Single Minute Exchange of Design Using Autodesk Inventor

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Class summary

Single Minute Exchange of Design provides a way to cut design time to a single digit using iLogic and the Visual Basic (VBA) application programming interface to Autodesk Inventor software. We cover basic programming in VBA and iLogic. We explain how to combine VBA and iLogic for design automation

Key learning objectives

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

- Perform basic programming
- Use the Visual Basic interface to Inventor
- Use an iLogic component in Inventor
- Automate a design in Inventor

Introduction

SMED (Single Minute Exchange of Die)



Shigeo Shingo
(新郷 重夫 1909 – 1990)

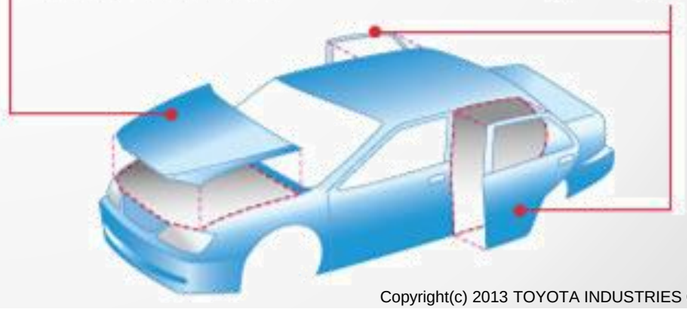
He find a way to change a dies in a single minute



Hood Outer Trim Die



Rear Door Outer Draw Die



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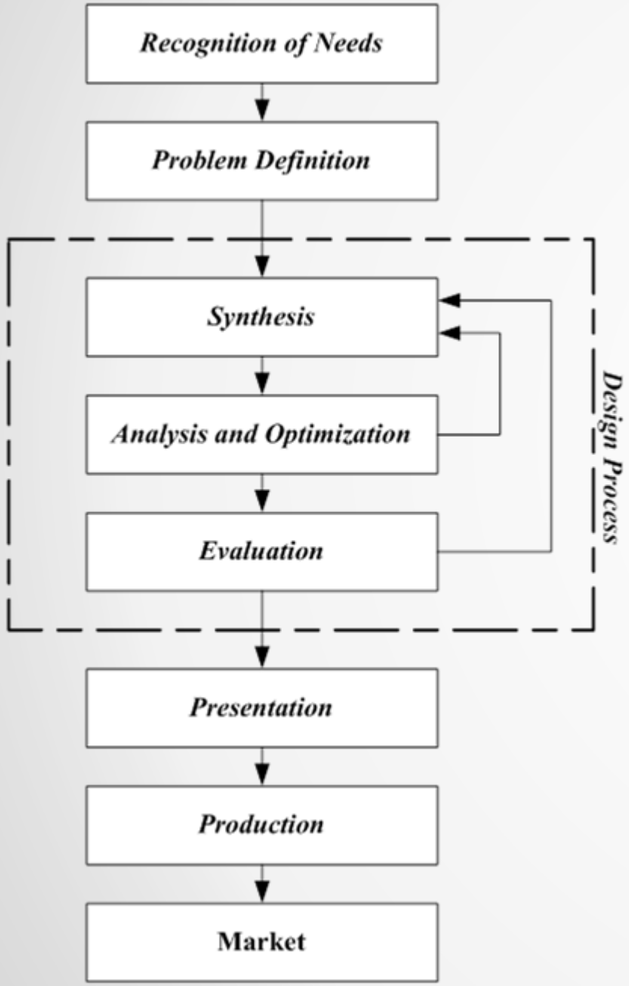
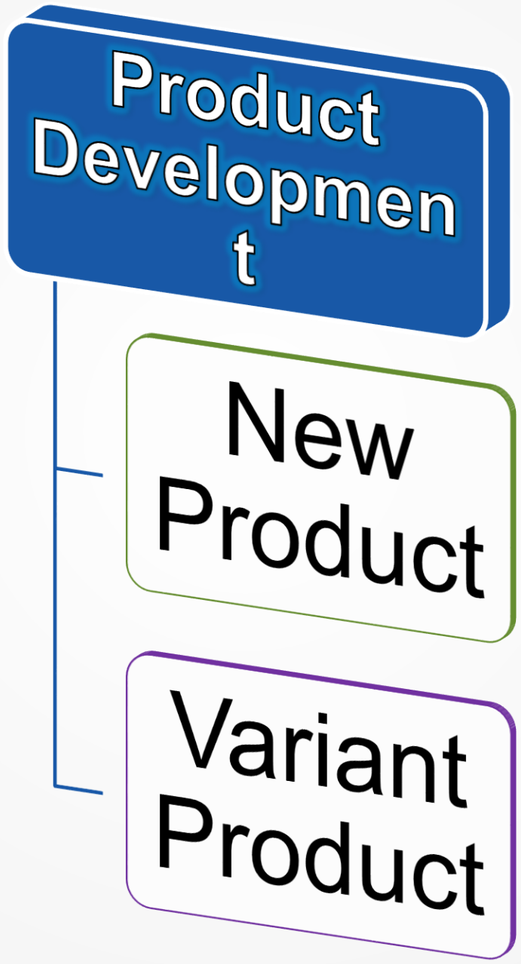
How About in Design Process??



SMEDg
(Single Minute Exchange of Design)

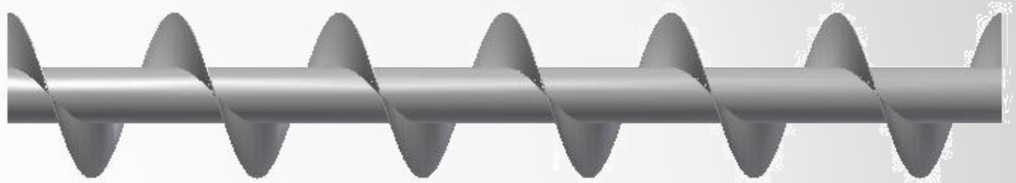
How to change a design parameter in a single minute

Product Development

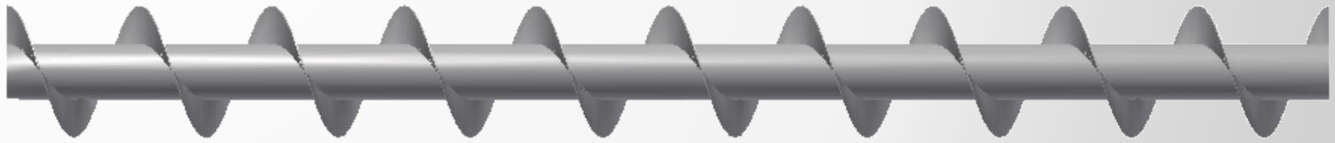


Variant Product

The first product has parameter:
Length: 1500 mm
Outer Diameter: 250 mm

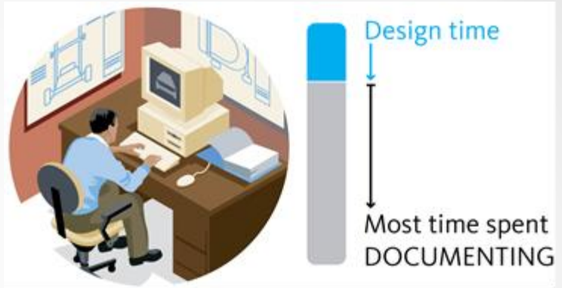


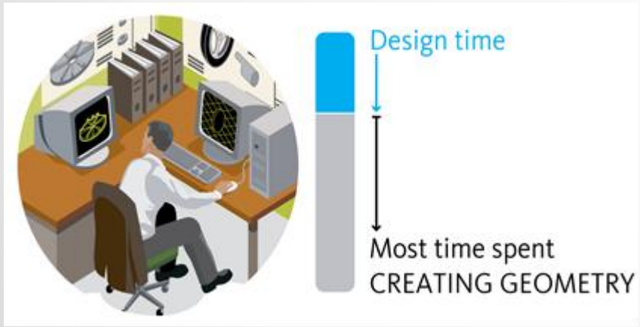
Second product has parameter:
Length: 2000 mm
Outer Diameter: 200 mm



Evolution of Design

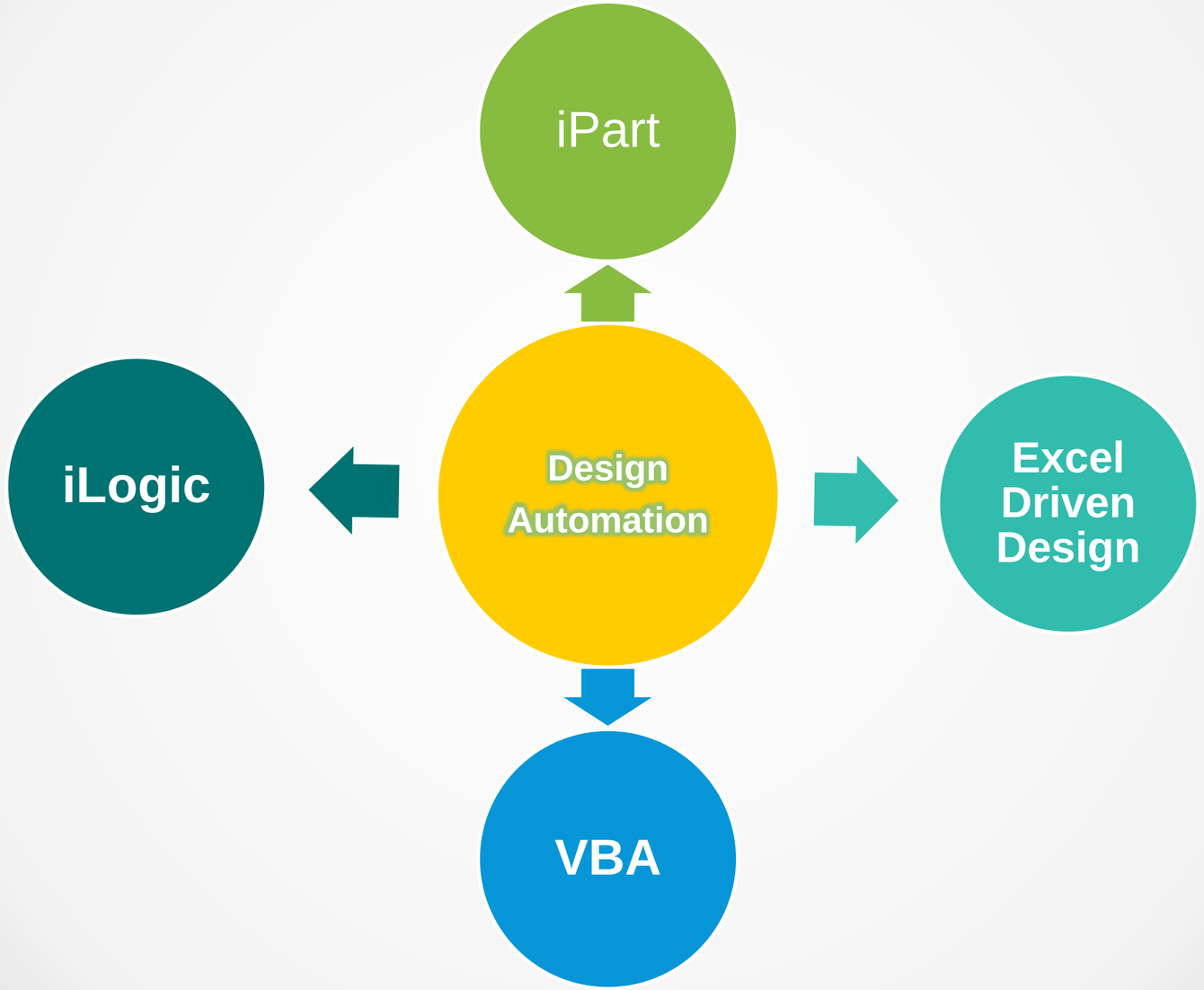




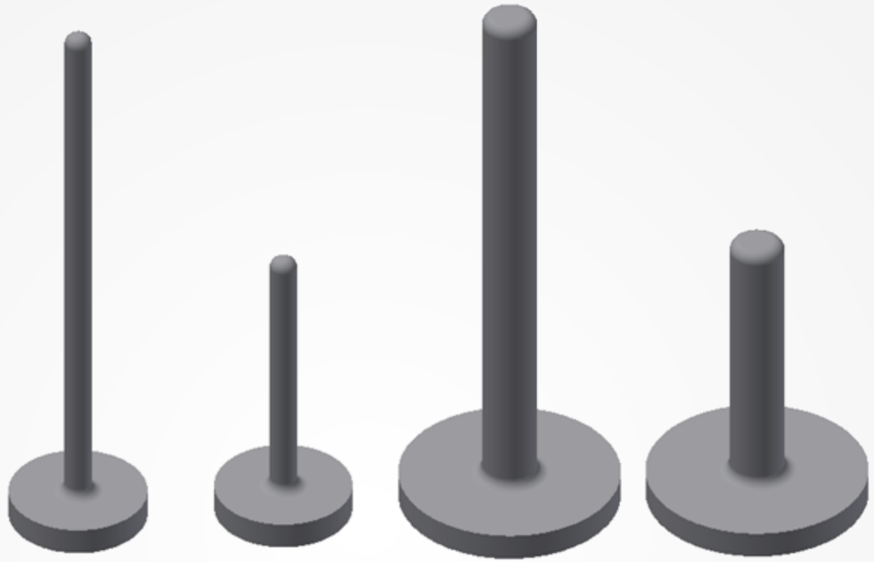


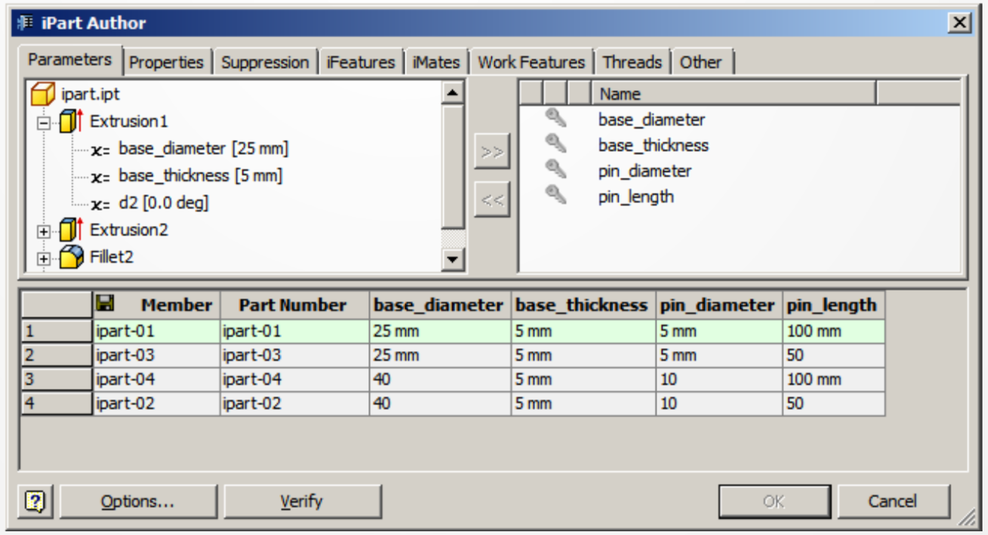


Design Automation in Autodesk Inventor

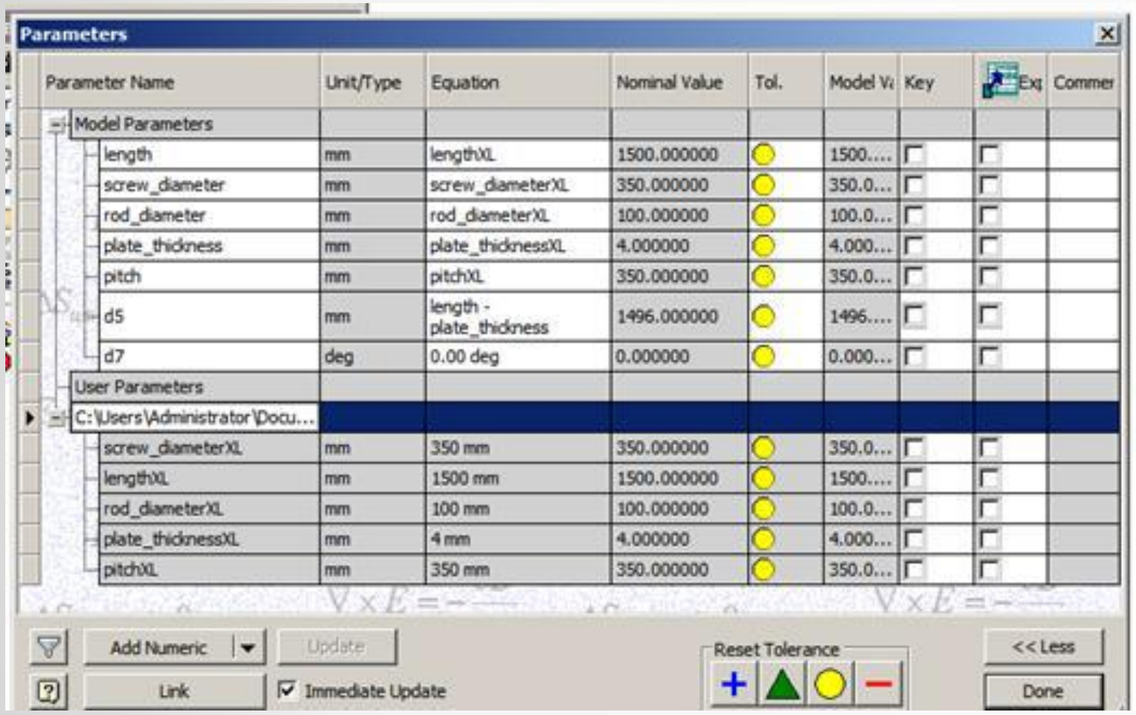


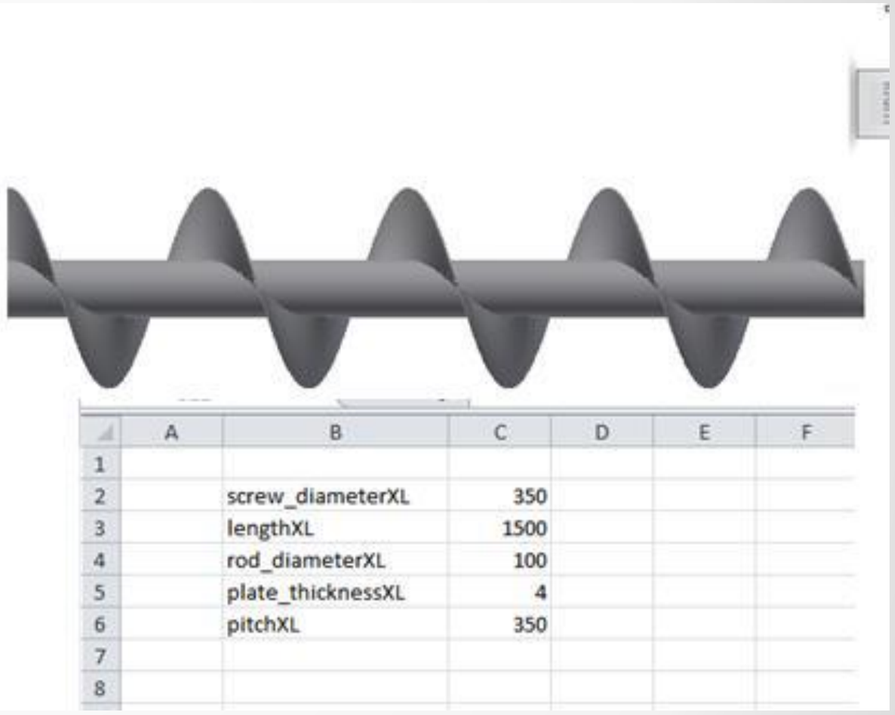
iPart




iPart Author
Parameters
Properties
Suppression
iFeatures
iMates
Work Features
Threads
Other
ipart.ipt
Extrusion1
base_diameter [25 mm]
base_thickness [5 mm]
d2 [0.0 deg]
Extrusion2
Fillet2
Name
base_diameter
base_thickness
pin_diameter
pin_length
Member | Part Number | base_diameter | base_thickness | pin_diameter | pin_length
1 | ipart-01 | ipart-01 | 25 mm | 5 mm | 5 mm | 100 mm
2 | ipart-03 | ipart-03 | 25 mm | 5 mm | 5 mm | 50
3 | ipart-04 | ipart-04 | 40 | 5 mm | 10 | 100 mm
4 | ipart-02 | ipart-02 | 40 | 5 mm | 10 | 50
Options...
Verify
OK
Cancel

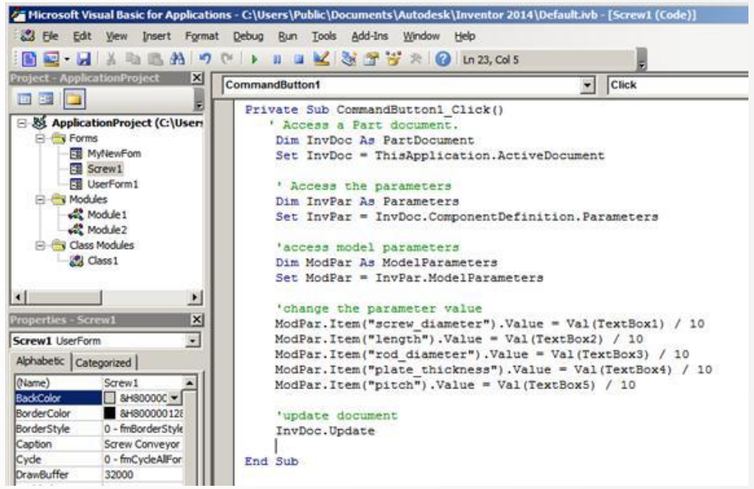
Excel Driven Design





VBA

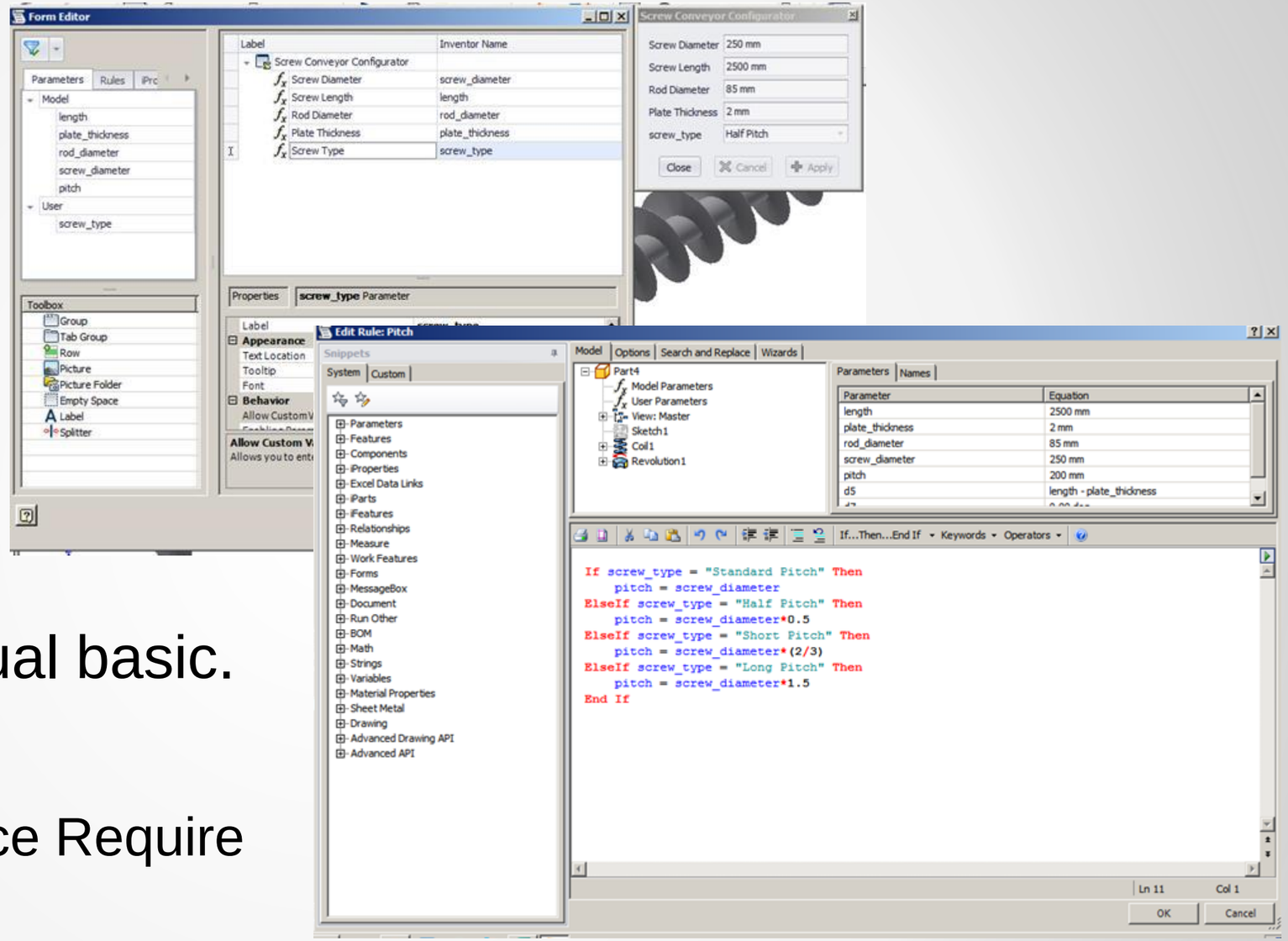
Visual Basic for Applications (VBA) is a programming environment developed by Microsoft and licensed by other companies to integrate into their applications.



INVENTOR

AUTOCAD

iLogic

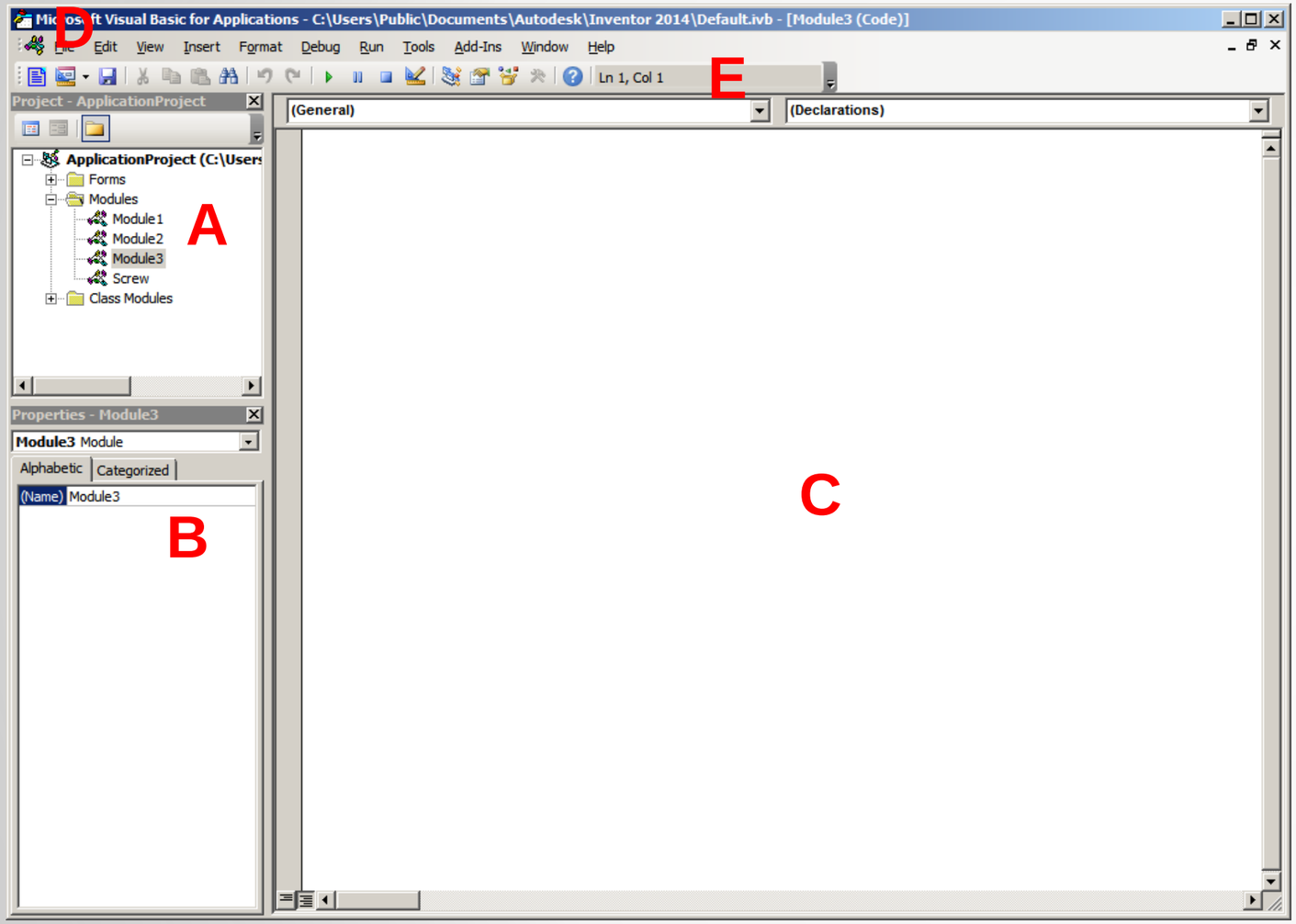


iLogic is a simplified visual basic.

No Programming Experience Require

VBA (Visual Basic Application)

VBA Editor



A The Project Explorer displays a list of projects and project items in Autodesk Inventor documents.

B The Properties window displays a list of the properties for the selected item.

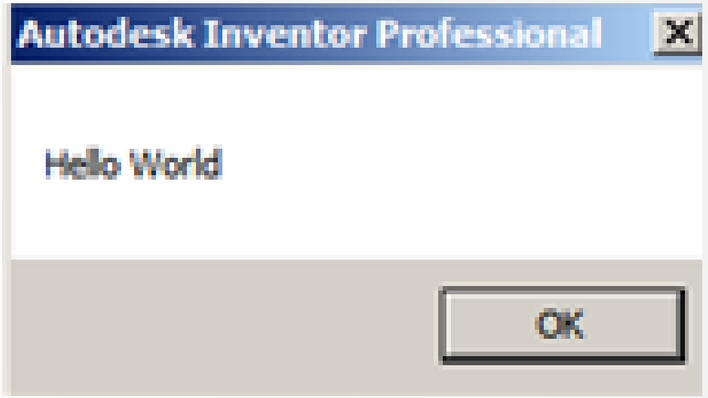
C The Code window

D The menu bar displays the commands you use to build, run, and debug your program

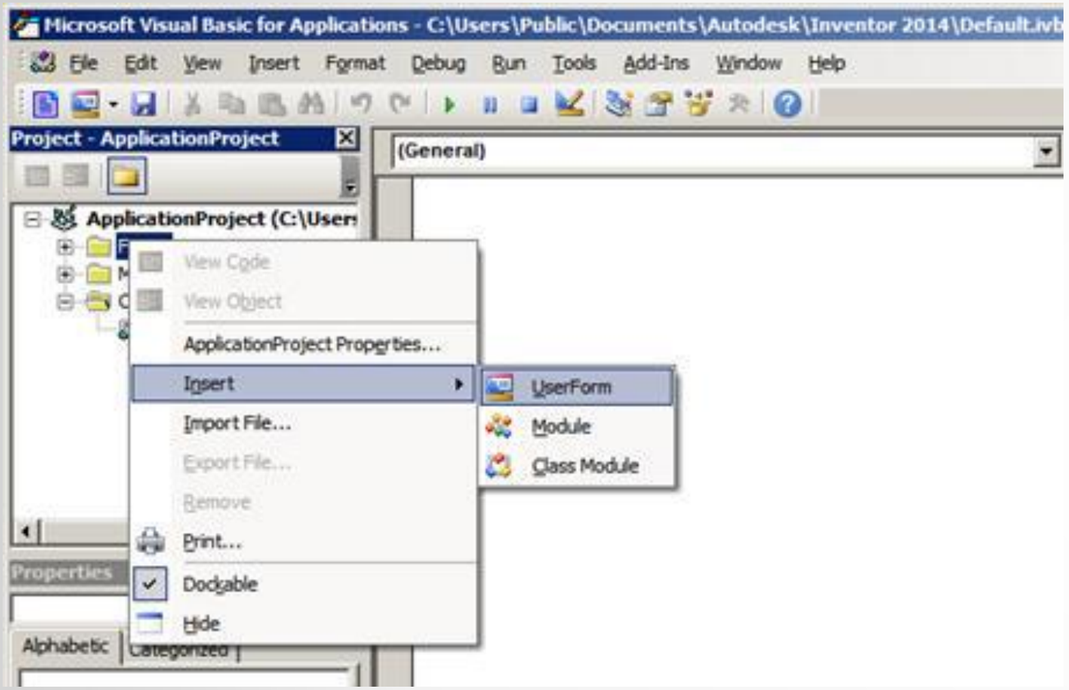
E The toolbar provides quick access to commonly used commands in the development environment.

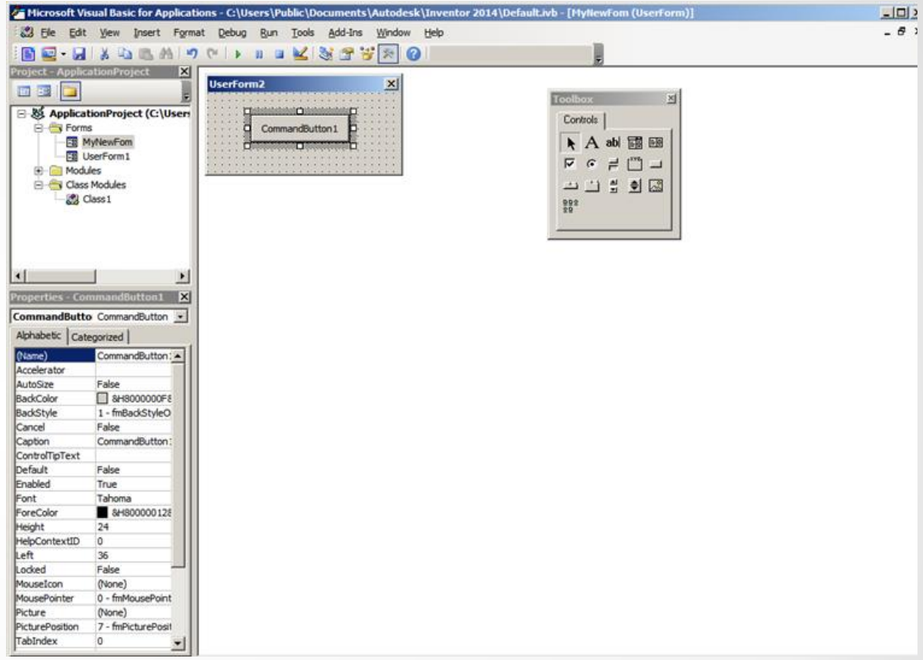
Our First Simple Programming

```
Public Sub HelloWorld()
    MsgBox ("Hello World")
End Sub
```



Make a program using user form

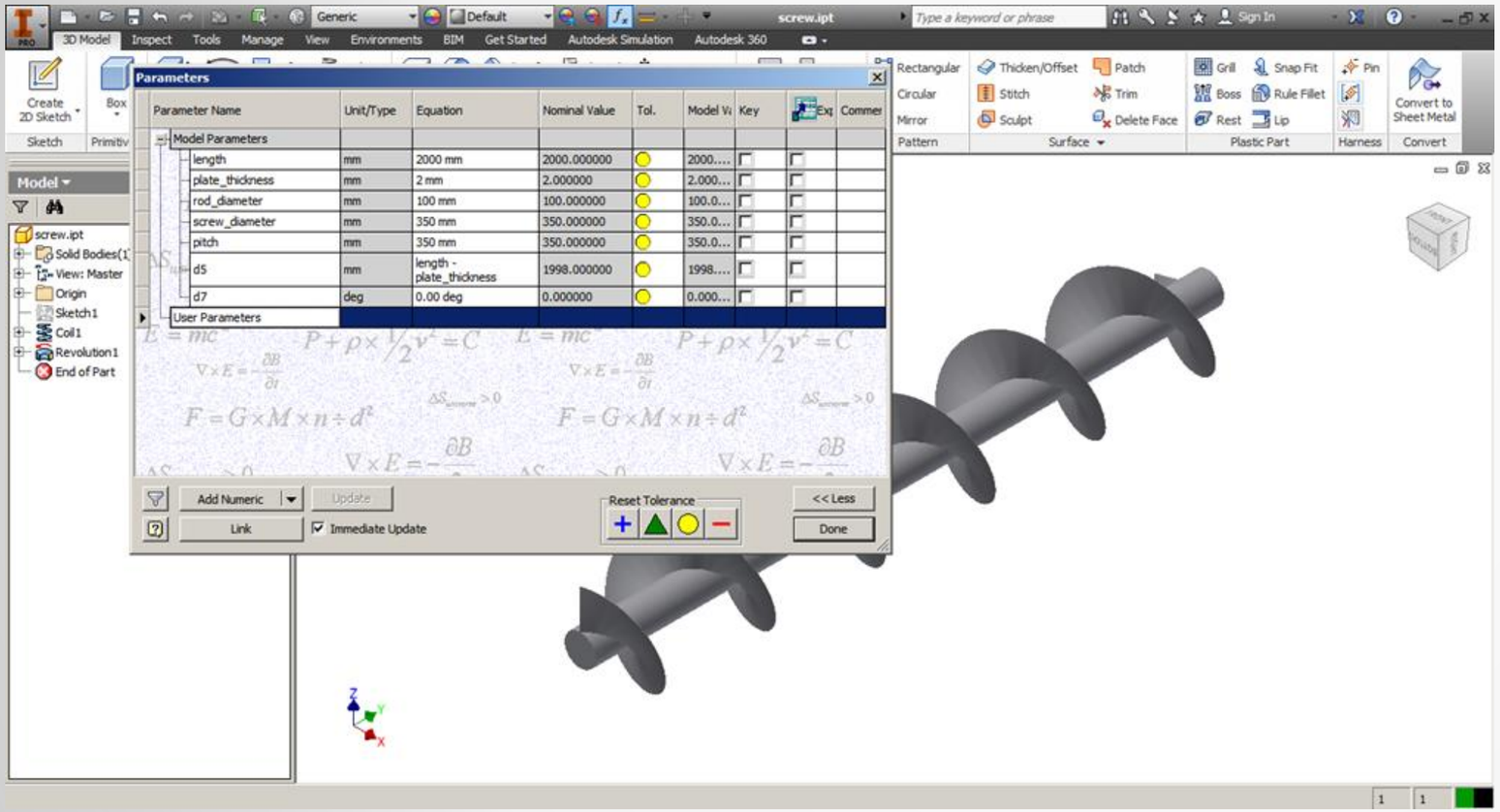





```
Private Sub CommandButton1_.click()
        MsgBox (“Hello World")
End Sub
```



How to setting a parameter Using VBA (Parameter Table)



screw_diameter : 350 mm

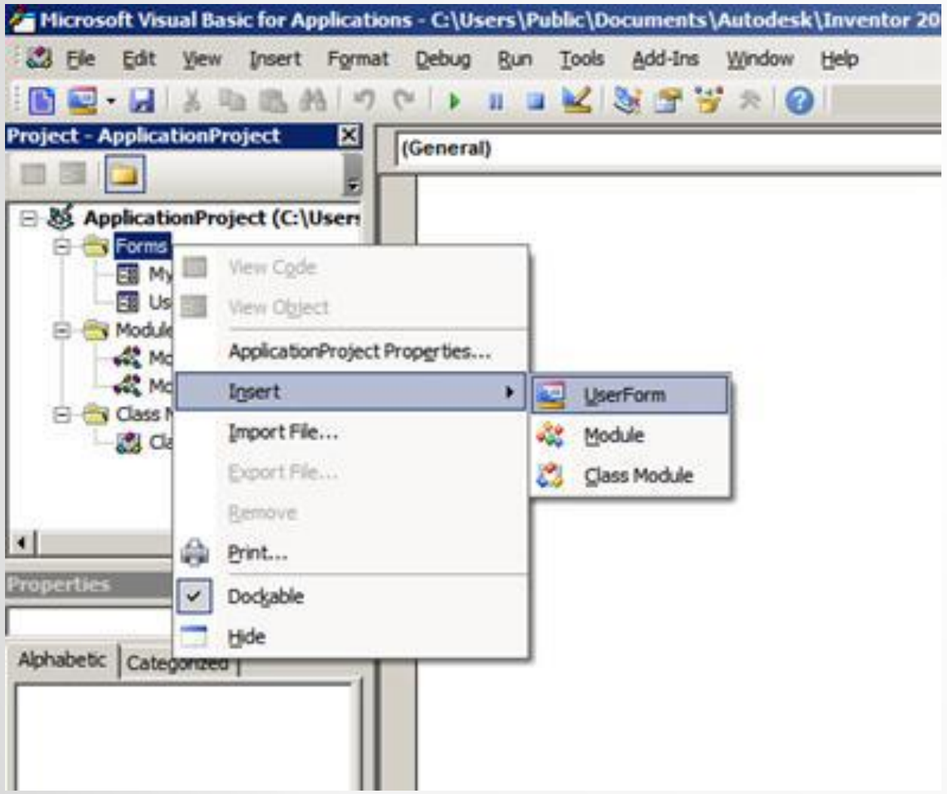
length : 2000 mm

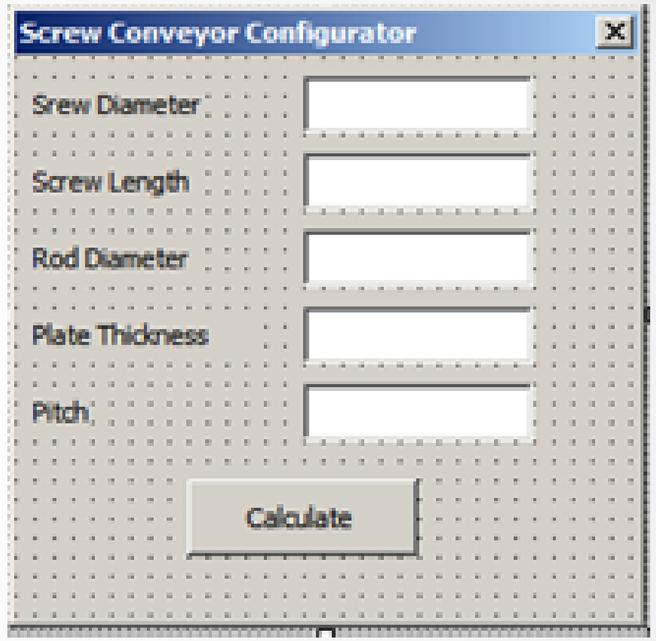
rod_diameter: 100 mm

plate_thickness: 2 mm

pitch: 350 mm

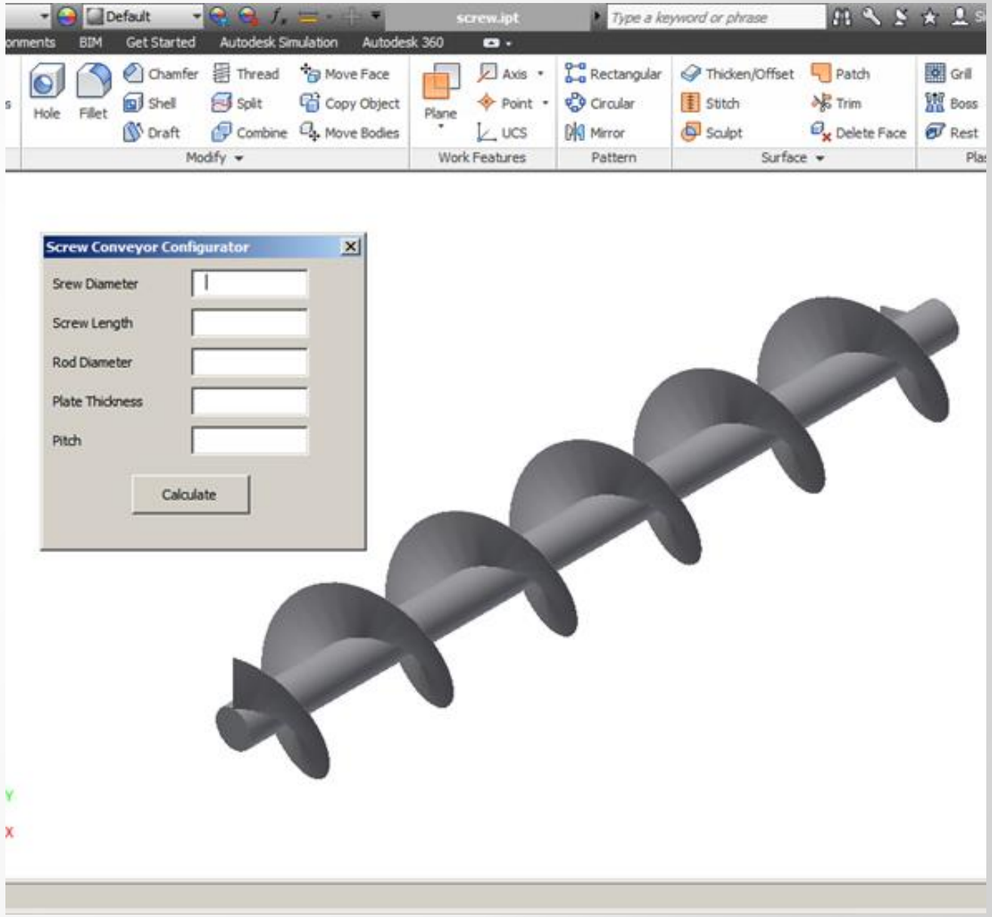
How to setting a parameter Using VBA (User Form)





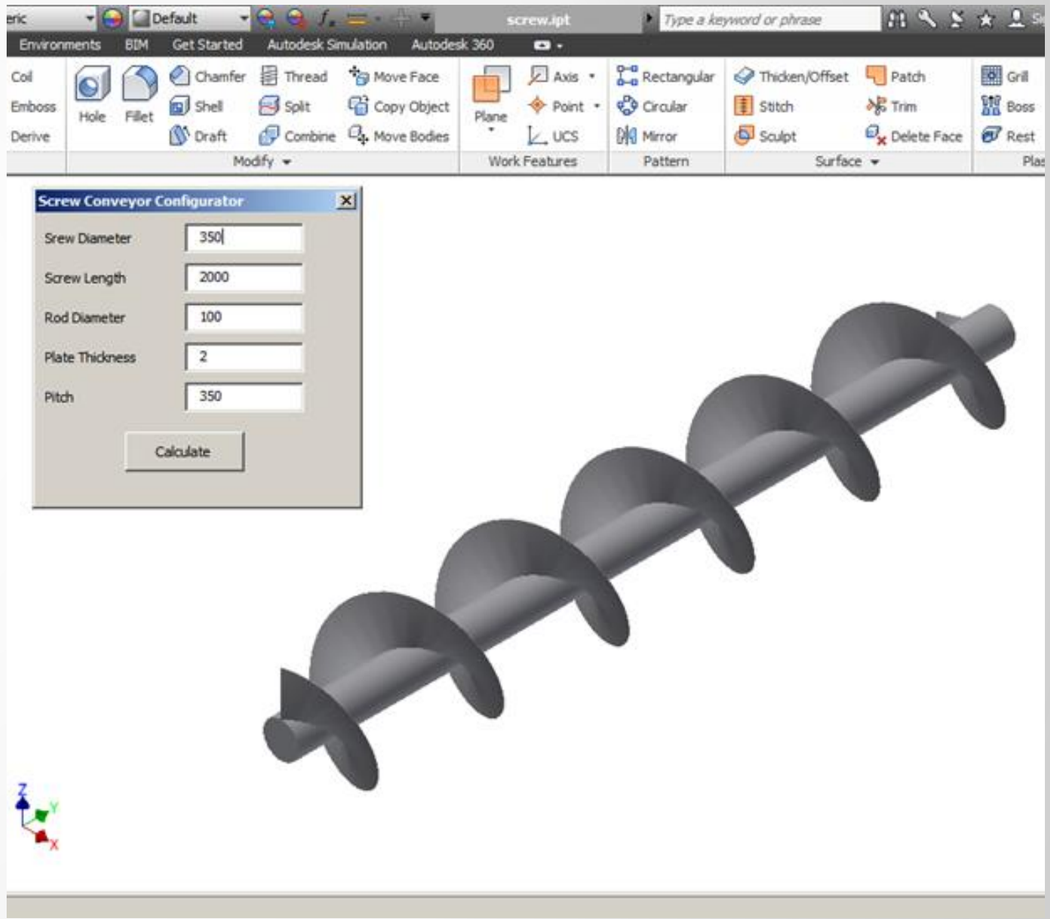
How to setting a parameter Using VBA (Main Code To Setting Parameter)

```
Private Sub CommandButton1_Click()
' code to ccess a Part document.
        Dim InvDoc As PartDocument
        Set InvDoc = ThisApplication.ActiveDocument
' code to access the parameters
        Dim InvPar As Parameters
        Set InvPar = InvDoc.ComponentDefinition.Parameters
' code to access model parameters
        Dim ModPar As ModelParameters
        Set ModPar = InvPar.ModelParameters
‘ code to change the parameter value
        ModPar.Item("screw_diameter").Value = Val(TextBox1) / 10
        ModPar.Item("length").Value = Val(TextBox2) / 10
        ModPar.Item("rod_diameter").Value = Val(TextBox3) / 10
        ModPar.Item("plate_thickness").Value = Val(TextBox4) / 10
        ModPar.Item("pitch").Value = Val(TextBox5) / 10
' code to update document
        InvDoc.Update
End Sub
```

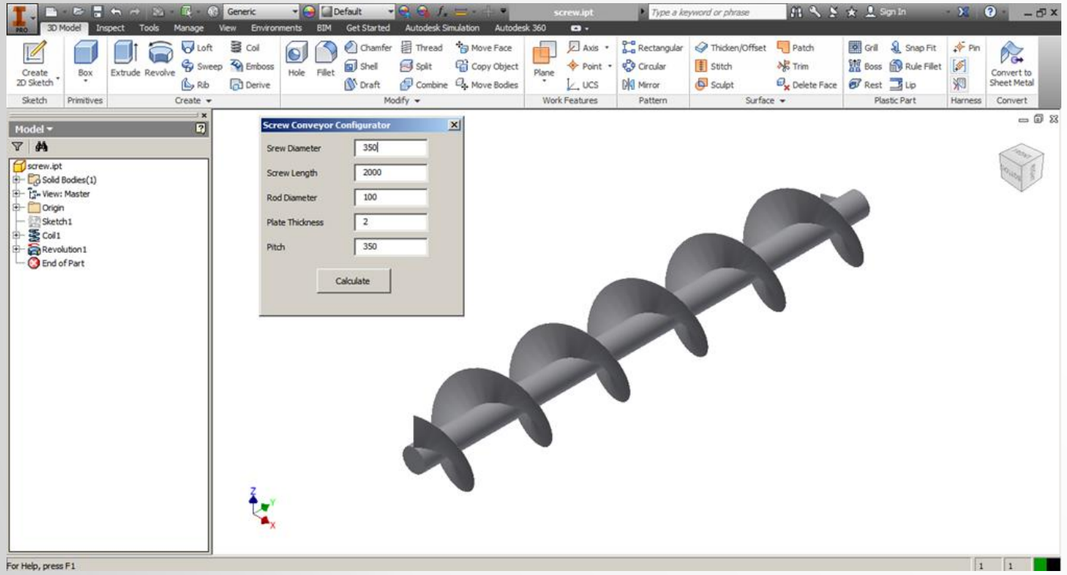


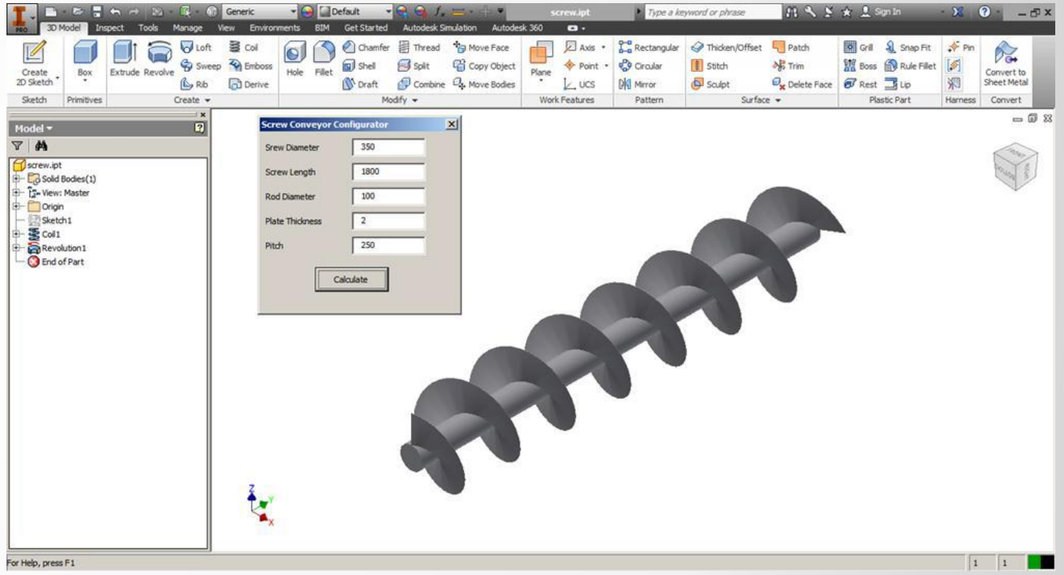
How to setting a parameter Using VBA (Code To Read Parameter Value)

```
Private Sub UserForm_Activate()
' code to ccess a Part document.
        Dim InvDoc As PartDocument
        Set InvDoc = ThisApplication.ActiveDocument
' code to access the parameters
        Dim InvPar As Parameters
        Set InvPar = InvDoc.ComponentDefinition.Parameters
' code to access model parameters
        Dim ModPar As ModelParameters
        Set ModPar = InvPar.ModelParameters
‘ code to read the parameter value
        TextBox1.Text = (ModPar.Item("screw_diameter").Value) *
        10
        TextBox2.Text = (ModPar.Item("length").Value) * 10
        TextBox3.Text = (ModPar.Item("rod_diameter").Value) * 10
        TextBox4.Text = (ModPar.Item("plate_thickness").Value) *
        10
        TextBox5.Text = (ModPar.Item("pitch").Value) * 10
End Sub
```



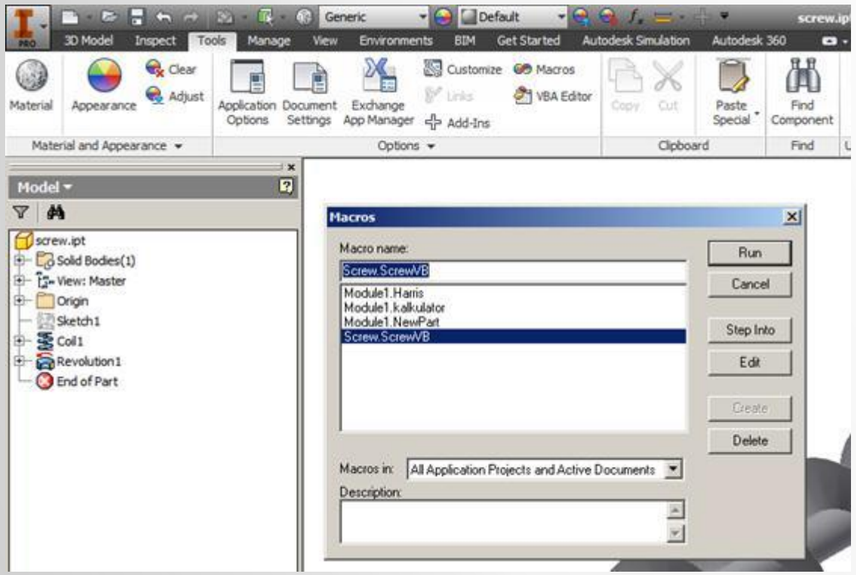
How to setting a parameter Using VBA (Run The Program)

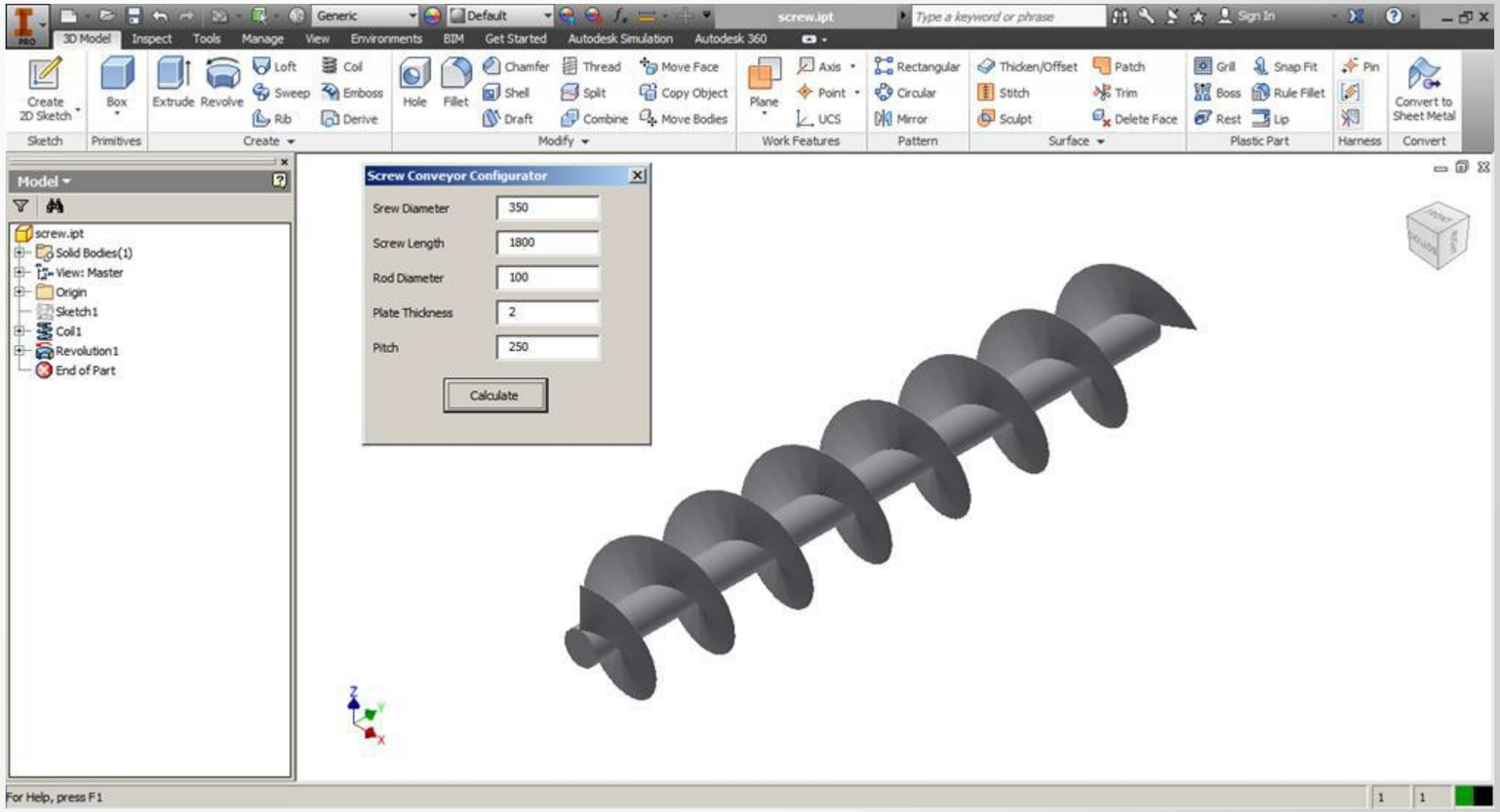




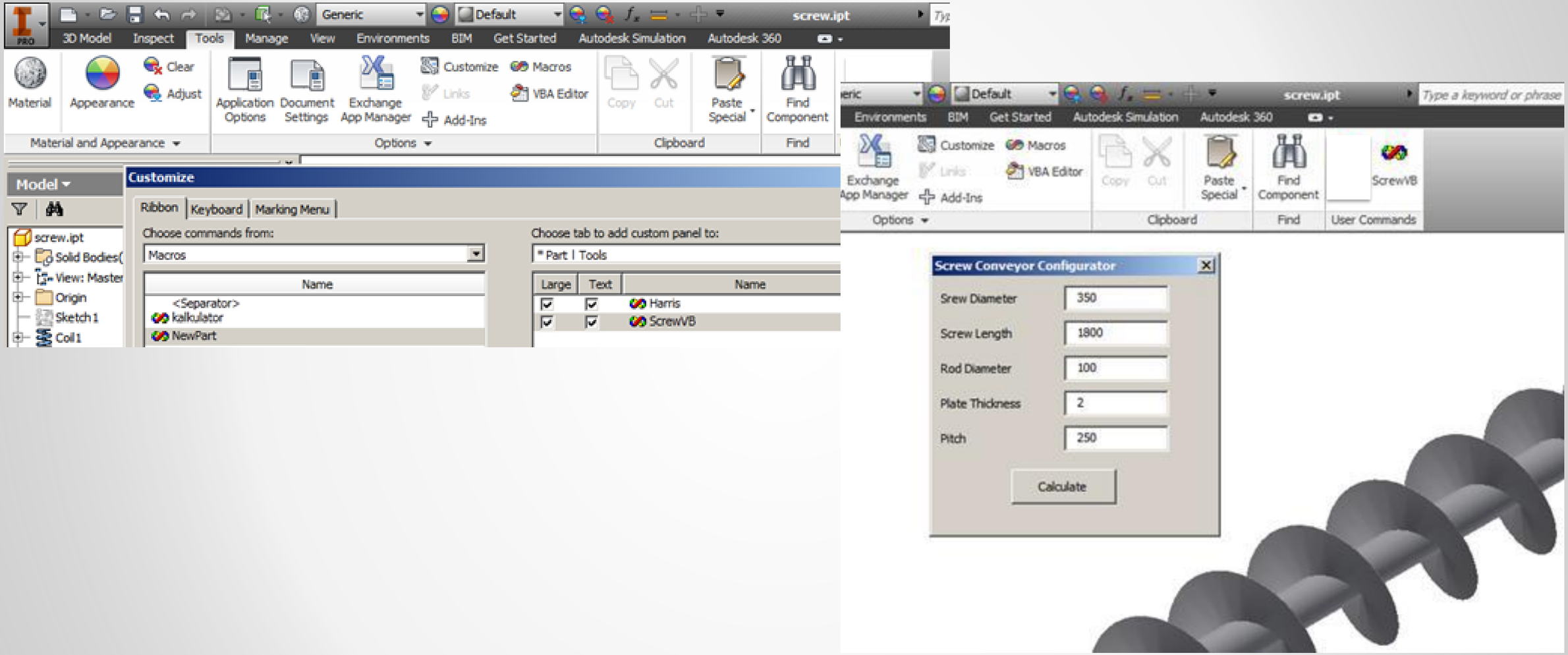
How to setting a parameter Using VBA (Run The Program)

```
Public Sub ScrewVB()
        Screw1.show
End Sub
```

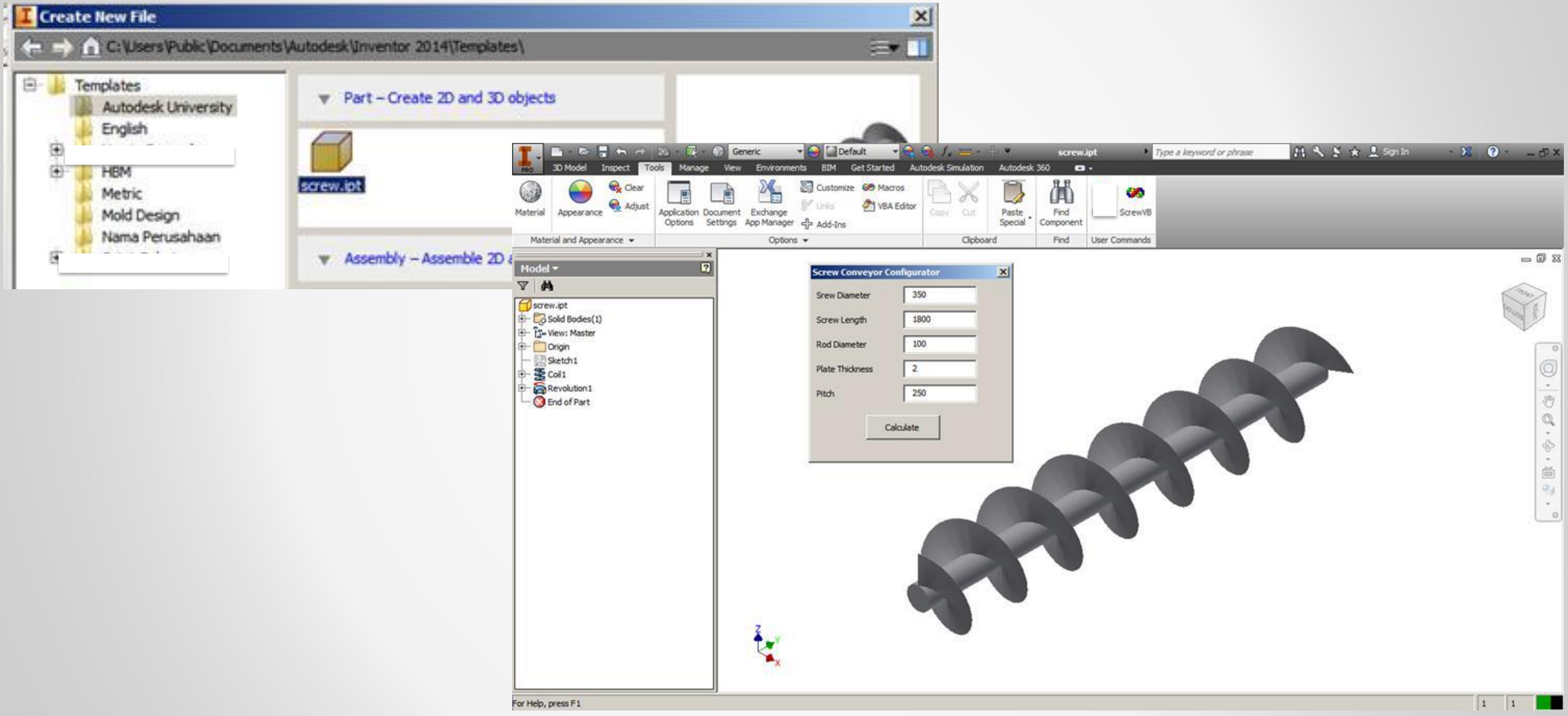




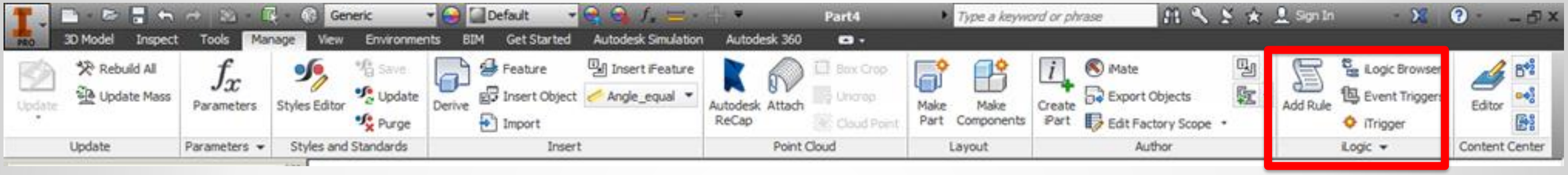
To make a custom Ribbon Menu



To make a design template

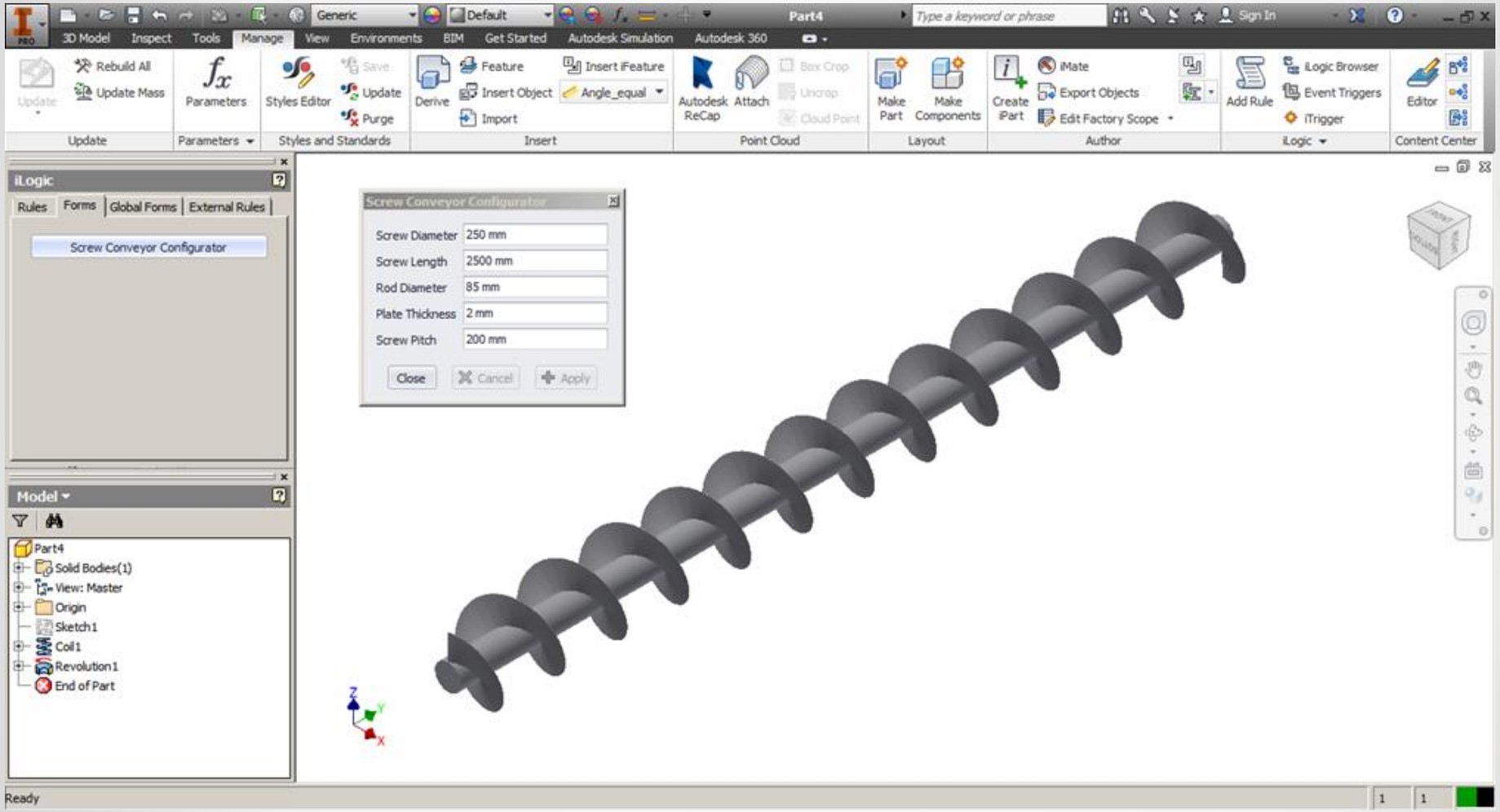


iLogic Menu

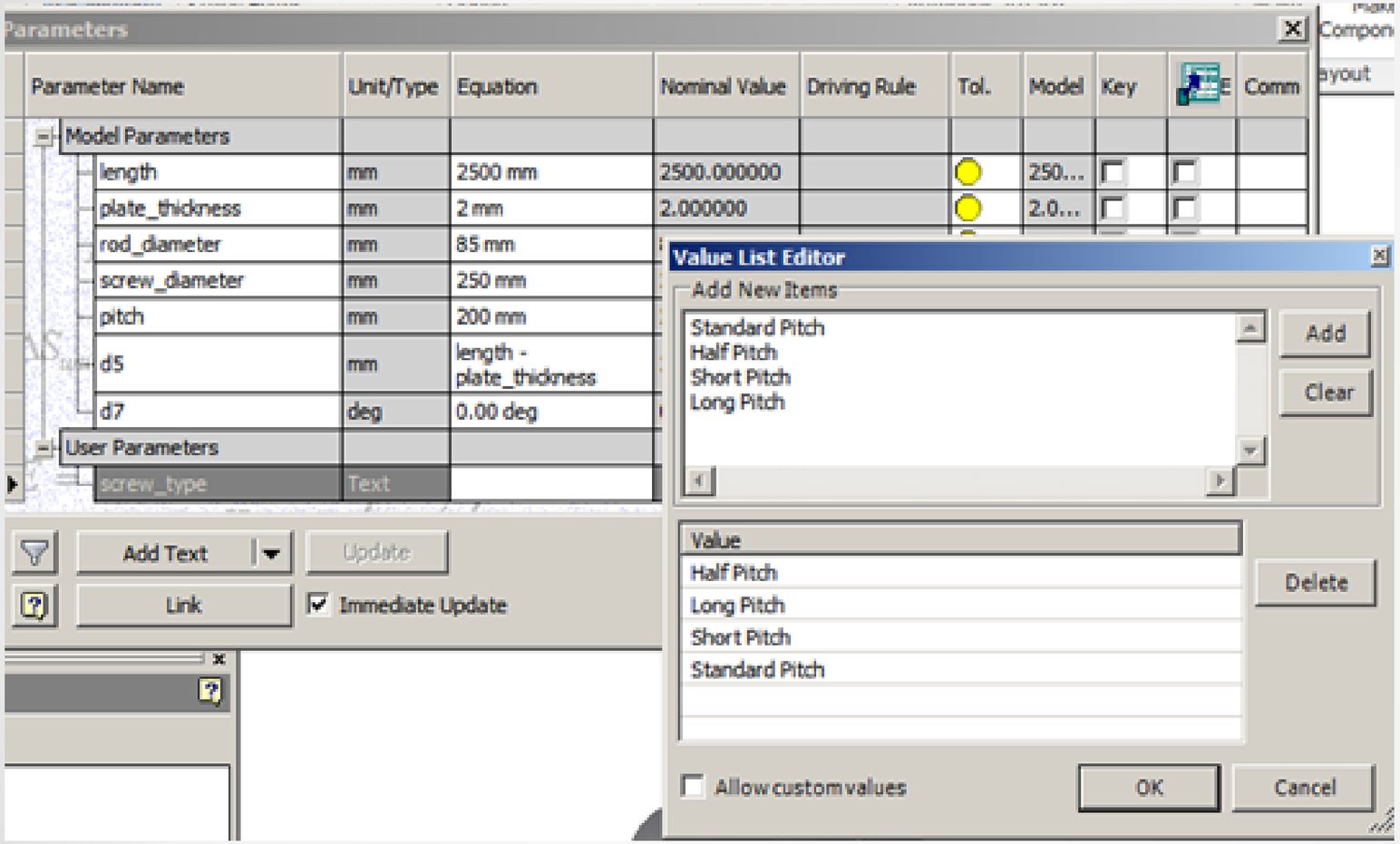

Generic
Default
Part4
Type a keyword or phrase
Sign In
3D Model
Inspect
Tools
Manage
View
Environments
BIM
Get Started
Autodesk Simulation
Autodesk 360
Rebuild All
Update Mass
Update
Parameters
Styles Editor
Save
Update
Purge
Derive
Feature
Insert Object
Import
Insert iFeature
Angle_equal
Autodesk ReCap
Attach
Box Crop
Uncrop
Cloud Point
Make Part
Make Components
Create iPart
iMate
Export Objects
Edit Factory Scope
Add Rule
iLogic Browser
Event Triggers
iTrigger
Editor
Update
Parameters
Styles and Standards
Insert
Point Cloud
Layout
Author
iLogic
Content Center


iLogic
Rules
Forms
Global Forms
External Rules
Part4

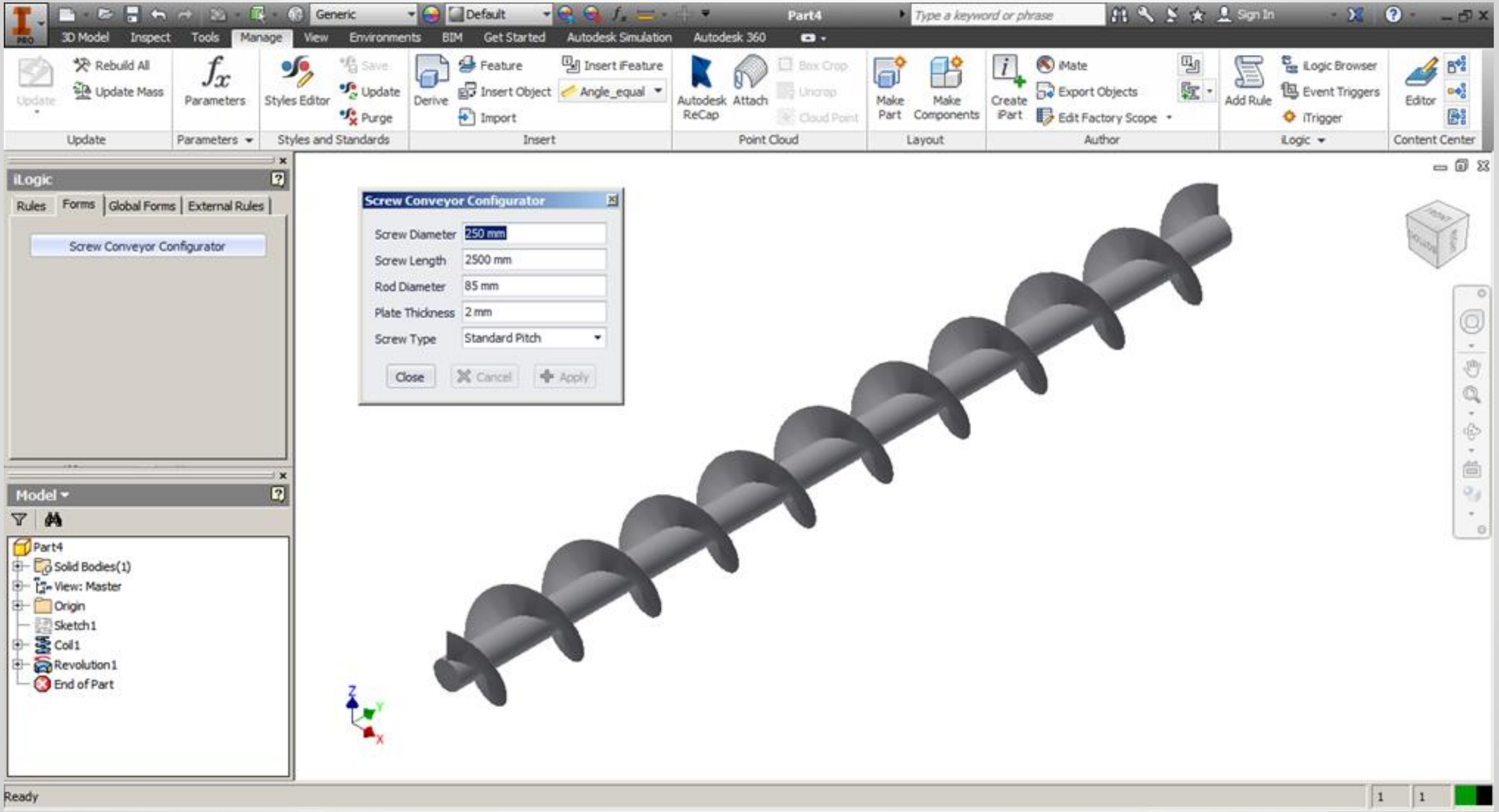
How To Set Parameter Using iLogic



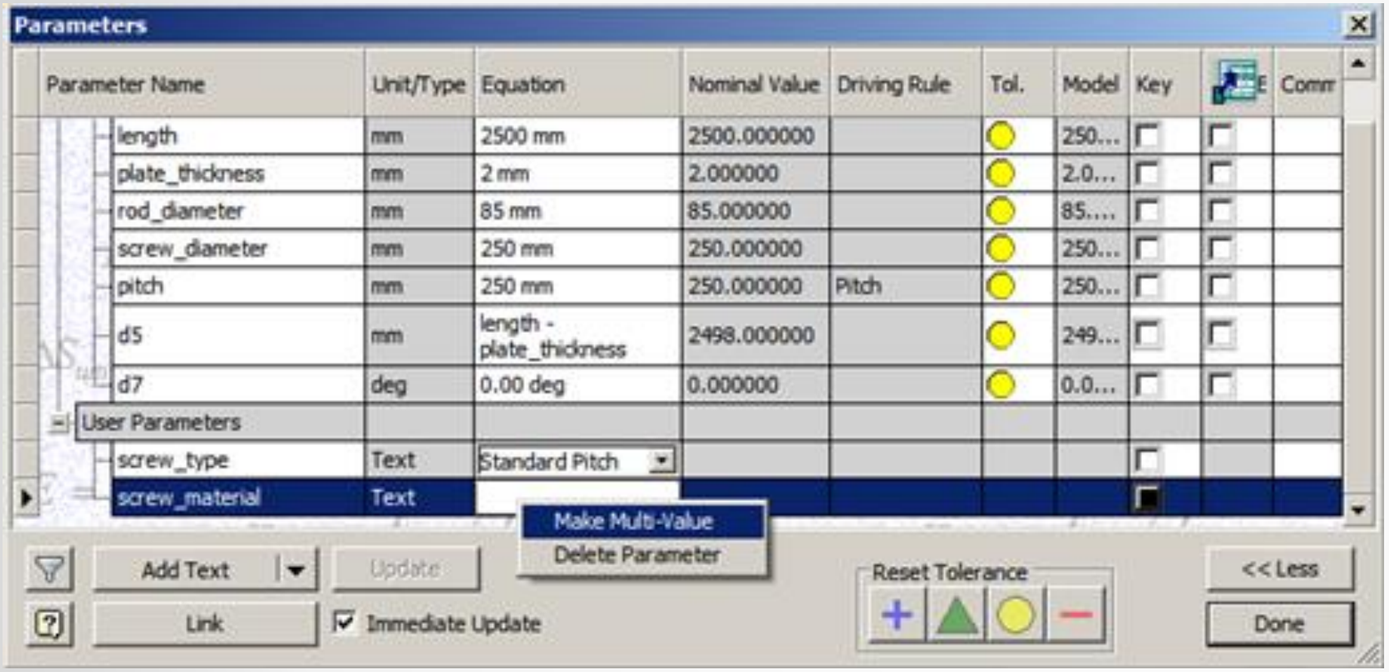
Rule Based Parameter in iLogic

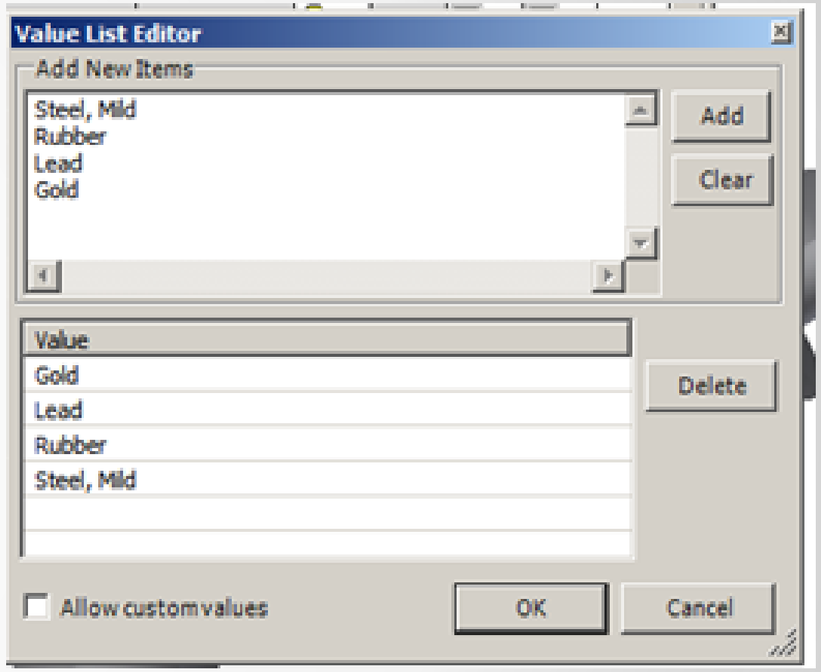


Rule Based Parameter in iLogic



Access iProperties using iLogic

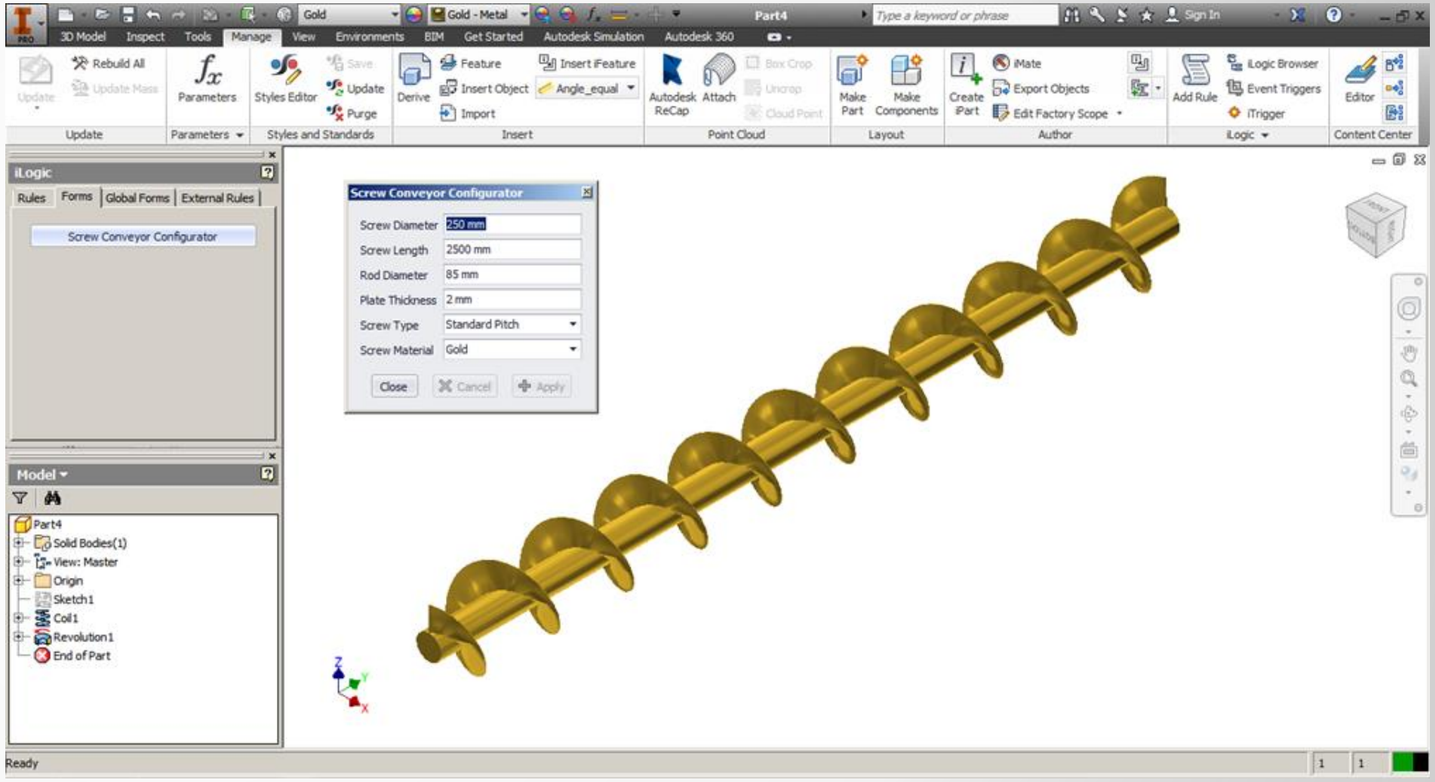




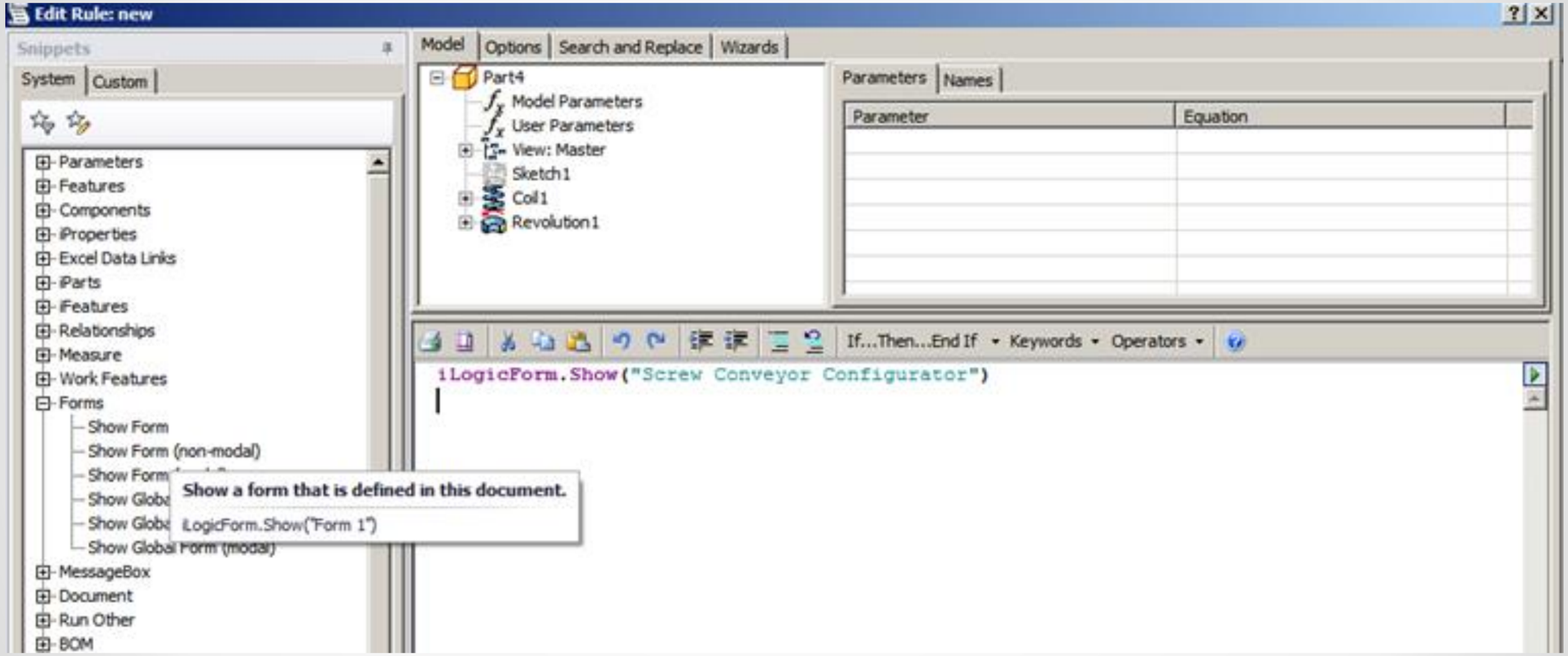
Access iProperties using iLogic (Change Material Properties)

```
If  screw_material="Gold" Then
        iProperties.Material="Gold"
ElseIf  screw_material="Steel, Mild" Then
        iProperties.Material="Steel, Mild"
ElseIf  screw_material="Rubber" Then
        iProperties.Material="Rubber"
ElseIf  screw_material="Lead" Then
        iProperties.Material="Lead"

End If
```

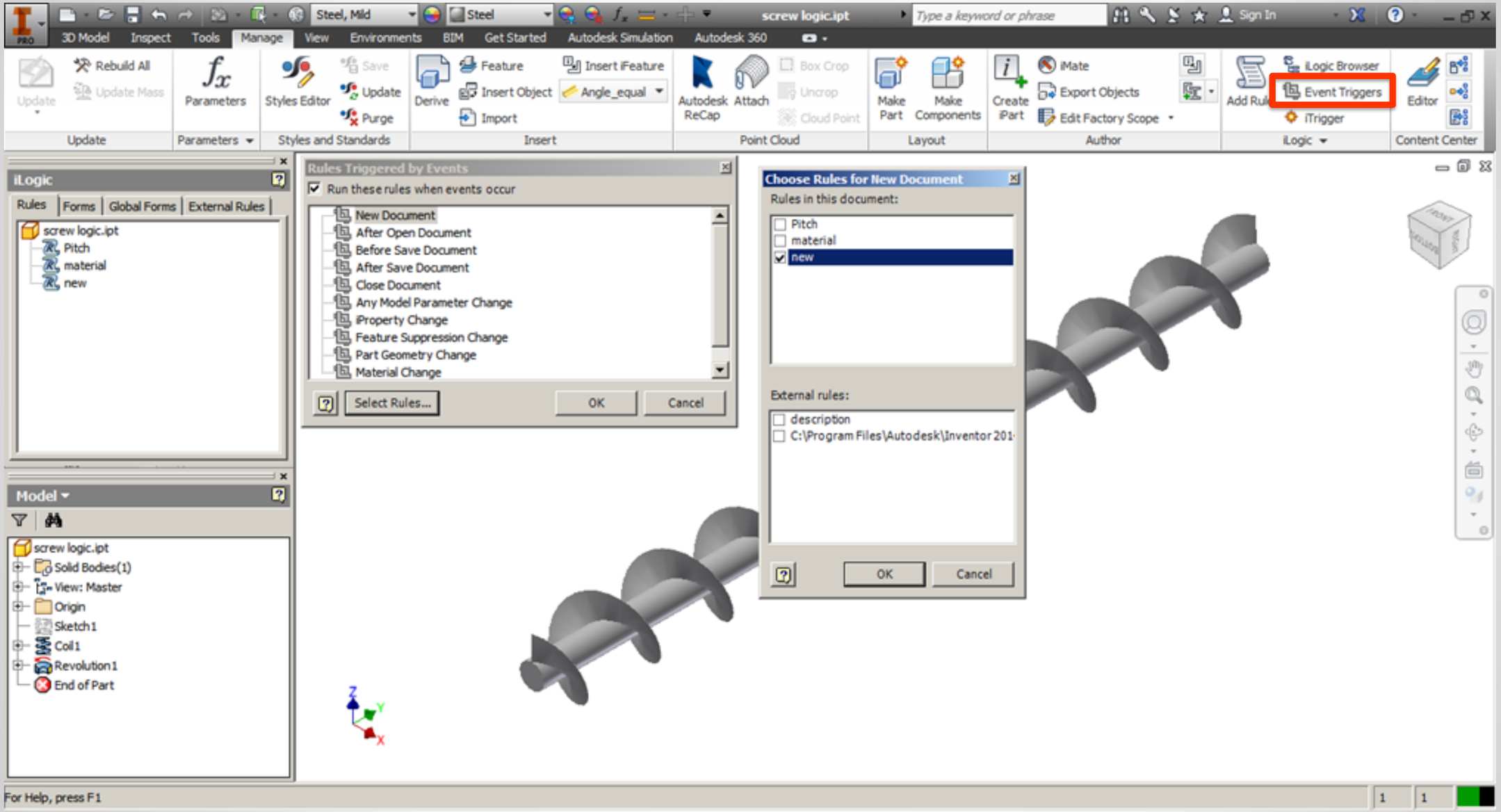


Event Trigger

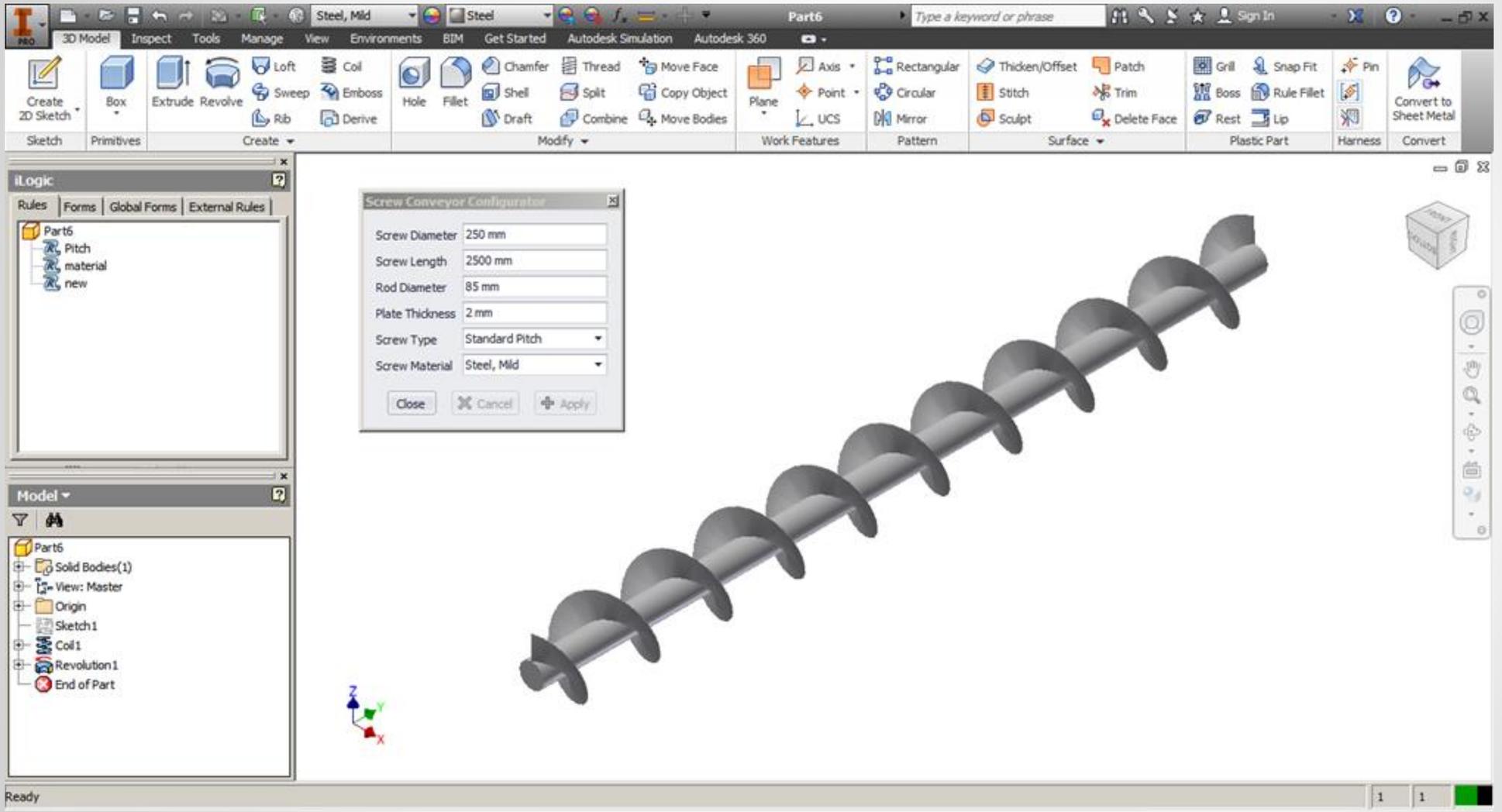


iLogicForm.Show (“Screw Conveyor Configurator"**)**

Event Trigger



New Product Development Using iLogic





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