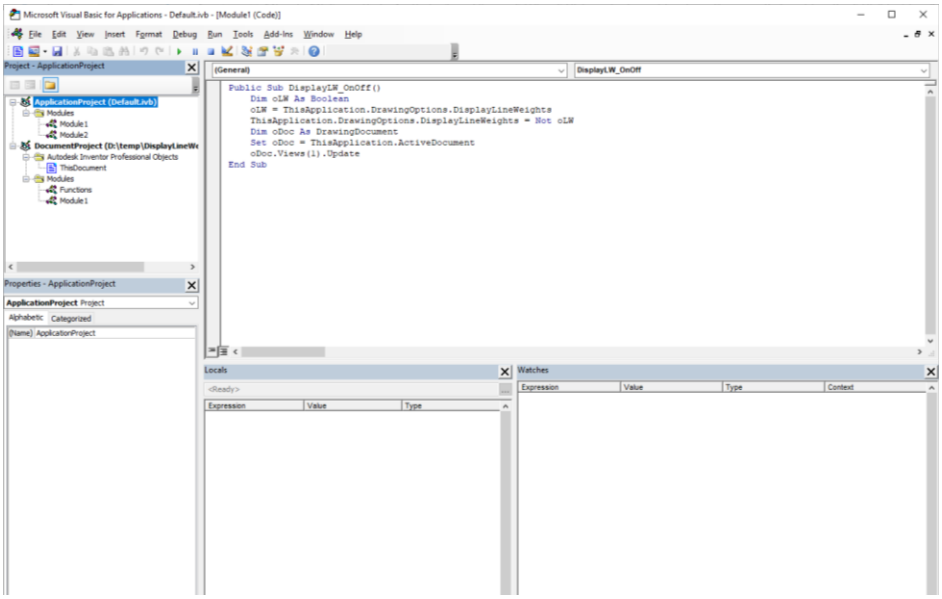
**Step 1: Create a new VBA Macro**

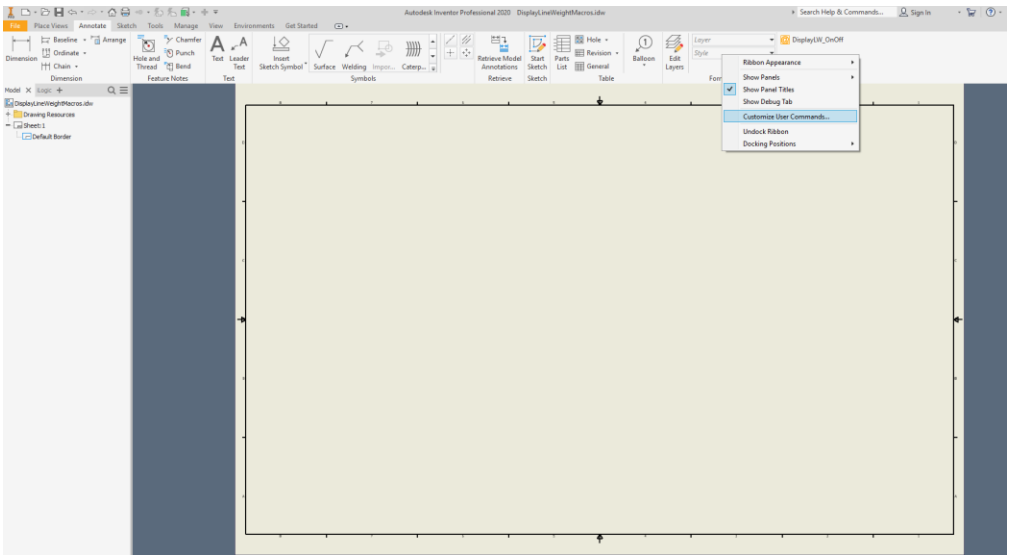
1. Launch VBA Editor
2. Goto ApplicationProject
3. Activate Module 1 or insert new Module
4. Insert the following lines of text into the text window (Shown below) in blue
5. Save
6. Close

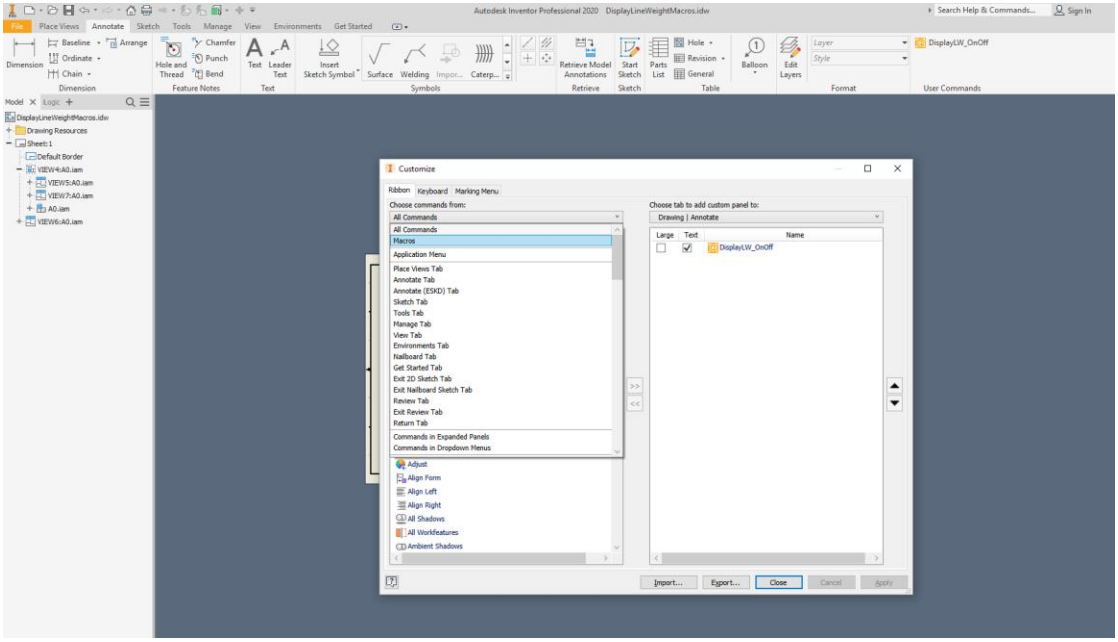
```
Public Sub DisplayLW_OnOff()
    Dim oLW As Boolean
    oLW = ThisApplication.DrawingOptions.DisplayLineWeights
    ThisApplication.DrawingOptions.DisplayLineWeights = Not oLW
    Dim oDoc As DrawingDocument
    Set oDoc = ThisApplication.ActiveDocument
    Dim oViews As Views
    Set oViews = oDoc.Views
    For Each oView In oViews
        oView.Update
    Next
End Sub
```

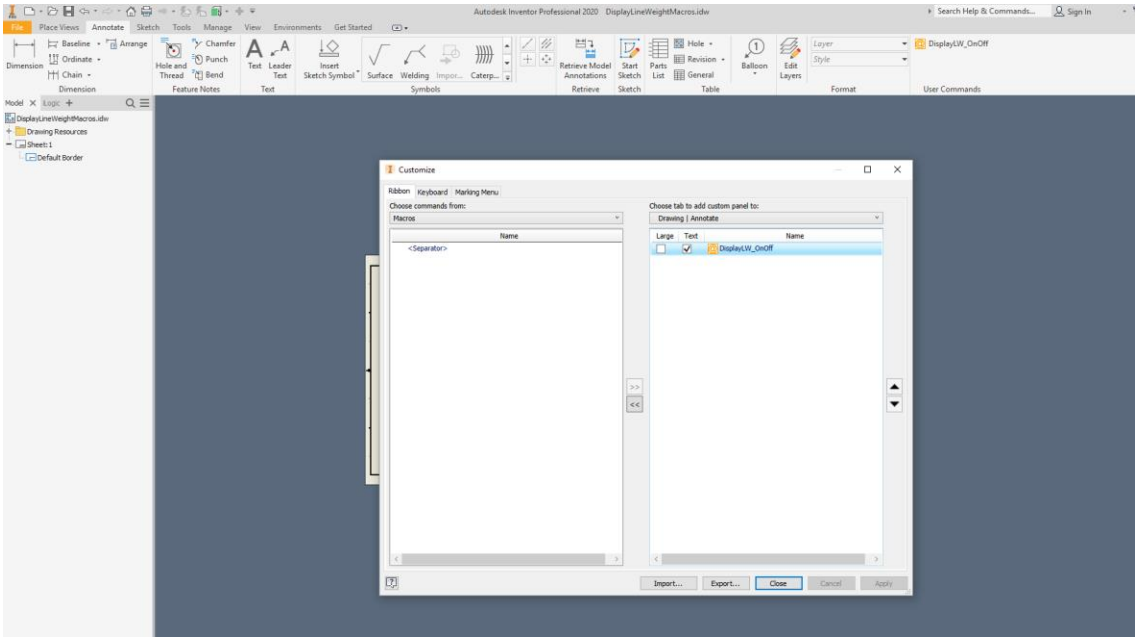


**Step 2: Create a new Ribbon Button**

1. Right Click on the Ribbon Panel
2. Customize User Commands
3. Pull down “All Commands dropdown” and Select Macros
4. Select DisplayLW_OnOff and check Text
5. Click DisplayLW_OnOff – works like a toggle switch
6. If you have multiple views you would need to just adjust the code to change




Autodesk Inventor Professional 2020 DisplayLineWeightMacros.idw
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Bend
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Text
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Surface
Welding
Caterp...
Symbols
Retrieve Model Annotations
Retrieve
Start Sketch
Sketch
Parts List
Hole
Revision
General
Table
Balloon
Edit Layers
Layer
Style
Format
DisplayLW_OnOff
User Commands
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Logic
DisplayLineWeightMacros.idw
Drawing Resources
Sheet:1
Default Border
Customize
Ribbon
Keyboard
Marking Menu
Choose commands from:
Macros
Name
<Separator>
Choose tab to add custom panel to:
Drawing | Annotate
Large
Text
Name
DisplayLW_OnOff
Import...
Export...
Close
Cancel
Apply