Inventor and Configurator 360 Lab Session

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Learning Objectives

This hands-on lab will introduce users to the iLogic fundamentals that are the building blocks to automating design within an Inventor model.

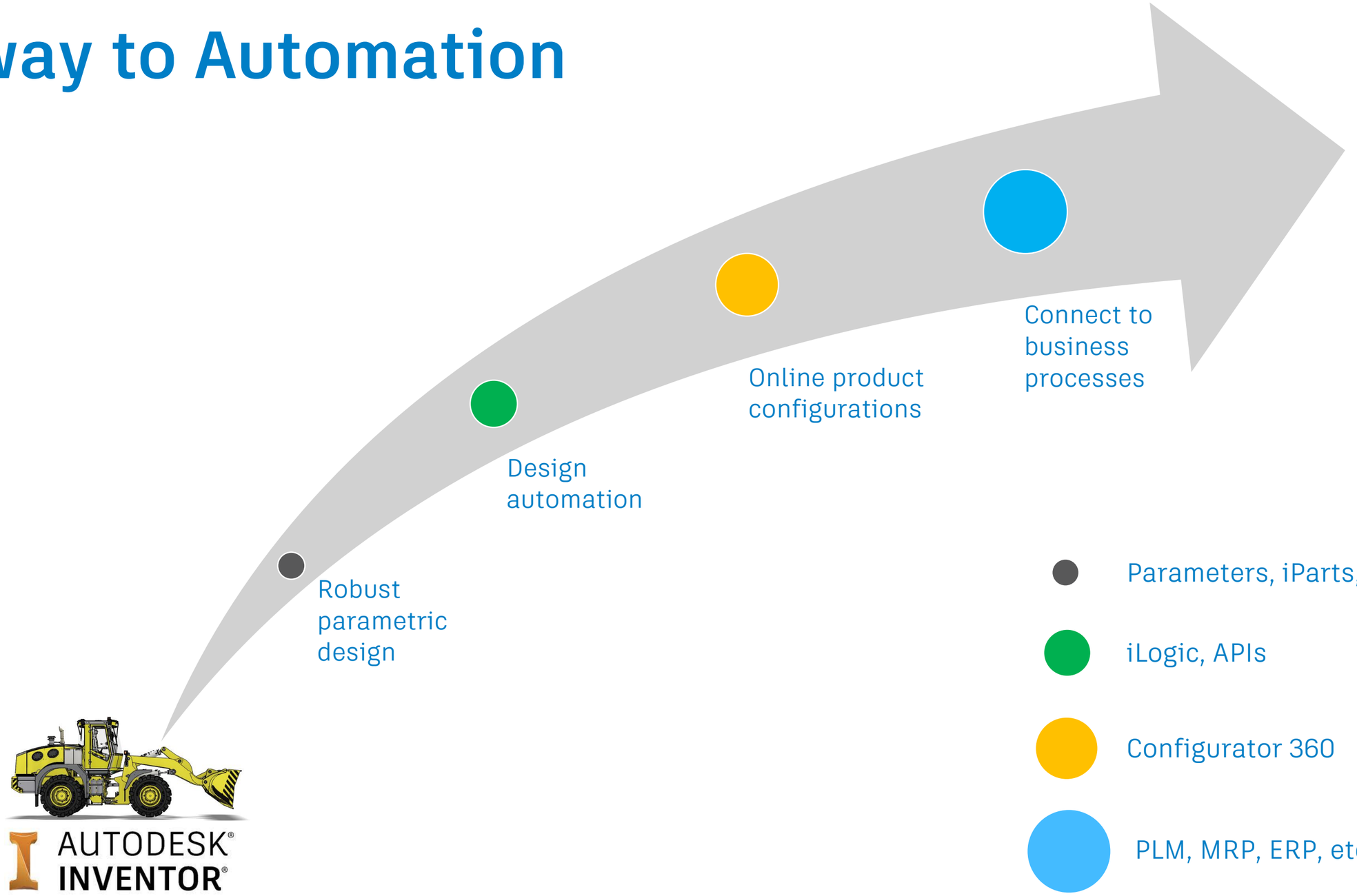
Topics to be covered are:-

- Understand how to create iLogic rules.
- Understand the different ways iLogic can control 3D models.
- Learn some best practices when utilizing iLogic.
- See how iLogic can be used for Design Automation with Configurator 360.

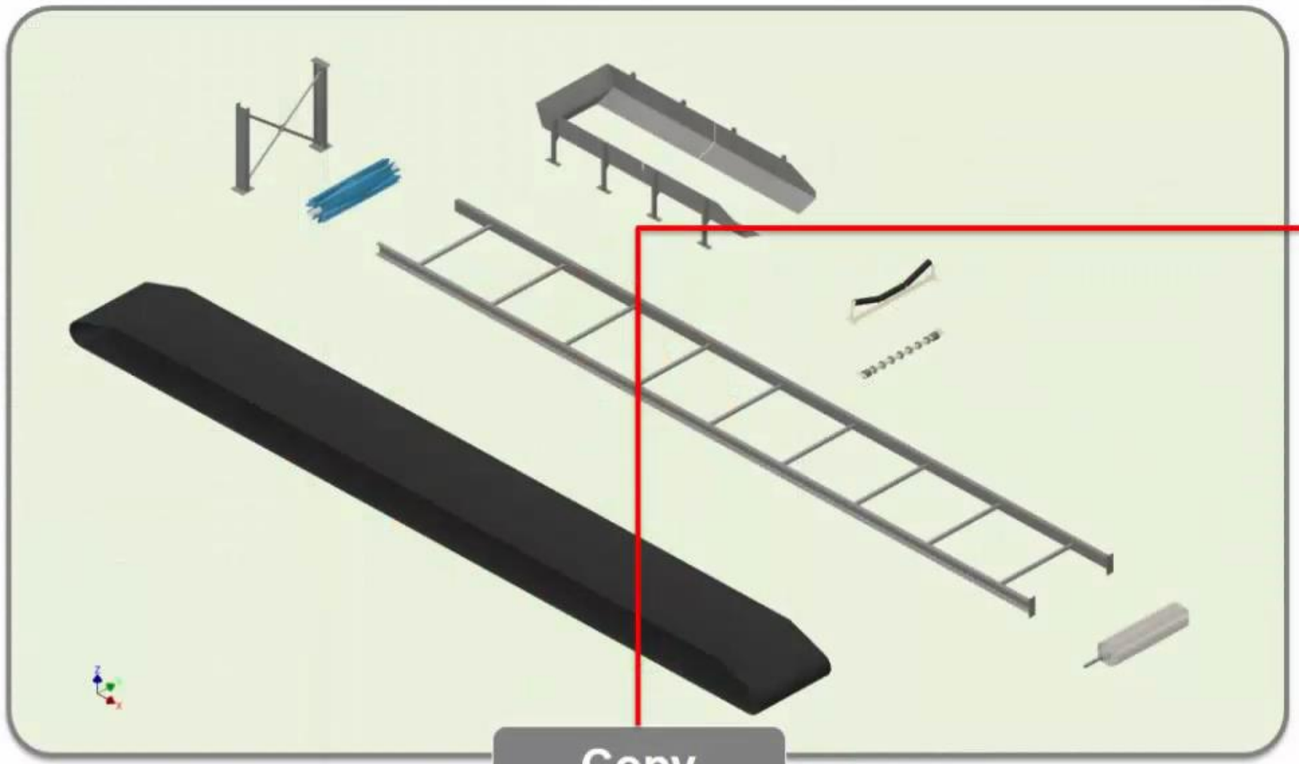
Inventor iLogic – What does it offer

- Customer Need
  - Simple way to control complex design variants and automate tedious design tasks
- Inventor iLogic
  - Rules-based design with no programming expertise required
  - More design intelligence embedded directly into your digital models.

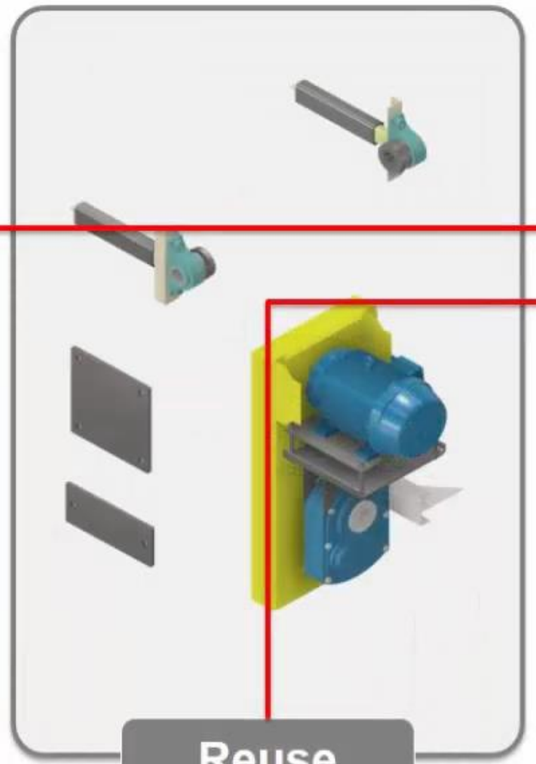
Gateway to Automation



1 Parametric Relationships



Copy



Reuse

Classification
Design Specific
Non-Design Specific

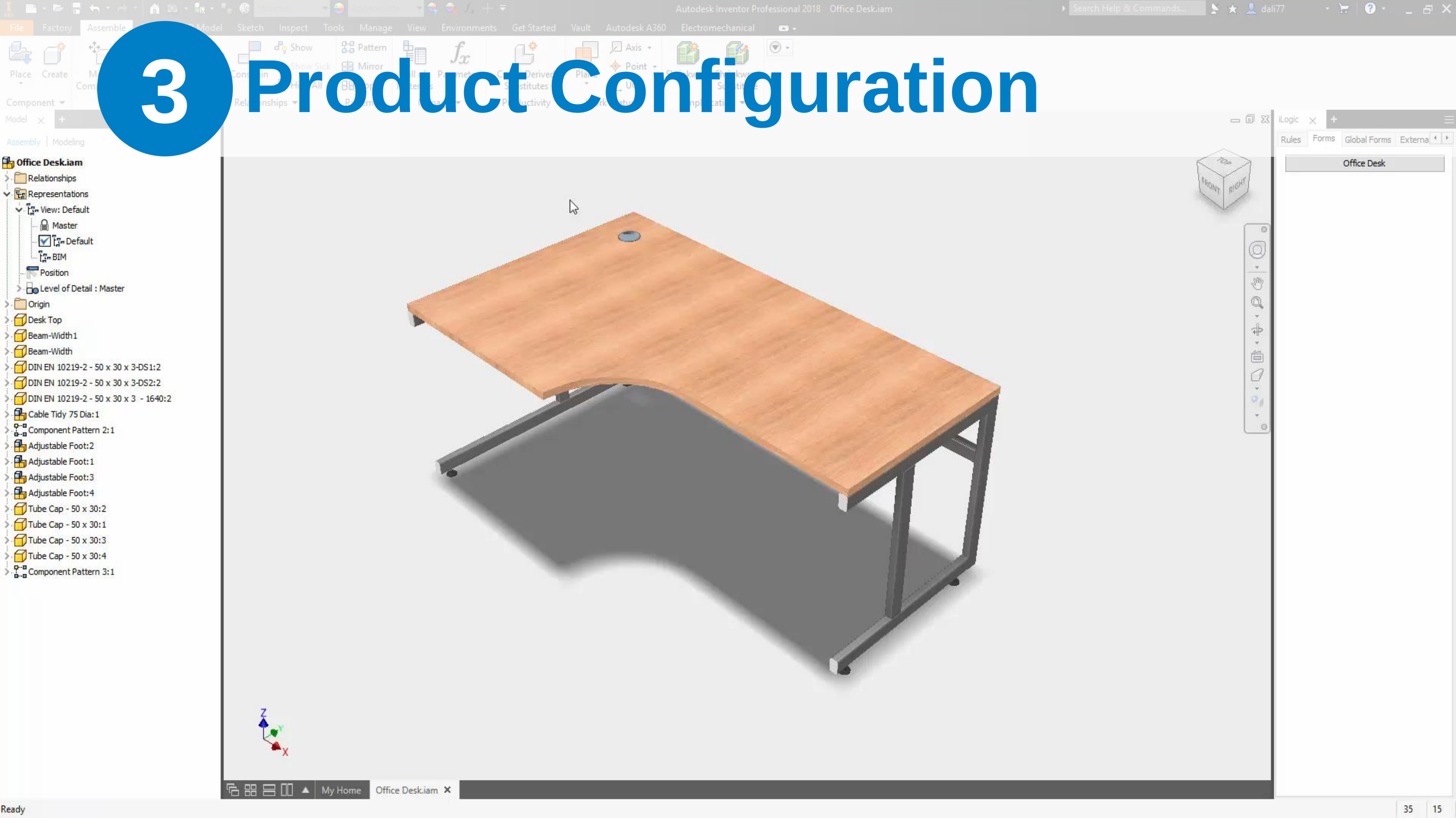
2 Standards Enforcement

This example shows two types of enforcement.

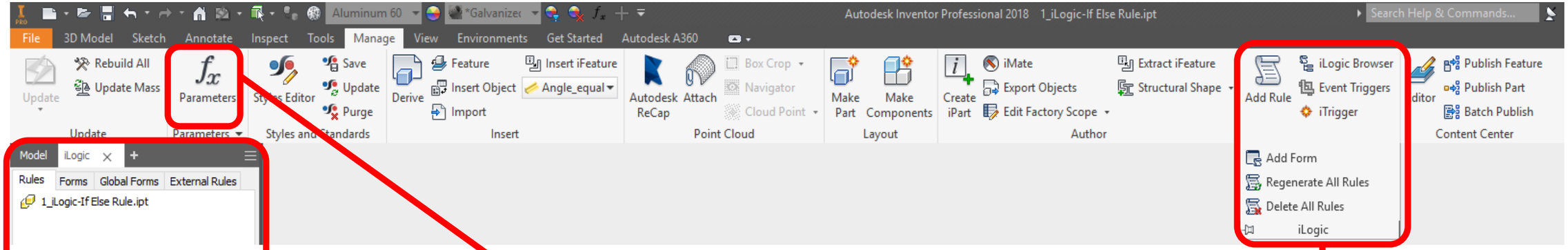
#1 Standardise the width to 1m.
If not possible adjust the last cubicle to suit.

#2 No arrangement less than 2m is allowed.

3 Product Configuration



iLogic Getting Started – Manage Tab



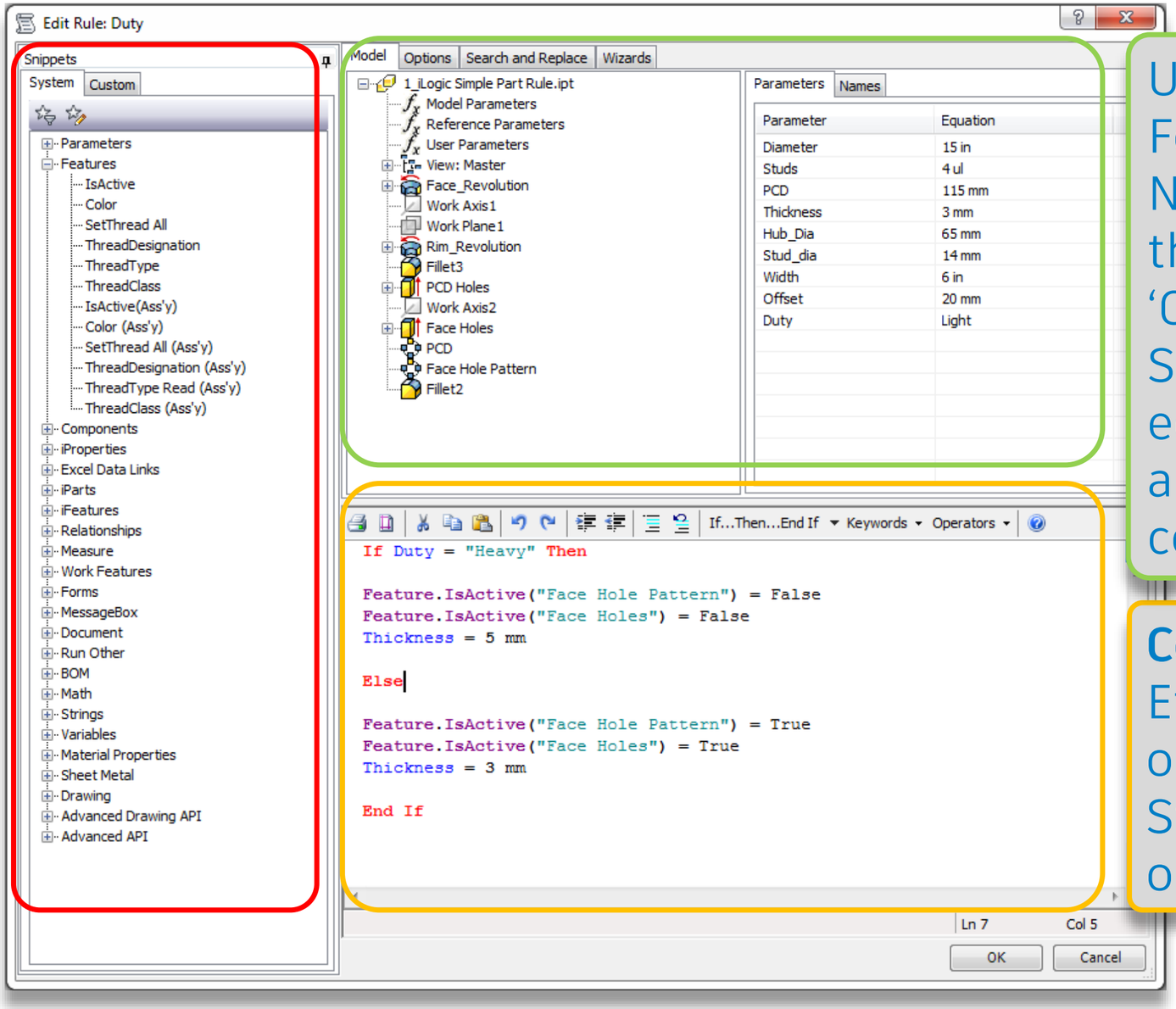
Use the iLogic browser to Create and Edit Rules within the Rule Editor, Create & Edit Forms and link to External.

Use the Parameters to Create & Edit Number, Text, True/False and Multi List Parameters.

Turn the iLogic browser on/off & apply triggers to execute iLogic based upon an event. Such as run on Open or Save.

iLogic Getting Started – The iLogic Rule Editor

Snippet Browser
Lookup all fundamental iLogic control options and utilise or save standard snippet collection.



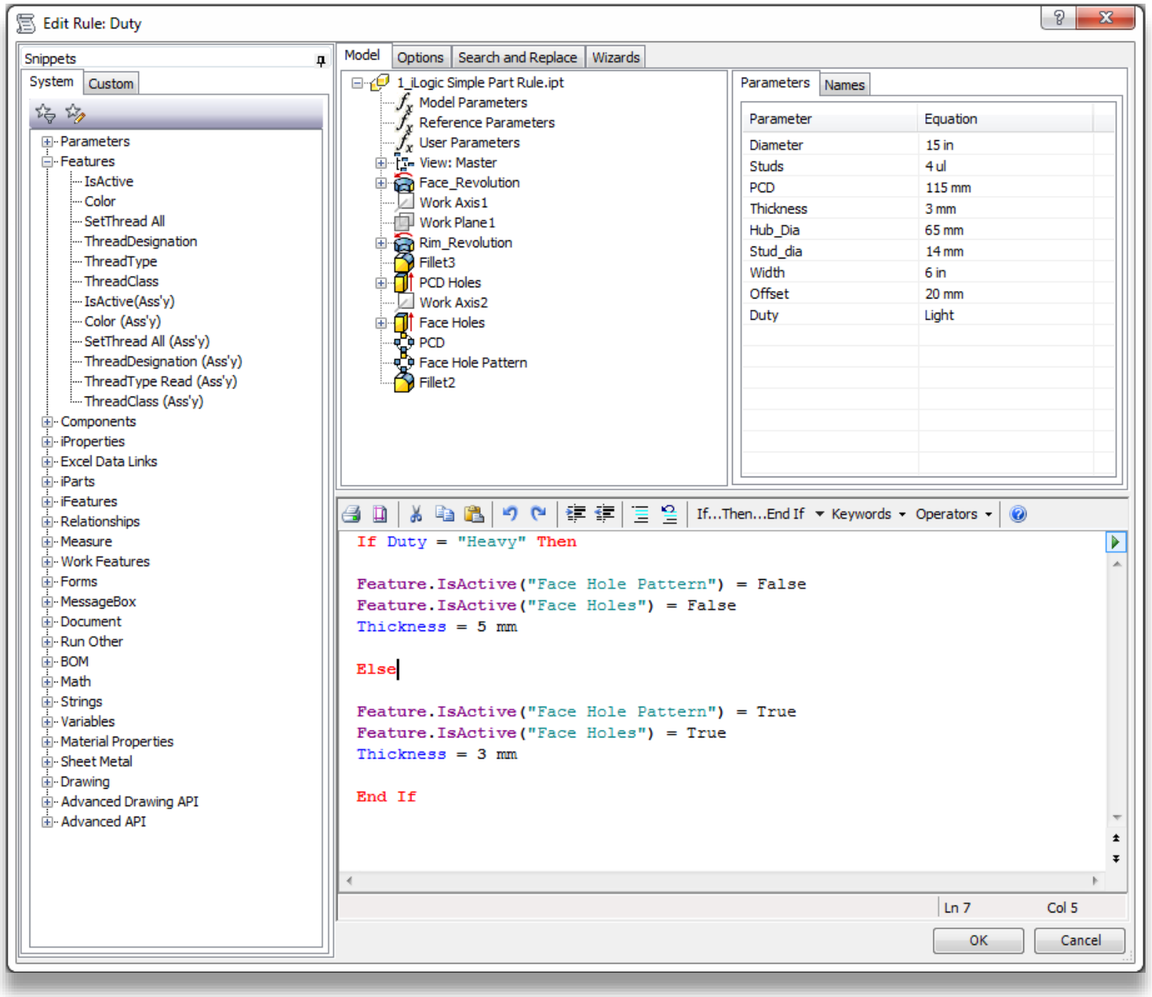
Use to identify Feature and Part Names including the ability to 'Capture Current State' which lists all editable options and creates iLogic code below.

Code Editor.
Either use snippets or 'Capture Current State' from Feature or Part name

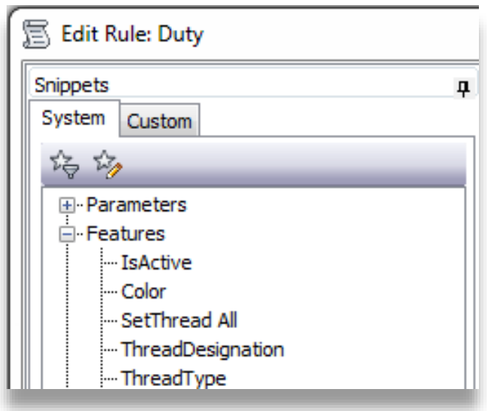
Fundamentals of Rules Authoring

- Rules determine & drive the conditional behaviour of your Inventor designs.

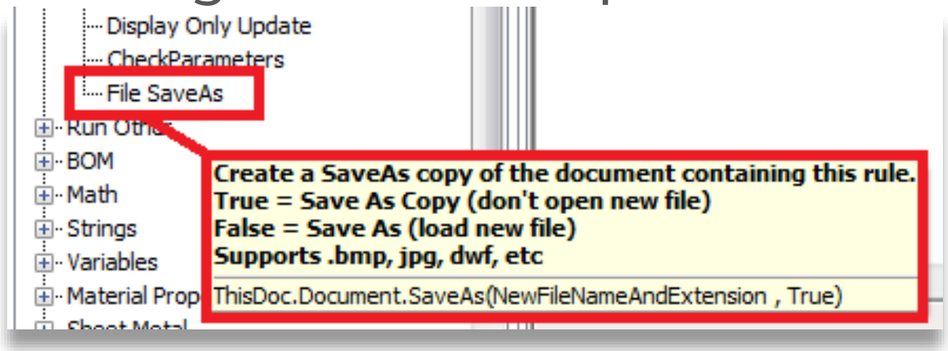
- By default rules are embedded, saved and stored directly in the Inventors part and assembly documents.



Building an iLogic Statement

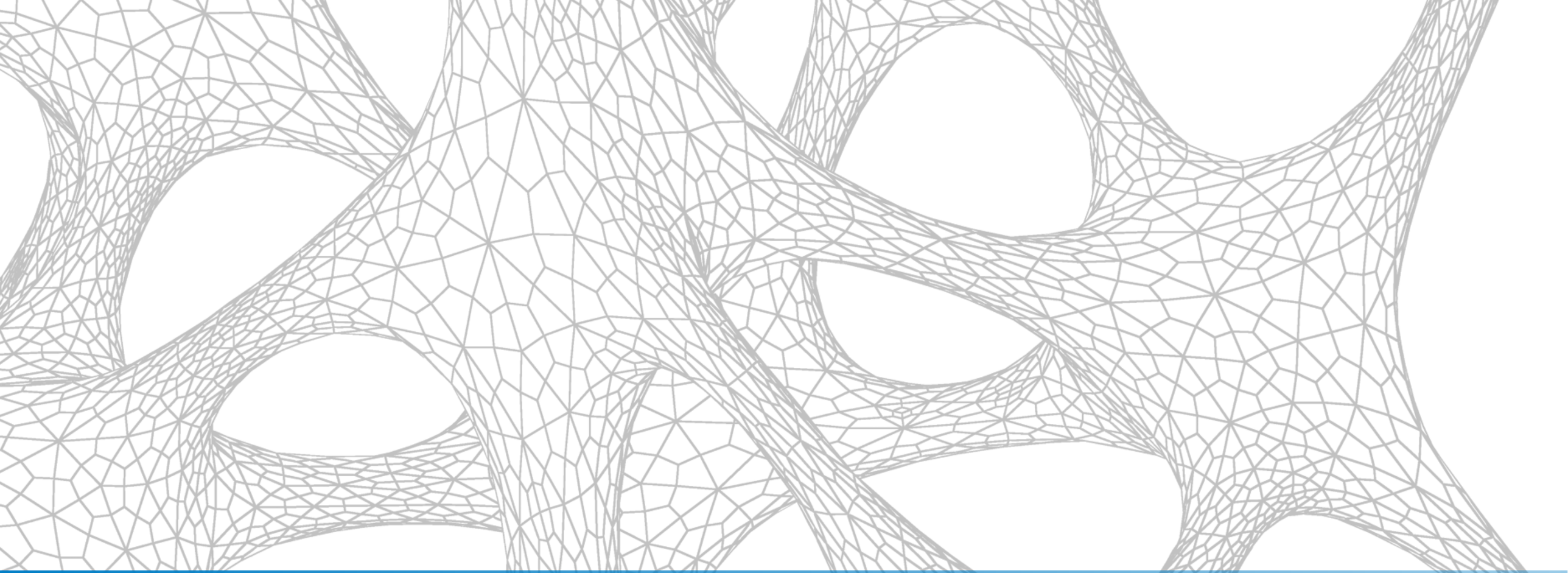


- Using the iLogic rule editor you can look at all the available out of the box rule snippets.
- All rules will give a tooltip to their usage.



- Once inserted you can then modify and add to suit.

```
ThisDoc.Document.SaveAs(NewFileNameAndExtension , True)
edited to...
ThisDoc.Document.SaveAs("..\iLogicExport.sat" , True)
```



Hands On Examples

ALL FILES ARE LOCATED UNDER:
C:\Datasets\Demir Ali & Peter Van Avondt

Exercise #1 : Simple Part Conditional Rule

Utilising the **If / Else / End If** statement

- The purpose of exercise is to be able to configure a rule that changes the model and it's parameters based upon a text value selection.

Requirement

*"We would like to configure our design to allow a user to select whether the wheel is "**Heavy**" or "**Light**" Duty. This should control whether the wheel face holes are suppressed and the thickness is changed to suit"*

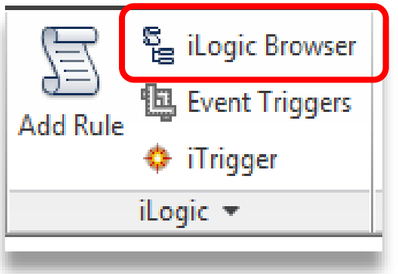
Workflow User Benefit

You can define 'parameters' that are much more obvious for the user to identify and modify.

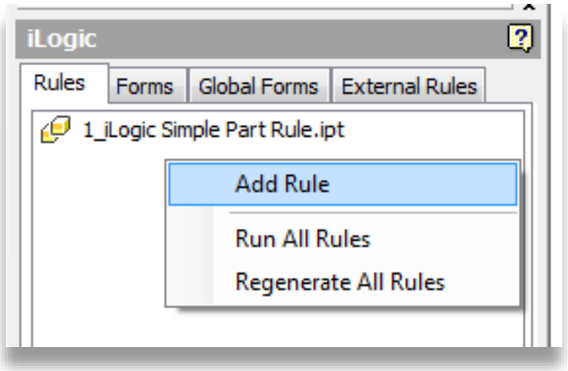
#1 How To : Simple Part Conditional Rule

1. Open file “**1_iLogic Simple Part Rule.ipt**”
2. Create a new **Text** Parameter called ‘**Duty**’ & make it a MultiValue parameter by specifying two parameters called **Heavy** and **Light.**

3. Turn on the iLogic Browser available from the manage Tab.
4. Right Mouse click in the iLogic Browser and select ‘Add Rule’.



5. Enter a suitable rule name and type in the following and once complete click OK.



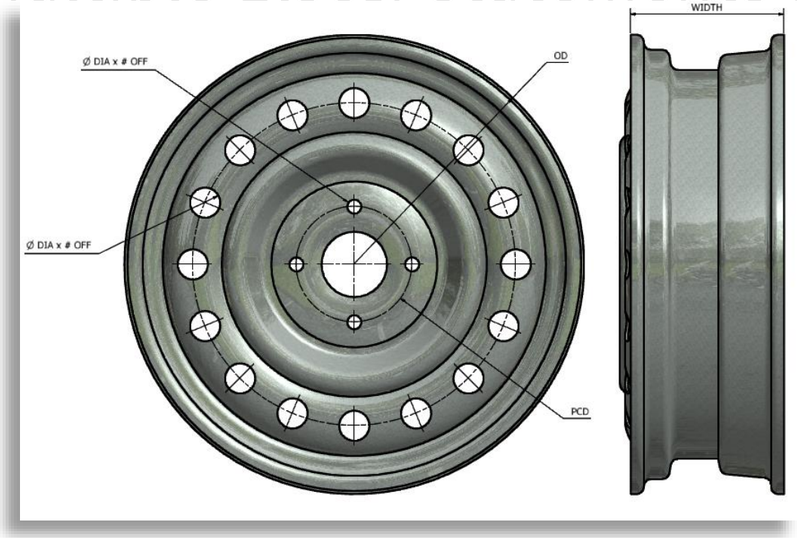

```
If Duty = "Heavy" Then
Holes = True
Thickness = 5 mm
Else
Holes = False
Thickness = 3 mm
End If
Feature.IsActive("Face Holes") = Holes
Feature.IsActive("Face Hole Pattern") = Holes
```

6. Once complete, you can change the text parameter and the model will update accordingly.

Exercise #2 : Conditional Rule Combination

When to utilise the Case and Select statement

- The purpose of exercise is to be able to configure a rule that can deal with a greater variance of options.
- Sometimes writing a single rule that has multiple ElseIf statements makes it difficult to read, write and manage.



Requirement

"We would like to configure our design to allow a user select the wheel size and usage which in turn controls if the face holes exist, the number of holes AND the wheel mounting holes diameter, number off and PCD."

Workflow User Benefit

You can define a Case statement that encompasses multiple configuration options into a single condition.

#2 How To : Conditional Rule Combination

1. Open file “**2_iLogic-Case Select Rule.ipt**”.
2. Create a new rule called “**Size**”.
3. Create the rule as shown.
4. Once complete you now have a component that a user can select a **Duty** and **Size** and the design will update to suit.

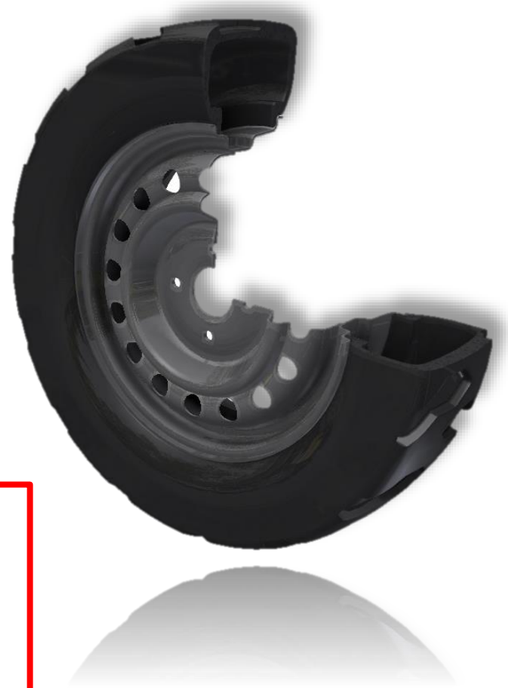
```
Select Case Diameter
        Case 13 in, 14 in
                Holes = 14 ul
                Studs = 4 ul
                PCD = 115 mm
        Case 15 in, 16 in
                Holes = 16 ul
                Studs = 5 ul
                PCD = 120 mm
        Case 17 in, 18 in
                Holes = 18 ul
                Studs = 5 ul
                PCD = 125 mm
End Select
```

Exercise #3 : Drive Parameters from the Assembly

- The purpose of exercise is to be able to configure a rule to control part parameters from within the assembly.
- This is a fundamental requirement to configure designs from a central source.

Requirement

“We would like to be able to open OR copy a design then configure all parts within the assembly in one hit.”



Workflow User Benefit

Rather than editing files individually, users can quickly build relationships between one file and another in any direction. Allowing access to associative change and quickly ensuring the necessary changes are pushed through and in sync.

#3 How To : Drive Parameters from the Assembly

1. Open file “**3_iLogic-Assembly Parameter Rule.iam**”
   - **Note: The Part instance names have been overridden. This ensures when files are copied or replaced the rule will still compute as the instance name is fixed.**
2. Create a new rule called ‘**Parameters**’
3. We can now tell iLogic that the assembly parameters will be driving the part parameters OR vice versa.
4. We will also add two lines that ensure the models are updated once the change has happened.
5. Once complete, you can change the necessary parameters and the assembly model will update accordingly.

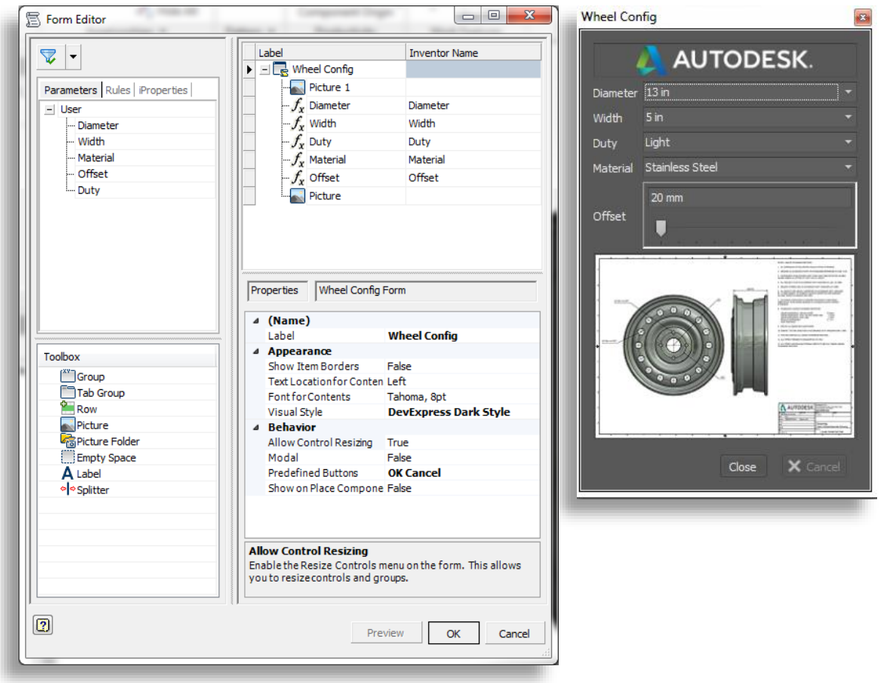
```
'This indicates a commented out line that is ignored.
Parameter("Wheel", "Diameter") = Diameter
Parameter("Tyre", "Diameter") = Diameter
‘Set Wheel and Tyre width
Parameter("Wheel", "Width") = Width
Parameter("Tyre", "Width") = Width
‘Set wheel duty for Light or Heavy Duty
Parameter("Wheel", "Duty") = Duty
'Update the files as per normal Inventor update
RuleParametersOutput()
InventorVb.DocumentUpdate()
```

Exercise #4 : Creating an iLogic Form

- The purpose of exercise is to be able to create a simple form to control the assembly.
- In addition we can use the form to create an entry form for iProperties. Using an Event Trigger this can be run at a suitable event such as file open/close/save, etc

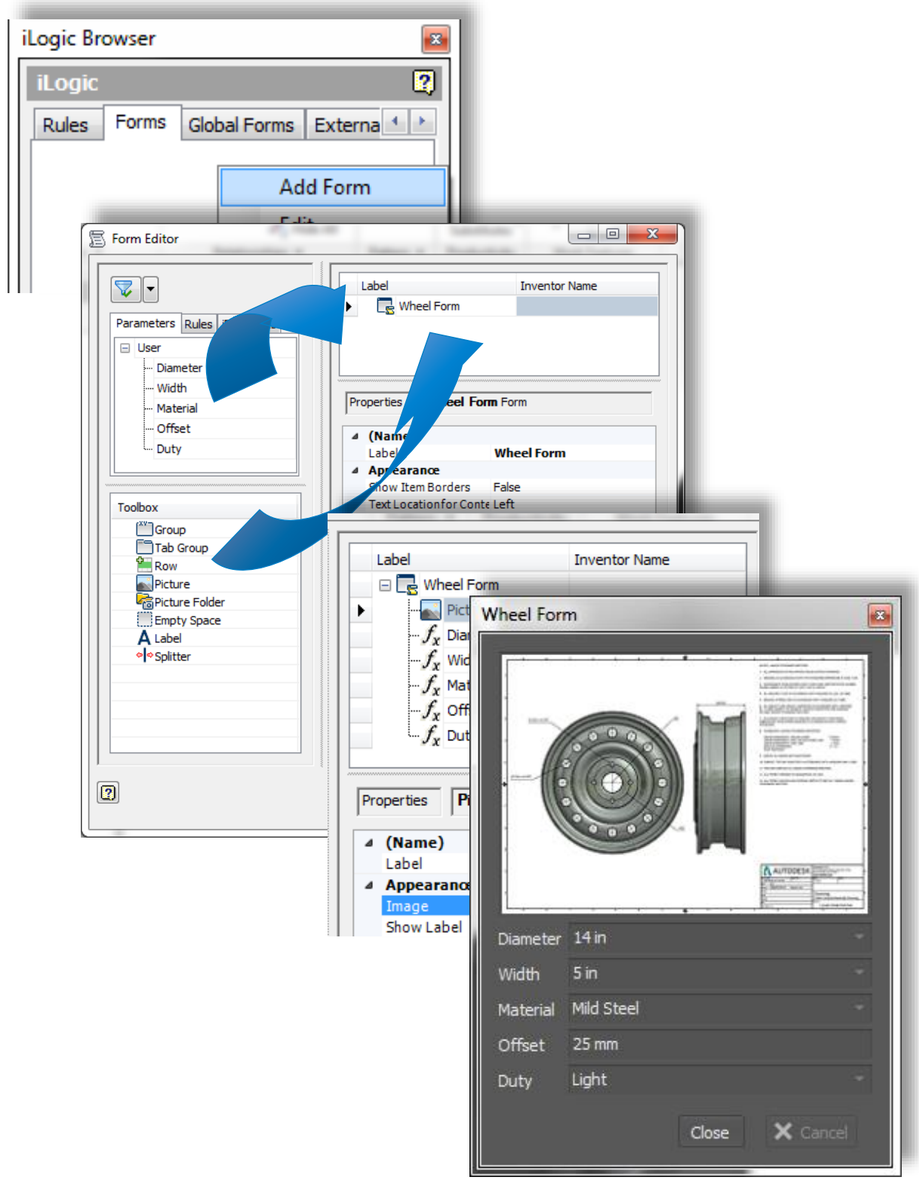
Requirement

“We would like to have an easy to configure product with a bespoke entry form and the ability to drive properties based upon the selected option”



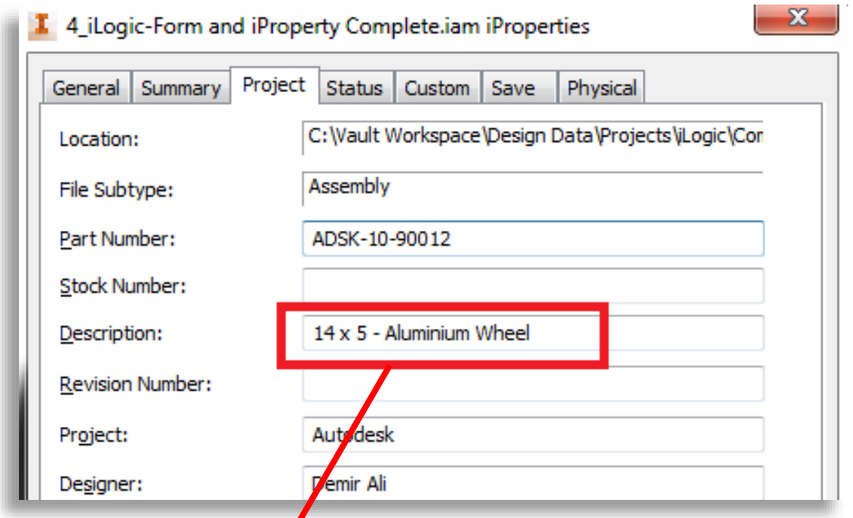
#4 How To : Creating an iLogic Form

1. Open file “**4_iLogic-Form Creation.iam**”
2. Within the iLogic Browser right mouse click and select ‘**Add Form**’
3. The dialog box will show all the Key parameters and a preview of the generated form.
4. Here we give the form a name and display style. You can then drag the relevant parameters into the right side of the dialog.
5. In addition you can drag additional display controls from the **Toolbox** into the form layout. A typical example is an image.
6. When is dragged in, you can then select it a browse to an image (samples located in dataset folder) to embed it or change the style of a field from text to slider, etc.



Exercise #5 : Creating intelligent iProperties (Optional)

- Whether related to a configuration or just to make data more consistent, quite often companies want to configure the designs iProperties to update and match the current setup.



- This is especially useful when cut lengths need to be part of the stock number, etc.

The description of “14 x 5 – Aluminium Wheel” is made up of:-

“<Parameter> <Text> <Parameter> <Text> <iProperty> <Text>”

#5 How To: Creating intelligent iProperties (Optional)

- Utilising iLogic to update a files iProperties makes it easy and very quick to combine any value/parameter/property to generate an iProperty definition.
- It also gives us the opportunity to reformat as necessary

```
'As document is in Millimeters all units are reported in mm so this will convert Diameter and Width to Inches @ 0 decimal places
pdia = Round(Diameter / 25.4, 0)
pwidth = Round(Width / 25.4, 0)

'The below concatenates the converted iLogic parameters with plain text
iProperties.Value("Project", "Description") = "" & pdia & " x " & pwidth & " - " & Material & " Wheel"
```

- Adding & <parametername> & into a text string allows you to concatenate parameter values such as length into a field or message box.

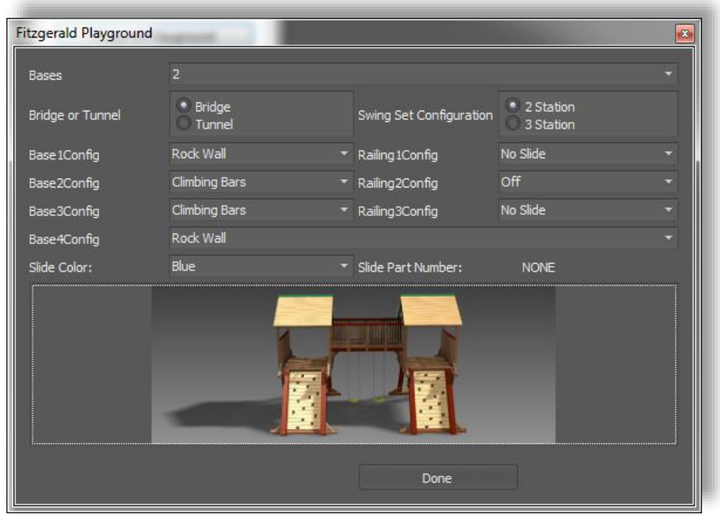
How is iLogic being used by our customers?

Control component sizes.

Suppress parts & Sub-assemblies.

Change component quantity.

Update drawings to suit.



Adjust assembly constraints.

Automate SaveAs to different file formats.

Swap parts & sub-assemblies for alternatives.

Compute and update iProperties.

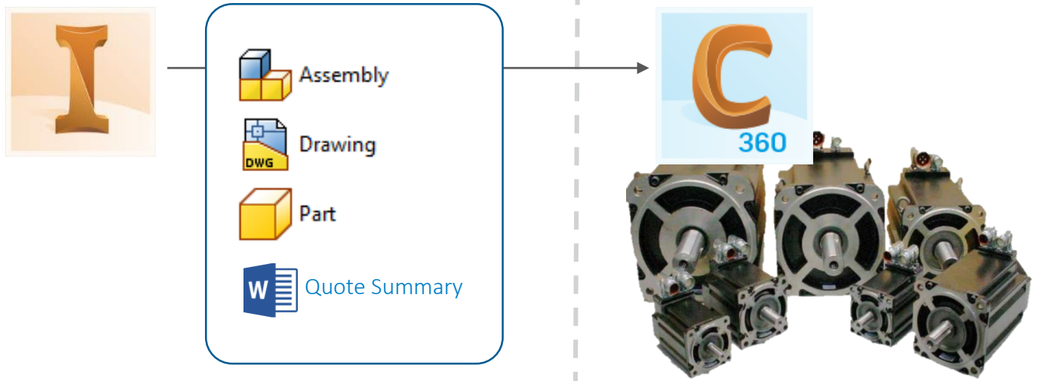
Upload to Configurator360.

Configurator C360

Autodesk Configurator 360

Online configurations made easy

- Cloud-based 3D product configurator for Autodesk® Inventor® designs
- Accessible by customers & sales teams via a web enabled device
- Improve bid response time & accuracy to help win more business, profitably



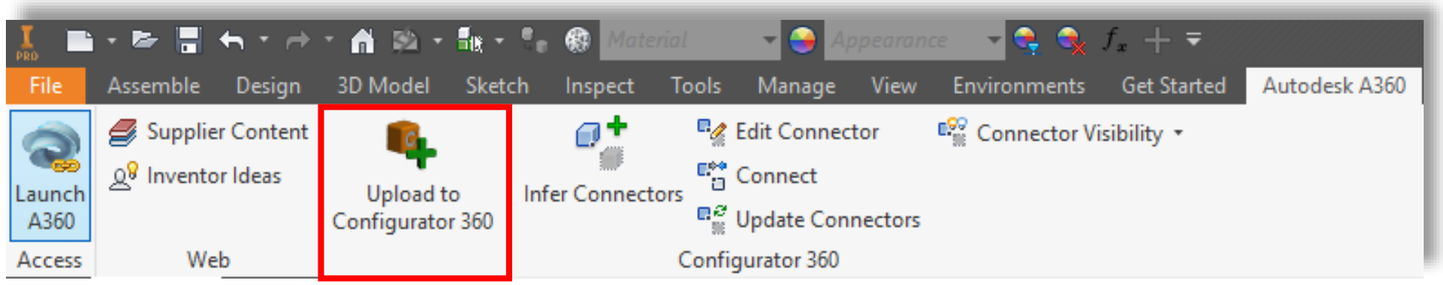
20+

2D & 3D downloadable file formats to chose from.

Design → Configure → Download → Request Quote

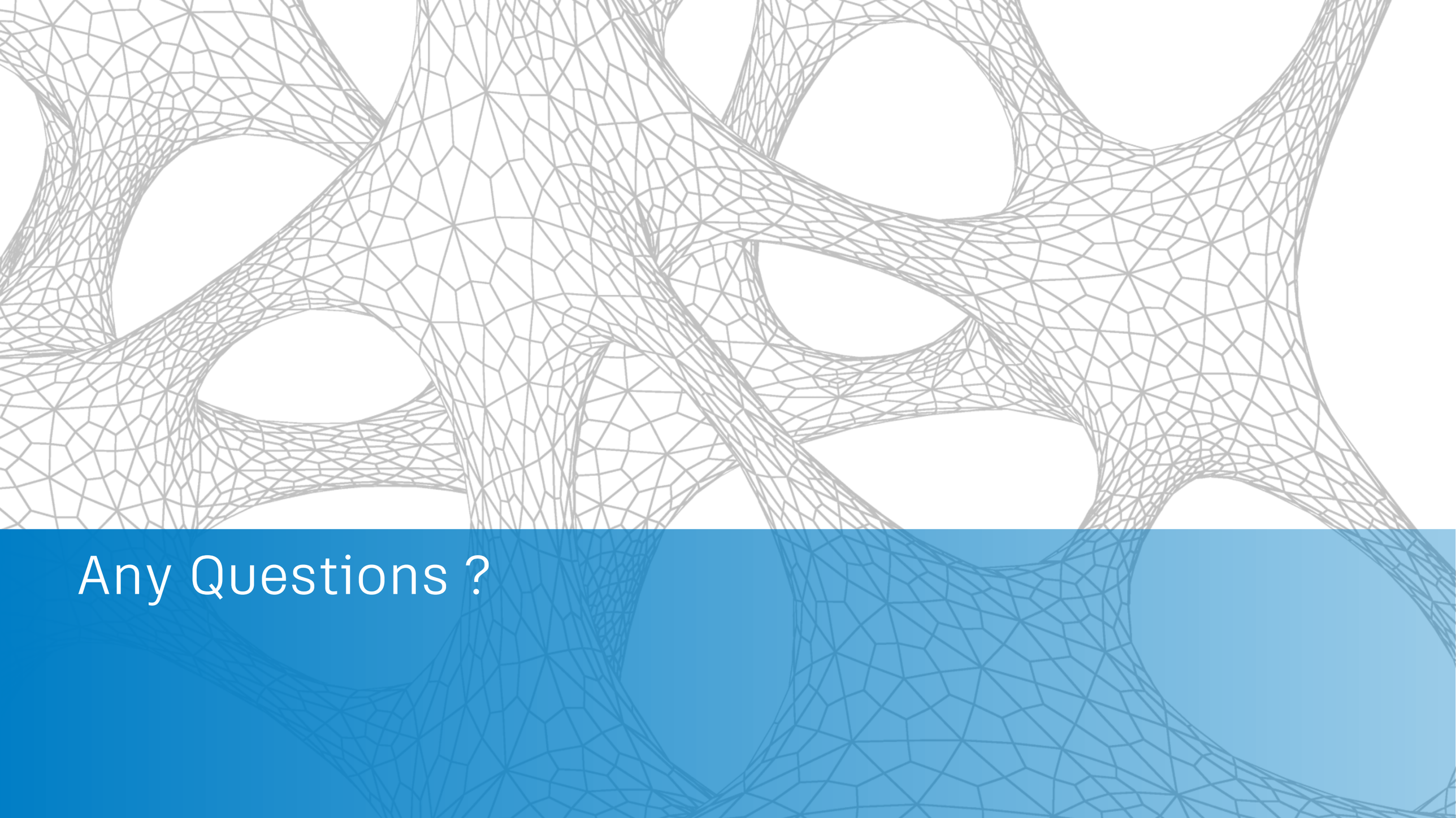
Autodesk Configurator 360 – Trying it out

- Go to **configurator360.autodesk.com**
- Sign in with your **AutodeskID**. (This will give you a 30 day trial.)
- Upload and test Inventor models using the option from the **AutodeskA360** ribbon.



- You can control the branding, content, formats, parameters, security, etc, BUT with Configurator360 it's as insanely simple to make your content available to configure online.

Configurator360 Demonstration



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