old code by typing or copying the following code in the code editor window:

```
If Classification = "Light_Duty" Then
    Thread_Dia = 52 mm
    Feature.ThreadDesignation("Thread1") = "M52x5"
    OA_Length = 295 mm
    Outer_Dia = 80 mm
    Thread_Length = 40 mm
ElseIf Classification = "Normal_Duty" Then
    Thread_Dia = 62 mm
    Feature.ThreadDesignation("Thread1") = "M62x4"
    OA_Length = 325 mm
    Outer_Dia = 95 mm
    Thread_Length = 55 mm
ElseIf Classification = "Heavy_Duty" Then
    Thread_Dia = 72 mm
    Feature.ThreadDesignation("Thread1") = "M72x6"
    OA_Length = 355 mm
    Outer_Dia = 105 mm
    Thread_Length = 60 mm
End If
```

- In addition to the previous conditions you have added two additional modifications.
- The parameter "Thread_Dia" will be modified by a parameter change.
- The feature "Thread1" will be modified by a parameter change.
- The code syntax to modify the thread feature was taken from the iLogic Snippet Library.

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- The Snippet library contains dozens of rule code samples to help you get started with the proper syntax for many other functions.
- Click "**OK**" to dismiss the iLogic Rule Editor
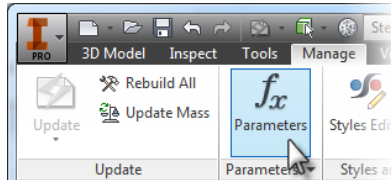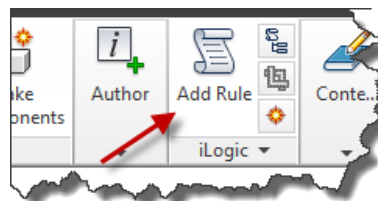
Test the rule
- Launch the Parameter Editor


- Change the value of the "Classification" parameter.
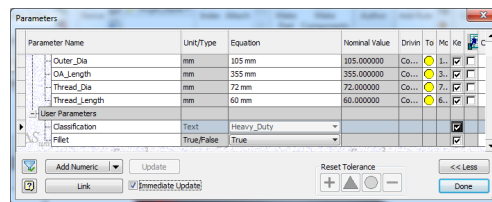
- Notice how the shaft changes in response to the parameter change according to the rule.

You will now create a rule that modifies several Inventor iProperties when the "Classification" parameter is changed.
- On the Manage Tab click "**Add Rule**" on the iLogic tile.

- Name the new rule "**iProp_Rules**" and select **OK** on the Rule Name dialog box
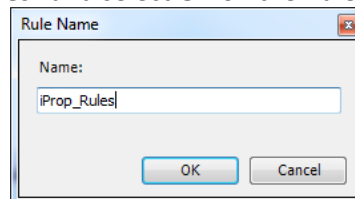
- The new rule will appear in the iLogic Browser and the iLogic Rule Editor Dialog box will appear.
- **Type or copy** the following code into the code editor window:

```
If Classification = "Light_Duty" Then
    iProperties.Value("Project", "Part Number") = "295-L"
    iProperties.Value("Project", "Description") = "Light Duty Pin"
ElseIf Classification = "Normal_Duty" Then
```

```
  iProperties.Value("Project", "Part Number") = "325-N"
  iProperties.Value("Project", "Description") = "Normal Duty Pin"
ElseIf Classification = "Heavy_Duty" Then
  iProperties.Value("Project", "Part Number") = "355-H"
  iProperties.Value("Project", "Description") = "Heavy Duty Pin"
End If
```
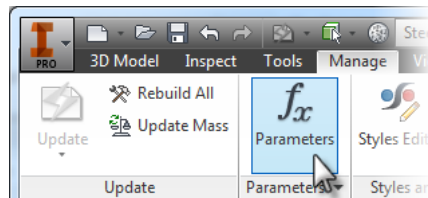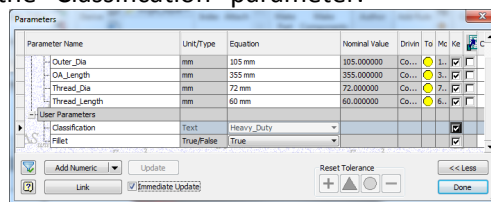
- This rule will evaluate the "Classification" parameter and modify the Part Number and Description iProperties accordingly.
- The code syntax to modify the iProperties was taken from the iLogic Snippet Library.
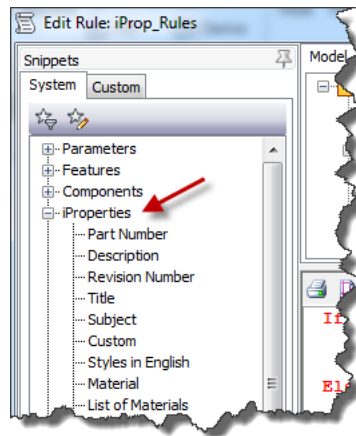
- Click "**OK**" to dismiss the iLogic Rule Editor.

Test the rule.
- Launch the Parameter Editor.

- Change the value of the "Classification" parameter.

- The shaft changes in response to the parameter change according to the rule.
- The iProperties of the shaft are updated according to the rule.
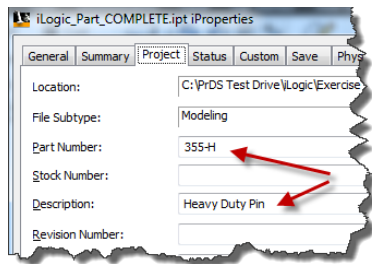
Close the file and do not save any changes.

Exercise2: Assembly Based Automation

In this exercise, you develop an iLogic rule to configure several components of an assembly. The intent is to show how iLogic can be used to drive rules and modify model parameters from the assembly level.

For this exercise, make sure that *PrDS Test Drive.ipj* is set as the active project in Inventor prior to starting the steps.

1. In Autodesk Inventor, Navigate to: \iLogic\Exercise_2 and open ***iLogic_Assy_START.IAM***

2. On the Manage Tab select "**Parameters**" to view the model parameters dialog box.
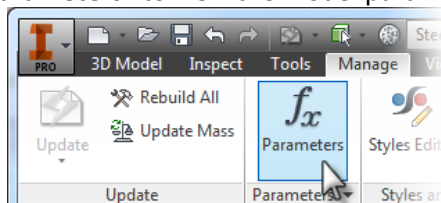
A multi-value parameter called "Assy_Config" contains three values: "Light_Duty", "Normal_Duty" and "Heavy_Duty".
- The values for the parameter will be used to configure the components of the assembly.
- The Shaft will change according to the rules from Exercise 1.
- The Flange Ring will respond accordingly.
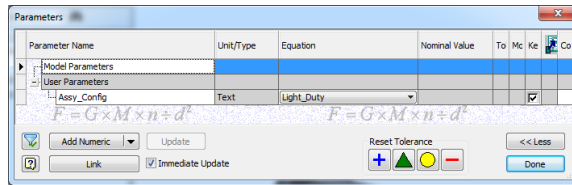
- Select "**Done**" to dismiss the Parameters dialog.

You will now create a rule that modifies the Shaft and the Flange Ring when the "Assy_Config" parameter is changed.

- On the Manage Tab click "Add Rule" on the iLogic tile
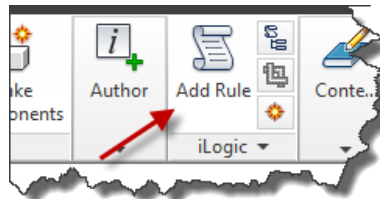
- Name the new rule "**Assy_Config**" and select **OK** on the Rule Name dialog box
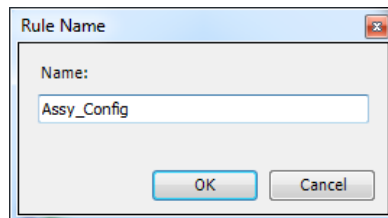
- The new rule will appear in the iLogic Browser and the iLogic Rule Editor Dialog box will appear.
- **Type or copy** the following code into the code editor window:

```
If Assy_Config = "Light_Duty" Then
    Parameter("PK46-25-00-003-Complete:1", "Classification") = "Light_Duty"
    Parameter("PK46-25-00-002-Complete:1", "Inner_Dia") = 70 mm
ElseIf Assy_Config = "Normal_Duty" Then
    Parameter("PK46-25-00-003-Complete:1", "Classification") = "Normal_Duty"
    Parameter("PK46-25-00-002-Complete:1", "Inner_Dia") = 85 mm
ElseIf Assy_Config = "Heavy_Duty" Then
    Parameter("PK46-25-00-003-Complete:1", "Classification") = "Heavy_Duty"
    Parameter("PK46-25-00-002-Complete:1", "Inner_Dia") = 95 mm
End If
```
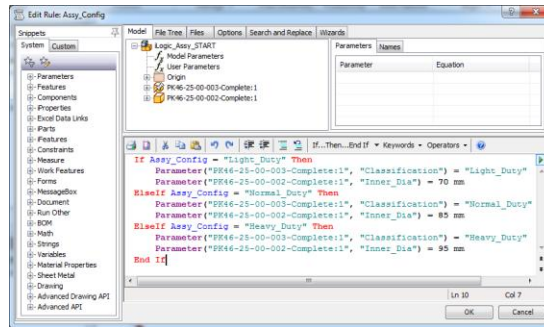
- The code will evaluate the condition of the "Assy_Config" parameter.
- The "Classification" parameter in the Shaft component will be changed accordingly (thus the rule you built in Exercise 1 will make adjustments to several other parmeters.
- The "Inner_Dia" parameter of the Flange Ring will be adjusted accordingly.
- Select **OK** to dismiss the Rule Editor.

Test the rule.

- Launch the Parameter Editor.
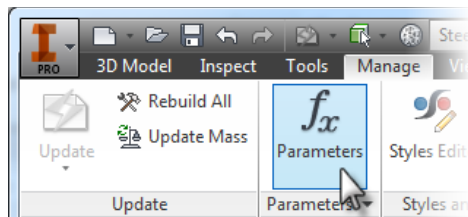
- Change the value of the "Assy_Config" parameter.
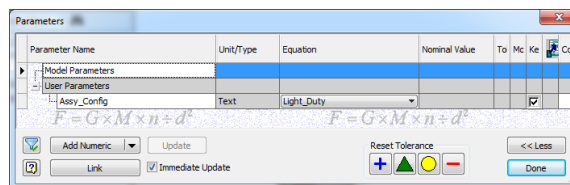
- The parameter is evaluated and the components are modified according to the embedded rule code.

iLogic provides a powerful method for capturing internal standards and industry knowledge within the digital prototype, allowing the user to quickly make large-scale changes at the assembly level.
Close the file and do not save any changes.

Exercise3: Assembly Configuration

In this exercise, you develop an iLogic rule to make an assembly quickly switch between two different subassemblies based on a desired configuration. You will also create a simple iLogic form as a user interface for the assembly configuration.

For this exercise, make sure that *PrDS Test Drive.ipj* is set as the active project in Inventor prior to starting.

2. In Autodesk Inventor, Navigate to: \iLogic\Exercise_3 and open ***Assy,Chassis, Front_START.IAM***
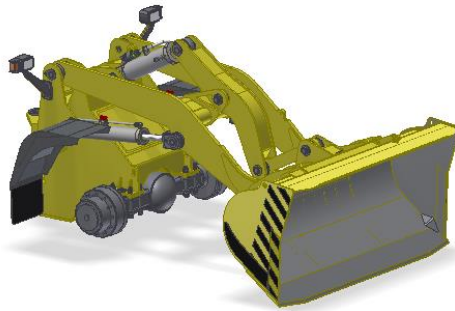
3. ILogic requires a custom "Level of Detail" representation in order to manage component suppression.
   - A custom Level of Detail named "iLogic" has been created for you.
   - Make sure it is the active LOD.
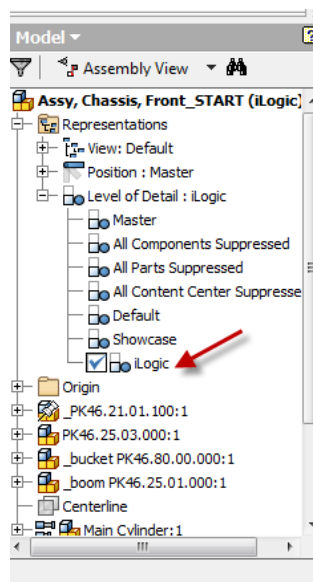
4. This assembly contains two different bucket configurations.

- "Scoop" style
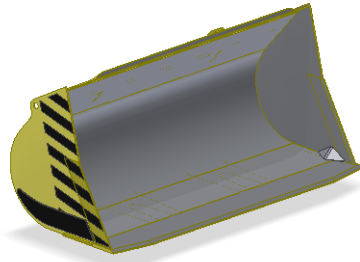
- "Rake" style
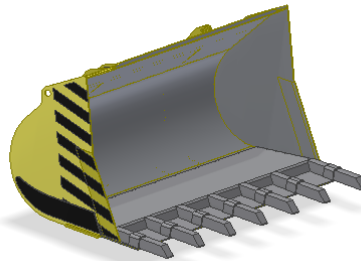
5. When one of these subassemblies is active the other will be suppressed.
   - Note: The suppressed bucket will still be part of the assembly model but will be invisible and will be marked as "reference" in the Bill of Material and therefore will not be included in a drawing Parts List.
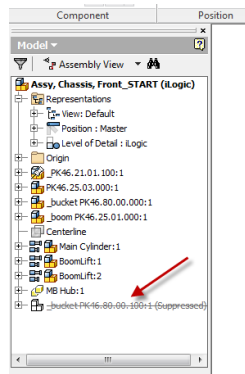

6. On the Manage Tab select "**Parameters**" to view the model parameters dialog box.
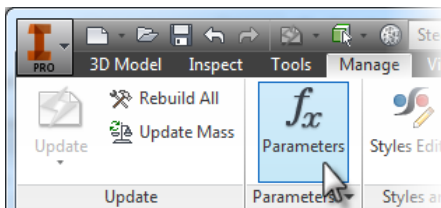

- A multi-value parameter called "Chassis_Config" contains two values: "Scoop Bucket" and "Rake Bucket".

- The values for this parameter will be used to configure the bucket subassembly used in the assembly.
- A multi-value parameter called "Position" contains three values: "Master", "Raised" and "Lowered".
- The values for this parameter will be used to activate different position representations of the chassis.
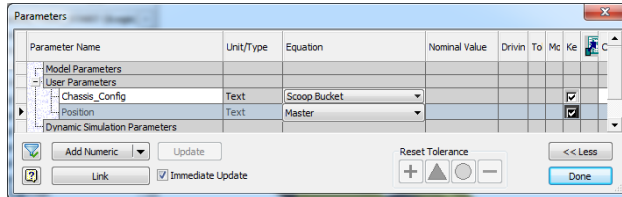
7. A rule that drives the Position Representation parameter has been created for you.
   - The code used in this rule was derived from advanced information found in the Inventor Application Programing Interface.
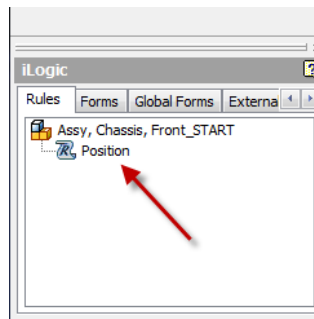
8. You will now create a rule that determines the bucket assembly to be used when the "Chassis_Config" parameter is modified.
   - On the Manage Tab click "Add Rule" on the iLogic tile
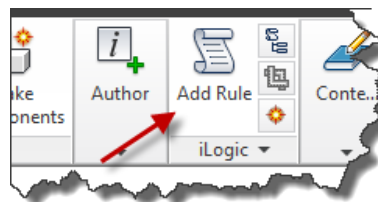
   - Name the new rule "**Chassis_Rule**" and select **OK** on the Rule Name dialog box
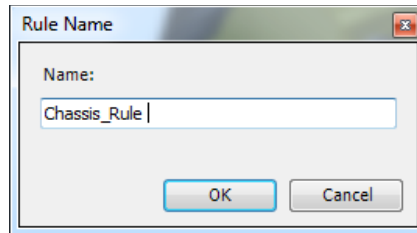
- The new rule will appear in the iLogic Browser and the iLogic Rule Editor Dialog box will appear.
- **Type or copy** the following code into the code editor window:

```
If Chassis_Config = "Rake Bucket" Then
Component.IsActive("_bucket PK46.80.00.100:1") = True
Component.IsActive("_bucket PK46.80.00.000:1") = False
Else
Component.IsActive("_bucket PK46.80.00.100:1") = False
Component.IsActive("_bucket PK46.80.00.000:1") = True
End If
```
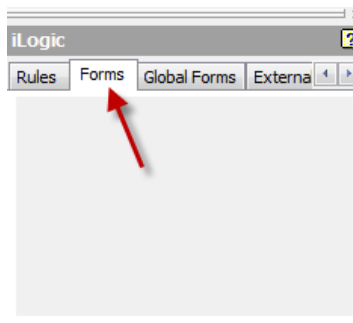
- The code will evaluate the "Chassis_Config parameter and manage the suppression state of the bucket subassemblies.
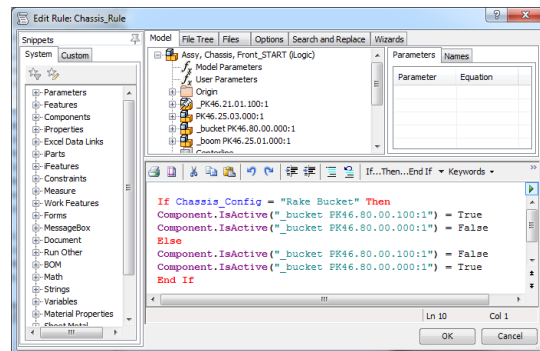
- The code syntax to modify suppression was taken from the iLogic Snippet Library.
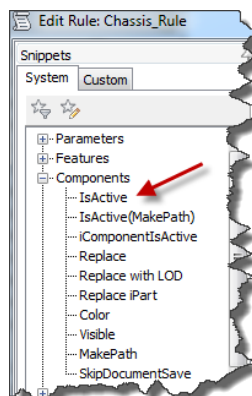
- Click "**OK**" to dismiss the iLogic Rule editor
9. Next you will create an iLogic form to drive the two parameters.
   - Activate the Forms Tab on the iLogic Rules Browser.

- Right-Mouse-Click in the iLogic Browser and select "**Add Form**".
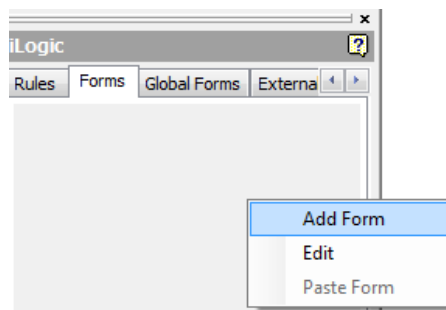
- The iLogic Form Editor appears and enables you to build a form using existing rules, parameters and iProperties.
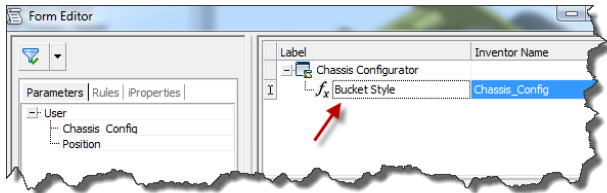- The iLogic Form Editor also spawns a preview of your form so you can see you changes in real-time.
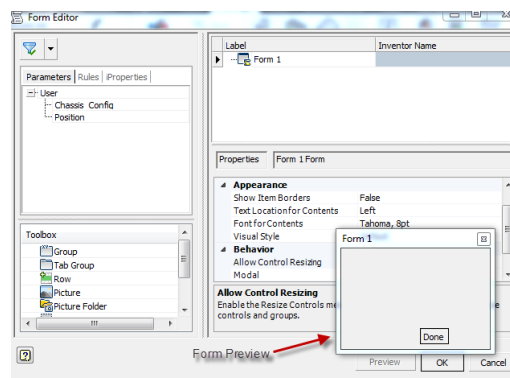
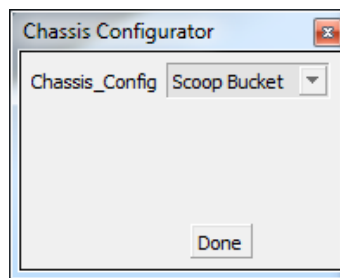10. In the iLogic form editor window, change the name of the form to "**Chassis Configurator**"
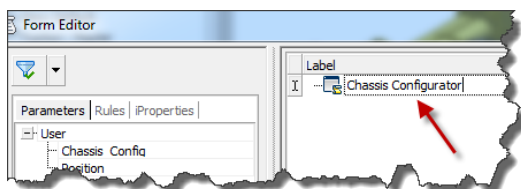
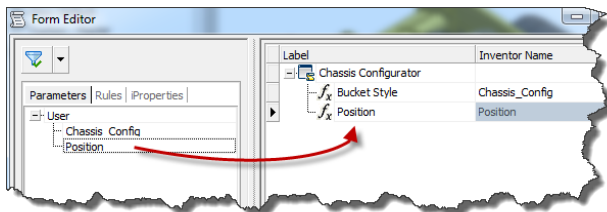11. Drag the "Chassis_Config" text from the Parameters window into the form editor window.
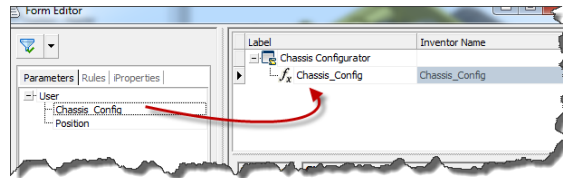
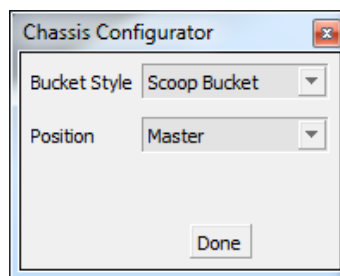12. Change the label to read "**Bucket Style**".
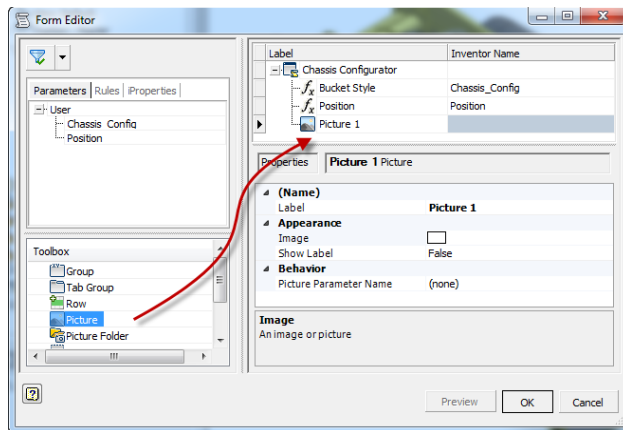

13. Note the changes in the form preview.

14. From the Parameters window drag "Position" into the form editor.
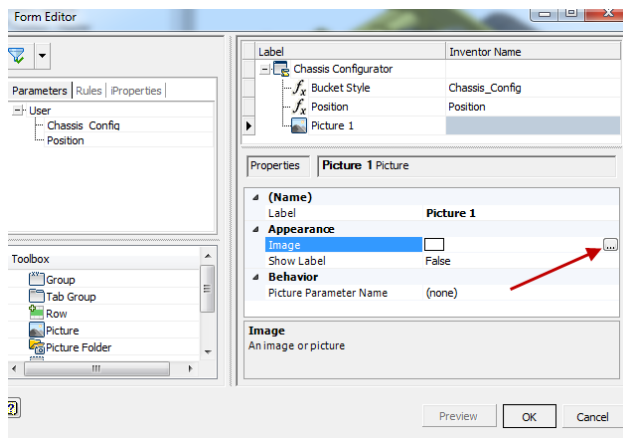
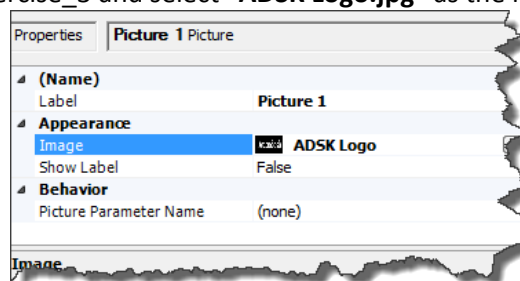

15. Note the change in the form preview.

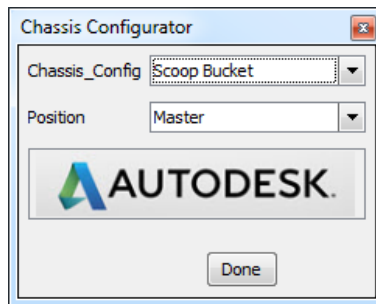16. Drag "Picture" from the Toolbox Window into the form editor

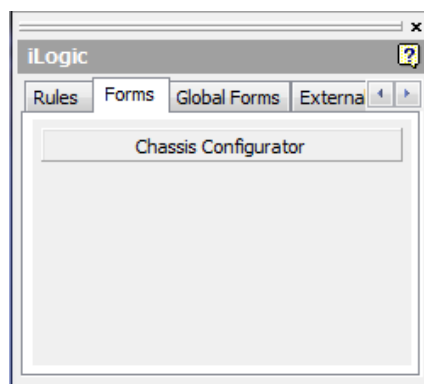17. Define the image for the picture in the form element properties window.

18. Navigate to //iLogic/Exercise_3 and select "**ADSK Logo.jpg**" as the image for your form.
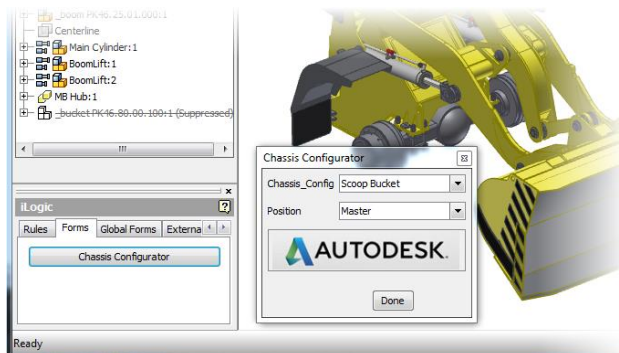

19. Note the change in the form preview.

20. Select "OK" to dismiss the iLogic Form Editor. The form button will appear in the iLogic browser.

21. Select the Chassis Configurator button to launch your new form.

22. You can now use this form to quickly toggle between bucket configurations and positional representations.
23. Close the file and do not save changes.